Acta Polytechnica **63**(1):65–74, 2023
https://doi.org/10.14311/AP.2023.63.0065

HEAT TRANSFER CHARACTERISTICS OF DOUBLE PIPE HEAT EXCHANGER HAVING EXTERNALLY ENHANCED INNER PIPE

Balasubramanian Vijayaragavan, Suyambu Pandian Asok, Chandrasekar Ramalekshmi Shakthi Ganesh*

Mepco Schlenk Engineering College, Department of Mechanical Engineering, Sivakasi, Virudhunagar District-626005, Tamilnadu, India

* corresponding author: shakthi.jan25@gmail.com

Abstract. An investigation on enhancement of heat transfer is carried out for a double pipe heat exchanger in which the outer wall of the inner pipe is provided with circumferential labyrinth passages. Rectangular and triangular cavities with fixed labyrinth tooth thickness, height, and pitch are considered and the effect of added labyrinth structures on the heat transfer characteristics is discussed. A two-dimensional steady numerical simulation is carried out using ANSYS-FLUENT software. The flow Reynolds number equals to 20 000 and 43 000 for the hot and cold fluids, respectively, while other fluid properties are constant. From the numerical analysis carried out in this work, it is identified that the added labyrinth passages in the heat exchange surface improve the heat transfer rate and can reduce the length of the heat exchanger. Numerical predictions agree well with the results obtained from the experiment conducted.

Keywords: Double pipe heat exchanger, heat transfer, numerical analysis, rectangular cavity.

1. Introduction

Heat exchange between hot and cold fluid plays a significant role in many applications. Heat transfer enhancement techniques implemented in heat exchangers reduce the running costs of the device by saving energy. Out of the various types of heat exchangers available, the double pipe heat exchanger shown in Figure 1 is a simple design with two concentric pipes, whose performance can be substantially improved by various enhancements. It is also suitable for high temperature and high pressure applications due to its small diameters. The literatures regarding these enhancements are primarily experimental in nature, with gases as working fluids [1, 2].

The heat transfer enhancement techniques in a double pipe heat exchanger are classified into passive and active techniques. Passive methods can work without any external aids by a geometric modification such as varying the cross section of the inner pipe [2–8]. Active methods require external stimulation such as vortex creation by rotating the inner pipe [9]. The helically corrugated inner pipe with alternating ridges and grooves in a double pipe heat exchanger was investigated using steady numerical simulations [2]. The new design improved the heat transfer by a factor of three as compared to the smooth-walled inner pipe.

The heat transfer characteristics of blossom shaped fins on the inner tube were investigated experimentally and numerically at a constant air inlet temperature [3]. The increase in the number of fins resulted in a more uniform distribution of the temperature field. The reasons for the performance improvement of the double pipe heat exchanger with helical fins and vortex generator were found to be the increase in heat transfer surface, reduction of hydraulic diameter, secondary flow caused by the helical channel and the vortices, destabilisation of fluid flow and intensification of turbulence induced by the vortex generators [4]. The inner twisted square duct with air as working fluid showed a considerable enhancement in the heat transfer in both the laminar and turbulent regimes [5]. Experimental investigations of a double pipe helical heat exchanger with a copper fin in the annulus section showed an improvement in the overall heat transfer coefficient calculated by replacing the hydraulic diameter with an equivalent diameter [6].

In the present study, an improvement in the performance of a double pipe heat exchanger is achieved by incorporating a labyrinth path in the annular region, which enhances the surface area of the inner tube externally. Thus, the labyrinth structure increases the surface area of heat transfer. During the flow through this enhanced structure, the fluid is made to pass through cavities and the corresponding land, resulting in vortex generation in each cavity. The recirculation caused by the vortex increases the heat transfer rate of the hot fluid by increasing the number of restrictions [10]. The labyrinth structure introduced in the outer surface of the inner pipe is shown in Figure 2.

In the conventional double pipe heat exchangers, the annular area is kept larger to allow the cold fluid flow to gain heat from the hot fluid flowing in the inner pipe. The innovation in the present study is the improvement in heat transfer by enhancing the annular flow path with the annular flow area much lesser than the pipe flow area and leaving the pipe flow undisturbed. Even a small amount of increase in

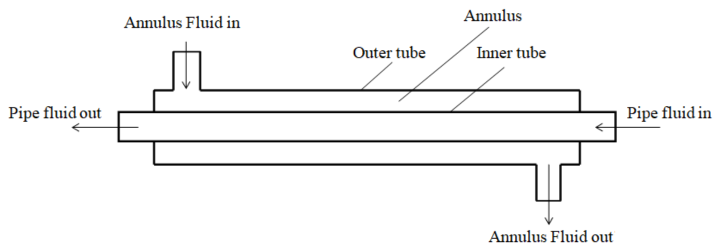

Figure 1. Schematic of the double pipe heat exchanger.

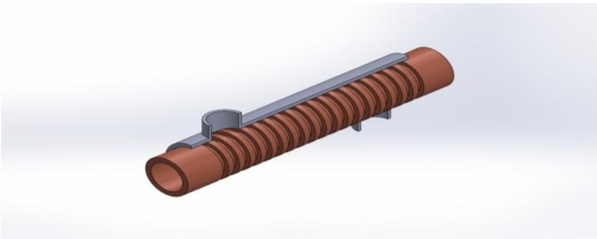

Figure 2. Heat exchanger with labyrinth.

the rate of the heat transfer by surface modification can result in a considerable improvement in process efficiency and economy. The dimensions of the heat exchanger before modification are given in Figure 3.

