CFD analysis of double pipe counter flow heat exchanger subjected to varying cold fluid velocity in flat inner pipe

1Vishal Gaurav Rai, 2Ashish Muchrikar
1M. Tech scholar, 2Prof. & hod department of mechanical engineering
1Corporate Institute of Science and Technology Bhopal,
2Corporate Institute of Science and Technology Bhopal

Abstract - **In this work a 3 dimensional CFD simulation is performed on counter flow arrangement pipe. The main objective of the work is to simulate a concentric pipe flow with heat transfer characteristics with different cross section of inner pipe to examine the effect of the change in cross section on the Nu number of flow. The work is validated against the existing literature with concentric arrangement. The cross section of inner pipe is changed according to the aspect ratio of existing literature and Re of annulus is increased from 2000 to 11000 to see the effect on inner fluid Nu. The Re of hot fluid is kept constant at 7000 and cold fluid Re is taken as 2000, 5000, 8000 and 11000. No significant change in heat transfer coefficient is observed with respect to change in Re for fixed aspect ratio but it is observed that as we go toward lower aspect ratio heat transfer coefficient increased in significant amount and is maximum by 38.47% at Re of 2000. It is also seen that the change is higher for lower Re of cold fluid which is flowing in annulus. For Nu the trend is not same as heat transfer coefficient for different aspect ratio keeping the Re fixed and found maximum at aspect ratio of 0.72 by 6.27% as compared to the previous case with hot fluid at Re of 7000.**

keywords - **Heat transfer, Nusselt number, Aspect ratio, effectiveness, Double pipe, Turbulent flow**

I. INTRODUCTION

Heat exchanger is an important device used in a wide range of daily engineering process. It find its application in various sectors of industries as HVAC systems, power plants and many more. Various types of heat exchanger is available in market and more is under development phase. Also existing one are continuously improved for better efficiency.

It is widely accepted that high efficiency of heat exchangers contributes significantly to overall energy and cost savings. Therefore, continuous improvement of performance for double-pipe heat exchangers is of vital importance from both scientific and engineering points of view. However, due to vast variations of geometry, operation, and fluid properties, etc., systematic investigation of characteristics on fluid flow and heat transfer to design more efficient double-pipe heat exchangers is still a hot spot in recent years. Some of the techniques used to enhance the heat transfer are practiced as increasing the turbulency, altering the Re of flow etc. the various geometrical parameters are also changed to see to see the effect on heat transfer and one of the is being discussed in this article where the aspect ratio is already set up in the previous literature. In the present article we will change the Re of cold fluid to observe the effect on hot fluid thermal characteristics. Among available candidates for geometry of inner pipes, flat shape presents a feasible while simple variation of the circular counterpart, where much attention can be paid to. However, to the best of our knowledge, there has been no work reported on the effect of flat inner pipe geometry as a passive factor on heat transfer, and our understanding of the mechanisms by which the use of flat inner pipes can enhance heat transfer performance while introducing minimum extra pressure drop are not clear. Thus, this study is oriented to close this gap.

The change in the cross section is been practising to enhance the performance of the heat exchanger also. In this way the hydraulic diameter of the pipe can be changed so that it can affect the characteristic length term in the Nu number expression thus enhancing the Nu for same perimeter of the pipe. Plate and tube type heat exchanger also are in common practice to enhance the heat transfer as such type of heat exchangers are used in automotive cooling system. The fins in such type of heat exchangers plays an important factor in pressure drop and heat transfer characteristics. In case of forced convection it is important to check the combined performance in terms of pressure drop and heat transfer flux.

II. LITERATURE REVIEW

Smith Eiamsa-ard et. al [1] In the extant work, heat transfer and friction characteristics were experimentally investigated, employing down strips implanted in a concentric tube heat exchanger. The downward strip was implanted into the pipe to engender turbulent flow which facilitated to upsurge the heat transfer degree of the tube. The current rate of the tube was in a range of Reynolds number between 6000 and42,000. The turbulent flow devices were consisted of (1) the down projection with forward or backward provisions, and (2) the down projection with numerous inclined angles (θ=15°, 25° and 30°), implanted in the inner hose of the heat exchanger. In the experiment, hot water was streamed through the inner tube whereas cold water was flowed in the annulus.

