VIKAS KANNOJIYA,[1] RAHUL GAUR, PUSHPENDER YADAV, RIYA SHARMA

PERFORMANCE INVESTIGATION OF A DOUBLE PIPE HEAT EXCHANGER UNDER DIFFERENT FLOW CONFIGURATION BY USING EXPERIMENTAL AND COMPUTATIONAL TECHNIQUE

Heat exchangers are widely employed in numerous industrial applications to serve the heat recovery and cooling purpose. This work reports a performance analysis of a tube in tube heat exchanger for different flow configuration under variable operating conditions. The experimental investigation was performed on a U-shaped double pipe heat exchanger set up whereas Commercial Computational Fluid Dynamics code FLUENT along with k-ε turbulence modeling scheme was implemented for the simulation study. The flow solution was achieved by implementing k-ε turbulence modeling scheme and the simulation findings were compared with the experimental results. The experimental findings were in good agreement with the simulation results. The counter-flow configuration was found to be 29.4% more effective than the co-current one at low fluid flow rate. Direct relationship between heat transfer rate and flow rate is observed while effectiveness and LMTD showed inverse relationship with it. The significance of inlet temperature of hot and cold stream has been evaluated, they play crucial role in heat exchange process.

Nomenclature

C_p	specific heat of fluid
$\dot{m}$	mass flow rate
T	temperature
Q	rate of heat transfer
ε	dissipation rate of fluid turbulence
k	turbulence kinetic energy

[1]*Department of Mechanical and Industrial Engineering, IIT Roorkee, India.*
Email: vikas.passion.singh@gmail.com

LMTD   logarithmic mean temperature difference
CFD   computational fluid dynamic,
K   thermal conductivity
NTU   number of transfer unit
U   overall heat transfer coefficien,
C   capacity rate ratio
$\in$ effectiveness
A   area

1. Introduction

Several industrial units like thermal power plants, nuclear reactors, food processing, pharmaceutical industry, air-conditioning systems etc. require some heating, cooling and energy recovery methods. Heat exchangers are extensively used for the above-mentioned processes, as they exchange energy between two fluids separated by a solid boundary through conduction and convection. The idea of an efficient heat exchanger is based on maximum heat transfer with minimum area required.

There are numerous ways to classify heat exchangers, but with reference to the flow arrangements they can be only categorized into co-current, counter and cross flow heat exchangers. The heat transfer process in a heat exchanger is a complex one due to its dependency on several parameters, like working fluids, fluid mass flow rates, geometric characteristics of the exchanger etc. [1–4]. Moreover, evaluating the flow characteristics and temperature profiles across the heat exchanger pipe length is very difficult in such experimentation. The advancement of computational tools helps one to study such phenomena with ease and in a cost-effective manner. Recently, many researchers are adopting Computational Fluid Dynamics (CFD) to study the complex scientific processes.

The performance analysis of a counter flow heat exchanger has been studied by Narayanan et al. [5]. They reported a relationship between effectiveness, NTU and thermal resistances. Chato et al. [6] developed several models to predict the performance of a co-current flow heat exchanger. These models serve the purpose of creating an effective tool to evaluate and compare the performance of a heat exchanger. Khaled et al. [7] reported a thermal analysis of a heat exchanger. They studied the influence of fluid velocity, temperature and flow rates. They concluded that non-uniformity in fluid flow rate decreases the thermal performance of the heat exchanger. Several researchers employed CFD technique to analyze the heat transfer phenomenon in a heat exchanger [8–11].

Dang et al. [12] studied the heat transfer behavior of a 32 mm long microchannel heat exchanger for two different flow configurations. They conducted experimental and numerical analysis to investigate the influence of operating conditions, like mass flow rate and fluid stream temperature, on the heat trans-

fer rate of heat exchanger. They found counter-flow configuration to be more effective than the parallel one, as heat flux for the former one was reported to be 1.2 times higher than the later. However, less information regarding the operating parameters was reported. Mohanty et al. [13] reported a simulation based study of heat transfer in tube in tube-type heat exchanger. The study was carried out by using CFD code FLUENT 14.0. They studied the heat transfer phenomenon under different flow configuration. They observed the enhancement in heat transfer rate at the expense of pressure drop. Twisted tape inserts over the full width enhanced the heat transfer rate by 36%, as compared to plain tube.