By incorporating the enhanced structure on the outer surface of the inner tube, the heat transfer by the fluid flowing in the annular region is improved [11–13]. The enhancement with a triangular geometry can provide a stable flow recirculation in the cavity [14]. Hence, the triangular cavity can be a good option for heat transfer studies in addition to the rectangular cavity, which is popularly used. The details of the rectangular and triangular cavity structures incorporated in the outer surface of the inner tube are shown in Figure 4 and Figure 5, respectively. The surface enhancers efficiently augment the heat transfer rate due to the induced turbulence intensity [15]. Heat transfer decreases with the increase in the thickness and pitch of the external surface enhancers to be introduced in the heat exchangers [11]. Hence, the thickness and pitch of the tooth surface to be machined are fixed as 4 mm and 8 mm, respectively. In the present work, water is selected as the working fluid and a two-dimensional numerical analysis is carried out. The investigation on the heat transfer characteristics of the double pipe heat exchanger is carried out both numerically and experimentally with Reynolds number values up to 43 000.

2. Numerical simulation

CFD software is largely used to study the fluid flow and heat transfer by solving the mathematical equations with the help of a numerical analysis. It is equally helpful in designing a heat exchanger system as well as in troubleshooting by suggesting design modifications [16]. The two-dimensional geometric model of the double pipe heat exchanger is developed in SOLIDWORKS designing tool and imported into the ANSYS FLUENT in IGS format. Due to the symmetric nature of the heat exchanger, only the top half of the heat exchanger is modelled, and an axisymmetric condition is applied in the ANSYS FLUENT solver. The existing problem is thus simulated by using a two-dimensional heat transfer analysis [17–19]. Triangle elements are applied for all the three regions. The axisymmetric heat exchanger models for plain profile, profile with rectangular, and triangular labyrinths, whose meshes are shown in Figure 6, 7 and 8, respectively, are developed. Region 1 represents the hot fluid domain; Region 2 represents the copper solid domain and Region 3 represents the cold fluid domain. The body sizing and refinement conditions differ for each region.

Zero heat flux boundary condition is applied to the outer wall and solid side walls, assuming to be perfectly insulated [10]. The solid fluid interfaces are no-slip walls where the heat flux and the temperature are continuous across the boundary line [1, 3]. A grid independent test was conducted to check the dependence of the solution on the quality of the mesh. Pressure-based solver and k-epsilon turbulence model are chosen for the study [17, 20]. The k-epsilon models are categorised in two equation models in which both the turbulent length and the time scale are determined by solving two separate transport equations. Out of the three different k-epsilon models in ANSYS FLUENT, the standard k-epsilon model is the most commonly used model for simulating a turbulent flow [21]. The inlet temperature and mass flow rate of the cold and hot fluids are specified. At the outlet, the mass flow outlet condition is specified for both the hot and cold fluids. Table 1 gives the details of the solution methods used. An important aspect in the numerical analysis is to arrive at a convergence of the solution. The convergence criterion for continuity, velocity, and k-epsilon equations are reduced to 0.001, and for the energy equation, the value is set to 0.0001.

The differential equations constituting the mathematical model for the physical problem under study is given below.

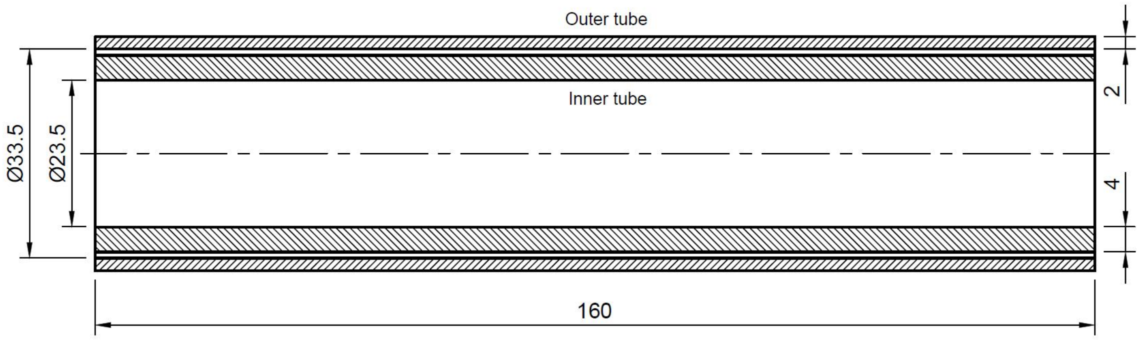

FIGURE 3. Dimensions of the plain heat exchanger.

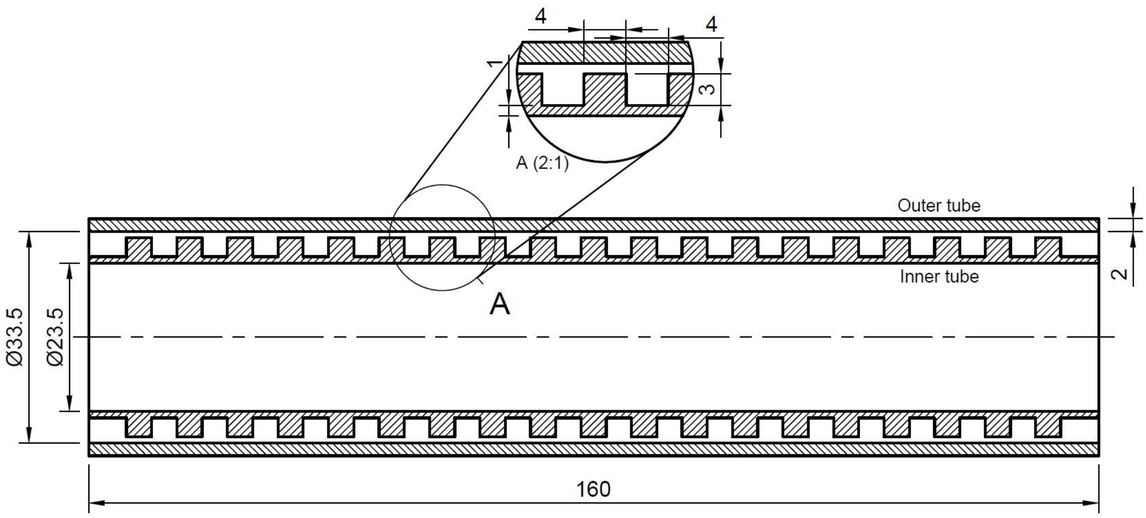

FIGURE 4. Dimensions of rectangular labyrinth heat exchanger.