M. Sheikholeslami et. Al [2] The current article investigates the sway of using fins and nano sized tackles on recital of discharging system. Various figures for nanoparticle have been measured. Cold fluid currents in both inner and outer deposits and middle deposit is full of PCM. To create a careful prime of designing heat stowage based on unvarying solidification, two aspect has been scrutinized; length of fins and profile factor. Temperature and solid fraction dispersals were reported at different time steps. The homogeneous model for Nano fluid has been protracted by incorporating various figures of CuO nanoparticles. The mathematical archetypal has been obtainable in the form of PDE's, which were unravelled using Galerkin FEM.

Weerapun Duangthongsuk et. al [3] This item accounts an experimental learning on the forced convective heat transfer and stream characteristics of a nano fluid consisting of water and 0.2 vol.% TiO2 nanoparticles. The heat transfer coefficient and friction influence of the TiO2–water nano fluid sinuous in a horizontal double-tube counter flow heat exchanger under turbulent current situations are investigated. The Degussa P25 TiO2 nanoparticles of about 21 nm diameter are used in the extant study. The outcomes manifestation that the convective heat transfer coefficient of nano fluid is marginally higher than that of the sordid liquid by about 6–11%.

M.M. Sarafraz et. al [4] In the contemporary work, an experimental study was focussed to tally the heat transfer coefficient of a liquid metal blend encompassing gallium, indium and tin (Ga-In-Sn) under dissimilar heat fluxes esoteric a compact heat exchanger invigorated with rectangular micro-passages. The microchannel was contrived from Cu/Zn alloy by means of computer numerical control machining (CNC) to bring out a reasonable heat transfer. The experiments were presented at 200–350 _C and for some random motion mass current rateof 0.1–1.5 gr/s .Pressure globule and also temperature silhouette lengthwise with the length of the microchannel were recurrently restrained and it was recognised that Ga-In-Sn eutectic had a credible thermal retort at temperatures >200 _C. Also, the pressure drip value diminished with an upsurge in the temperature of the structure

. Agarwal and Rao [5] experimentally explored thermal and frictional features of oil current in a double-pipe heat exchanger encompassing warped tape supplements with constant wall temperature. Their outcomes show that friction factor can be amplified by 3.13–9.71 times linked to straight pipes, while Nusselt number are raised by 2.28–5.35 and 1.21–3.70 times with respect to straight pipes at relentless current rateand power, respectively.

Tan et al. [6] used warped oval tubes in its place of artless circular pipes to produce a shell-and-tube heat exchanger. Conferring to their fallouts, the pressure drop. Sun and Zhang [19] conducted a numerical study on the impact of large vortices on thermal and flow physiognomies of turbulent airflows in a fin-and-tube heat exchanger. Their upshots expose that heat transfer coefficient in bigger pipes is upper than that in minor pipes by 20–30%.

Pethkool et al. [7] used a crenelated pipe as the central pipe of a double-pipe heat exchanger to scrutinise its impressions on heat transfer rate and pressure drop. They found that substantial growths in heat transfer and pressure drop are gained with the upsurges in penetration and torsional phase of the ridged pipe.

Sadighi et al. [8] used ridged pipes as together inner and outer pipes and studied their effects on heat transfer rate and pressure drop. Their grades indicate that the persuaded turbulence perceptibly increases heat transfer rate and pressure drop. Generally, these lessons clearly validate that changing geometrical formation is an effective and viable way to upsurge thermal performance of double-pipe heat exchangers. Nevertheless employment of complex alteration such as introduction of warped tape inserts and ridged pipes as deliberated in the above-mentioned literature shows that heat transfer rate can be improved, the trade-off is that pressure drop is also dramatically increased, which requires much higher power consumption.

R. Bhadouriya et. al [9] have implied that the flaccid way to amend geometry of inner duct is a conceivable choice for this perseverance. For example, linked to a traditional annulus, a twisted square channel demonstrates much better presentation under the same trifling conditions. Also, introduction of a fin tube as an inner pipe and increasing its ellipticity can boost heat transfer rate extraordinarily. Amid accessible candidates for geometry of internal pipes, flat figure grants a viable while simple disparity of the circular equivalent, where much thoughtfulness can be paid to optimize the solution set.

