International Journal of Current Trends in Engineering and Technology
www.ijctet.org, ISSN: 2395-3152
Volume: 09, Issue: 04 (July -August, 2023)

CFD Modeling of a Double Pipe Counter Flow Heat Exchanger

Avinash Kumar Singh, Pankaj Shrivastava
Department of Mechanical Engineering,
Corporate Institute of Science & Technology, Bhopal, Madhya Pradesh, India
savinashkumar2601@gmail.com, ps0203@gmail.com

Abstract— *This study examines the influence of variations in the inner pipe's cross-sectional area on the Nusselt number of flows in a concentric pipe arrangement with heat transfer properties. A three-dimensional Finite Volume Method (FVM) simulation was conducted on a counter-flow arrangement pipe. The inner pipe's cross-sectional area was adjusted according to the aspect ratio discussed in the existing literature, and the annulus's Reynolds number (Re) was varied from 2000 to 11000. The findings indicate that the Nusselt number increases as the aspect ratio and Reynolds number increase. The simulation results were cross-referenced and validated against relevant literature concerning concentric arrangements. The experiments maintained the Reynolds number of the hot fluid at 7,000 while gradually adjusting the Reynolds number of the cold fluid to 2,000, 5,000, 8,000, and 11,000. The results demonstrate that altering the Reynolds number for a fixed aspect ratio does not significantly impact the heat transfer coefficient. However, reducing the aspect ratio considerably enhances the heat transfer coefficient, reaching a peak of 38.47% at a Reynolds number of 2,000. Furthermore, this effect is more pronounced for the cold fluid in the annulus with lower Reynolds numbers.*

Keywords: *Effectiveness, aspect ratio, Nusselt number, Turbulent flow, double pipes*

I. INTRODUCTION

Heat exchangers play a crucial role in various engineering processes, including HVAC systems and power plants, and continuous efforts are being made to enhance their efficiency. Improving heat exchanger performance is of utmost importance to achieve energy and cost savings.

In order to develop efficient double-pipe heat exchangers, a comprehensive study of fluid flow and heat transfer properties is essential. However, due to the complex nature of factors like geometry, operation, and fluid properties, this remains an ongoing area of research. Researchers employ various approaches to enhance heat transfer, such as turbulence manipulation and modification of the Reynolds number. The impact of different geometric features on heat transport is also investigated.

This article specifically focuses on the influence of the aspect ratio on heat transfer efficiency. The objective is to examine how altering the Reynolds number (Re) of the cold fluid affects the thermal characteristics of the hot fluid. While circular inner pipes are commonly used in heat exchangers, this article explores the possibility of utilizing a flat inner pipe shape as an alternative. No previous studies have explored the passive effect of a flat inner pipe shape on heat transfer, and the mechanisms underlying its potential for improving heat transfer performance while reducing additional pressure drop are not yet understood. This research aims to bridge this knowledge gap.

Modifying the cross-sectional shape of pipes is another commonly employed approach to enhance heat exchanger performance. By adjusting the hydraulic diameter of the pipe, the characteristic length term in the Nusselt number equation can be influenced, resulting in increased Nusselt values for a given pipe perimeter. Plate and tube heat exchangers, particularly in automotive cooling systems, often utilize fins to enhance heat transfer. The inclusion of fins in these heat exchangers significantly affects both the pressure drop and the heat transfer properties. Evaluating the overall performance of forced convection is crucial, considering both the pressure drop and the heat transfer flux.

II. LITERATURE REVIEW

Jiyang Li et al. [1] conducted an experimental study to measure the air-side pressure drops and total heat transfer coefficients of a finless heat exchanger with and without a vortex generator, as well as five all-aluminium parallel multi-port heat exchangers. The experiments were conducted under dry, wet, and frosting/defrosting conditions. Three multi-port heat exchangers had louvered fins with fin pitches of 1.2 mm, 1.4 mm, and 1.6 mm, while two exchangers had slit fins with fin pitches of 1.2 mm and 1.4 mm. The study aimed to examine the effects of fin pitch and type on air-side pressure drop and total heat transfer performance in fin-tube heat exchangers.