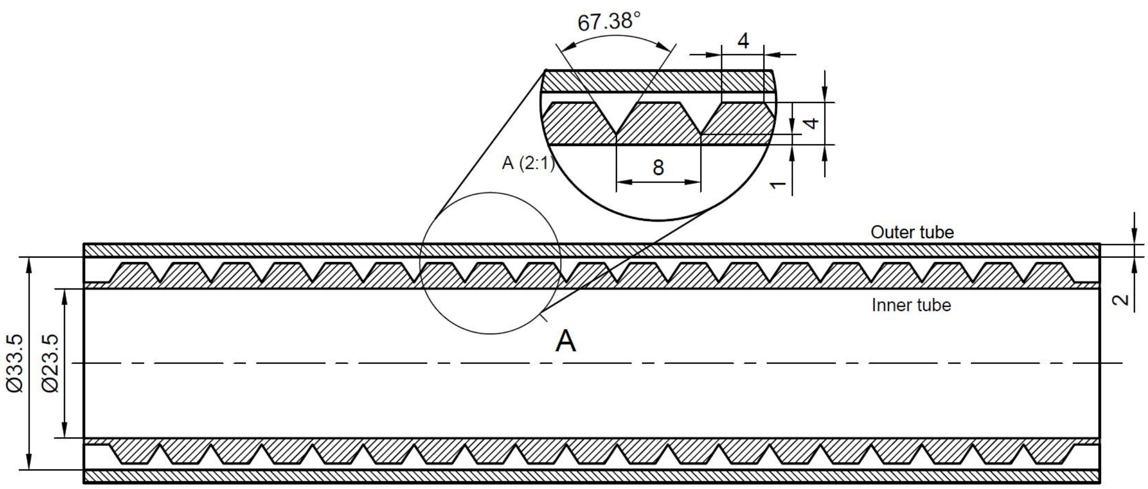

FIGURE 5. Dimensions of the triangular labyrinth heat exchanger.

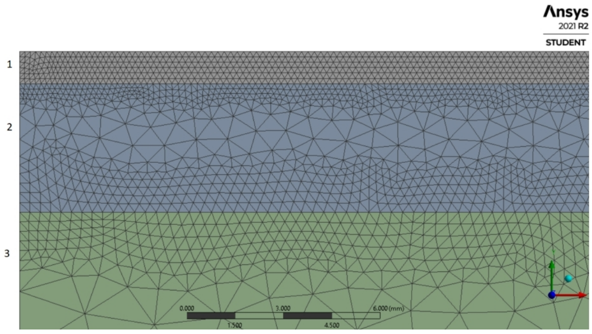

FIGURE 6. Mesh picture for plain heat exchanger model.

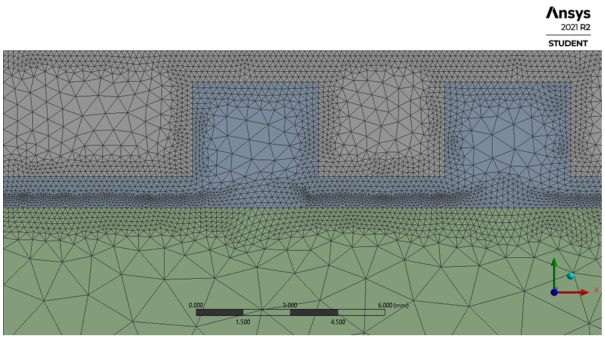

FIGURE 7. Mesh picture for heat exchanger with rectangular labyrinth.

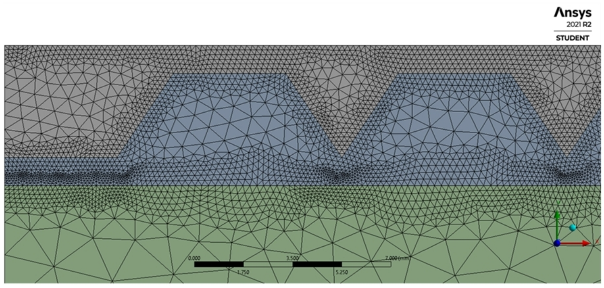

FIGURE 8. Mesh picture for heat exchanger with triangular labyrinth.

Item	Condition Taken
Solution Method	Pressure-Velocity Coupling
Scheme	Coupled
Gradient	Least squares cell based
Pressure	Second order
Momentum	Second order upwind
Turbulent kinetic energy	First order upwind
Specific dissipation rate	First order upwind
Energy	Second order upwind

TABLE 1. Details of the solution method.

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0. \tag{1}$$

Conservation of momentum:

$$\frac{\partial(\rho u_j)}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} = \rho f_i + \frac{\partial \sigma_{ji}}{\partial x_j}. \tag{2}$$

Conservation of energy:

$$\frac{\partial(\rho e)}{\partial t} + \frac{\partial(\rho e u_j)}{\partial x_j} = -\rho f_i u_i + \frac{\partial(\sigma_{ji} u_i)}{\partial x_j} - \frac{\partial q_j}{\partial x_j} \tag{3}$$

The established mathematical model is solved by a finite volume method where the computational domain is divided into small volumes, and then the discrete algebraic equations of the above model are solved by iteration. The residuals of the velocity, energy, and k-epsilon equations are small and a sample residual convergence plot for the plain double pipe heat exchanger is shown in Figure 9.