P.K. Swamee et al. [10] this study is oriented to close this gap. In this study, computational fluid dynamics (CFD) simulation was conducted to investigate the effects of flat inner pipes on performance of double-pipe heat exchangers. Compared with experimental and theoretical approaches, CFD can clarify the complex flow and heat transfer patterns inside double-pipe heat exchangers with much more details but rather reduced cost. Actually, CFD has been largely employed in scientific investigations and engineering applications of double-pipe heat exchangers.

A. Erek, B et al. [11] in this study plate and tube heat exchanger is used to observe the effect of fins on the heat transfer characteristic and pressure drop characteristics of the domain. For the purpose CFD simulation was performed and it was observed that fins also has significant effect on the pressure drop characteristics. Fin height has increased the pressure drop and enhanced the heat transfer value. A lesser tube thickness resulted into greater heat transfer.

K. Sharifi et al. [12]. In the following, geometrical and configuration information of the studied double-pipe heat exchanger with flat inner pipe is first described. Then, conservation equations used in the CFD simulation and associated boundary conditions are briefly discussed. After validating the numerical simulation and confirming grid independence, fluid flow and heat transfer characteristics in each designed configuration are presented and discussed. Finally, effects of geometry of flat inner pipes on pressure drop, temperature distribution, and thermal performance are investigated under a range of Reynolds number.

Do Huu-Quan et al. [13] studied the different cross section arrangement for inner flattend pipe to observe the effect of changing the aspect ratio of the hot fluid carrying pipe on the heat transfer coefficient and average Nu. Effectiveness is also seen to be enhanced with respect to aspect ratio defined in the study. They created an expression for the performance index to account for the both heat transfer and pressure drop characteristics. The current study in an extended study of this literature.

III. OBJECTIVE

The objective of the present work is to simulate a 3 dimensional double pipe heat exchanger with varying inner pipe cross section to see the effect on the thermal characteristics of the flow. In this regard following points to be discussed:

1 The change in heat transfer coefficient of hot fluid with respect to change in aspect ratio and Re of cold fluid.
2 The change in Nu of hot fluid with respect to change in aspect ratio and Re of cold fluid.
3 To validate the present work with the existing literature.
4 To analyze the temperature and pressure contours to see the behavior of the fluid.
5 To observe the effectiveness of heat exchanger with the changing parameters.

IV. METHODOLOGY

Domain description

The 3 dimensional double pipe counter flow arrangement is created in ANSYS design modeler with inner pipe diameter of 25 mm and outer pipe diameter of 40 m. the inner pipe thickness is taken as 1mm and material taken is aluminium. The aspect ratio of cases are taken from Do Huu-Quan et. al. the length of is 500 mm.

The computational fluid dynamics analysis is carried out using ANSYS fluent for our objective study. The response strictures have been taken from the base paper. The prevailing equations such as continuity equation, momentum equation, energy equations, K equation and ε equations are used to perform this computational analysis.

Governing Equations

The equation for preservation of mass,

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

Where S_m= mass included to the continuous phase or any handler sources.

Momentum Conservation Equations

Conservation of momentum in an inertial reference frame is described by

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

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

Energy Equation

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

$k-\in$ model

The turbulence kinetic energy, k, and its rate of dissipation, ∈, are obtained from the following transport equations:

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

and

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

Definition of performance indicators

In this section, several important parameters for definition of ther - mal and hydrodynamic performances inside the heat exchanger are defined. Overall heat transfer rates of the hot fluid flow in the inner pipe and the cold fluid flow in the annulus are

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

where $\dot{m}$ is inlet mass current ratefor the hot fluid, C_{ph} is specific heat capacity of the hot fluid, $\dot{m}$ is inlet mass current ratefor the cold fluid, and C_{pc} is specific heat capacity of the cold fluid. In addition, T_{in} and T_{ouf} show temperatures at inlet and outlet. Therefore, mean heat transfer rate is defined as

$Q = (Q_h + Q_c)/2$ (7) Ratio between actual and maximum achievable heat transfer rates represents effectiveness, which is calculated as

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

where $Q_{m\alpha x} = (\dot{m}C_p)_{mox}(T_{hin} - T_{cin})$

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

Mean convective heat transfer coefficients on the inner pipe and annulus can be obtained as

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

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

where $\overline{h}$ represents mean convective heat transfer coefficient in the inner pipe, $\overline{T}$ is mean temperature of the inner pipe wall, $\overline{q}$" is mean thermal flux on the inner pipe wall, and $\overline{h}$ is mean convective heat transfer coefficient in the annulus. Thus, mean Nusselt numbers in the inner pipe and annulus are

