International Journal of Machine Tools and Maintenance Engineering

E-ISSN: 2707-4552
P-ISSN: 2707-4544
IJMTME 2023; 4(1): 28-38
Received: 28-09-2023
Accepted: 30-10-2023

Hussein Hayder Mohammed Ali
Northern Technical University /Technical College of Engineering, Kirkuk, Iraq

Corresponding Author:
Hussein Hayder Mohammed Ali
Northern Technical University /Technical College of Engineering, Kirkuk, Iraq

Theoretical study to evaluate the performance of a double-tube heat exchanger using an inner convoluted tube and nanofluid (CuO)

Hussein Hayder Mohammed Ali

Abstract
Heat exchangers are mechanical devices that enable the transfer of thermal energy between two fluids that possess dissimilar temperatures while simultaneously preventing any intermingling between the fluids. Researchers in this study test how well a double-tube heat exchanger works by using both straight and convoluted tubes and a nanofluid that has CuO nanoparticles in it along with water. The inner tube under consideration is a continuous and convoluted object with a length of 1000 millimeters. It exhibits a smooth surface while also possessing a convoluted configuration. This convoluting is characterized by a total of 59 rotations, with each rotation having a height of 5 millimeters and being separated from the adjacent rotation by a distance of 0.8 millimeters. The physical model of the test part of the double tube heat exchanger utilizes two horizontal concentric cylinders to provide a concentric annular gap. The working fluid utilized in the cold and hot field reverse flow is a combination of water and nanofluid. The concept of the double tube heat exchanger's 3D model is designed, followed by a simulation using ANSYS FLUENT 2023 R1. Subsequently, the mesh is created. This study necessitates a total of 350 iterations. The findings are displayed for a variety of Reynolds numbers spanning from 6000 to 11000, specifically focusing on the convoluted tube's burr number of 95. The parameters examined and compared include pressure drop, friction coefficient, cold exit temperature, and average Nusselt number.

Keywords: Theoretical study, double-tube heat exchanger, convoluted tube, nano fluid

Introduction
Numerical methods have become the norm for dealing with issues of varying complexity in many fields of application. Computational fluid dynamics (CFD) use these techniques to model fluid flow in real-world systems by resolving single-phase equations. Inequalities of Navier-Stokes [1-4]. Heat exchangers have numerous uses in industry and engineering. By using heat transfer augmentation techniques, heat exchanger capacity can be greatly improved, resulting in a smaller and cheaper heat exchanger. One can classify enhancement methods as passive or active. The additional energy that is required to improve heat transfer is drawn from sources already present in the methodology, like expanded or treated surfaces, making passive techniques like these completely power independent. Active methods, such mechanical assistance and surface vibration that require an external power source have been determined to be [5]. Spiral fluted tubes are among the most useful improved tubes for an array of technical purposes, such as heat exchangers. Sang Chun Lee *et al.* [6] present the findings of an exploratory research into the heat transmission and efficiency of pressure drop (ΔP) of a tube with a spiral indentation. At Res between 500 and 5000, working fluids have included ethylene-glycol-water mixtures and pure water. The results indicated an increase in heat transfer coefficient of more than eight times for the spirally indented tube compared to the straight tube. Pethkool *et al.* [9] conducted a field experiment of the helically corrugated tube's heat transfer efficiency, friction coefficient (f), and thermal performance. It was investigated what might occur to the tubes' properties if the pitch-to-diameter ratios (P/DH = 0.18) and the rib-height-to-diameter ratios (e/DH = 0.02, 0.04, and 0.06) were modified. In this study, a range of 5500 to 60,000 Re was used in water. According to the data, the Nusselt number, friction factor, and thermal efficiency factor all improve as the pitch ratio (P/DH) and the rib-height ratio (e/DH) are raised. The modified tube achieved a thermal performance factor of 2.33 with a pitch-to-diameter (P/DH) ratio of 0.27 and a rib-height-to-diameter (e/DH) ratio of 0.06. Hassanzadeh, S., and S. Hossainpour [10] innovative 3D numerical method study of the friction factor and heat transmission coefficient in helical

corrugated tubes. All tubes were 500 mm long, 24 mm in diameter on the interior, and had a 0.02, 0.04, or 0.06 rib height to diameter ratio, as well as a 0.6, 0.8, or 1.2 rib pitch to diameter ratio. With a Re between 250,000 and 800,000, water served as the working fluid. According to the numerical findings, raising the rib height and decreasing the rib pitch improvements in heat transmission and minimized friction. Recently, nanofluids have been utilized to enhance heat transfer. Farajollahi *et al.* [11] examined experimentally in a straight tube was the heat transfer behavior of a-CuO/water and TiO_2/water nanofluids. The nanoparticle volume concentrations (c') of - CuO/Water and TiO_2/Water nanofluids are, respectively, 0.3-2% and 0.15-0.75%. Results showed that 0.5% vol. a- CuO/Water and 0.3% vol. TiO_2/Water nanofluids outperformed plain water by a factor of convolutedly 56% in terms of heat transfer coefficient. Rabienataj Darzi *et al.* [12] conducted a numerical investigation of the flow of a nanofluid consisting of CuO and water through a helically-corrugated tube. The Re ranged from 10,000 to 40,000, and the corrugating pitch was 16, 16, and 25 mm; the height was 1, 0.67, and 1 mm. The greatest rise in Nusselt number was 58% for a concentration of 4% of nanoparticles by volume, and 21% for a concentration of 2%. There is little evidence in the literature to suggest that nanofluids flowing inside a spiral fluted tube would significantly improve a heat exchanger's performance. Furthermore, numerical research on the current heat exchanger is lacking. As a result, this study aimed to compare improved heat exchange efficiency as a result of a curved twisted tube design with and without nanofluid through the use of experimental and numerical methods. The work continues with illustration of the convective heat transfer equations as well as boundary conditions for a thermal transfer annulus. The thermophysical characteristics and other dimensionless parameters of water are taken into account. This section also includes a discussion on mesh generation. At the interface between the fluid's surface and the container, unique solutions are introduced that satisfy the boundary conditions, when compared to experimental research, CFD simulation has the advantages of being cheap, taking a reasonable amount of time, solving multiple questions at once, and having a high level of protection. While CFD allows engineers to tackle a broad variety of problems involving heat and mass flow with numerical solutions, fluency is used to analyze the impact of utilizing a convoluted inner tube in the annular gap of a twin-tube heat exchanger. Conditions (turbulent), flow structure (three volumetric flows), and methods (forced convection) are discussed. We turned to CFD computers to help them solve one of the most challenging problems people face.

A description concerning how numerical simulations run
Using the program "SOLIDWORKS", the 3D geometry of the investigated two-pipe heat exchanger was built. Hexahedrons and triangular cells were then carefully incorporated into the model geometry to partition the domain into subdomains, cells, or elements [13]. The current study performed a numerical evaluation of a twin tube heat exchanger using a CFD code (ANASYS Workbench-2023 (R1)). Different physical settings can be configured in FLUENT software, including incompressible, compressible, non-viscous, viscous, laminar, and turbulent [14]. Figure (1) is a geometric depiction of the typical procedure for CFD simulation work. There are essentially three actions that occur in the numerical code:

1. Pretreatment: This stage includes linking and boundary requirements description.
2. At this level, analysing, the governing equations are integrated and calculated.
3. In the final step, known as "post-processing," results and analysis are made available to those in attendance.