Many researchers analyzed the heat transfer phenomena in heat exchanger under different operating conditions. However, the detailed report on the heat transfer rate influencing dimensionless parameters, such as LMTD, NTU and effectiveness, are still missing in the literature. Therefore, this study analyzes the performances of heat exchanger by considering the effect the mentioned dimensionless numbers. The investigations were carried out for co-current and counter-current type of heat exchangers by using experimental and computational technique. The experimentation was performed on a heat exchanger test rig set-up while CFD code FLUENT was optimized for computational solutions [14]. The analysis includes the evaluation and comparison of all the vital parameter influencing the heat transfer rate.

2. Experimental analysis

2.1. Experimental setup

Fig. 1 illustrates the experimental set-up used to evaluate the performance of different flow configuration in a tube in tube-type heat exchanger. The test section was a brass u-shaped tube, having two 1 m long straight sections. The hot fluid flows through the inner pipe (6.3 mm diameter), whereas the outer pipe (8 mm diameter) allows cold fluid through it. The test rig was provided with proper flow control valve to adjust the direction of flow regimes. When the valve V3 and V4 were open, then the flow configuration was parallel, whereas valve V5 and V6 created the flow in counter direction. The pipe line was wrapped with 17 mm thick insulation. An electric heater is also provided with the set-up to generate hot fluid. Magnetic drive centrifugal pumps were used to pump hot and cold streams. Two rota-meters measuring the flow rate of hot and cold fluid, respectively, were also mounted on the set-up. Five thermocouples (accuracy at 0.1°C) were inserted at different positions to sense the change in thermal state of the fluid. The schematic of experimental setup for counter current configuration in 2-D is shown in Fig. 2.

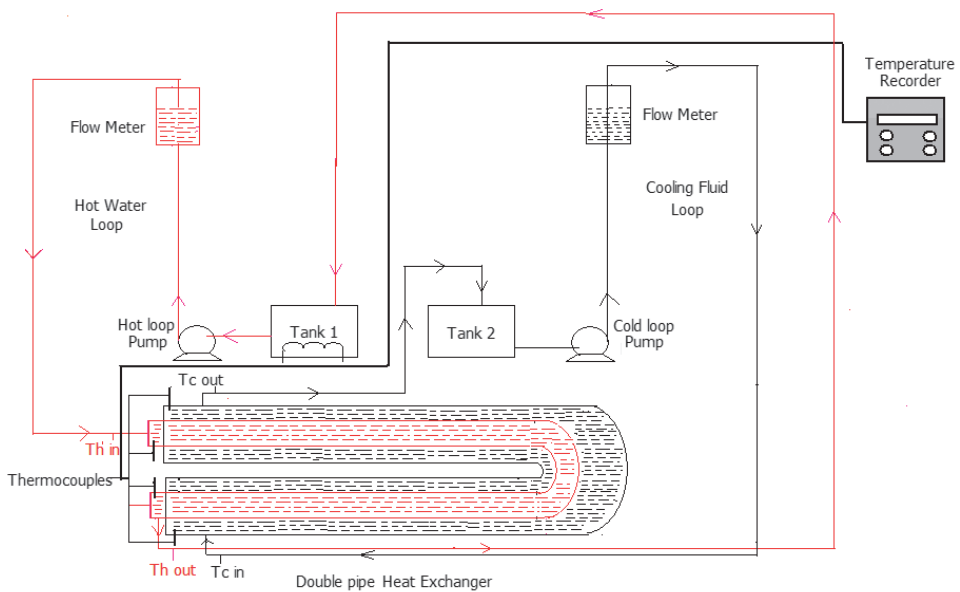Fig. 1. Experimental set-up of tube in tube heat exchanger

Fig. 2. Experimental set-up for counter flow configuration in 2-D

2.2. Solid-liquid property

The experimentation was performed on a tube in tube-type heat exchanger made of brass. Pure water was used for both hot and cold flow regimes. A brief detail of the solid-liquid property is presented in Table 1.

Table 1.

