Paper ID: 431

Numerical Analysis of Heat Transfer Enhancement of Shell and Helical Tube Heat Exchanger

MD Eshrak Jamil[1*], Md Iftekhab Uddin[1], Prasanjit Das[1]
[1]Chittagong University of Engineering and Technology, Raozan, Chattogram, Bangladesh-4349
E-mail: ejamil926@gmail.com

Abstract

In many industrial operations, from electricity generation to air conditioning, heat exchangers are one of the most essential devices to exchange between fluids. This research explores a comprehensive study on the design optimization of a shell and helical tube heat exchanger using Computational Fluid Dynamics (CFD) simulations with both a viscous k-epsilon realizable model and a laminar model. The primary objective is to enhance the heat exchanger's efficiency and performance by systematically analyzing various geometric parameters such as coil diameter, pitch, shell length, shell diameter and operational parameters such as mass flow rate, heat flux, temperature of hot and cold inlet and outlet. The investigation aims to provide valuable insights into the fluid flow and heat transfer characteristics such as heat transfer rate and overall heat transfer coefficient within the heat exchanger, enabling engineers to make informed design decisions for improved thermal performance.

Keywords: FSI, CFD, Shell and Helical Coil HX, Parametric Analysis.

1. Introduction

Heat exchangers are essential components in a wide range of industrial applications, including chemical processing, power generation, food processing, HVAC systems, and refrigeration. They facilitate the transfer of heat between two or more fluids, which is critical for maintaining thermal balance in systems. Heat transfer in these devices occurs primarily through conduction and convection, with radiation generally being negligible. Optimizing heat exchangers for efficient thermal performance is crucial, given their widespread use and energy-intensive nature.

Shell and tube heat exchangers are one of the most popular designs due to their simplicity, durability, and versatility across a range of temperatures and pressures. In recent years, the shell and helical tube heat exchanger has gained attention for its compact design and enhanced heat transfer capabilities. The helical coil configuration, compared to conventional straight tube designs, induces secondary flows that increase turbulence, leading to better convective heat transfer. This feature makes the helical design an excellent choice for applications where high heat transfer efficiency is required in limited space.

Numerous studies have focused on optimizing heat exchanger designs to maximize performance. Kewin Titus et al. (2020) conducted a comparative study on different shell configurations in helical coil heat exchangers, demonstrating the effectiveness of copper helical coils for enhanced heat transfer. Sivalakshmi S. et al. (2020) explored the use of helically looped fins to further enhance heat exchanger performance, showing that finned designs improve the heat transfer coefficient significantly. Similarly, Ali Najm Abed (2017) emphasized the role of geometric modifications, such as helical coils, in improving thermal efficiency in industrial applications.

Moreover, the use of nanofluids has emerged as a promising method to improve the thermal performance of heat exchangers. Mohammed Zaboli et al. (2019) analyzed the effects of incorporating nanoparticles like Al_2O_3 and CuO into the working fluid, finding that nanofluids can significantly enhance the thermal conductivity of the fluid, thus improving the overall heat transfer efficiency. However, the use of nanofluids is still an area requiring further investigation, particularly in terms of long-term performance and stability in practical applications.

The application of Computational Fluid Dynamics (CFD) has transformed heat exchanger research, allowing for detailed numerical simulations of fluid flow and heat transfer without the need for expensive experimental setups. Studies such as those by Seyed Faramarz Ranjbar et al. (2018) and J.S. Jayakumar et al. (2018) have employed CFD to explore the effects of geometric parameters like coil pitch, tube diameter, and coil radius on the performance of helical heat exchangers. These studies have shown that CFD is a powerful tool for understanding complex heat transfer phenomena and optimizing heat exchanger designs.

Building upon this body of knowledge, the current study focuses on the CFD-based design optimization of a shell and helical tube heat exchanger. The objective is to investigate the impact of various geometric parameters,

including helix pitch, tube diameter, and shell configuration, on heat transfer performance. By simulating different configurations, the study aims to optimize the heat exchanger design for maximum thermal efficiency while considering economic and environmental factors. The findings will provide valuable insights into improving the performance of heat exchangers for a variety of industrial applications, contributing to more sustainable and energy-efficient designs.

In summary, this research addresses a critical need for optimized heat exchanger designs by leveraging advanced CFD techniques to investigate key design parameters. The insights gained from this study will help enhance the efficiency of shell and helical tube heat exchangers, making them more effective in real-world applications such as chemical processing, power generation, and refrigeration systems.