3. Equations used in theoretical calculation

Reynolds number is given by the relation,

$$Re = \frac{u D_h}{\Upsilon} \tag{4}$$

and the hydraulic diameter D_h is given by the difference between the inner diameter of the outer tube and the outer diameter of the inner tube [4]. For the plain profile, Nusselt number is given by

$$Nu = \frac{h D_h}{k} \tag{5}$$

and the heat transfer coefficient

$$h = \frac{Q}{(A \cdot \Delta T_{hot})}. \tag{6}$$

For the tooth profile, Nusselt number [3] is given by

$$Nu = 0.0405 Re^{0.608} \frac{H_f}{D_h}^{-1.03} \frac{W_f}{D_h}^{-0.908}. \tag{7}$$

The Nusselt number of the plain model is in the range of 0.5, whereas the Nusselt number of the tooth model is about 6 times larger. The lower Nusselt number for the plain model indicates a laminar flow with the domination by conduction whereas the higher Nusselt number for the tooth model indicates a transition zone in the flow path due to the presence of alternating teeth and cavities. Logarithmic mean (LMTD) of the temperature difference of the inlet and outlet is calculated both for the CFD and experimental values. The heat transferred based on the hot and cold fluids is calculated and the average heat transfer rate is used for the calculation of the overall heat transfer coefficient [6, 16]. The length of the model is kept constant for plain, rectangular, and triangular profiles. The overall heat transfer coefficient is calculated based on the temperature data, and flow rates based on the following equation [22].

$$U = \frac{q}{(A \cdot LMTD)} \tag{8}$$

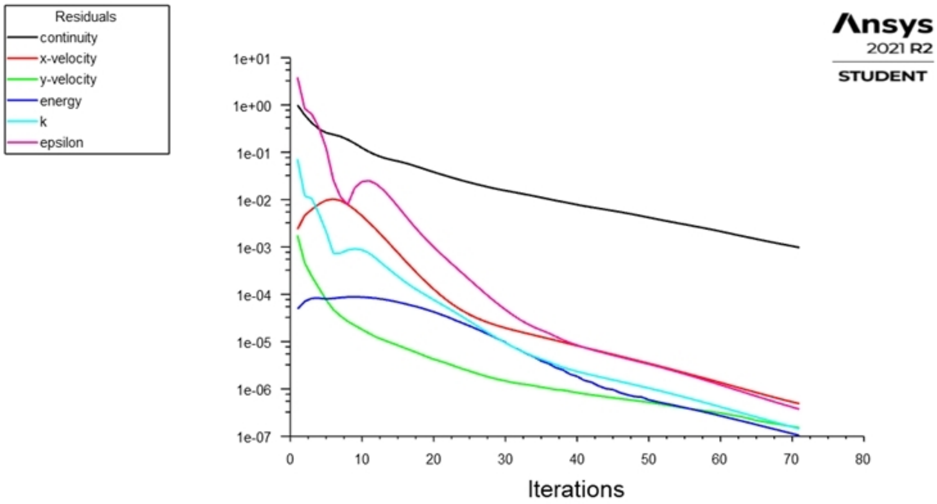

Figure 9. Convergence plot for plain heat exchanger.

S.No.	Specifications	Dimension [mm]
1	Inner diameter of the inner tube (d)	23.5
2	Thickness of the inner tube (ti)	4
3	Inner diameter of the outer tube (D)	33.5
4	Thickness of the outer tube (to)	2
5	Length of the heat exchanger (L)	160

Table 2. Dimensions of the heat exchanger tubes.

$$LMTD = \frac{(\Delta T_2 - \Delta T_1)}{\ln\left(\frac{\Delta T_2}{\Delta T_1}\right)} \tag{9}$$

4. Experiment details

Experiments are conducted for the parallel flow arrangement maintaining the mass flow rate of hot water at $0.4\,\mathrm{kg\,s^{-1}}$ in the annular region and the mass flow rate of cold water at $0.8\,\mathrm{kg\,s^{-1}}$ in the inner pipe. This corresponds to a Reynolds number of 20 000 and 43 000, respectively, when calculated based on the inlet velocity and the tube diameter [1]. The experiment setup consists of an inner tube made of copper where cold water is pumped from a storage tank. The outer tube is made of galvanized iron (GI) which has provision for hot water entry and exit. The hot water flows through the narrow annular gap between the inner copper pipe and the outer GI pipe. The dimensions of the heat exchanger tubes are stated in Table 2.

Temperatures are measured at the entry and exit points of hot and cold fluids with the help of sensors and their output signals are fed to the system through Arduino. The flow rates of the hot and cold fluids are measured by water flow sensors and their output signals are also logged by Arduino codes. The inlet temperature of the hot fluid is maintained at 350 K and the inlet temperature of the cold fluid is maintained at 295 K. The final assembly of the double pipe heat exchanger is shown in Figure 10.

Figure 10. Assembly of the inner and outer pipe.

Applications with low mass flow rates give rise to a mixed condition in which the effect of forced convection and free convection are of an equal importance [23]. The flow rate in the annular region is kept small in order to initiate the mixed convection to achieve a better heat transfer. As the rectangular cavities show a better performance in the numerical analysis, the annular flow path in the heat exchanger is modified by machining rectangular grooves in the inner copper tube to investigate the improvement in the heat transfer coefficient [6]. The outer surface of the inner tube becomes geometrically unique with alternating ridges and grooves. The experiments are repeated with the same conditions for the plain profile and the rectangular cavity profile, which are shown in Figure 11 and Figure 12, respectively.

Figure 11. Outer surface of the inner pipe with plain profile.

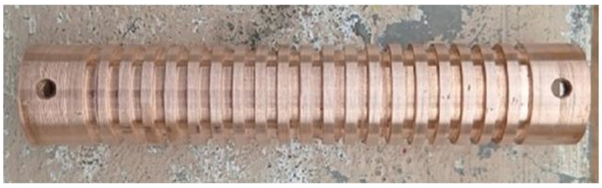

Figure 12. Outer surface of the inner pipe with rectangular profile.