Material used for the experimentation

Parameter	Solid	Fluid
Material	Brass	Water
Density (kg/m^3)	7800	1000
Specific Heat (KJ/kg K)	0.379	4.181
Conductivity (W/m K)	109	0.59

3. Data processing

Experimental data were used for the calculation of logarithmic heat transfer difference, heat transfer, overall heat transfer coefficient, effectiveness, NTU. The thermo-physical properties were evaluated by considering the bulk mean temperature of fluid.

The analytical relations for heat transfer are available in several literatures [15–18]. The following equations are used for the calculation of several heat transfer components.

The rate of heat transfer from hot fluid is:

$$Q_h = \dot{m}_h C_{ph}[T_{ho} - T_{hi}]. \tag{1}$$

The rate of heat transfer into cold fluid (Q_c) can be written as:

$$Q_c = \dot{m}_c C_{pc}[T_{Co} - T_{Ci}], \tag{2}$$

where $\dot{m}_c$ and $\dot{m}_h$ represent the mass flow rate of cold fluid and hot fluid, T_{Co} and T_{Ci} are the temperature of cold fluid at inlet and outlet section of the heat exchanger, whereas T_{ho} and T_{hi} are the hot fluid outlet and inlet temperature, respectively.

The logarithmic mean temperature difference can be expressed as:

$$\theta_m = \frac{\theta_1 - \theta_2}{\ln \dfrac{\theta_1}{\theta_2}}. \tag{3}$$

It may be noted that for counter-flow configuration $\theta_1 = [T_{hi} - T_{Ci}]$ and $\theta_2 = [T_{ho} - T_{Co}]$ whereas for the case of counter flow $\theta_1 = [T_{hi} - T_{Co}]$ and $\theta_2 = [T_{ho} - T_{Ci}]$.

The effectiveness of heat exchanger is calculated by NTU technique and can be written in the following way:

$$\epsilon_{parllel\,flow} = \frac{1 - e^{-(1+C)NTU}}{1 + C}, \tag{4}$$

$$\epsilon_{counter\,flow} = \frac{1 - e^{-(1-C)NTU}}{1 - Ce^{-(1-C)NTU}}, \tag{5}$$

where C is the capacity rate ratio,

$$C = \frac{(\dot{m}C_p)\text{minimum}}{(\dot{m}C_p)\text{maximum}} \tag{6}$$

and NTU is the number of transfer unit

$$\text{NTU} = \frac{UA}{(\dot{m}C_p)\,\text{min}}. \tag{7}$$

4. Computational study

4.1. Geometric details

In this work, ANSYS code FLUENT was utilized for the simulation study of heat transfer phenomenon in the heat exchanger. The geometry was created in ANSYS design modeler and further discretized into 514565 smaller tetrahedral elements after suitable grid sensitivity analysis, as described in Table 2. Fig. 3 represents the meshing on the geometry for the parallel flow configuration. The boundary conditions, namely mass flow rate inlet, pressure outlet and wall, were applied to the fluid flow domain.
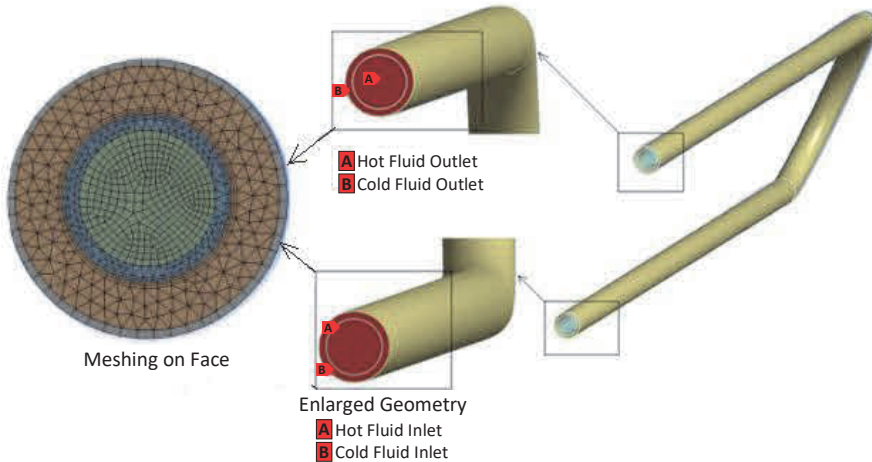

Fig. 3. Detailed view of geometry

Table 2.