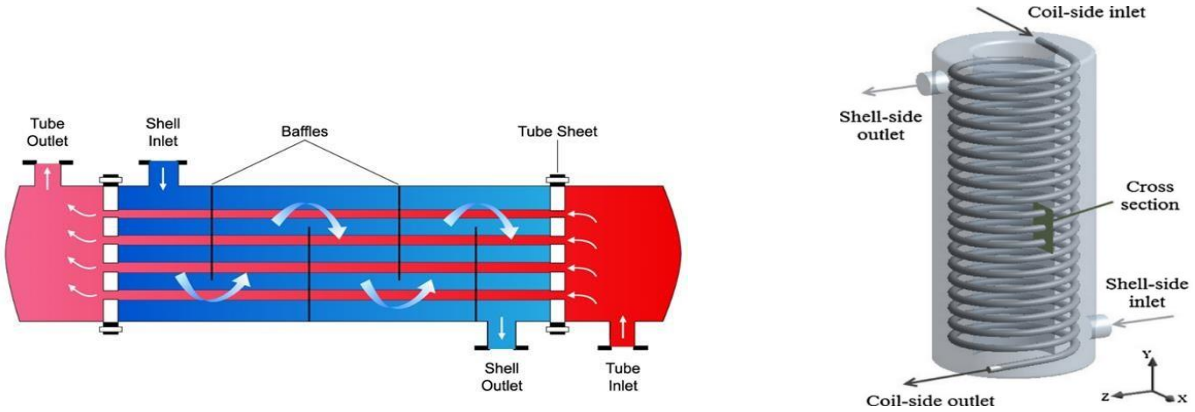


Fig. 1. Shell and tube heat exchanger

2. Methodology

2.1 Governing Equations

The simulation of heat transfer and fluid flow in a shell and helical tube heat exchanger is based on the following governing equations:

Continuity Equation (Mass Conservation):

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\vec{\vartheta}\right) = 0 \tag{1}$$

Momentum Conservation (Navier-Stokes Equations):

$$\rho \frac{\partial \vec{\vartheta}}{\partial \mathrm{t}} + \rho(\vec{\vartheta}.\nabla)\vec{\vartheta} = -\nabla \mathrm{p} + \mu\nabla^2\vec{\vartheta} + \vec{F} \tag{2}$$

Energy Equation:

$$\frac{\partial}{\partial \mathrm{t}}(\rho \mathrm{E}) + \nabla.(\vec{\vartheta}(\rho \mathrm{E} + \mathrm{p})) = \nabla.\left(\mathrm{k}\nabla \mathrm{T} + \tau.\vec{\vartheta}\right) \tag{3}$$

Where E is the total energy, k is the thermal conductivity, and T is the temperature. This equation accounts for heat conduction, convection, and energy transport due to fluid motion.

To evaluate heat exchanger performance, the overall heat transfer coefficient is calculated using the logarithmic mean temperature difference (LMTD) method:

$$\Delta T_{LMTD} = \frac{\Delta T_2 - \Delta T_1}{ln(\frac{\Delta T_2}{\Delta T_1})} \tag{4}$$

Where ΔT_2 and ΔT_1 are the temperature differences between hot and cold fluids at the inlet and outlet.

The total heat transfer rate is given by:

$$Q = m_s(T_{hot\ in} - T_{hot\ out}) \tag{5}$$

The overall heat transfer coefficient U0 is then calculated as:

$$U_0 = \frac{Q}{A_0 \Delta T_{LMTD}} \tag{6}$$

2.2 Geometry Design

The geometry of the shell and helical tube heat exchanger is modeled using ANSYS Design Modeler, beginning with the creation of the outer shell as a cylindrical structure representing the external casing. The helical tube path inside the shell is generated using the "Helix" feature, with parameters such as pitch, diameter, and number of coils defined to match design specifications. The tube's cross-sectional profile, typically circular, is then swept

along the helical path to create the 3D tube geometry. Inlet and outlet ports for the hot and cold fluids are added to simulate the entry and exit points of the fluids, and finally, the outer shell and helical tubes are merged using Boolean operations to form a single, continuous model ready for meshing and CFD simulation.