5. Results and discussion

The heat exchanger performance can be improved by making the surface area larger, which is possible by increasing the diameter or the length of the heat exchanger. In this work, the heat exchanger length is increased and the effect on heat transfer characteristics is analysed. The initial condition considered is the hot fluid in the annulus at $0.4\,\mathrm{kg\,s^{-1}}$ and cold fluid in the inner pipe at $0.8\,\mathrm{kg\,s^{-1}}$. At the reduced length of $0.16\,\mathrm{m}$, the temperature drop of the hot fluid is equal to the temperature rise of the cold fluid (Figure 13). The length is increased to $0.5\,\mathrm{m}$ and then to $1.5\,\mathrm{m}$. When the length of the heat exchanger increases, the temperature drop of the hot fluid is higher than the temperature rise of the cold fluid though the flow rate of the cold fluid is higher.

The annular fluid is now changed to cold at $0.8\,\mathrm{kg\,s^{-1}}$ flow rate and pipe fluid to hot at $0.4\,\mathrm{kg\,s^{-1}}$ (Figure 14). Even now, the temperature drop of the hot fluid is higher than the temperature rise of the cold fluid, and the difference further increases with increased length.

When the flow rate of the hot and cold fluids is interchanged, the temperature rise of the cold fluid is higher than the temperature drop of the hot fluid (Figure 15).

The above results confirm that the heat exchange by the annular fluid is higher when the annular flow area is reduced in the manifold as compared to the inner pipe area. Hence, the annular fluid is fixed as hot water with a flow rate of $0.4\,\mathrm{kg\,s^{-1}}$ and the pipe fluid is fixed as cold water with a flow rate of $0.8\,\mathrm{kg\,s^{-1}}$ with the length of the heat exchanger fixed at $0.16\,\mathrm{m}$ to achieve a better heat transfer.

The calculation of the overall heat transfer coefficient is based on the inner surface of the tube, since the calculation based on the outer tube area may provide biased results [6]. The comparison of the overall heat transfer coefficient obtained from the numerical analysis is shown in Figure 16. The overall heat transfer coefficient of the rectangular labyrinth profile is twice than that of the plain profile and the value for the triangular labyrinth profile is in between the rectangular labyrinth and plain models. Hence, only the plain and rectangular labyrinth profiles are considered for experimental testing. The temperature plot for the plain and rectangular profiles is shown in the Figures 17 and 18, respectively.

Three replicates of testing are carried out for both the plain and rectangular cavity configurations. Temperature data are recorded every 10 seconds and the temperature used for the comparison is taken at a stable condition. Initially, the plain copper tube is placed inside the outer GI pipe and the temperature sensors are mounted on the inlets and outlets of the hot and cold pipes. One flow sensor is connected for each hot and cold pipes and the output signals of all the sensors are connected to the Arduino module. The experiment results support the findings from the numerical analysis that the tooth with a rectangular cavity labyrinth profile is having better heat transfer characteristics as compared to the plain profile. Figure 19 provides the comparison of numerical and experiment results of the plain and rectangular labyrinth heat exchangers.

Though the overall heat transfer coefficient is underestimated by the CFD analysis, this approach remains non-detrimental as the experimental testing shows a better exchanger performance than predicted [24]. When compared to the conventional heat exchanger with a finned inner tube, a significant reduction in pressure drop is achieved without affecting the heat exchanger performance. This result indicates that, for the double pipe heat exchanger model, it is an innovative way to provide vortex generators through labyrinth structures for the improvement of heat transfer. In other words, using the heat transfer enhancement technique presented in this paper, the modified double pipe heat exchanger can be a good choice for a situation that requires large quantity of heat to be removed in a limited heat exchanger volume.

6. Conclusion

Increased heat exchange can be achieved by increasing the heat transfer surface area, increasing the heat transfer coefficient, or creating turbulence in the flow of hot fluids. The increase in the exchange surface area by increasing the heat exchanger size directly increases the cost of the fabrication. The increase in the heat transfer coefficient requires increase in the flow velocity and Reynolds number, which needs modification in the process parameters. Hence the improvement is achieved by modifying the outer surface of the inner tube by creating turbulence in the flow. In the present work, three different lengths ($0.16\,\mathrm{m}$, $0.5\,\mathrm{m}$ and $1.5\,\mathrm{m}$) are considered for CFD analysis in order to understand the effect of length on the heat exchange by annular fluid. Experiments are conducted for hot fluid mass flow rate of $0.4\,\mathrm{kg\,s^{-1}}$ in the annular region, and cold fluid mass flow rate of $0.8\,\mathrm{kg\,s^{-1}}$ in the inner

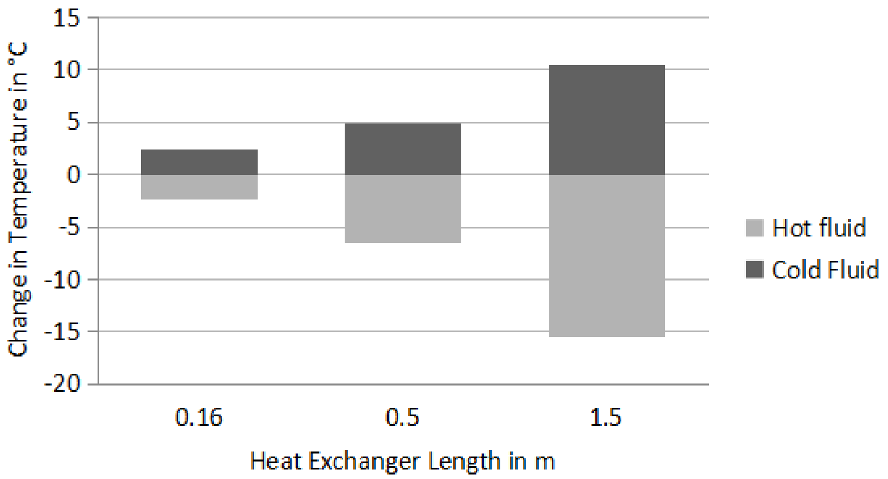

FIGURE 13. Effect of increasing the heat exchanger length (Hot fluid in annulus at $0.4\,\mathrm{kg\,s^{-1}}$).