The results revealed that the heat transfer performance of a finless heat exchanger with a longitudinal vortex generator (LVG) was approximately 40% lower than that of a fin-tube heat exchanger in dry conditions. However, the pressure drop was comparable between the two. In wet conditions, the heat transfer coefficient was the same, but the finless heat exchanger demonstrated superior drainage efficiency and lower pressure drop. During frosting/defrosting, the heat transfer coefficient was comparable for both types, but the finless heat exchanger exhibited significantly lower pressure drop. At the end of each icing phase, the pressure drop only reached 50 Pa.

Overall, the findings indicated that a finless heat exchanger with a vortex generator performs worse than a fin-tube heat exchanger in dry conditions, but it outperforms the latter in wet and frosting/defrosting conditions.

K. Torii et al. [2] proposed a new method to enhance heat transfer and reduce pressure loss in a fin-tube heat exchanger with circular tubes at relatively low Reynolds numbers. The method involved utilizing delta winglet-type vortex generators arranged in a novel configuration known as the "common flow up" setup. This structure effectively delayed separation, reduced form drag, and eliminated the zone of poor heat transfer near the tubes' wake. Experimental studies validated the effectiveness of this augmentation method. When the winglets were placed in staggered tube banks, heat transfer increased by 30% to 10%, and pressure loss decreased by 55% to 34% for Reynolds numbers ranging from 350 to 2100. In the case of in-line tube banks, the augmentation ranged from 20% to 10%, while pressure loss decreased by 15% to 8%.

Jorge A. et al. [3] developed an algorithmic mathematical model to simulate steady-state behavior in gasketed plate heat exchangers with various configurations. The model took into account factors such as the number of channels, number of passes on each side, fluid locations, feed connection locations, and channel-flow type. The objectives of the model were to investigate the impact of configuration on exchanger performance and to optimize the configuration. The simulation results included temperature profiles in all channels, thermal efficiency, distribution of the total heat transfer coefficient, and pressure drops. The model also explored the assumption of a constant total heat transfer coefficient.

In a numerical study conducted by Hamed Arjmandi et al. [4], researchers investigated the effects of combining vortex generators, twisted tape turbulators, and Al2O3-H2O nanofluid as the base fluid in a twin-pipe heat exchanger. The study focused on analyzing the heat transfer rate and pressure drop behavior. The researchers used response surface methodology (RSM) based on central composite design (CCD) to determine the optimal shape of the combined vortex generator and twisted tape turbulator. They evaluated twenty different cases with varying pitch ratios, angles, and Reynolds numbers to maximize the Nusselt number while minimizing the friction factor. The results indicated that the pitch ratio had a significant influence on both the Nusselt number and friction factor, resulting in a fivefold improvement in efficiency compared to the original design. Additionally, reducing the angle of the vortex generators in the new combined turbulator increased the Nusselt number and friction factor.

According to a study by R. Bhadouriya et al. [5], modifying the geometry of the inner duct in a flexible manner, such as using a twisted square channel, can be a viable alternative to traditional annulus designs, leading to superior performance under similar conditions. Another effective approach is to introduce a fin tube as the inner pipe and increase its ellipticity, which significantly enhances the heat transfer rate. Among the various options for internal pipe geometry, using a flat shape provides a feasible and straightforward variation of the circular counterpart, offering potential for optimization.

A. Erek et al. [6] conducted a study using computational fluid dynamics (CFD) to investigate the impact of fins on heat transfer characteristics and pressure drop in a plate and tube heat exchanger. The presence of fins was found to have a significant influence on the pressure drop characteristics, with increasing fin height resulting in higher pressure drop and improved heat transfer performance. Additionally, reducing the tube thickness was found to enhance heat transfer.

K. Sharifi et al. [7] studied the geometry and configuration aspects of a double-pipe heat exchanger with a flat inner pipe. The researchers discussed the conservation equations used in their computational fluid dynamics (CFD) simulation, as well as the relevant boundary conditions. After ensuring grid independence and validating the numerical simulation, the study reported and evaluated the fluid flow and heat transfer parameters for each proposed configuration. The investigation aimed to understand the impact of the flat inner pipe design on pressure drop, temperature distribution, and thermal performance across a wide range of Reynolds numbers.