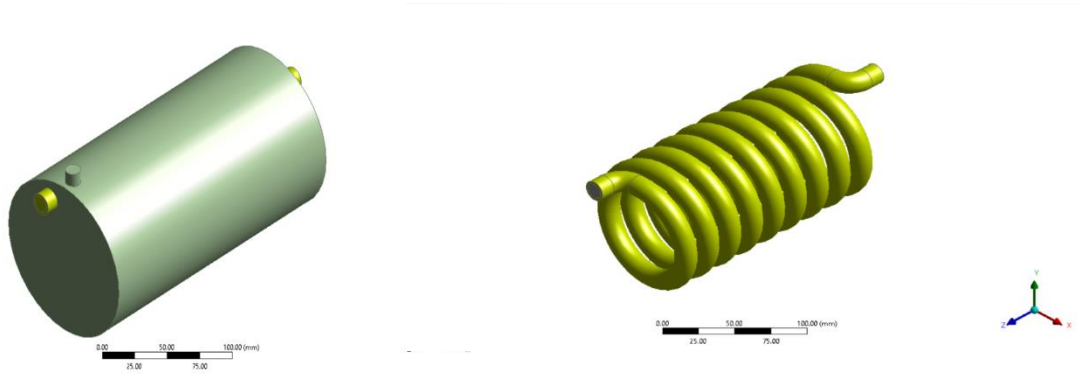


Fig. 2. Geometric model of shell and helical tube heat exchanger

2.3 Boundary Condition

The boundary conditions for the shell and helical tube heat exchanger are established to accurately simulate the fluid flow and heat transfer within the system. The cold fluid enters through the inlet port at a temperature of 10°C with a mass flow rate of 0.19 kg/s, while the hot fluid enters through its inlet port at a temperature of 90°C and a mass flow rate of 0.169 kg/s. The outlet ports are set to allow for the exit of the fluids at their respective exit temperatures, which will be calculated based on the heat exchange occurring within the exchanger. Additionally, no-slip boundary conditions are applied to the tube walls to account for the viscous effects of the fluids, ensuring that the fluid velocity is zero at the solid-fluid interface, while the heat transfer between the fluid and the tube walls is modeled using convective heat transfer coefficients.

2.4 Numerical Model

The numerical setup for the CFD analysis of the shell and helical tube heat exchanger involves several key components to ensure accurate and efficient simulation. Initially, the geometry is imported into ANSYS Fluent after being meshed in ANSYS Meshing, where conformal meshing is utilized to improve accuracy and reduce computational time. A structured mesh with an appropriate element size of 0.215 m is created, resulting in 213,774 nodes and 865,747 elements. The numerical model is based on the governing equations of fluid flow and heat transfer, which include the Navier-Stokes equations for momentum conservation and the energy equation for thermal energy conservation. The simulation is performed under steady-state conditions, and the SIMPLE algorithm is employed for pressure-velocity coupling. Turbulence is modeled using the k-epsilon model, which allows for the simulation of turbulent flow characteristics. Time step controls and convergence criteria are established to ensure that the solution converges to a steady state, allowing for accurate extraction of heat transfer rates and temperature distributions within the heat exchanger. The results are validated against existing literature and experimental data to confirm the reliability of the numerical setup.