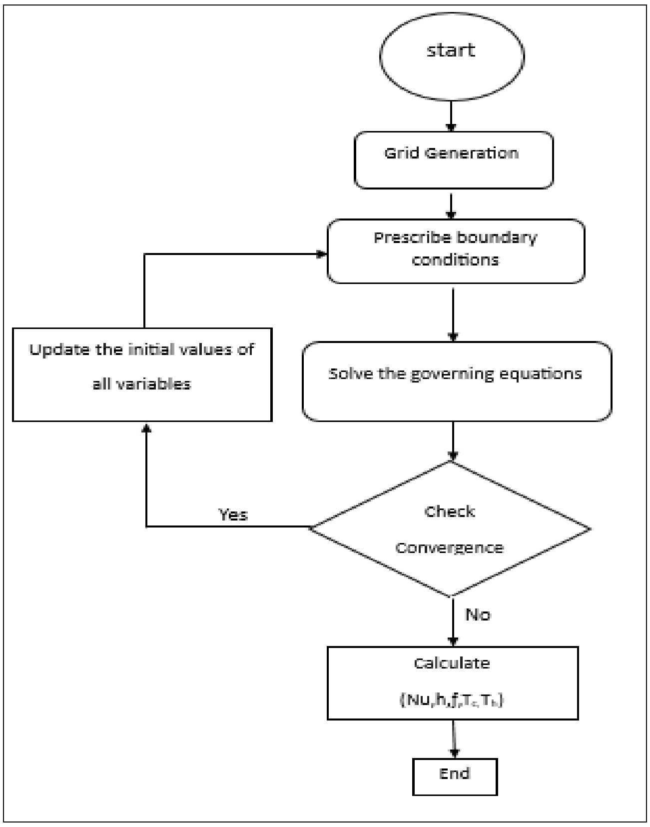
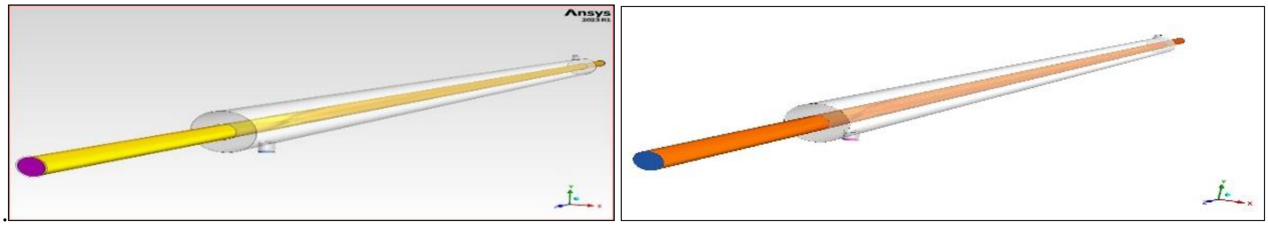


Fig 1: Numerical Simulation Flow Chart.

Description of the physical model and the issue
Double-pipe heat exchangers with counter-flow are the conventional design, as seen in Figure (2). By using a convoluted inner tube. Each of the concentric tubes began with water as the base fluid, which was then replaced with nanofluid (CuO) at that point, hot water flows in the annular space, and cold fluid flows through the inner tube. The fluids on either side of the wall exchange heat as they move past one another. A straight and convoluted inner tube measuring 1000 millimeters in length has 59 twists, each 5 millimeters high, and 0.8 millimeters apart. Two horizontal concentric cylinders are employed to create a concentric annular gap in the physical model of the test section of the double tube heat exchanger. The cold and hot field reverse flow working fluid is water and nanofluid. The double tube is designed with the outer diameter of the inner tube (copper), the length (1000 mm) and the thickness (0.8 mm). The ratio between the outer diameter of the inner tube (copper) and the outer diameter of the tube (plastic) is (D_{in} /d_{out}) = 0.211 the thickness is neglected. The outer tube to reduce losses and assumes that the thermophysical properties depend on temperature. Fig. 2 is an illustration of the computational domain for this study.



(a) Straight inner tube. (b) Convoluted inner tube.

Fig 2: This investigation's physical geometry (a) straight (b) convoluted.

Theoretical Analysis

As can be seen in Figure (3), the cold fluid (water) then replaced with CuO nanofluid with (0.1) % concentration flowing in the tube side of the current study imbibed heat from the hot water flowing at the side of the shell. The current heat exchanger analysis has been performed on the assumptions of a steady state condition, an insulated heat exchange outer surface, and no phase changer. The rate of thermal transfer through the current heat exchanger can be computed under these conditions by [15]:

$$Q_{\text{con.}} = \dot{m} c_p (T_{\text{out}} - T_{\text{in}}) \text{ and } Q_{wall} = \bar{h} A_s (T_s - T_b) \quad (1)$$

This study calculates the properties of the sample fluid at the average $T_{\text{exit, cold}}$. It can be determined using the following formula:

$$T_m = \frac{T_{\text{in}} + T_{\text{out}}}{2} \quad (2)$$

The fluid's The mathematical equation to calculate the heat transfer coefficient is as follows:

$$h = \frac{\overline{m} c_p (T_{\text{out}} - T_{in})}{A_s (T_S - T_b)} \quad (3)$$

Where

$$A_s = \pi D_i L \quad (4)$$

Due to the complicated conical tube with flutes on the inside, the concept of "volume-based diameters" is employed in place of conventional diameters. It is determined as [16]:

$$D_{vi} = \sqrt{\frac{4Vol}{\pi L}} \quad (5)$$

The interior Nusselt number can then be calculated as [17]:

$$N_u = \frac{h D_i}{K_f} \quad (6)$$

The (f) is written as [18]:

$$f = 2\Delta P \times \frac{D_h}{\rho_{f_c} u_{m^2} L} \quad (7)$$

The thermal performance factor is indicated by [19]

$$PEC = \frac{\left(\frac{Nu_{nf}}{Nu_f}\right)}{\left(\frac{f_{nf}}{f_f}\right)^{\frac{1}{3}}} \quad (8)$$

Thermo-Physical Properties of Nanofluid

The properties can be obtained as follow:

$$m_p = \frac{\emptyset \times \rho_p \times \left(\frac{m_f}{\rho_f}\right)}{(1-\emptyset)} \quad (9)$$

Nanofluid thermal conductivity can be estimated by:

$$k_{nf} = k_f \left(\frac{k_s + 2k_f - 2\emptyset (k_f - k_s)}{k_s + 2k_f + \emptyset (k_f - k_s)} \right) \quad (10)$$

Using the formula for mixtures, the density of nanofluids can be calculated [15]:

$$\rho_n = (1 - \emptyset)\rho_f + \emptyset \rho_P \quad (11)$$

The nanofluid's specific heat can be calculated using the following mixture rule:

$$CPnf = \frac{(1 - \emptyset)CPbf * \rho bf + \emptyset * \rho p * Cpp}{\rho nf} \quad (12)$$

The nanofluid's dynamic viscosity can be calculated by [18]:

$$\mu_{nf} = \mu_{bf}(1 + 2.5\emptyset) \quad (13)$$

Numerical Simulation

This simulation was performed by utilising the Ansys FLUENT 2023 R1 software suite. The current heat exchanger employs the principles of continuity, momentum, and energy conservation equations to analyse fluid dynamics and heat transfer phenomena. The heat exchanger currently under consideration is designed with the computer-aided design software Inventor 2023, as depicted in Figure 3.