Similarly, Do Huu-Quan et al. [8] conducted a study exploring different cross-sectional arrangements for the flattened inner pipe. The objective was to investigate how changing the aspect ratio of the hot fluid-carrying pipe influenced the heat transfer coefficient and average Nusselt number (Nu). The investigation demonstrated that altering the aspect ratio improved efficiency. Additionally, the researchers introduced a performance index that considered both heat transfer and pressure drop characteristics.

III. OBJECTIVE

This research aims to do a Finite volume simulation of a twin pipe heat exchanger while altering the cross sections of the inner pipe. The purpose of this inquiry is to examine the effect on the thermal parameters of the fluid flow. The following topics will be covered:

1. Evaluation of the hot fluid's heat transfer coefficient in relation to alterations in the cold fluid's aspect ratio and Reynolds number (Re).
2. The Nusselt number (Nu) of the hot fluid is examined in connection to changes in the aspect ratio and Reynolds Number of the cold fluid.
3. Validation of the current study by comparison with previous literature.
4. Temperature and pressure contour analysis to get an understanding of fluid dynamics.
5. Evaluation of the heat exchanger's effectiveness under various parameter settings.

By addressing these issues, we hope to better understand the thermal performance of a double-pipe heat exchanger with varied inner pipe cross sections.

IV. METHODOLOGY

Domain description

ANSYS Design Modeler was used to create the 3-dimensional double pipe counter flow layout. The inner pipe diameter was 25 mm, while the outside pipe diameter was 40 mm. The inner pipe was composed of aluminium and had a thickness of 1 mm. The aspect ratios employed in the study were adopted from the research conducted by Do Huu-Quan et al. The heat exchanger's total length was set at 500 mm.

A finite volume analysis (FVM) was performed using ANSYS Fluent to achieve the study's goal. The governing equations, such as the continuity equation, momentum equation, energy equation, k-equation, and ∈-equation, were used in the study. In order to guarantee that the simulations were reliable and consistent, the boundary conditions and numerical parameters were chosen based on the information offered in the original publication.

Governing Equations

The formula for mass preservation,

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\,\vec{v}) = S_m$$

Where S_m= the mass added to the permanent phase or any handler sources.

Momentum Conservation Equations

The characteristics of momentum conservation in an inertial reference frame include

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

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}) - \frac{2}{3}\nabla.\vec{v}I\right]$$

Energy Equation

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}(\alpha_k\rho_k E_k) + \nabla.\sum_{k=1}^{n}(\alpha_k\vec{v}_k(\rho_k E_k + p)) = \nabla.(k_{eff}\nabla T) + S_E$$

***k*−∈ model**

The following transport equations are used to calculate the kinetic energy of the turbulence, k, and its rate of dissipation, ∈ :

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho \in - Y_M + S_k$$

and

$$\frac{\partial}{\partial t}(\rho \in) + \frac{\partial}{\partial x_i}(\rho \in v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\in}\right)\frac{\partial \in}{\partial x_j}\right] + C_{1\in}\frac{\in}{k}(G_k + C_{3\in}G_b) - C_{2\in}\rho\frac{\in^2}{k} + S_\in$$

Performance indicators definition

The definitions of various key variables that affect the thermal and hydrodynamic performances of the heat exchanger are provided in this section. It concentrates on the total heat transfer rates between the cold fluid in the annulus and the hot fluid in the inner pipe.

$$Q_h = \dot{m}C_{ph}(T_{in} - T_{out})_h$$
$$Q_c = m_c C_{pc}(T_{out} - T_{in})_c$$

The product of the intake mass current rate (ṁ) for the hot fluid, the specific heat capacity (Cph) of the hot fluid, the inlet mass current rate (ṁ) for the cool fluid, and the specific heat capacity (Cpc) of the cold fluid is the mean heat transfer rate in this context. The calculation also considers the temperatures at the intake (Tmin) and output (Tout).

$$Q = (Q_h + Q_c)/2$$

Effectiveness is the ratio of the greatest attainable heat transfer rate to the actual heat transfer rate.

$$\gamma = \frac{Q}{Q_{\max}}$$

where $Q_{m\sigma x} = (\dot{m}C_p)_{mox}(T_{h\dot{m}} - \tau_{c\dot{m}})$

in which $(\dot{m}C_p)_{mox}$ shows the higher value between $\dot{m}C_{pc}$ and $\dot{m}C_{ph}$.