The details of the grid independence analysis

S. no.	Maximum no. of elements	Heat transfer rate (Watt)
1.	1,34,437	2865
2.	2,27,450	2784
3.	3,05,495	2748
3.	4,01,758	2716
4.	4,85,552	2695
5.	5,14,565	2687

4.2. Boundary conditions

The simulation study for the heat exchanger was carried out by using CFD code FLUENT. The fluid flow in the heat exchanger posses turbulent attributes and was modeled by using Standard k-ε turbulence modeling scheme. Table 3 represents the brief description of boundary conditions implemented for the simulation study. The simulation study was carried out on Intel Xenon E51607v2 3.0 having 2.95 GHz processor with 16 GB RAM, the convergence was achieved when the residuals droped to 10^{-5}.

Table 3.

Boundary conditions for simulation study

Type	Description	Input
Model	Turbulence scheme	• Realizable k-ε • Near wall treatment: enhanced wall treatment
Energy equation	Description	• On
Material	Description	• Flowing fluid (Water) • Solid Domain (Brass)
Operating conditions	Gravitational acceleration and operating pressure	• 0 • 101325 Pa
Boundary conditions	Inlet	• Mass flow rate inlet, temperature
	Outlet	• Pressure outlet, temperature
	Wall	• Stationary • No slip • Roughness constant: 0.5 µm
Solution control	Pressure-velocity coupling	• Simple algorithm
Solution initialization	Description	• Compute from all zones

5. Results and discussion

5.1. Influence of mass flow rate

Experiments have been conducted to evaluate the heat exchanger performance under variable operating conditions. Fluid mass flow rate have been varied for both the flow configurations from 0.02–0.10 kg·m^{-3} in the step of 0.2 kg·m^{-3}. The influence of mass flow rate on the effectiveness for both type of heat exchanger is plotted in Fig. 4. It can be noted from the figure that at low fluid flow rate the counter flow configuration is approximately 29.4% more effective than the parallel one. However, the effectiveness of counter-flow configuration encounters significant downfall at higher flow rates (0.6–0.10 kg/s) and becomes close to the effectiveness of parallel flow configuration.
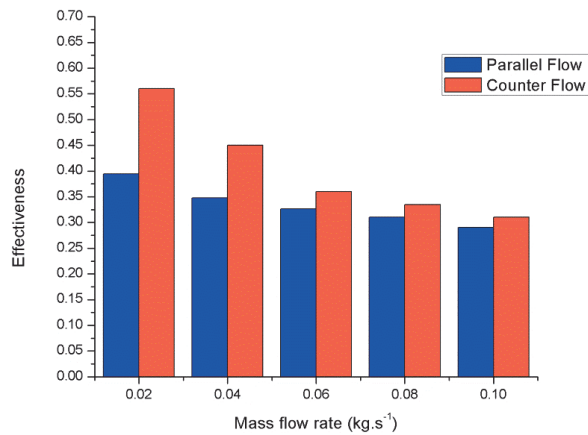
Fig. 4. Influence of mass flow rate on the effectiveness of parallel
and counter flow heat exchanger

The overall heat transfer coefficient has a vital role in heat transfer rate. Fig. 5 represents the variation in the overall heat transfer coefficient with the fluid flow rates for the counter-flow configuration. It can be observed here that the overall heat transfer coefficient linearly increases with the mass flow rate. The value of overall heat transfer coefficients, based on the external radius, is found to be 66% higher than that of parallel configuration at a flow rate of 0.02 kg/s.

The logarithmic mean temperature difference (LMTD) is an essential parameter to determine the temperature driving force of any heat exchanger. The LMTD for both the flow configuration has been calculated under different flow rates and is represented in Fig. 6. It can be observed that LMTD, for both the exchangers, dropped with the increase in mass flow rate. Moreover, LMTD for parallel flow heat exchanger lags behind with counter flow by 11.5% at 0.10 kg/s mass flow rate.