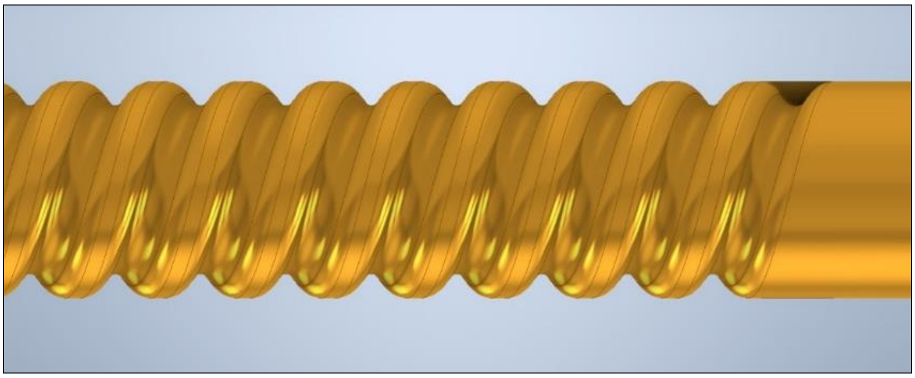


Fig 3: Convoluted inner tube heat exchanger.

Conservation equations and boundary conditions

The present heat exchanger assumes that the flow characteristics exhibit steady-state behaviour, include a Newtonian fluid, occur in 3D, and are characterised by turbulence. The subsequent equation represents the governing principles of continuity, momentum, and energy [15].

Continuity Equation is:

$$\left(\frac{1}{r}\frac{\partial(ru_r)}{\partial_r} + \frac{1}{r}\frac{\partial u_\emptyset}{\partial_\emptyset} + \frac{\partial u_z}{\partial_z}\right) = 0 \tag{14}$$

Momentum equations are
Radial-Momentum

$$\left(u_r\frac{\partial u_r}{\partial_r} + \frac{u_\emptyset}{r}\frac{\partial u_r}{\partial_\emptyset} - \frac{u_\emptyset^2}{r} + u_z\frac{\partial u_r}{\partial_z}\right) = -\frac{\partial p}{\partial r} + \\ +\mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \emptyset^2} + \frac{\partial^2 u_r}{\partial z^2} - \frac{u_r}{r^2} - \frac{2}{r^2}\frac{\partial u_\emptyset}{\partial \emptyset}\right) \tag{15}$$

Tangential-Momentum

$$\rho_{fc}\left(u_r\frac{\partial u_\emptyset}{\partial \partial_r} + \frac{u_\emptyset}{r}\frac{\partial u_\emptyset}{\partial \theta} + u_z\frac{\partial u_\emptyset}{\partial z} + \frac{u_r u_\theta}{r}\right) = -\frac{1}{r}\frac{\partial P}{\partial \theta} + \\ +\mu_{fc}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\emptyset}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_\emptyset}{\partial \emptyset^2} + \frac{\partial^2 u_\emptyset}{\partial z^2} - \frac{u_\emptyset}{r^2} + \frac{2}{r^2}\frac{\partial u_r}{\partial \emptyset}\right) \tag{16}$$

Axial-Momentum

$$\rho_{fc}\left(u_r\frac{\partial u_z}{\partial r} + \frac{u_\emptyset}{r}\frac{\partial u_z}{\partial \emptyset} + u_z\frac{\partial u_z}{\partial z}\right) = -\frac{\partial P}{\partial z} + \\ +\mu_{fc}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial_r}\right) + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \emptyset^2} + \frac{\partial^2 u_z}{\partial z^2}\right) \tag{17}$$

Energy Equation is

$$\left(u_r\frac{\partial T}{\partial r} + \frac{u_\theta}{r}\frac{\partial T}{\partial \emptyset} + u_z\frac{\partial T}{\partial z}\right) = \alpha_{f,c}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 T}{\partial \emptyset^2} + \frac{\partial^2 T}{\partial z^2}\right) \tag{18}$$

Where μ_{f_c} means the dynamic viscosity.

- ρ_{fc} is the density of water.
- P is the pressure.
- u is the velocity-vector.
- T is the working fluid temperature.
- $\alpha_{f,c}$ is the thermal diffusivity.

To solve the governing equations, the employed boundary conditions must be meticulously chosen for the physical domain under study. On the internal pipe and annulus surfaces, the boundary conditions of the components of velocity are assumed to be no-slip conditions (u_r = u = u_z = 0) [19].

As stated previously, the hot fluid passes through the heat exchanger's shell, while the cold fluid flows through the inner tube in the opposite direction. Therefore, at the inlet of the inner tube and the central annulus, a uniform velocity (Um) and boundary conditions (BC) are applied at the inner tube outlet and the casing gap ($P_{\text{out}} = P_{\text{gage}}$). The inbound cold fluid is at a temperature of (20°C), while the incoming hot water is at a temperature of (65°C).

In order to reduce losses, the girth of the outer tube has also been neglected. The specifications of convective heat transfer between the inner tube fluid and jacket fluid of the heat exchanger through the separating wall are presented in Table (1).

Table 1: Double-tube heat exchanger applied boundary conditions

Boundaries	Flow-BC	Heat transfer- BC
Inlet cold	Developing, Re=6000 11000	$T_{c,in} = 20°C$
Inlet hot	Developing, Re=1900-2000	$T_{h,in} = 65°C$
Outlet	Pressure outlet, zero	$P_{\text{out}} = 0$
Outer wall of annulus	No slip condition	Adiabatic wall

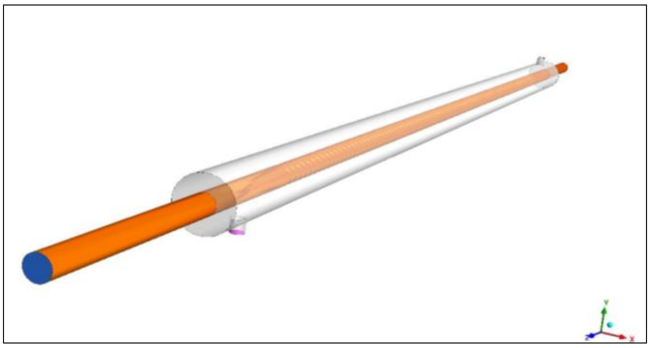


Fig 4: Boundary conditions and physical geometry.

In order to calculate the standard Navier-Stokes and energy equations for the double-pipe heat exchanger, some simplifying assumptions must be made.
The points that follow are the most important presumptions made:

1. Stable, 3D flow modeling.
2. The fluid is a single phase, incompressible, forced turbulent flow system.
3. Properties of the thermophysical fluid remain constant.
4. Negligible radiant heat transmission and no slippage.