Following are the mean convective heat transfer coefficients for the inner pipe and annulus:

$$\overline{h_{pipe}} = \frac{-\overline{q_w'}'}{0.5(T_{hin} + T_{ho}) - \overline{T_w}}$$

$$\overline{h_{annulus}} = \frac{-\overline{q_w'/}}{\overline{T_w}, -0.5(T_{cin} + T_{co})}$$

where $\bar{h}$ is the mean thermal flux on the inner pipe wall, $\bar{T}$ is the mean temperature of the inner pipe wall, $\bar{q}$'' is the mean convective heat transfer coefficient in the annulus, and $\bar{h}$ is the mean convective heat transfer coefficient in the inner pipe.

As a result, the following formula may be used to get the mean Nusselt values in the inner pipe and annulus:

$$\overline{Nu_{pipe}} = \frac{\overline{h_{pipe}}D_{hpipe}}{k_c}$$

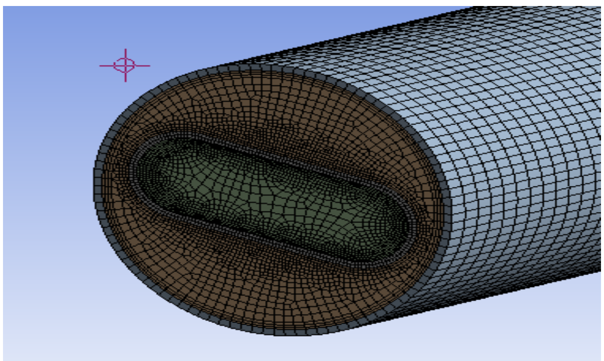
$$\overline{Nu_{annulus}} = \frac{\overline{h_{annulus}}D_{hannulus}}{k_c}$$

where k_c is the fluids' thermal conductivity and D_h is their hydraulic diameter.

Pressure differentials at the entrance and outflow on either side are used to calculate the friction coefficient.

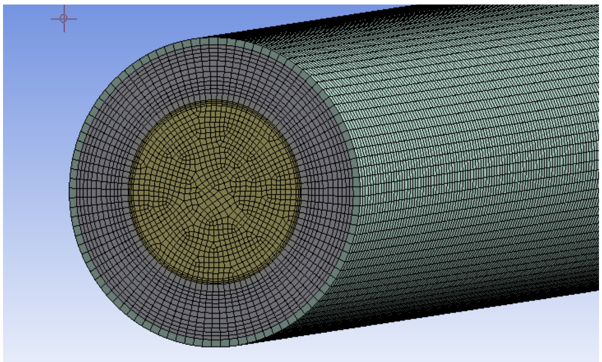
Meshing Details:

Case 1 in the simulation had 712,840 nodes and 622,250 cells, whereas Case 4 had a total of 958,569 nodes and 844,400 cells.



(a)

For example, 1, the average orthogonal quality was 0.99, suggesting a high-quality mesh ideal for producing reliable outcomes. The mesh elements utilized in the simulation were predominantly hexagonal, which helped to improve computational efficiency and speed up the calculations.



(b)

Figure 1 Meshing of (a) case 1 and (b) case 4

Boundary circumstances:
Water was chosen as the study's working fluid because it had the following characteristics: density (ρ) = 1000 kg/m^3, kinematic viscosity (v) = 1×10^(-3) m^2/s, and specific heat capacity (cp) = 4182 J/kg-K. The cold water intake was set at 293 K, while the hot water inlet was set to 333 K. The standard k- ε model with increased wall treatment was the turbulence model applied in the simulation to represent the effects of the boundary layer correctly. The inlet boundaries were set as velocity inlets, while the outlet boundary was set as a pressure outlet. The walls of the inner pipe were thermally coupled, whereas the exterior wall was considered to be insulated with no heat flux.