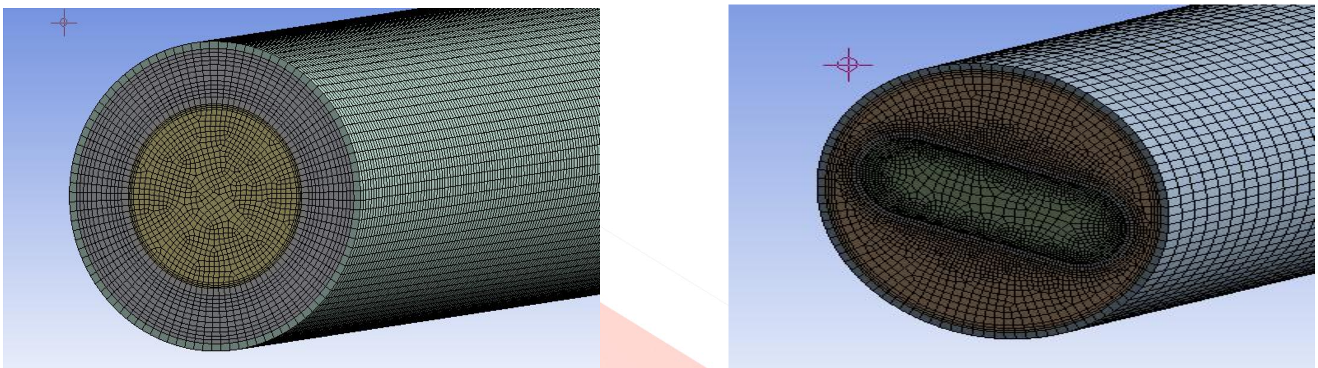
$$\overline{Nu_{pipe}} = \frac{\overline{h_{pipe}} D_{hpipe}}{k_c}$$

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

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

Friction coefficient is evaluated using pressure difference at the inlet and outlet on each side.

Meshing Details:



(a) (b)

Fig. 1 Meshing of (a) case 1 and (b) case 4

The total number of nodes and cells for case 4 is 958569 and 844400 respectively and for case 1 it is 712840 and 622250 respectively. The average orthogonal quality for the case 1 is 0.99 which is best for capturing the accurate result. Mesh elements are hexagonal thus enhancing the calculation speed.

Boundary conditions:

The fluid taken in the study is water with water with these properties: $\rho = 1000 kg/m^3$, $v = 1\times 10^{-3} m^2/s$, $c_p = 4182$J/kg-K. hot water inlet temperature is 333K and cold inlet temperature is 293K. the turbulence model is taken as standard k-∈ model with enhanced wall treatment to account for the boundary layer effect. Both inlet is taken as velocity inlet and outlet as pressure outlet. Inner pipe walls are coupled and outer wall is given 0 heat flux to make it insulated wall.

V. RESULT AND DISCUSSION

Re of the cold fluid is changed from 2000 to 11000 while hot fluid Re is kept constant at 7000.the base case for validation is taken for all aspect ratio and Re of hot fluid is varied as in previous literature. The case is validated with Do Huu-Quan et. al. and found to be well matching and a maximum error of 9.82% is observed for heat transfer coefficient. Following chart is plotted for comparision of the case with existing literature. Solution control method was taken as SIMPLEC to resolve the equation faster.