Mesh Generation

The caliber of the geometry has a significant impact on the precision of the numerical results and how long does it take to arrive at the end of a numerical simulation. Hence, the assessment of mesh quality and quantity is crucial during the process of mesh generation. A higher mesh density is observed to yield more precise outcomes, albeit at the expense of increased computational time and memory usage. Therefore, to ensure stability and accuracy, the verification of results is conducted using the minimum number of meshes feasible.
The utilization of hybrid grids, which integrate diverse element like triangles and quadrilaterals in 2D besides tetrahedra, hexahedra, prisms, and pyramids in 3D, is a fundamental attribute of proficiency. The selection of the appropriate mesh type to utilise is contingent upon the specific application at hand. In the realm of basic geometries, it has been shown that quadrilateral and hexahedral meshes tend to yield solutions of superior quality with a reduced number of cells when compared to triangular and tetrahedral meshes. Conversely, when dealing with complex geometries, triangular and tetrahedral meshes necessitate a greater amount of effort in the meshing process [15].
This study utilises a tetrahedral mesh type, with the mesh and structure of the existing model being illustrated in pictures (5). Throughout the duration of this study, a substantial quantity of cells has been gathered, and it has been determined that an average of 5M cells is the most favourable number in relation to both the most accurate numerical forecast of the current model and the highest number of iterations executed prior to the solver concluding. The present study necessitates a total of 350 iterations, as depicted in Figure 6.

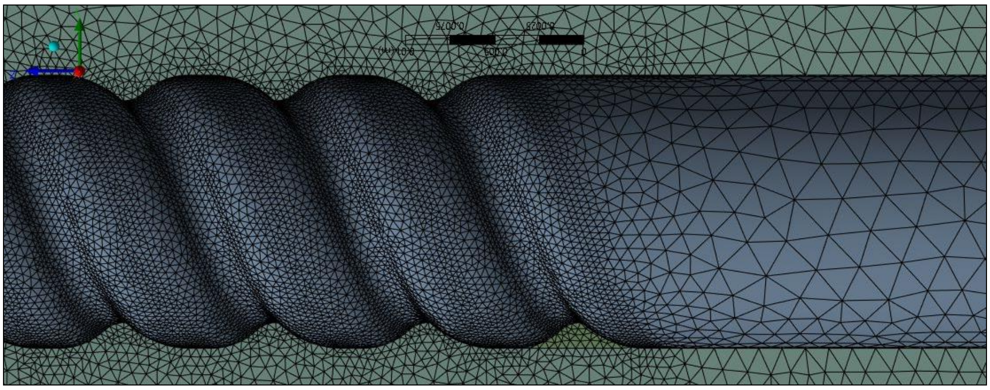


Fig 5: Meshing of the model.

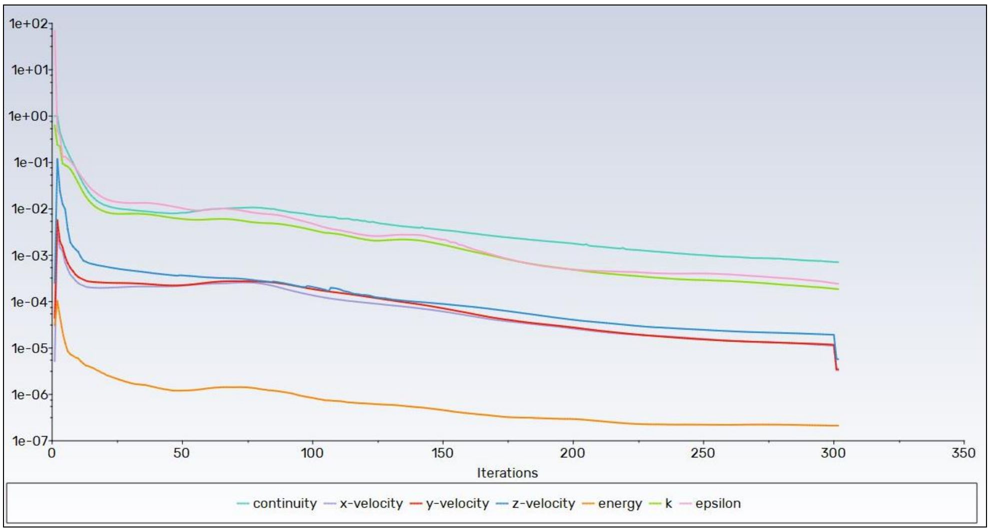
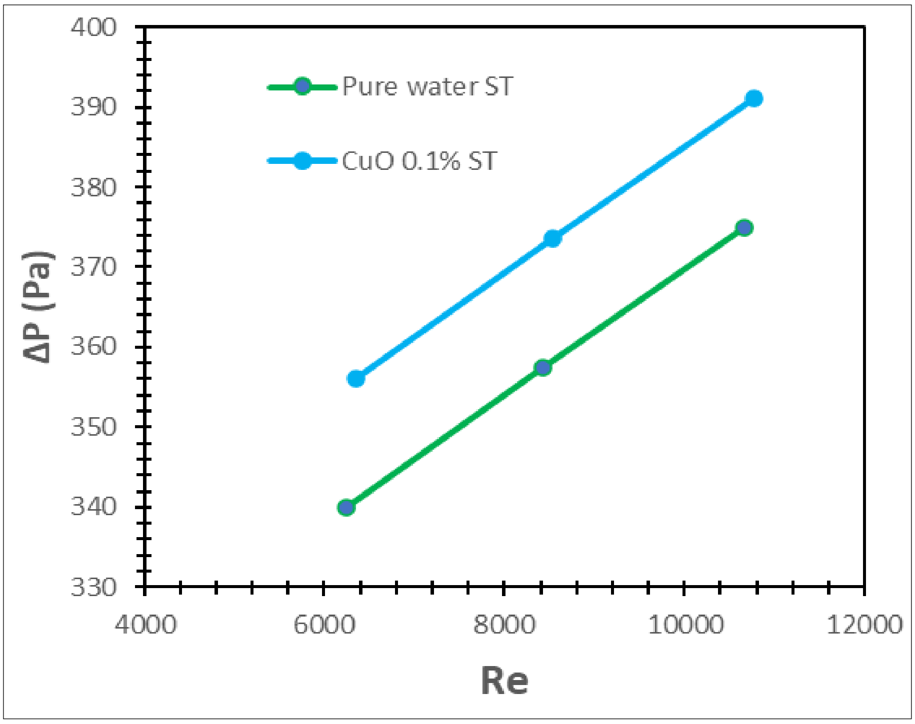


Fig 6: The present study's computer model remains.