The heat transfer between hot and cold fluid for both the flow configurations is examined at different flow rates by using experimental and computational tech-
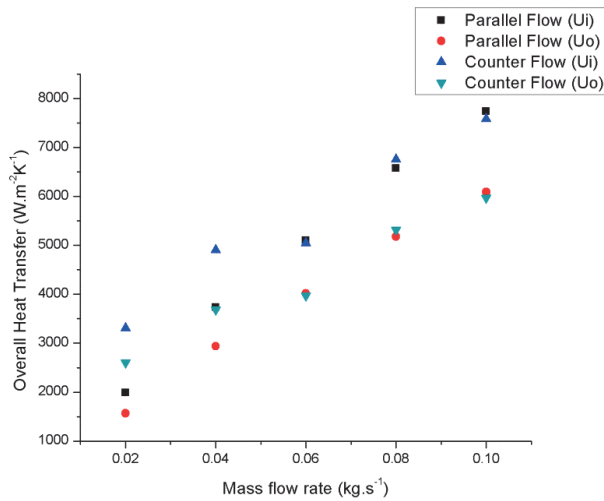
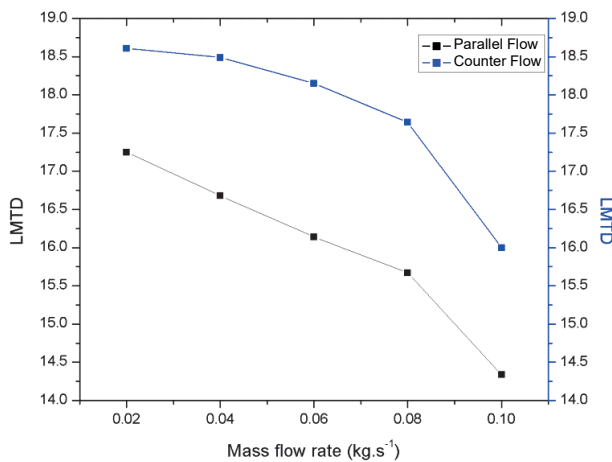Fig. 5. Variation of overall heat transfer rate with mass flow rate

Fig. 6. Effect of mass flow rate on LMTD

nique. The relationship between heat transfer rate and mass flow rate of water is plotted in Fig. 7. It can be noted from the figure that the heat transfer rate for both the heat exchangers increases with the mass flow rate. The counter-flow configuration has higher heat exchanging capabilities than the parallel flow unit. At low mass flow rate, counter-flow configuration transfers 43% more heat than the parallel one. However, at higher flow rates (0.10 kg/s) the heat exchanging ability for counter-flow heat exchanger leads parallel by only 9.3%. The simulation result shows trends similar to the experimental one, and the difference between them is less than 6%.
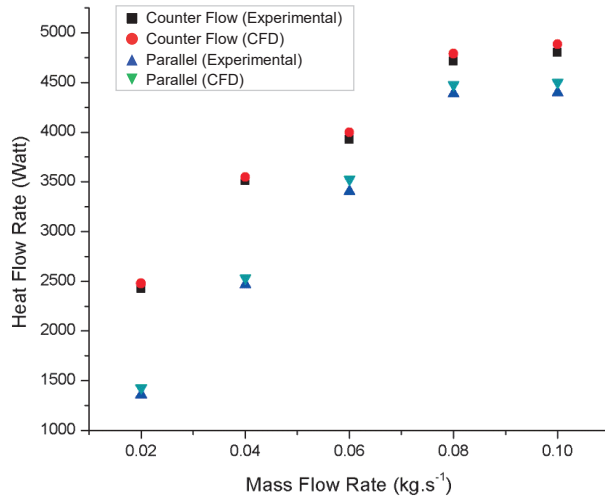
Fig. 7. Influence of mass flow rate on heat transfer