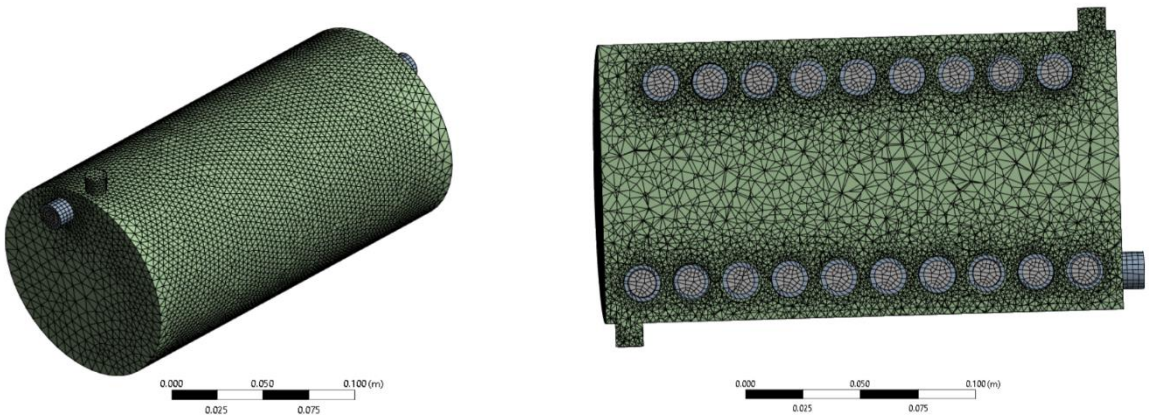


Fig. 3. Combined model and cross-sectional mesh view

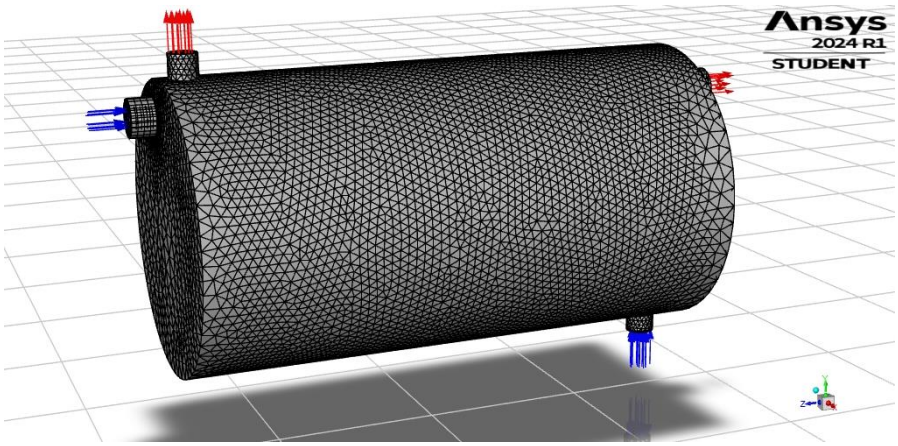


Fig. 4. Boundary condition setup view

3. Results and Discussion

3.1 Velocity Vectors and Streamline Contours

In numerical analysis, velocity vectors represent the direction and magnitude of fluid flow within the system. For shell and helical coil HX, the velocity vectors show how the fluid moves through the shell and coils which helps in understanding the distribution and behavior of the flow for efficient heat transfer.

In figure 5, we can see that hot fluid enters into the hot fluid domain helical pipe with higher temperature and leaves the pipe at a lower temperature. On the contrary, the velocity vectors also represent the cold fluid entering into the cold fluid domain shell side with a lower temperature and leaving the shell tube with a much higher temperature.

Streamlines shown in figure 6 provide significant insights into the behavior of fluid particles as they move through the heat exchanger. The velocity distribution and streamline path were shown in the yz plane of the geometry.