V. OUTCOME AND COMMENTS

While the hot fluid's Reynolds number (Re) stayed constant at 7000, the cold fluid's Re fluctuated between 2000 and 11000. The basic case took into account various aspect ratios throughout the validation process, and the hot fluid's Re was modified in accordance with prior research. A comparison with the research done by Do Huu-Quan et al. was undertaken to check the correctness of the results, and it revealed a good level of agreement. The largest inaccuracy in the heat transfer coefficient that could be measured was 9.82%. A comparison chart was created to show the findings of the current investigation against the previously published literature. The SIMPLEC solution control approach was used in the simulation, which sped up the resolution of the governing equations.

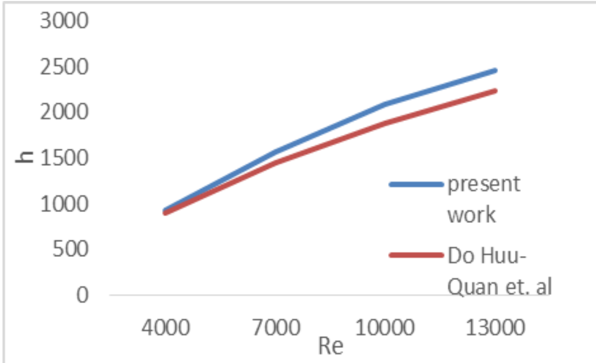


Figure 2 Comparison of mean heat transfer coefficient with existing data

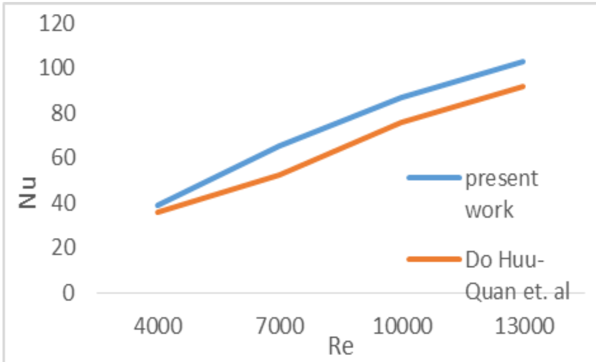
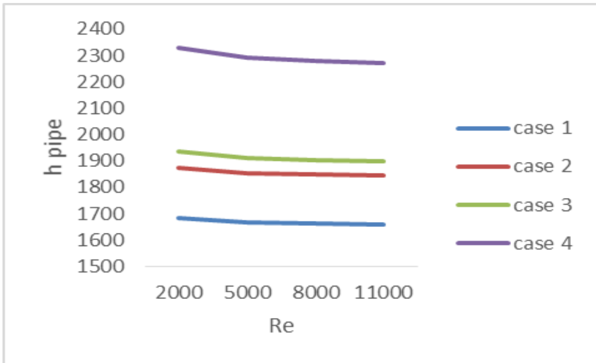
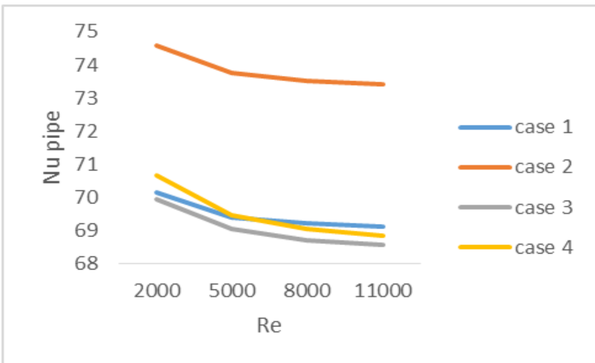