Fig. 8 represents the temperature contour plotted at the mid plane of the left straight section of u-shape heat exchanger. The contour describes the heat exchange process for a counter-current fluid flow where the hot fluid enters from the left (the central passage) while the cold water is coming from the opposite
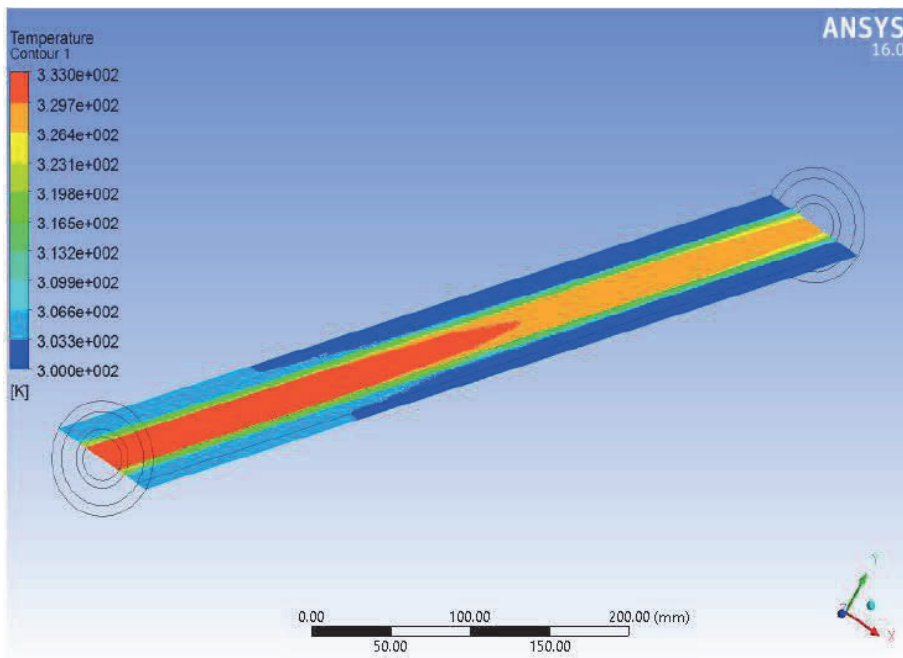

Fig. 8. Temperature contour at the left straight section of counter flow heat exchanger

end (the surrounding passage). The red color indicates the maximum temperature, while blue gives the indication of minimum one. The hot water temperature is decreasing in the direction of pipe length, as it is exchanging its heat with the surrounding cold water passage through an indirect contact. This process can be observed by the color change in the mentioned contour. On the other hand, the cold water temperature is increasing as it gains the heat from hot fluid.

5.2. Influence of hot water inlet temperature

Investigations have been carried out to evaluate the influence of inlet temperature of hot fluid on the performance of counter-flow heat exchanger. The variation in the inlet temperature of hot fluid is achieved by increasing the voltage of the heater, at the same time cold fluid enters the heat exchanger at a constant temperature for every experimental run. The variation of LMTD with inlet temperature of hot fluid stream is illustrated in Fig. 9. The LMTD attains higher value with the increase in inlet temperature of hot stream. The LMTD with hot fluid at 80°C inlet temperature is found to be 1.9 times higher than the fluid at 65°C temperature.
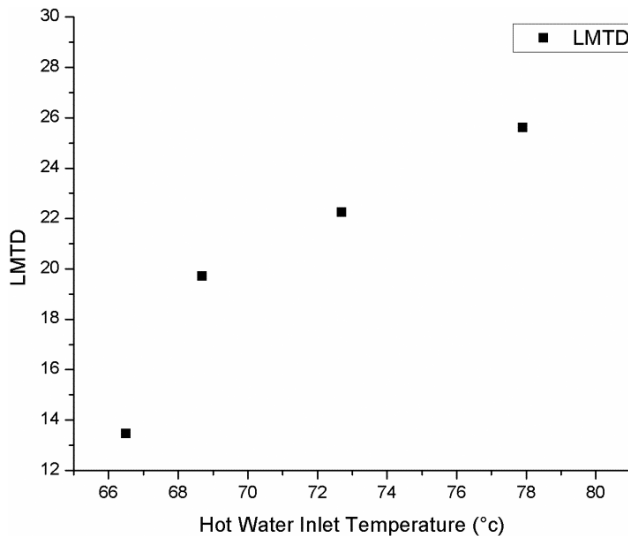
Fig. 9. Variation in LMTD with hot water inlet temperature

The influence of hot fluid inlet temperature on the heat transfer rate is represented in Fig. 10. The heat transfer rate increases with the warmer fluid and is found to be 19% higher for the fluid at 80°C temperature than that of at 65°C.