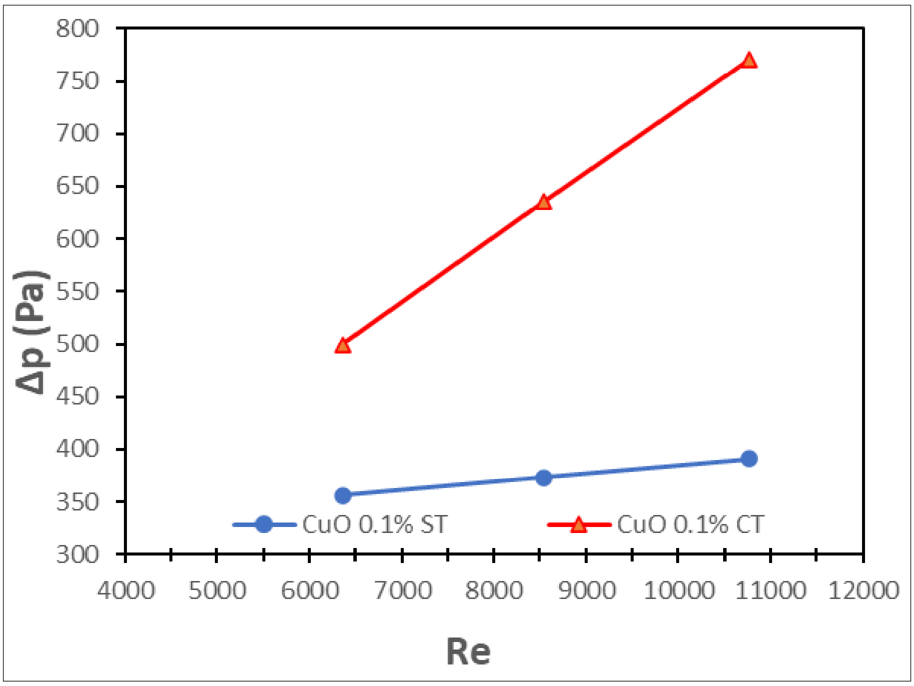
Results and Discussions

A comparison of the present numerical results of the turbulent fluid flow inside a double tube heat exchanger with respect to the convoluted and straight inner tube with a length of 1 meter is discussed. The results are presented for a range of Re from 5000 to 12000 with burrs number 95 for the convoluted tube, (ΔP), (f), $T_{cold,exit}$ and average Nusselt number.

Figure 7 describes the (ΔP) versus Re for numerical results. The values of (ΔP) increase dramatically as the Re increases, according to numerical findings. It becomes evident that (ΔP) values for pure water and nanofluids at 0.1% volume concentration in straight double-pipe heat exchangers differ significantly. Figure 7a shows that the maximum (ΔP) values was (391 Pa) in case of nanofluids at 0.1% volume concentration and Re = 10768 versus straight double-pipe heat exchanger. This results due to nanofluids has higher viscosity than pure water. Figure 7b shows that the maximum (ΔP) values was (770 Pa) in case of nanofluids at 0.1% volume concentration and Re = 10768 versus convoluted double-pipe heat exchanger. This higher (ΔP) due to flow disturbance and tortuosity.



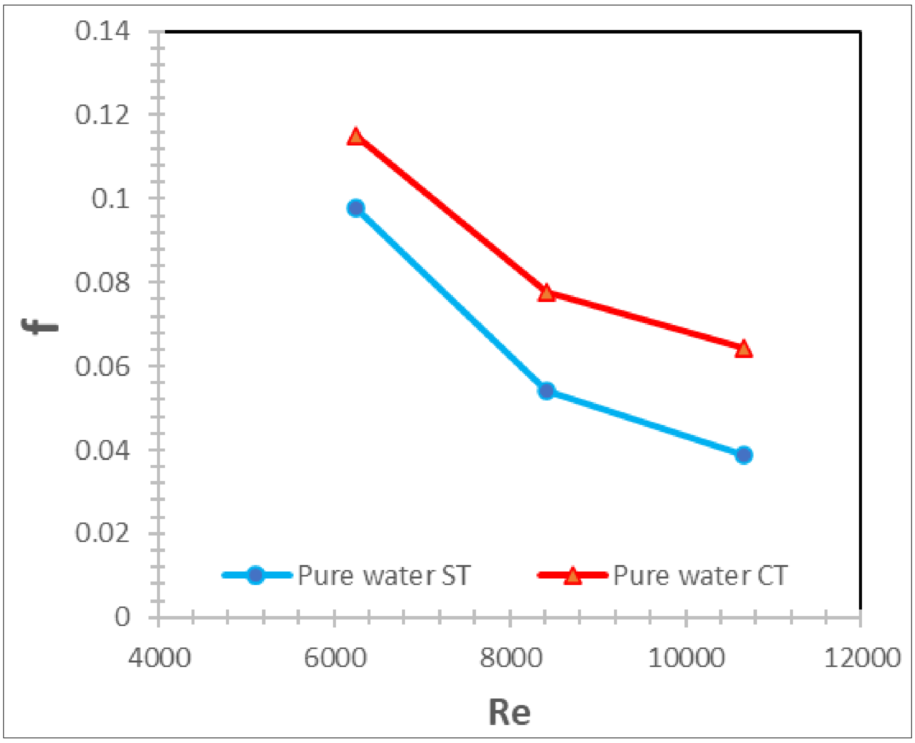
A) Both pure water and 0.1% CuO versus straight double-pipe heat exchanger.



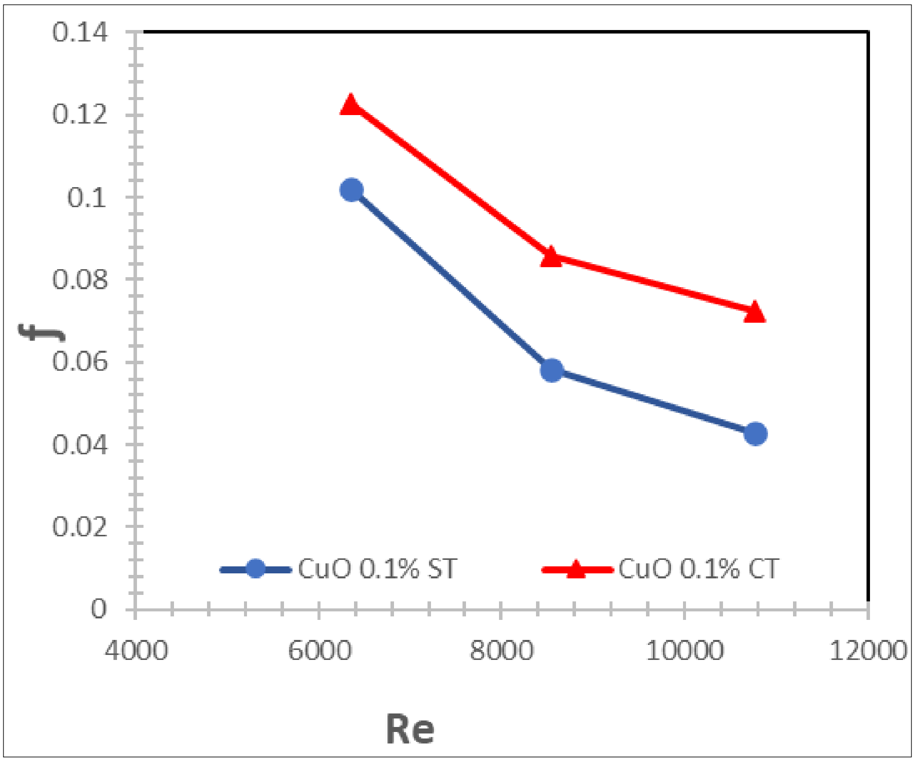
B) Both straight and convoluted double-pipe heat exchangers at 0.1% CuO

Fig 7: Variation of numerical (ΔP) versus Re.

Figure 8 illustrates the numerical results of (f) in relation to a broad range of Res. As depicted in Figure 8 the values of friction factor decrease dramatically as the Re increases, according to numerical findings. It becomes evident that friction factor values for straight and convoluted double-pipe heat exchangers differ significantly.



A) Straight and convoluted double-pipe heat exchangers for pure water.