Figure 3 Comparison of Nu with existing data



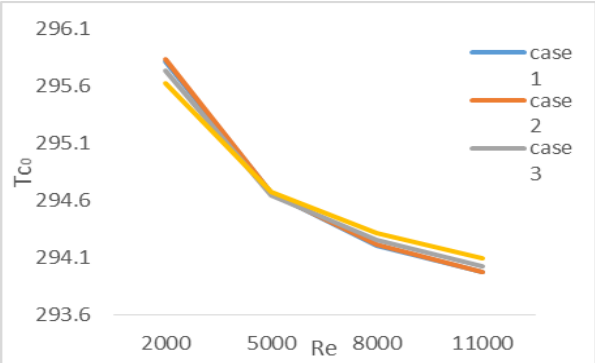
(a)



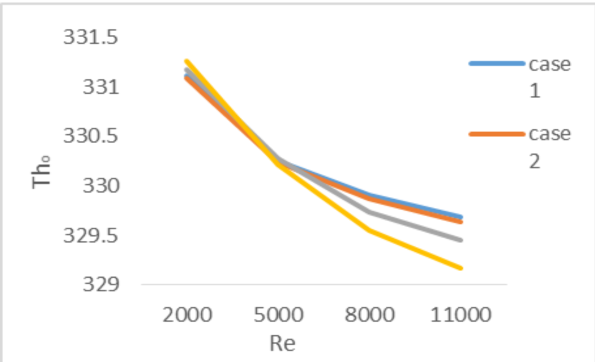
(b)

Figure 4 illustrates the variation of (a) the mean heat transfer coefficient and (b) the Nu for different cases.

Aspect ratios of 0.72, 0.53, and 0.37 were chosen for the modified examples 2, 3, and 4, correspondingly. Case 1 depicts a concentric twin pipe design with counterflow characteristics, which was taken from earlier literature. A chart was created after analysis of the mean heat transfer coefficient and Nusselt number (Nu).

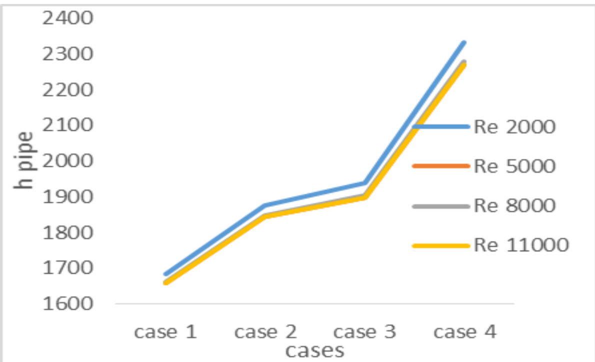


(a)



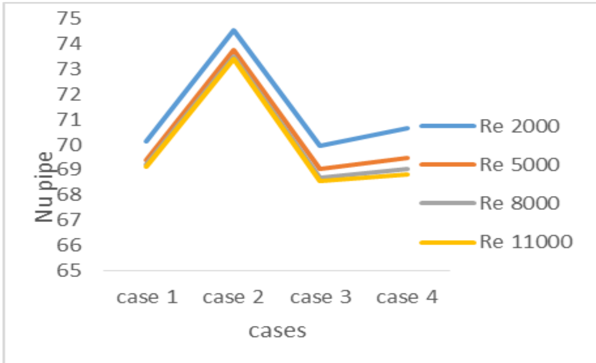
(b)

Figure 5 displays the variation of temperature for (a) the outlet of the cold fluid and (b) the outlet of the hot fluid across different cases.



(a)

According to the research, case 2 was predicted to have the greatest Nu value in this study because its Nu expression has the hydraulic diameter and heat transfer coefficient multiplication terms. The trend in the figure supported the ideal Nu value seen in case 2. The study also demonstrated patterns in heat transfer values, showing whether they rose or fell depending on certain aspect ratios. The graph below shows that modifications in the cold fluid's annulus's Reynolds number did not affect the pipe's heat transfer coefficient and Nu. However, significant variances were seen in connection to aspect ratio changes.



(b)

Figure 6 depicts the variation of (a) the mean heat transfer coefficient and (b) the Nu for different Reynolds numbers (Re).