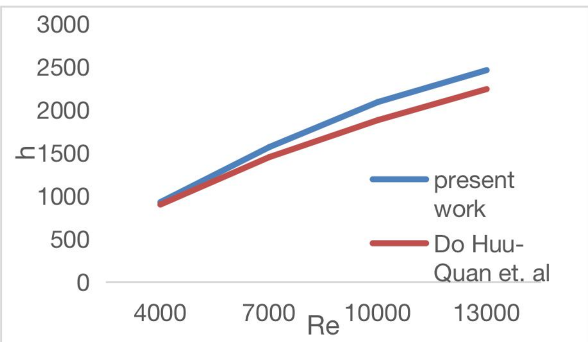


Fig. 1 Comparision of mean heat transfer coefficient with existing data

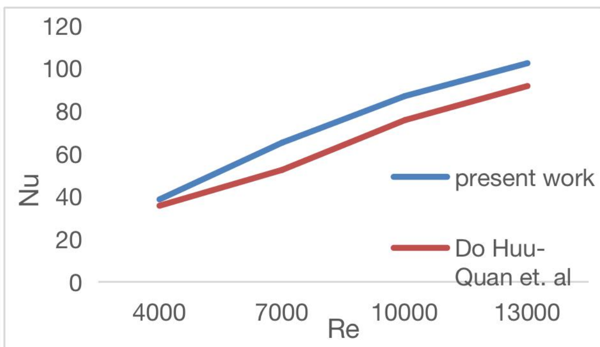
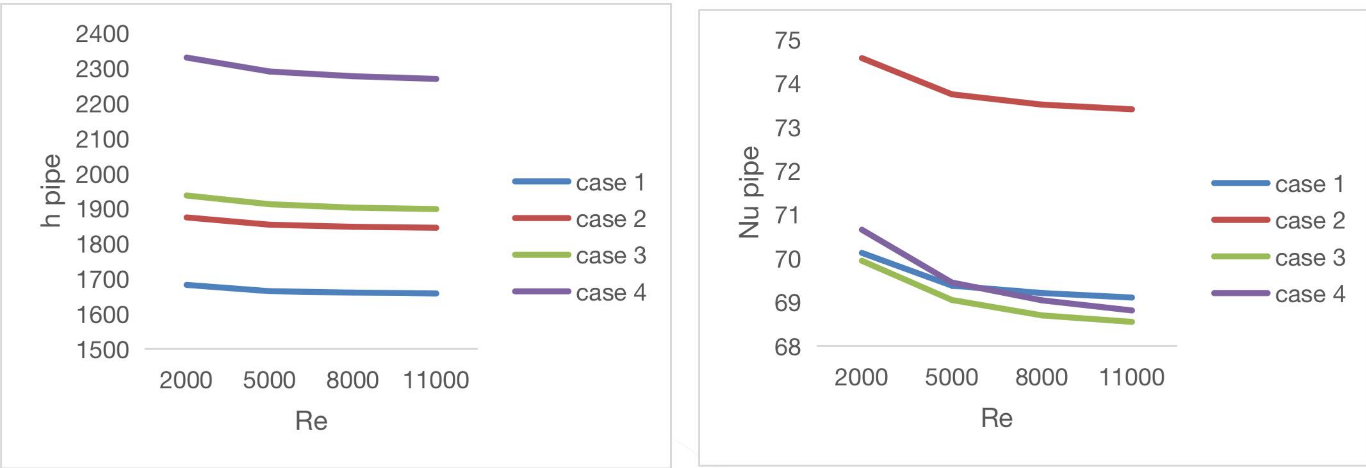


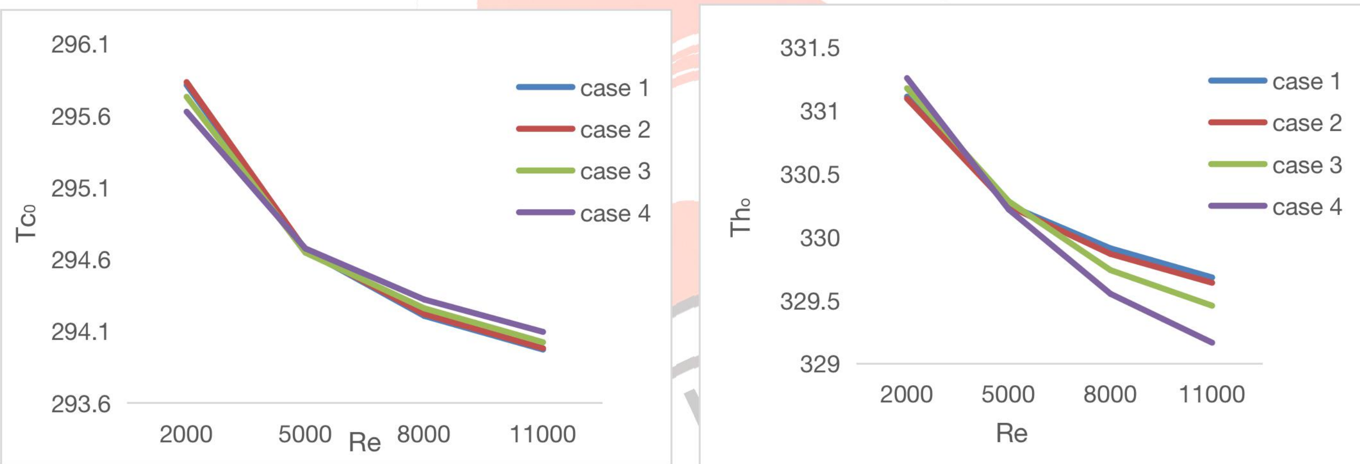
Fig. 2 Comparision of Nu with existing data