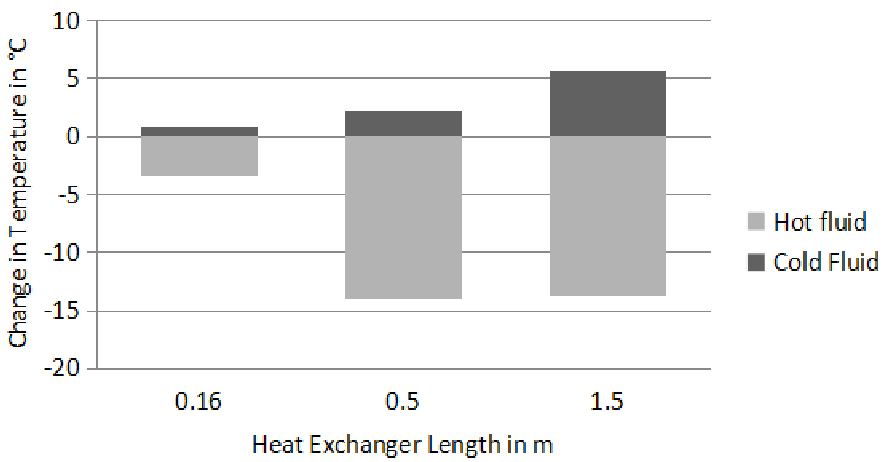

FIGURE 14. Effect of increasing the heat exchanger length (Hot fluid in inner pipe at $0.4\,\mathrm{kg\,s^{-1}}$).

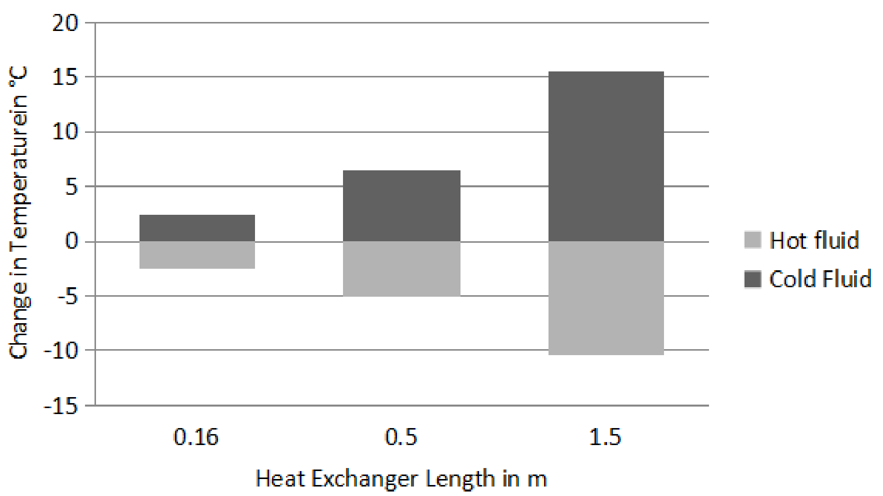

FIGURE 15. Effect of increasing the heat exchanger length (Hot fluid in inner pipe at $0.8\,\mathrm{kg\,s^{-1}}$).

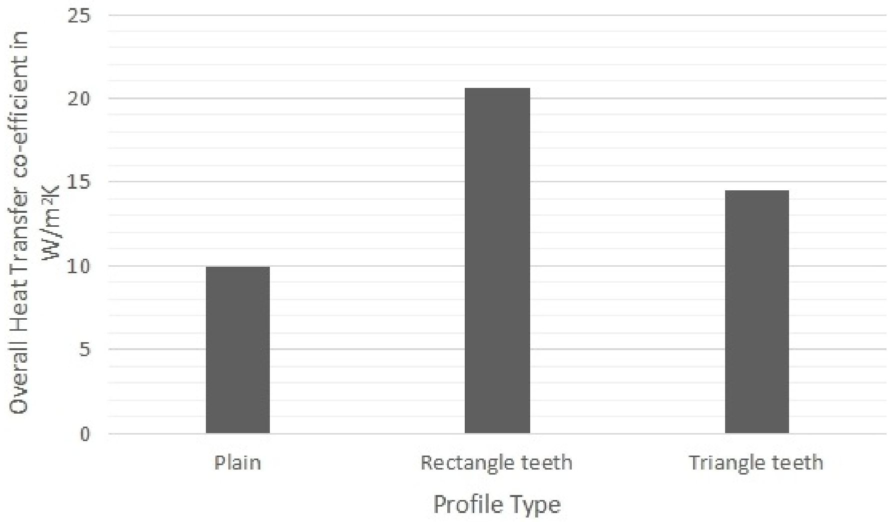

FIGURE 16. Comparison of overall heat transfer coefficient for CFD results.

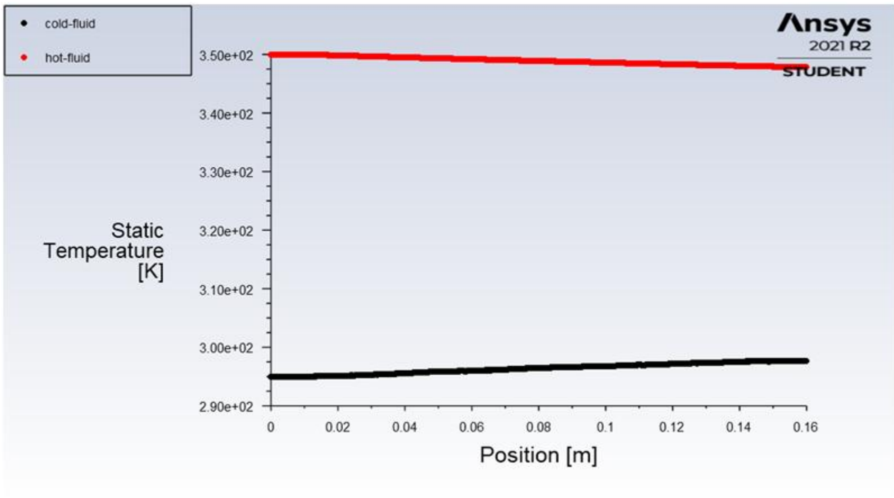

FIGURE 17. Temperature plot for plain heat exchanger.