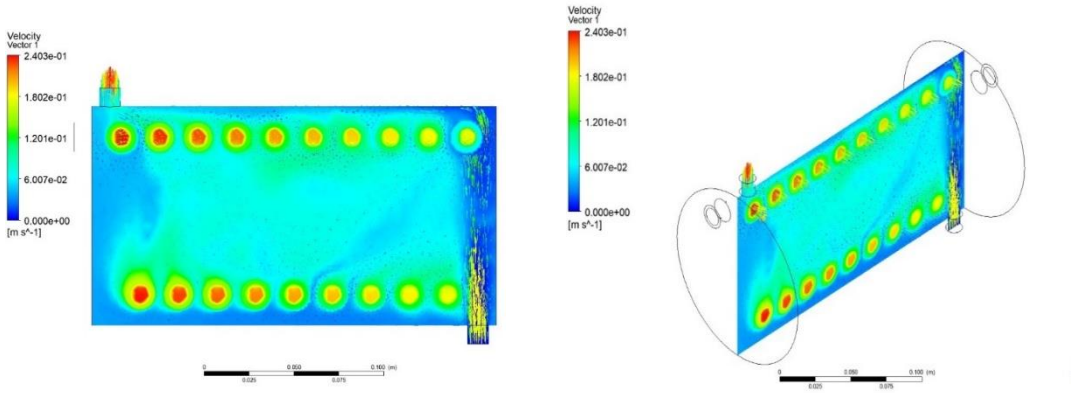


Fig. 5. Velocity Vector through YZ Plane

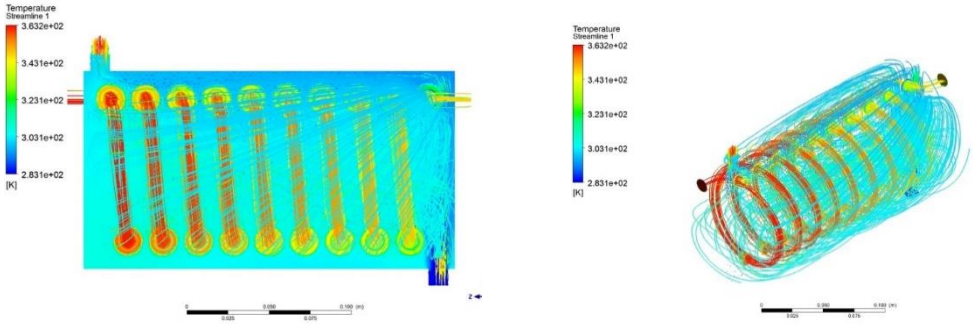


Fig. 6. Streamline Contours of the heat exchanger

3.2 Temperature Distribution and Pressure Contours

Temperature contours depict the distribution of temperature throughout the system. In the shell side, the contours typically show how the temperature changes as the fluid moves through the shell, while in the tube side they illustrate temperature changes along the tube length. For a helical tube, the contours show how the temperature and pressure varies within the helical coil.

From the boundary condition, the inlet temperature of cold fluid domain i.e. shell side inlet was taken as 10^0C or 283K. From the contour, we can see the temperature is increased in shell side outlet to maximum of 300.6K and minimum of 296.86K due to heat exchange throughout the system. In hot fluid domain i.e. helical coil tube the fluid enters at 90oC or 363K and leaves the system at minimum of 334.84K and maximum of 346.69K. This indicates that the cold fluid obtains a temperature increase of maximum temperature of 23.6K and hot fluid through the helical pipe drops the temperature of maximum 29.7K. The difference of temperature gain and drop occurred due to the heat exchange between the two fluid domains and inner helical pipe.