For modification in cases aspect ratio of 0.72, 0.53, and 0.37 are taken for case 2, 3 and 4 respectively. Case 1 is concentric double pipe arrangement with counter flow characteristics taken from previous literature. After analyzing the mean heat transfer coefficient and Nu following chart is plotted. it is expected that Nu for case 2 will be maximum in this study because the multiplication terms hydraulic diameter and heat transfer coefficient in the expression of Nu is maximum in case 2. The following trend of chart was plotted after analyzing the result. The study showed an optimum value of Nu for case 2. Furthermore the study revealing the trends of the heat transfer value being increasing or decreasing with respect to different aspect. The chart below showing that the heat transfer coefficient and Nu for pipe is not much affected with respect to change in the Re of cold fluid in annulus. But showing an appreciable change with respect to change in aspect ratio.



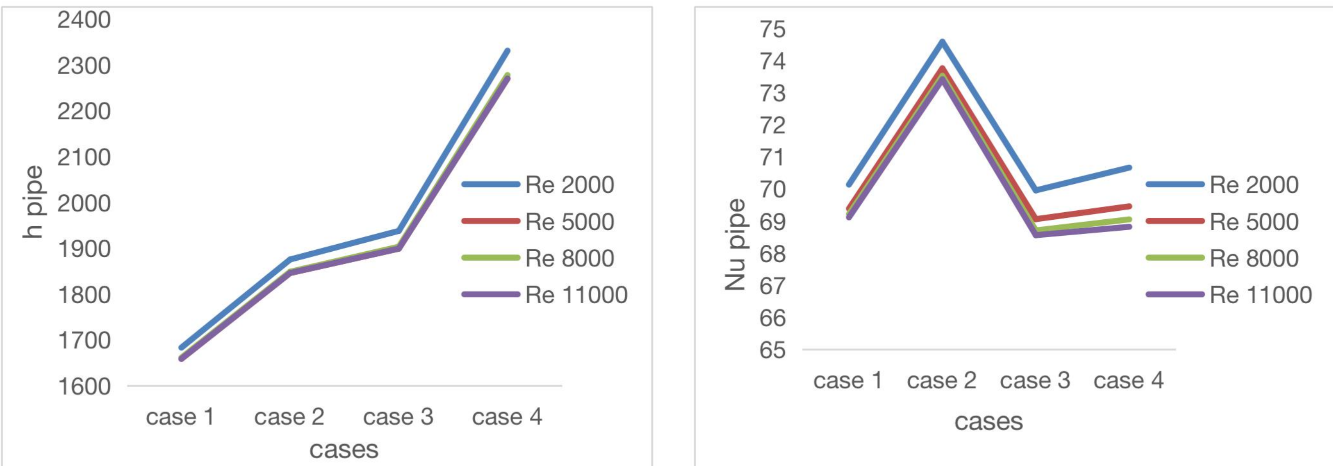
(a) (b)

Fig. 3 variation of (a) mean heat transfer coefficient and (b) Nu for different cases



(a) (b)

Fig. 4 Variation of temperature of (a) cold fluid outlet and (b) hot fluid outlet for different cases



(a) (b)

Fig. 5 Variation of (a) mean heat transfer coefficient and (b) Nu for different Re