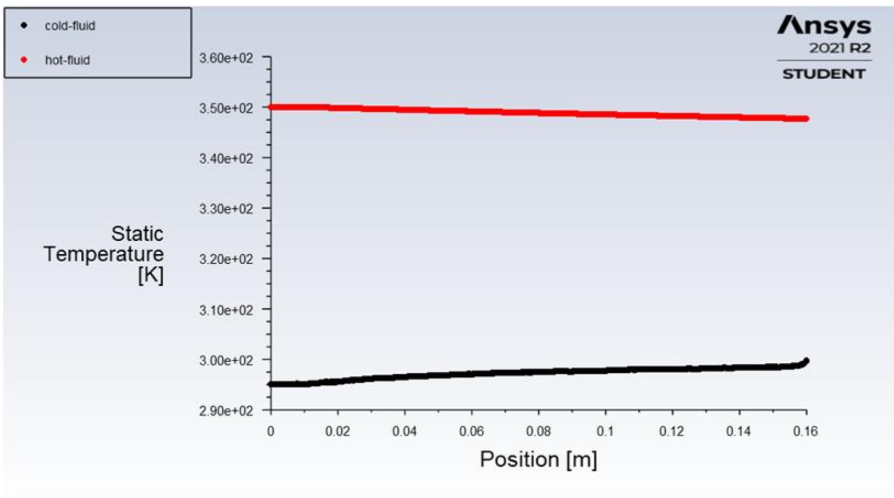

FIGURE 18. Temperature plot for rectangular labyrinth heat exchanger.

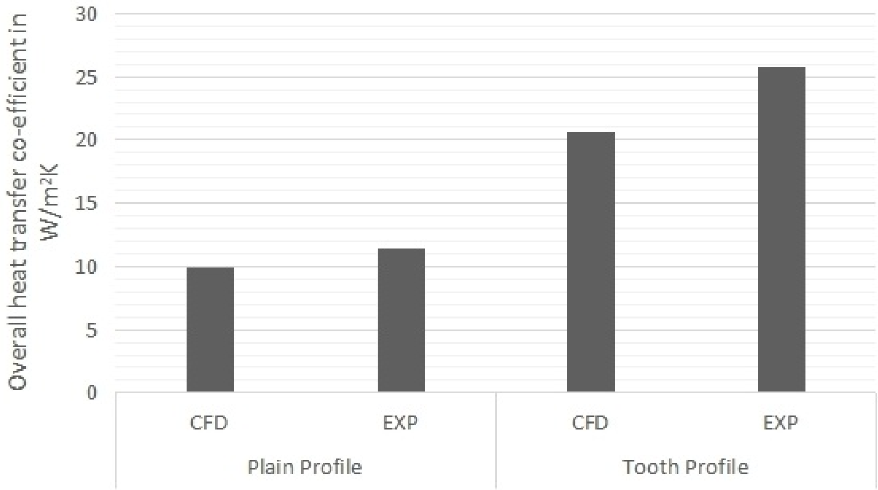

FIGURE 19. Comparison of experiment and numerical results.

pipe, with a constant length of 0.16 m. It is observed that heat transfer rate increases with reduced annular flow area. This effect is predominant when the flow rate of the pipe fluid is higher than the flow rate of the annular fluid and when the length of the heat exchanger increases. The heat transfer further improves when the plain annular flow path is replaced with labyrinth structures and rectangular cavity labyrinths are found to be better performing than the triangular cavity labyrinth. A two-dimensional axisymmetric model was analysed using ANSYS FLUENT software and the results are comparable with the experiment results.

List of symbols

značka popis značky [jednotka]
ρ density [kg m^{-3}]
u velocity [m s^{-1}]
D_h hydraulic diameter [m]
Υ kinematic viscosity [m^2 s^{-1}]
h heat transfer co-efficient [W m^{-2}K^{-1}]
k thermal conductivity [W m^{-1}K^{-1}]
Q heat transferred [W]
A heat exchanger area [m^2]
H_f height of the tooth [m]
W_f pitch of the tooth [m]
U overall heat transfer coefficient [W m^{-2}K^{-1}]
T temperature [K]
Re Reynolds number (Dimensionless)
Nu Nusselt number (Dimensionless)
ΔT_1 difference between hot and cold fluid temperatures at entry
ΔT_2 difference between hot and cold fluid temperatures at exit
$LMTD$ Logarithmic Mean Temperature Difference

References

[1] A. Lemouedda, A. Schmid, E. Franz, et al. Numerical investigations for the optimization of serrated finned-tube heat exchangers. *Applied Thermal Engineering* **31**(8-9):1393–1401, 2011. https://doi.org/10.1016/j.applthermaleng.2010.12.035

[2] J. M. Gorman, K. R. Krautbauer, E. M. Sparrow. Thermal and fluid flow first-principles numerical design of an enhanced double pipe heat exchanger. *Applied Thermal Engineering* **107**:194–206, 2016. https://doi.org/10.1016/j.applthermaleng.2016.06.134

[3] L. Duan, X. Ling, H. Peng. Flow and heat transfer characteristics of a double-tube structure internal finned tube with blossom shape internal fins. *Applied Thermal Engineering* **128**:1102–1115, 2018. https://doi.org/10.1016/j.applthermaleng.2017.09.026