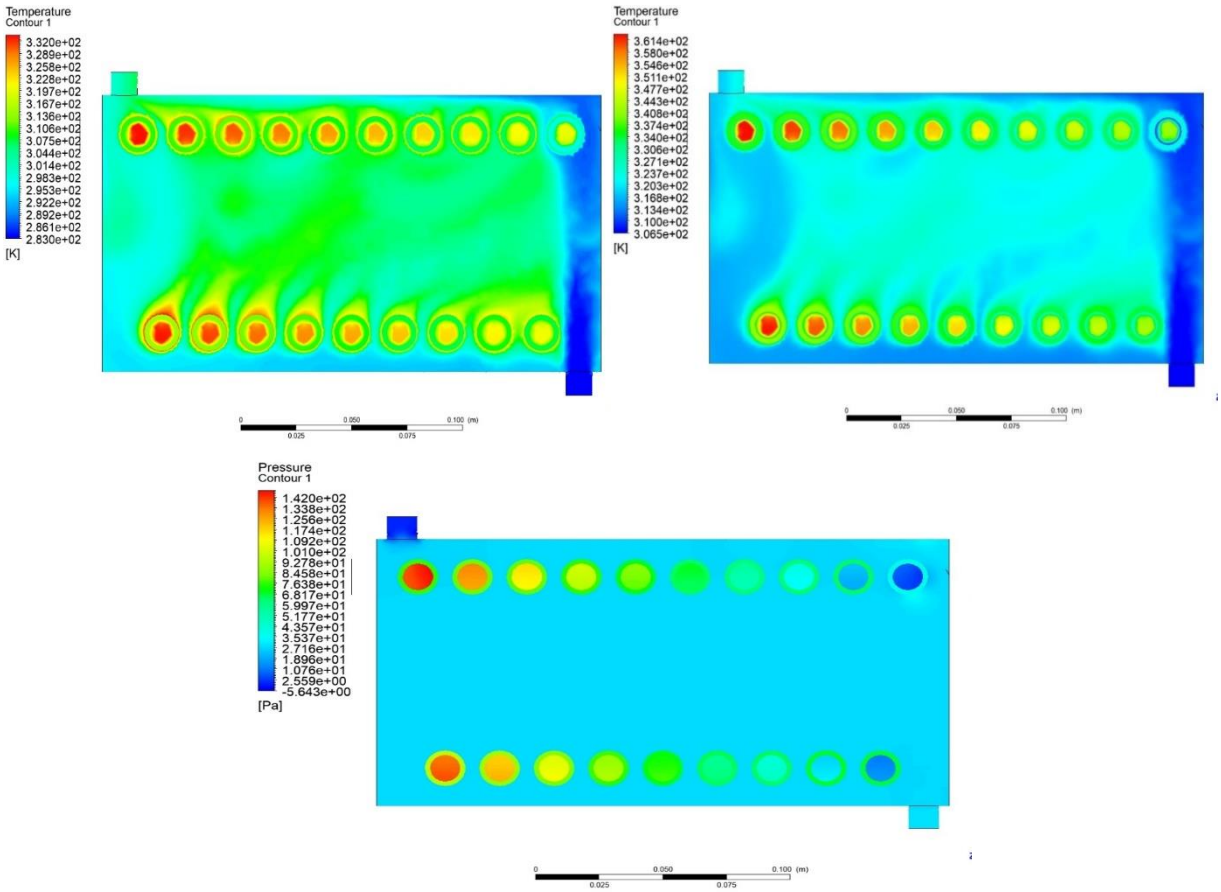


Fig. 7. (a) Temperature contour of cold fluid domain in YZ plane (b) Temperature contour of hot fluid domain in YZ plane (c) Pressure distribution contours

3.3 Parametric Analysis of The Heat Exchanger

Parametric analysis of the shell and helical tube heat exchanger involves varying parameters like flow rates, temperatures in both cold and hot fluid, coil diameter, shell diameter, shell length, pitch length, inner helical pipe diameter etc. to identify the variation of output parameters and results. The analysis can put emphasis on the overall performance and efficiency of the shell and helical tube heat exchanger. Here I have done some parametrical analysis on the heat transfer analysis of shell and helical coil tube heat exchanger to validate the influence of parameters such as shell diameter, coil diameter, shell length, pitch length etc. on the output results. The variation of output parameters such as heat transfer rate and overall heat transfer coefficient is shown on the analytical chart is given below. I studied the heat transfer analysis with several geometric parameters to inquire how the geometric parameters make influence on the output results. The analysis of the different design parameter is given below.

3.3.1 Effect of shell diameter in heat transfer

The diameter of shell side of the heat exchanger plays a vital role in the overall performance of the heat exchanger. Here we can see the output parameters, heat transfer rate and overall heat transfer coefficient is 1220.60 watt & 154.432 $Wm^{-2}k^{-1}$ respectively for the base geometric model as there was no change in parameters. Then we see the gradual decrement of heat transfer rate and overall heat transfer coefficient within the increase of shell diameter as expected. This is due to the increase of shell side area which decreases turbulence. The variation is shown through design point table 1 and graphically represented in figure 8.

Table 1. Parametric analysis of shell diameter

Design Number	Shell diameter (mm)	Q (watt)	U($Wm^{-2}k^{-1}$)
Design 1	115	1250.82	160.569
Design 2	120	1220.16	154.432
Design 3	125	1186.65	143.276
Design 4	130	1138.4	127.334

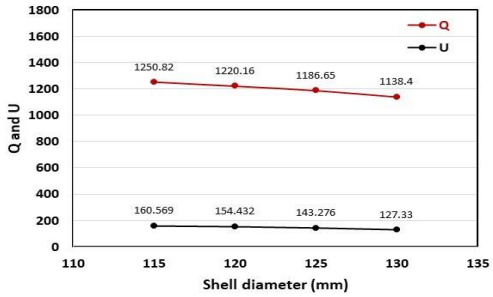


Fig. 8. Heat transfer variation with shell diameter

3.3.2 Effect of pipe diameter in heat transfer

The diameter of the helical pipe plays a vital role in the behavior of heat transfer of a shell and helical tube heat exchanger. As the diameter of inner pipe increases, the heat transfer rate and overall heat transfer coefficient also decrease. The maximum values of heat transfer rate 1288.2 W and overall heat transfer coefficient 164.948 $Wm^{-2}k^{-}1$ are found in maximum inner pipe diameter.

Table 2. Parametric analysis of pipe diameter

Design Number	Pipe diameter (mm)	Q (watt)	U($Wm^{-2}k^{-1}$)
Design 1	10	1288.22	164.948
Design 2	12	1136.65	143.276
Design 3	14	1151.88	143.9
Design 4	15	1108.83	138.097

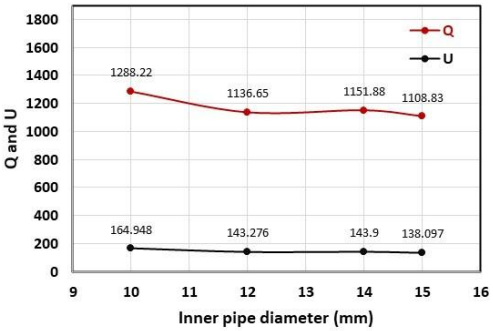


Fig. 9. Heat transfer variation with inner tube diameter

3.3.3 Effect of pitch length

Both heat transfer rate and overall heat transfer coefficient increases with the decremental of pitch length as it increases number of turns in the helical coil which increases the total heat transfer area. We can see the maximum heat transfer rate and overall heat transfer coefficient when the pitch length is minimum.