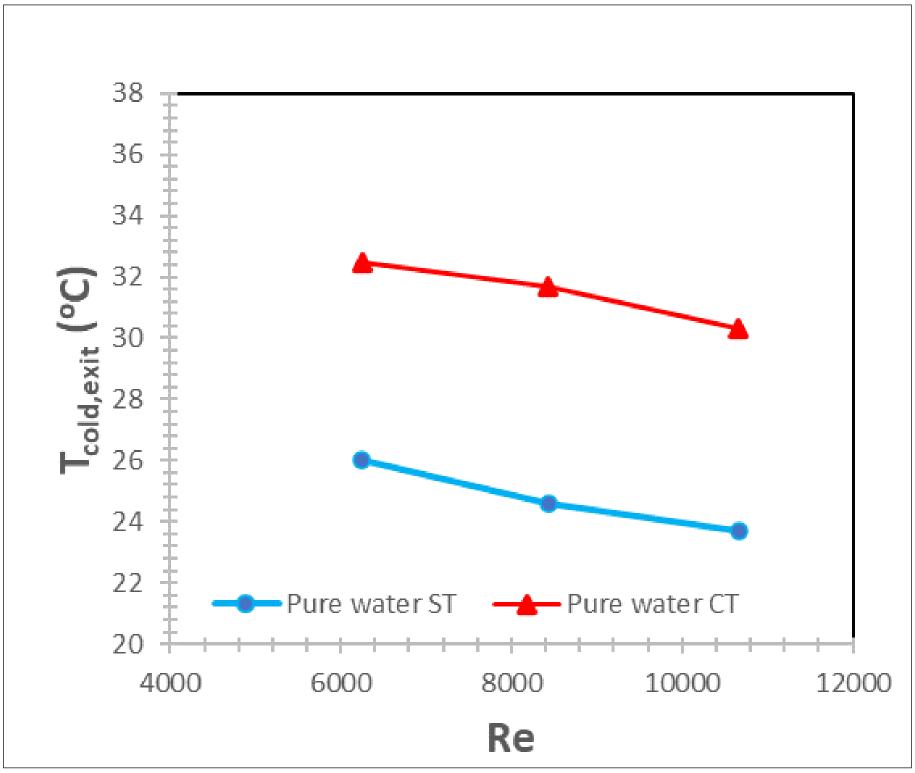


B) Straight and convoluted double-pipe heat exchangers for 0.1% CuO.

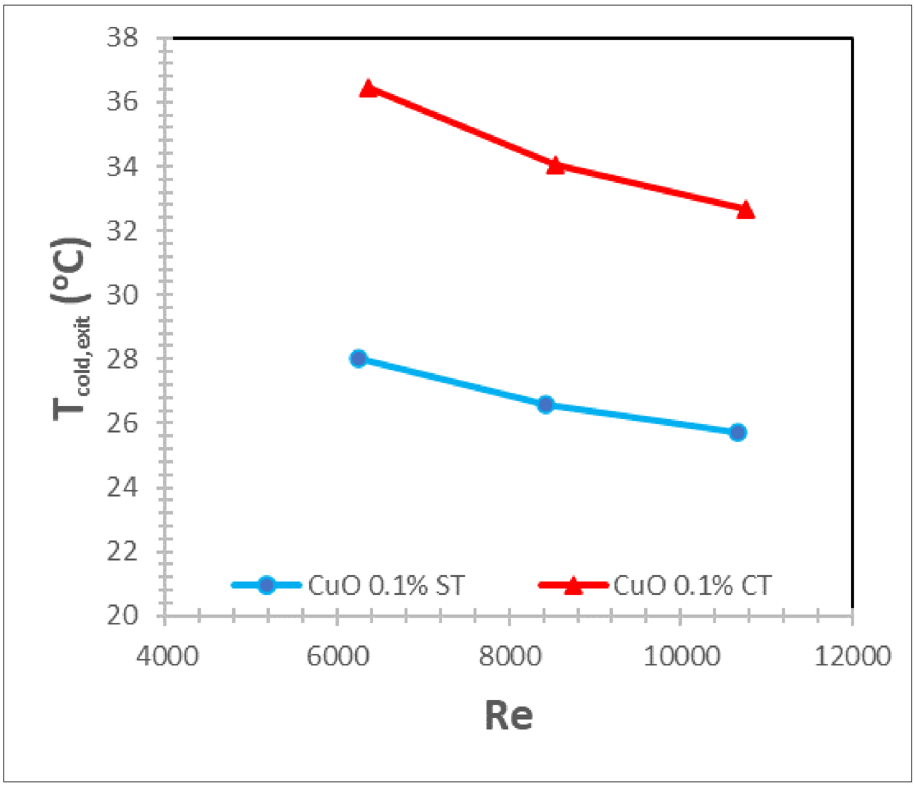
0

Fig 8: Variation of numerical friction factor versus Re.

Figure 9 demonstrates that the temperature values of the cold liquid outside the convoluted tube heat exchanger are greater than the temperatures of double straight tube heat exchanger due to the increased contact surface of exterior side of inner convoluted tube, allowing for improved heat transfer.



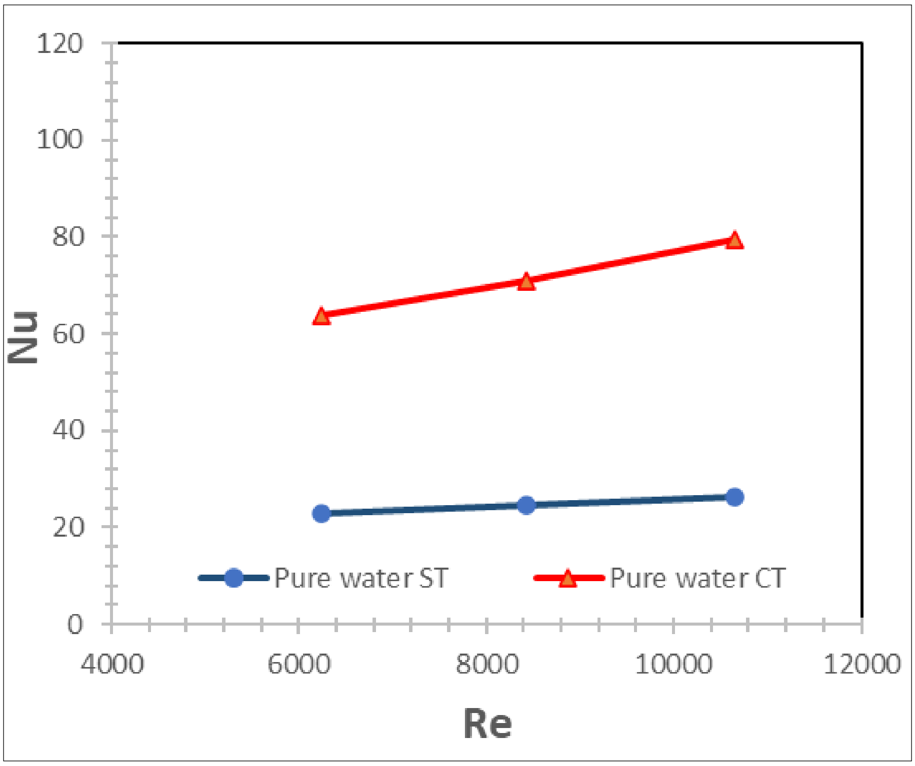
a) Straight and convoluted double-pipe heat exchangers for pure water.