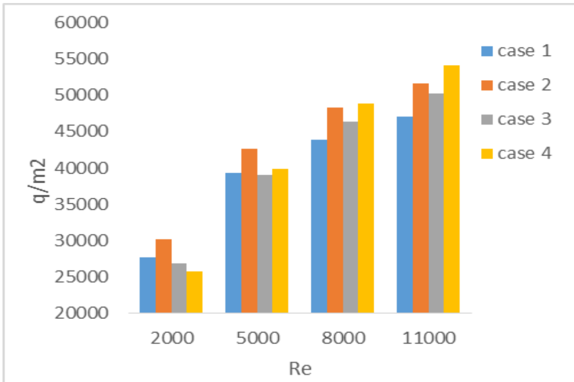
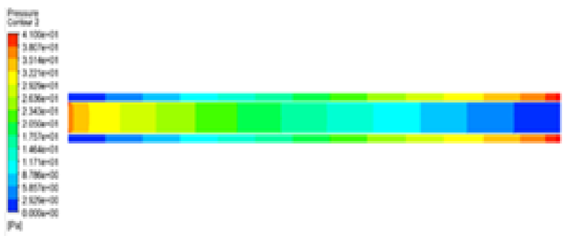


Figure 7 illustrates the variation of heat flux for different cases in relation to Reynolds numbers (Re).



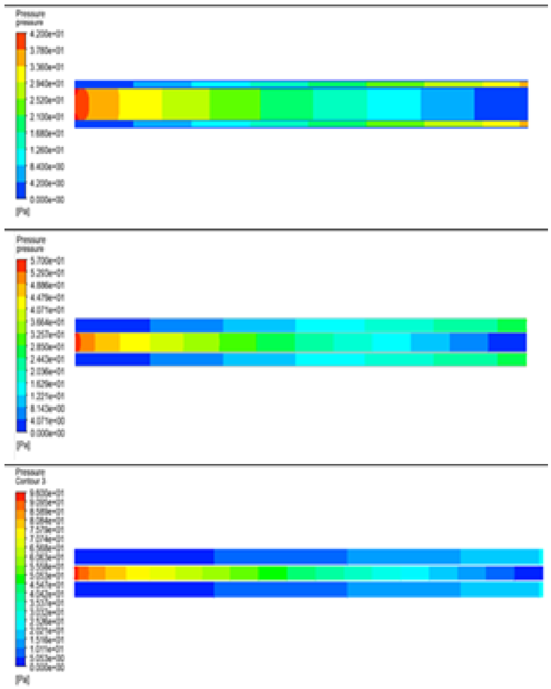


Figure 8 showcases the pressure contours for case 1, case 2, case 3, and case 4, respectively, at a Reynolds number (Re) of 2000.

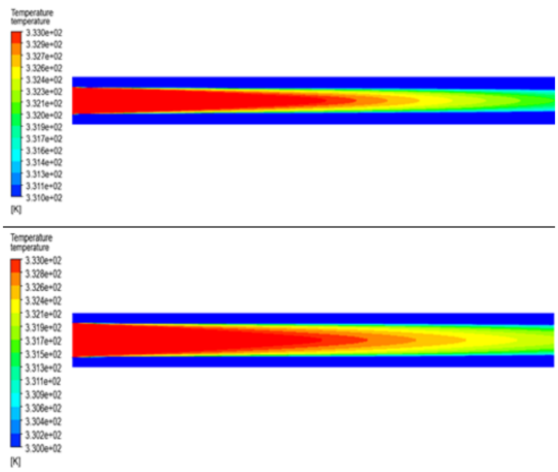


Figure 9 illustrates the temperature contours for case 1, case 2, case 3, and case 4, respectively, at a Reynolds number (Re) of 2000.

At a Reynolds number (Re) of 2000, Figure 9 shows the corresponding temperature contours for Cases 1, 2, 3, and 4. The figure's contours were created using a Reynolds number (Re) of 2000. The contours depict the formation of temperature layers and the movement of the flow itself.

VI. CONCLUSION

This study aimed to investigate the impact of changing the Reynolds number of the cold fluid on the Nusselt number and heat transfer coefficient of the hot fluid. After carefully analyzing the findings and charts, the following conclusions can be drawn:

When the cold fluid's Reynolds number (Re) was increased from 2000 to 11000, a modest reduction in the pipe's heat transfer coefficient was observed, ranging from 1.45% to 2.6% in all cases. This indicates that changes in the Reynolds number of the annulus fluid do not significantly affect the heat transfer coefficient of the pipe.