Table 3. Parametric analysis of pitch length

Design Number	Pitch length (mm)	Q (watt)	U($Wm^{-2}k^{-1}$)
Design 1	18	1278.18	152.637
Design 2	21.4	1136.65	143.276
Design 3	23	1089.54	137.594
Design 4	25	1013.734	132.571

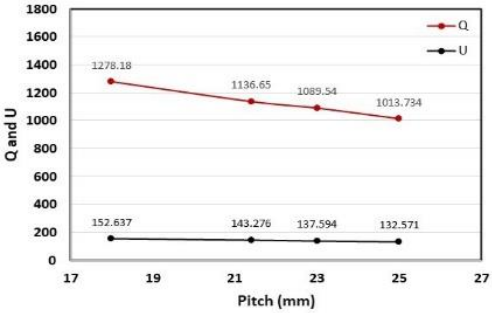


Fig. 10. Heat transfer variation with Pitch

3.3.4 Impact of shell length on heat transfer

From the plots we can see that heat transfer depends upon shell length, whereas we can see that overall performance and heat transfer coefficient decreases within the incremental of shell length firstly but it then started

increasing gradually within shell length. This is due to enlargement of shell side with respect to a smaller helical pipe that reduces overall heat transfer coefficient. The maximum heat transfer rate and overall heat transfer coefficient were in maximum shell length. The variation is shown in table 4 graphically represented at figure 11.

Table 4. Parametric analysis of shell length

Design Number	Shell length (mm)	Q (watt)	U($Wm^{-2}k^{-1}$)
Design 1	110	1169.93	145.931
Design 2	112.5	1136.65	143.276
Design 3	115	1172.96	147.536
Design 4	116	1189.62	149.806

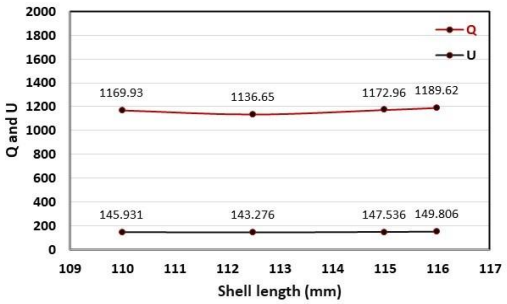


Fig. 11. Heat transfer variation with shell length

3.3.5 Effect of coil diameter on heat transfer

Larger coil diameters generally increase the overall heat transfer area for conduction which tends to force the heat transfer enhancement. On the other hand secondary flows promote the mixing of the fluid, increasing the heat transfer rate and overall heat transfer coefficient shown in table 5 and figure 12.

Hence from the analysis of the numerical experiment and following various graphical representations given above, we can assure that the performance of the heat transfer of the shell and helical coil heat exchanger can be maximized by decreasing shell diameter, increasing inner pipe diameter, decreasing shell length and decreasing coil diameter.

Table 5. Parametric analysis of coil diameter

Design Number	Coil diameter (mm)	Q (watt)	U($Wm^{-2}k^{-1}$)
Design 1	70	797.215	128.51
Design 2	76	947.594	134.665
Design 3	86	1136.65	143.276
Design 4	92	1254.77	159.158

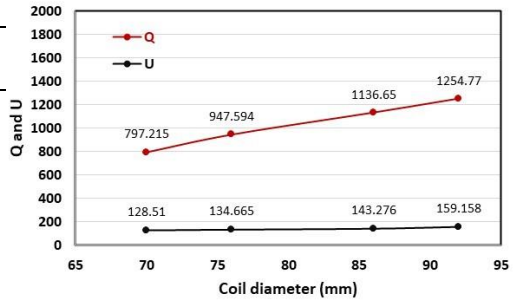


Fig. 12. Heat transfer variation with coil diameter

3.4 Comparison of K-Epsilon with Laminar Model

The computational fluid dynamics simulation was done in viscous k-epsilon model and then the simulation was again run in laminar model to compare the output results and computed values. Viscous k-epsilon realizable model solves two additional equations for turbulent kinetic energy (k) and its dissipation (epsilon) which are both zero in purely laminar model. That's why the computational results have some minor difference. The output result and computed values are compared in the following table.