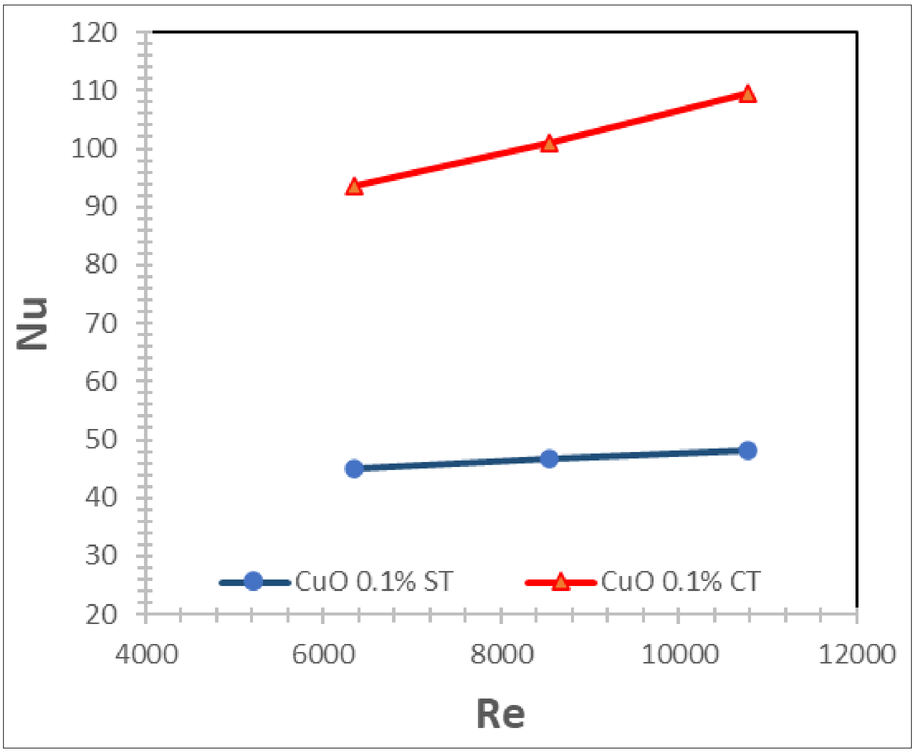
b) Straight and convoluted double-pipe heat exchangers for 0.1% CuO.

Fig 9: The variation of numerical $T_{cold,\ exit}$ versus Re.

Figure 10 represents the variation of numerical Nusselt number versus Res. It is obvious that the Nusselt numbers increase progressively with the increment of Re and with using a convoluted tube and a nanofluid (CuO). The spiral convoluted tube provides greater turbulence intensity near the wall through recirculation flow, which promotes excellent mixing of all water components and reduces the thermal boundary layer thickness. This case leads to enhancing heat transfer and result an increase in the rapidity at which heat transfer from warmer to cooler water. Figure 10b shows that the maximum (Nu) values was (110) in case of nanofluids at 0.1% volume concentration and Re = 10768 versus convoluted double-pipe heat exchanger.



a) straight and convoluted double-pipe heat exchangers for pure water.



b) straight and convoluted double-pipe heat exchangers for 0.1% CuO.

Fig 10: The variation of numerical Nusselt number versus Res.

Figure 11 illustrates the temperature and velocity contours along the test section for a straight tube heat exchanger at various axial distances. It can be seen from these figures that the maximal temperature occurs at 115 cm from the end of the tube (x=115 cm). This diagram's velocity contours demonstrate that the distribution of velocities at x= 15 cm is uniform. Also, fluid velocity increases at the tube's center and decreases progressively near the tube's wall.

Figure 12 illustrates the gradients in temperature and speed simulations, with and without nanofluids, of a heat exchanger comprised of a convoluted tube, at various axial distances. The test findings revealed that the utilisation of nanofluid led to enhanced heat transfer in comparison to the conventional straight tube heat exchanger. It was observed that the highest temperature was attained in the terminal section of the tube. The observed behaviour can potentially be attributed to the improved fluid interaction among the core circulation region and the wall flow zone, which the curved fluted tube assists to create. Enhancing the effective thermal conductivity of the nanofluid yields supplementary advantages. The velocity outlines that were displayed in this analysis illustrate that at x = 0 cm the speed distribution is uniform and decreases towards the tube wall while increasing further inside the tube.

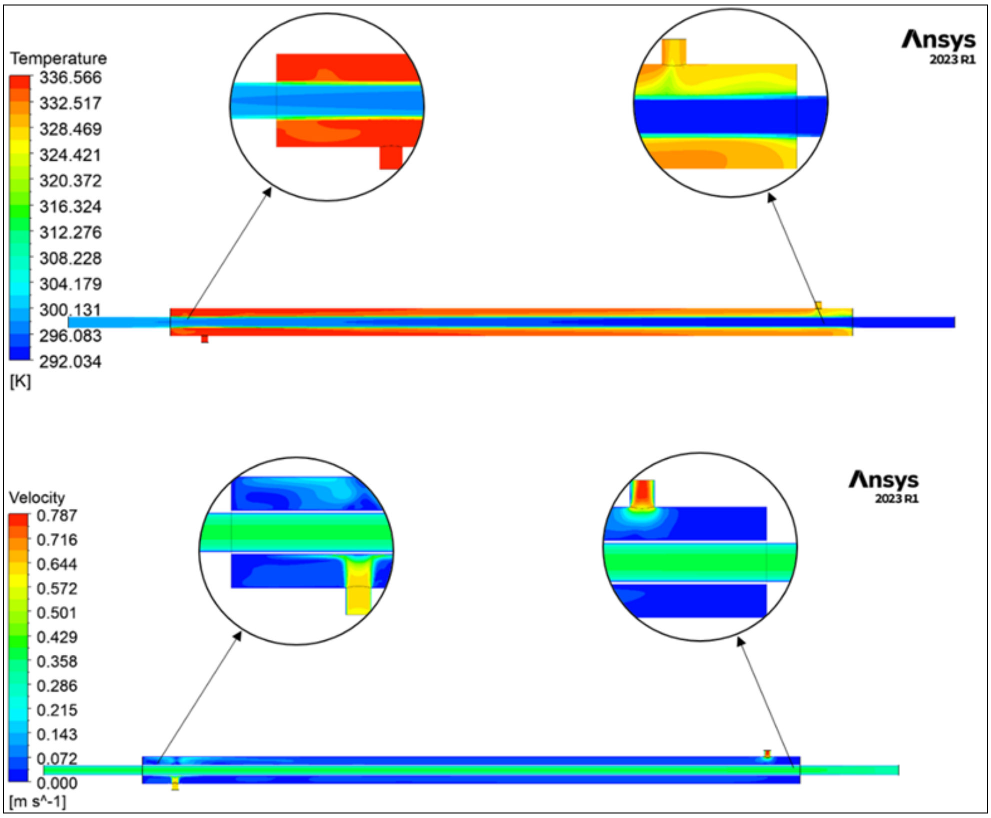


Fig 11: Test section temperature and speed curves for a straight double-pipe heat exchangers