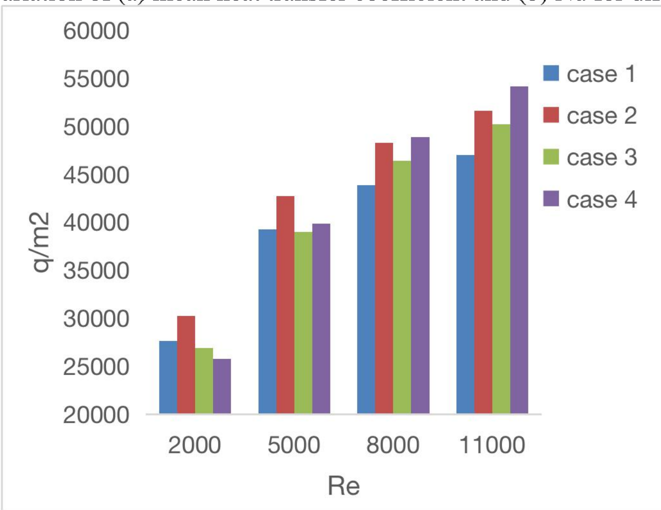


Fig.6 Variation of heat flux for different cases with respect to Re

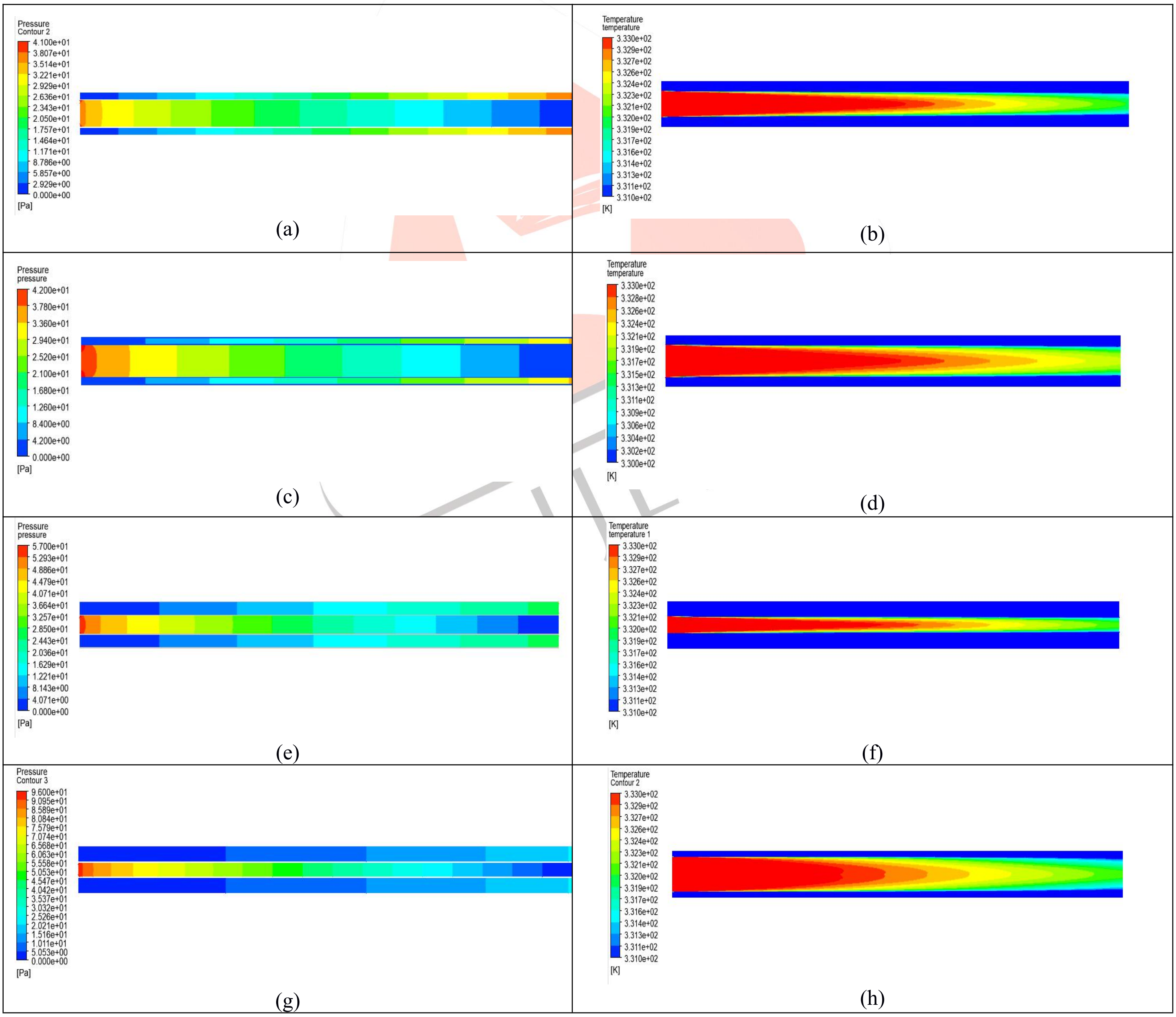


Fig. 7 contours of pressure (a), (c), (e), (g) and temperature (b), (d), (f), (h) for case 1, case 2, case 3 and case 4 respectively at Re 2000

The contours shown in fig are taken at Re of 2000. The contours are showing the flow development and temperature layers being developed under flow conditions.