On the other hand, the heat transfer coefficient of the pipe increased by 38.47% for Re 2000, 37.57% for Re 5000, 37.10% for Re 8000, and 36.85% for Re 11000 when comparing different scenarios at a constant Reynolds number. This suggests that the heat transfer coefficient tends to increase as the aspect ratio decreases, with a more pronounced increase observed for lower Reynolds number values. Similar trends were observed for the Nusselt number (Nu) with respect to Reynolds number fluctuations, although the degree of change varied. The highest Nu value was observed in case 2, which had the greatest multiplication of the heat transfer coefficient and hydraulic diameter in its Nu formulation. Notably, compared to the hot fluid example from the previous experiment at Re 7000, Nu increased by a maximum of 6.27%.

When examining the heat flux figures, it was found that increasing the Reynolds number from 2000 to 11000 resulted in percentage increases of 69.99% for case 1, 70.68% for case 2, 86.92% for case 3, and 110.21% for case 4. Additionally, the percentage increment was higher as the aspect ratio decreased, indicating that a lower aspect ratio enhances the heat transmission value.

The heat flow initially increased from case 1 to case 2 when comparing different examples at a given Reynolds number, and then it decreased in line with the Nu trend. Furthermore, the effectiveness of the heat exchanger arrangement was examined, revealing that raising the Reynolds number from 2000 to 11000 resulted in a decrease in effectiveness of 56.02% for case 1, 56.33% for case 2, 52.37% for case 3, and 38.19% for case 4. It was also observed that, at a constant aspect ratio, effectiveness increased with lower Reynolds number values.

References

[1] J. Li, C. Dang, E. Hihara, Heat transfer enhancement in a parallel, finless heat exchanger using a longitudinal vortex generator, Part B: experimental investigation on the performance of finless and fin-tube heat exchangers, Int. J. Heat Mass Transf. 128(2019) 66–75.

[2] K. Torii, K.M. Kwak, K. Nishino, Heat transfer enhancement accompanying pressure loss reduction with winglet-type vortex generators for fin-tube heat exchangers, Int. J. Heat Mass Transf. 45 (2002) 3795–3801.

[3] JAW Gut, J.M. Pinto, Modeling of plate heat exchangers with generalized configurations, Int. J. Heat Mass Transf. 46 (2003) 2571–2585.

[4] H. Sadighi Dizaji, S. Jafarmadar, F. Mobadersani, Experimental studies on heat transfer and pressure drop characteristics for new arrangements of corrugated tubes in a double pipe heat exchanger, Int. J. Therm. Sci. 96 (2015) 211–220.

[5] R. Bhadouriya, A. Agrawal, S.V. Prabhu, Experimental and numerical study of fluid flow and heat transfer in an annulus of the inner twisted square duct and outer circular pipe, Int. J. Therm. Sci. 94 (2015) 96–109.

[6] A. Erek, B. ¨Ozerdem, L. Bilir, Z. ˙Ilken, Effect of geometrical parameters on heat transfer and pressure drop characteristics of plate fin and tube heat exchangers, Appl. Therm. Eng. 25 (2005) 2421–2431.

[7] K. Sharifi, M. Sabeti, M. Rafiei, A.H. Mohammadi, L. Shirazi, Computational fluid dynamics (CFD) technique to study the effects of helical wire inserts on heat transfer and pressure drop in a double pipe heat exchanger, Appl. Therm. Eng. 128 (2018) 898–910.

[8] D. Huu-Quan, Ali Mohammad Rostami, Mozafar Shokri Rad, Mohsen Izadi, Ahmad Hajjar, Qingang Xiong, 3D numerical investigation of turbulent forced convection in a double-pipe heat exchanger with flat inner pipe, Applied Thermal Engineering 182 (2021)106-116

[9] Heat transfer textbook by Yunus A. Çengel

[10] https://en.wikipedia.org/wiki/Heat_exchanger