[4] L. Zhang, H. Guo, J. Wu, W. Du. Compound heat transfer enhancement for shell side of double-pipe heat exchanger by helical fins and vortex generators. *Heat and Mass Transfer* **48**:1113–1124, 2012. https://doi.org/10.1007/s00231-011-0959-5

[5] R. Bhadouriya, A. Agrawal, S. Prabhu. Experimental and numerical study of fluid flow and heat transfer in an annulus of inner twisted square duct and outer circular pipe. *International Journal of Thermal Sciences* **94**:96–109, 2015. https://doi.org/10.1016/j.ijthermalsci.2015.02.019

[6] D. Majidi, H. Alighardashi, F. Farhadi. Experimental studies of heat transfer of air in a double-pipe helical heat exchanger. *Applied Thermal Engineering* **133**:276–282, 2018. https://doi.org/10.1016/j.applthermaleng.2018.01.057

[7] W.-L. Chen, W.-C. Dung. Numerical study on heat transfer characteristics of double tube heat exchangers with alternating horizontal or vertical oval cross section pipes as inner tubes. *Energy Conversion and Management* **49**(6):1574–1583, 2008. https://doi.org/10.1016/j.enconman.2007.12.007

[8] X. Tang, X. Dai, D. Zhu. Experimental and numerical investigation of convective heat transfer and fluid flow in twisted spiral tube. *International Journal of Heat and Mass Transfer* **90**:523–541, 2015. https://doi.org/10.1016/j.ijheatmasstransfer.2015.06.068

[9] W. El-Maghlany, E. Eid, M. Teamah, I. Shahrour. Experimental study for double pipe heat exchanger with rotating inner pipe. *International journal of advanced scientific and technical research* **4**(2):507–527, 2012.

[10] E. Ozden, I. Tari. Shell side CFD analysis of a small shell-and-tube heat exchanger. *Energy Conversion and Management* **51**(5):1004–1014, 2010. https://doi.org/10.1016/j.enconman.2009.12.003

[11] A. K. Agrawal, S. Sengupta. Laminar fluid flow and heat transfer in an annulus with an externally enhanced inner tube. *International Journal of Heat and Fluid Flow* **14**(1):54–63, 1993. https://doi.org/10.1016/0142-727X(93)90040-T

[12] J. Taborek. Double-pipe and multitube heat exchangers with plain and longitudinal finned tubes. *Heat Transfer Engineering* **18**(2):34–45, 1997. https://doi.org/10.1080/01457639708939894

[13] R. Raj, N. S. Lakshman, Y. Mukkamala. Single phase flow heat transfer and pressure drop measurements in doubly enhanced tubes. *International Journal of Thermal Sciences* **88**:215–227, 2015. https://doi.org/10.1016/j.ijthermalsci.2014.10.004

[14] S. P. Asok, K. Sankaranarayanasamy, T. Sundararajan, et al. Pressure drop characteristics of water flow through static annular and triangular cavity labyrinth seals. *Engineering Applications of Computational Fluid Mechanics* **2**(4):482–495, 2008. https://doi.org/10.1080/19942060.2008.11015246

[15] 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. *International Communications in Heat and Mass Transfer* **35**(2):120–129, 2008. https://doi.org/10.1016/j.icheatmasstransfer.2007.07.003

[16] M. M. Aslam Bhutta, N. Hayat, M. H. Bashir, et al. CFD applications in various heat exchangers design: A review. *Applied Thermal Engineering* **32**:1–12, 2012. https://doi.org/10.1016/j.applthermaleng.2011.09.001

[17] C. V. M. Braga, F. E. M. Saboya. Turbulent heat transfer, pressure drop and fin efficiency in annular regions with continuous longitudinal rectangular fins. *Experimental Thermal and Fluid Science* **20**(2):55–65, 1999. https://doi.org/10.1016/S0894-1777(99)00026-6

[18] L. Sun, C.-L. Zhang. Evaluation of elliptical finned-tube heat exchanger performance using CFD and response surface methodology. *International Journal of Thermal Sciences* **75**:45–53, 2014. https://doi.org/10.1016/j.ijthermalsci.2013.07.021

[19] H. Huisseune, S. De Schampheleire, B. Ameel, M. De Paepe. Comparison of metal foam heat exchangers to a finned heat exchanger for low reynolds number applications. *International Journal of Heat and Mass Transfer* **89**:1–9, 2015. https://doi.org/10.1016/j.ijheatmasstransfer.2015.05.013

[20] M. I. Kim, Y. Lee, B.-W. Kim, et al. CFD modeling of shell-and-tube heat exchanger header for uniform distribution among tubes. *Korean Journal of Chemical Engineering* **26**:359–363, 2009. https://doi.org/10.1007/s11814-009-0060-7

[21] H. Nemati, M. A. Moghimi. Numerical study of flow over annular-finned tube heat exchangers by different turbulent models. *CFD Letters* **6**(3):101–112, 2014.

[22] T. J. Rennie, V. G. S. Raghavan. Experimental studies of a double-pipe helical heat exchanger. *Experimental Thermal and Fluid Science* **29**(8):919–924, 2005. https://doi.org/10.1016/j.expthermflusci.2005.02.001

[23] T. Maré, N. Galanis, I. Voicu, et al. Experimental and numerical study of mixed convection with flow reversal in coaxial double-duct heat exchangers. *Experimental Thermal and Fluid Science* **32**(5):1096–1104, 2008. https://doi.org/10.1016/j.expthermflusci.2008.01.002

[24] S. Freund, S. Kabelac. Investigation of local heat transfer coefficients in plate heat exchangers with temperature oscillation IR thermography and CFD. *International Journal of Heat and Mass Transfer* **53**(19-20):3764–3781, 2010. https://doi.org/10.1016/j.ijheatmasstransfer.2010.04.027