VI. CONCLUSION

The main objective of the work was to observe the effect of increasing the Reynolds number of the cold fluid on the Nusselt number and heat transfer coefficient of the hot fluid. After analyzing the results and charts following conclusive points are drawn:

1 Moving over from Re 2000 to 11000 heat transfer coefficient for pipe is decreased in the range of 1.45% to 2.6% for all cases showing that there is very less change in heat transfer coefficient of pipe with respect to change in the Re of annulus fluid. On the other hand for constant Re moving over from case 1 to case 4 the change in heat transfer coefficient of pipe is increased by 38.47% for Re 2000, 37.57% for Re 5000, 37.10% for Re 8000 and 36.85% for Re 11000. Thus it is concluded that the heat transfer coefficient is increasing with decrease in aspect ratio and the increment is higher at lower Re.

2 Similar patterns have been observed for particular case with change in Re but Nu variation is altered as it is maximum for case 2. It is expected because the multiplication of heat transfer coefficient and hydraulic diameter in the expression of Nu is maximum for case 2. There is a maximum increase of 6.27% in Nu as compared to the previous case with hot fluid at Re 7000.

3 After observing the heat flux value it is seen that moving over from Re 2000 to 11000 heat flux value is increased by 69.99% for case 1, 70.68% for case 2, 86.92% for case 3 and 110.21% for case 4. It is also seen that percentage increment is increasing as we go toward the lower aspect ratio hence it is concluded that decrease in aspect ratio increases the heat transfer value. For fixed Re moving over the cases it is concluded that first heat flux is increased from case 1 to case 2 then it decreases following the Nu trend.

4 After observing the effectiveness value of the arrangement it is seen that moving over Re from 2000 to 11000 effectiveness is decreased by 56.02% for case 1, 56.33% for case 2, 52.37% for case 3 and 38.19% for case 4. It is also seen that for a fixed aspect ratio effectiveness is better for low Re.

REFERENCES

[1] S. Eiamsa-ard, S. Pethkool, C. Thianpong, P. Promvonge, Turbulent flow heat transfer and pressure loss in a double pipe heat exchanger with louvered strip inserts, Int. Commun. Heat Mass Transfer 35 (2008) 120–129.

[2] M. Sheikholeslami, R.-U. Haq, A. Shafee, Z. Li, Y.G. Elaraki, I. Tlili, Heat transfer simulation of heat storage unit with nanoparticles and fins through a heat exchanger, Int. J. Heat Mass Transf. 135 (2019) 470–478.

[3] W. Duangthongsuk, S. Wongwises, Heat transfer enhancement and pressure drop characteristics of TiO2–water nanofluid in a double-tube counter flow heat exchanger, Int. J. Heat Mass Transf. 52 (2009) 2059–2067.

[4] M.M. Sarafraz, M.R. Safaei, M. Goodarzi, B. Yang, M. Arjomandi, Heat transfer analysis of Ga-In-Sn in a compact heat exchanger equipped with straight micro-passages, Int. J. Heat Mass Transf. 139 (2019) 675–684.

[5] S.K. Agarwal, M. Raja Rao, Heat transfer augmentation for the flow of a viscous liquid in circular tubes using twisted tape inserts, Int. J. Heat Mass Transf. 39 (1996) 3547–3557.

[6] X.-H. Tan, D.-S. Zhu, G.-Y. Zhou, L.-D. Zeng, Heat transfer and pressure drop performance of twisted oval tube heat exchanger, Appl. Therm. Eng. 50 (2013) 374–383.

[7] S. Pethkool, S. Eiamsa-ard, S. Kwankaomeng, P. Promvonge, Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube, Int. Commun. Heat Mass Transfer 38 (2011) 340–347.

[8] 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.

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

[10] P.K. Swamee, N. Aggarwal, V. Aggarwal, Optimum design of double pipe heat exchanger, Int. J. Heat Mass Transf. 51 (2008) 2260–2266.

[11] 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.

[12] 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.

[13] 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