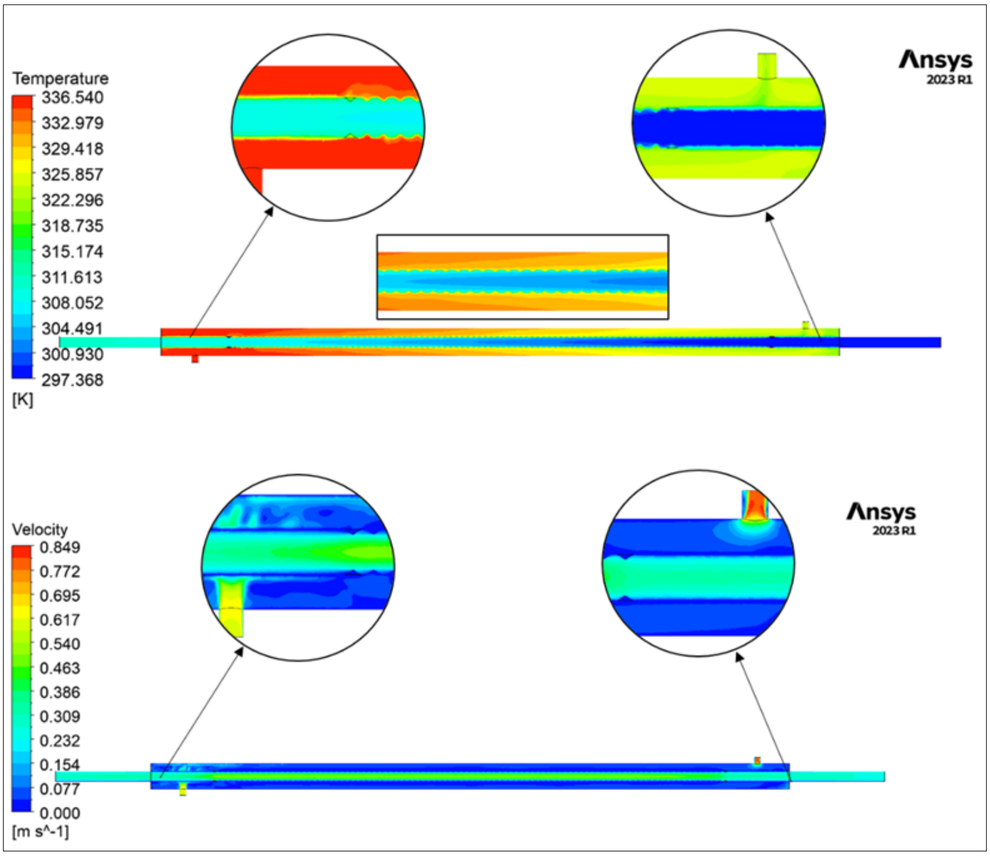


Fig 12: Test section temperature and speed curves for a convoluted double-pipe heat exchangers

Conclusions

From the results upon, the following conclusions are estimated

1. The test findings indicate that the convoluted tube exhibits a larger change in pressure drop in relation to the Reynolds number compared to the straight tube when water is used as the working fluid for both tubes. Moreover, the use of the 0.1% CuO nanofluid results in a more pronounced discrepancy in pressure drop variation between the convoluted and straight tubes.
2. The observed variance in the numerical friction factor with respect to Reynolds numbers is found to be larger for convoluted tubes compared to straight tubes when water is employed. Also, when the 0.1% concentration of Cou nanofluid is used, the change that is seen is the same for both the convoluted and convoluted tubes.
3. The drop in Reynolds numbers is associated with a decrease in the exit cold fluid temperature. In terms of heat exchange, the convoluted tube outperforms the straight tube for both water and Cou nanofluid (0.1%). This can be attributed to the longer course that the water or nanofluid follows within the convoluted tube.
4. The Nusselt number exhibits a positive correlation with the Reynolds number in both straight and convoluted pipe configurations when water and the CuO nanofluid are employed. the maximum (Nu) values was (110) in case of nanofluids at 0.1% volume concentration and Re = 10768 versus convoluted double-pipe heat exchanger.

References

1. Hussein ZA, Danook S, Mohamad B. Enhancement of thermal efficiency of nanofluid flows in a flat solar collector using CFD, Diagnostyka. 2022;23(4):1-9. https://doi.org/10.29354/diag/156384.
2. Qader F, Hussein A, Danook S, Mohamad B, Khaleel O. Enhancement of Double-Pipe Heat Exchanger Effectiveness by Using Porous Media and TiO2 Water, CFD Letters. 2023;15(4):31-42. https://doi.org/10.37934/cfdl.15.4.3142.
3. Qader F, Mohamad B, Hussein AM, Danook SH. Numerical study of heat transfer in circular pipe filled with porous medium. Pollack Periodica (published online ahead of print; c2023. https://doi.org/10.1556/606.2023.00869
4. Anderson D, Tannehill JC, Pletcher RH. Computational fluid mechanics and heat transfer. Taylor & Francis; c2016.
5. Sahu D, Dewangan DN. CFD Analysis of Shell and Tube Heat Exchanger using Nanofluid, Int. J Innov. Res. Sci. Eng. Technol, 2020, 9.
6. Eiamsa-Ard S, Thianpong C, Eiamsa-Ard P, Promvonge P. Thermal characteristics in a heat exchanger tube fitted with dual twisted tape elements in tandem, International Communications in Heat and Mass Transfer. 2010;37(1):39–46.
7. Liu S, Sakr M. A comprehensive review on passive heat transfer enhancements in pipe exchangers, Renewable and sustainable energy reviews. 2013;19:64-81.
8. Lee SC, Nam SC, Ban TG. Performance of heat transfer and pressure drop in a spirally indented tube, KSME International Journal. 1998;12:917-925.
9. Pethkool S, Eiamsa-Ard S, Kwankaomeng S, Promvonge P. Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube, International Communications in Heat and Mass Transfer. 2011;38(3):340–347.
10. Hossainpour S, Hassanzadeh R. Numerical investigation of tub side heat transfer and pressure drop in helically corrugated tubes; c2011.
11. Farajollahi B, Etemad SG, Hojjat M. Heat transfer of nanofluids in a shell and tube heat exchanger, Int J Heat Mass Transf. 2010;53(1-3):12–17.
12. Darzi AAR, Farhadi M, Sedighi K, Aallahyari S, Delavar MA. Turbulent heat transfer of Al2O3–water nanofluid inside helically corrugated tubes: numerical study, International Communications in Heat and Mass Transfer. 2013;41:68-75.
13. Matsson JE. An introduction to ANSYS fluent 2022. Sdc Publications; c2022.
14. Pletcher RH, Tannehill JC, Anderson D. Computational fluid mechanics and heat transfer. CRC press; c2012.
15. Bergman TL, Bergman TL, Incropera FP, Dewitt DP, Lavine AS. Fundamentals of heat and mass transfer. John Wiley & Sons; c2011.
16. Rousseau PG, Van Eldik M, Greyvenstein GP. Detailed simulation of fluted tube water heating condensers, International Journal of Refrigeration. 2003;26(2):232-239.
17. Xuan Y, Li Q. Heat transfer enhancement of nanofluids, Int J Heat Fluid Flow. 2000;21(1):58–64.
18. Ali HHM, Hussein AM, Allami KMH, Mohamad B. Evaluation of shell and tube heat exchanger performance by using ZnO/water nanofluids. Journal of Harbin Institute of Technology (New Series). 2023;30:1-13 https://doi.org/10.11916/j.issn.1005-9113.2023001
19. Versteeg HK, Malalasekera W. An introduction to computational fluid dynamics: the finite volume method. Pearson education; c2007.