Table 6. Viscous k-epsilon vs Laminar model

Output results and parameters	Viscous k-epsilon model	Laminar model
Total heat transfer rate (Q)	1136.65 watt	1560 watt
Overall heat transfer coefficient (U)	143.276 $Wm^{-2}k^{-1}$	213.55 $Wm^{-2}k^{-1}$
Cold outlet temperature (max)	300.691K	306.6 K
Cold outlet temperature (min)	296.88K	298.88 K
Hot outlet temperature (max)	346.69K	341.4 K
Hot outlet temperature (min)	334.836K	322.7 K
Surface Nusselt number (Nu) (area weighted average)	6.518276	2.084014
Surface Nusselt number (Nu) (integral)	3.02535	1.1374
Cell Reynolds number (Re)	15.33233	47.8245
Molecular Prandtl number (pr)	3.497592	6.99091

4. Conclusion

In this study, using computational fluid dynamics, analytical research has been done for shell and helically coiled tube heat exchanger. From literature, other researcher's works and studies influenced me a lot to understand the behavior of shell and helical coil tube heat exchanger. The primary objectives of my research were to investigate the heat transfer characteristics, to understand the impact of various parameters on performance. Characteristics of the fluid flow were also studied for constant temperature and constant wall heat flux conditions. The helical coil design demonstrated a higher heat transfer coefficient compared to conventional shell and tube heat exchanger. This can be attributed to the increased surface area and improved fluid dynamic within the helical coil. Different characteristic properties such as surface Nusselt number, molecular Prandtl number, cell Reynolds number were computed through fluent. Total heat transfer rate and overall heat transfer coefficient throughout the heat exchanger were calculated using their governing expressions.

Parameters such as coil diameter, pitch, shell diameter etc. made an impact on the overall performance of heat transfer rate and overall heat transfer coefficient. Optimization of these parameters led to enhanced heat transfer rate and overall heat transfer coefficient.

The research was limited by certain assumptions in the numerical model and the scope of experimental conditions. The fluid flow was assumed as laminar flow throughout the system. In future studies, authors can explore research by other fluid flow systems. Future research could focus on different materials, varying fluid properties and long-term operation stability. Additionally, adding fins and baffles in the tube surface can increase potential heat transfer area for increasing required heat transfer rate.

11. References

[1] Note on the instability of India-rubber tubes and balloons when distended by fluid pressure. (1891, December 31). Proceedings of the Royal Society of London, 49(296–301), 458–463.

[2] Kyriakides, S., & Yu-Chung, C. (1990). On the inflation of a long elastic tube in the presence of axial load. International Journal of Solids and Structures, 26(9–10), 975–991.

[3] Taylor, T. W., & Yamaguchi, T. (1994, February 1). Three-Dimensional Simulation of Blood Flow in an Abdominal Aortic Aneurysm—Steady and Unsteady Flow Cases. Journal of Biomechanical Engineering, 116(1), 89–97.

[4] Di Martino, E., Guadagni, G., Fumero, A., Ballerini, G., Spirito, R., Biglioli, P., & Redaelli, A. (2001, November). Fluid–structure interaction within realistic three-dimensional models of the aneurysmatic aorta as a guidance to assess the risk of rupture of the aneurysm. Medical Engineering & Physics, 23(9), 647–655.

[5] Józsa, T. I., & Paál, G. (2014, December). Boundary conditions for flow simulations of abdominal aortic aneurysms. International Journal of Heat and Fluid Flow, 50, 342–351.

[6] Fillinger, M. F., Marra, S. P., Raghavan, M., & Kennedy, F. E. (2003, April). Prediction of rupture risk in abdominal aortic aneurysm during observation: Wall stress versus diameter.

[7] Alhayani, A., Giraldo, J., Rodríguez, J., & Merodio, J. (2013, October). Computational modelling of bulging of inflated cylindrical shells applicable to aneurysm formation and propagation in arterial wall tissue. Finite Elements in Analysis and Design, 73, 20–29.

[8] Souche, A., & Valen-Sendstad, K. (2022, December). High-fidelity fluid structure interaction simulations of turbulent-like aneurysm flows reveals high-frequency narrowband wall vibrations: A stimulus of mechanobiological relevance? Journal of Biomechanics, 145, 111369.