The relationship between the effectiveness of the heat exchanger and NTU is plotted in Fig. 11. It can be noted that NTU has a direct influence on the effectiveness.
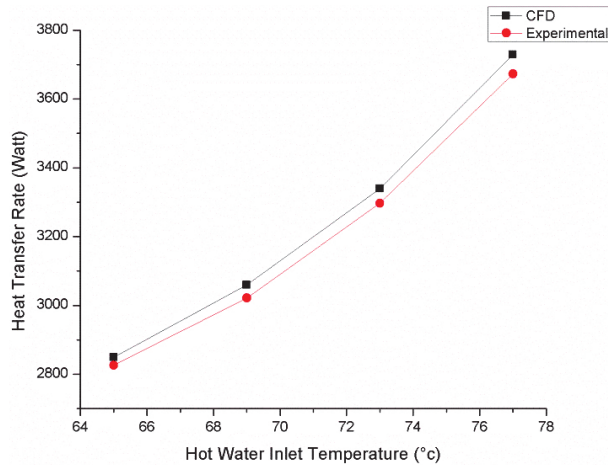
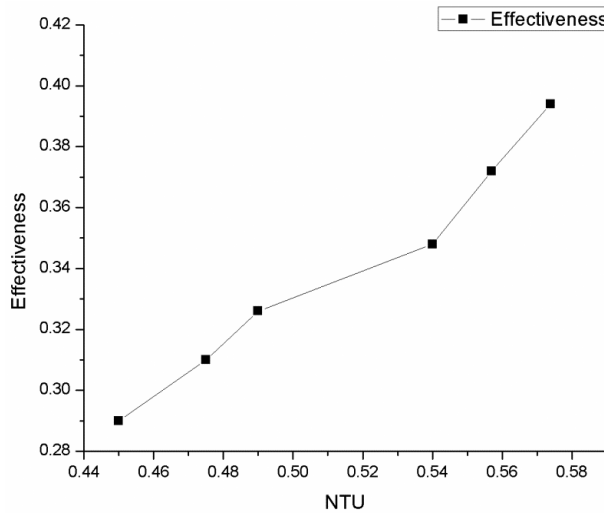Fig. 10. Effect of hot water inlet temperature on the heat transfer

Fig. 11. NTU verses effectiveness

5.3. Influence of cold water inlet temperature

To analyze the influence of cold stream inlet temperature on the performance of counter flow heat exchanger, cold water is fed to the heat exchanger at different temperature, whereas hot fluid allows it to flow at constant temperature throughout the experimental run. Fig. 12, illustrates the variation in Number of Transfer Unit (NTU) and the effectiveness of the heat exchanger with the cold fluid inlet temperature. It is noticed that both the effectiveness and NTU faces drop in its value with the increase in cold fluid temperature. The colder fluid significantly enhances

the effectiveness, for instance effectiveness when the fluid is at 23°C temperature is 4.5% higher than that when the fluid is at 33°C, whereas no such a significant enhancement is observed for NTU.
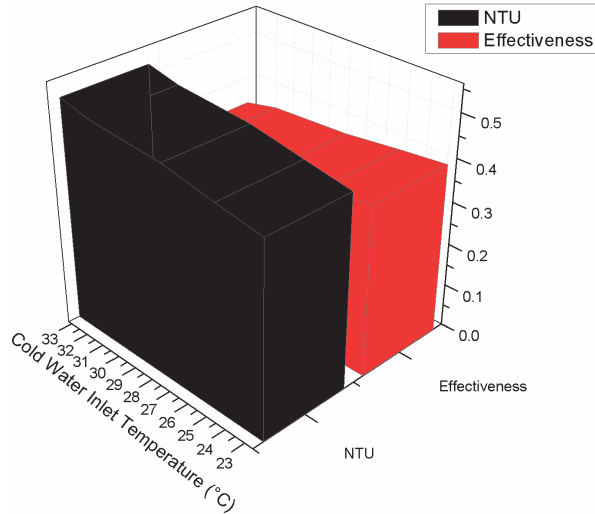

Fig. 12. Variation in NTU and Effectiveness with cold water inlet temperature

The fluid behavior inside the heat exchanger pipe is very difficult to predict experimentally, but the same can be easily achieved by CFD. The flow turbulence caused by the cold flow stream at 0.08 kg/s mass flow rate is plotted at the pipe wall and is represented in Fig. 13. It can be noted that the flow is highly turbulent at the
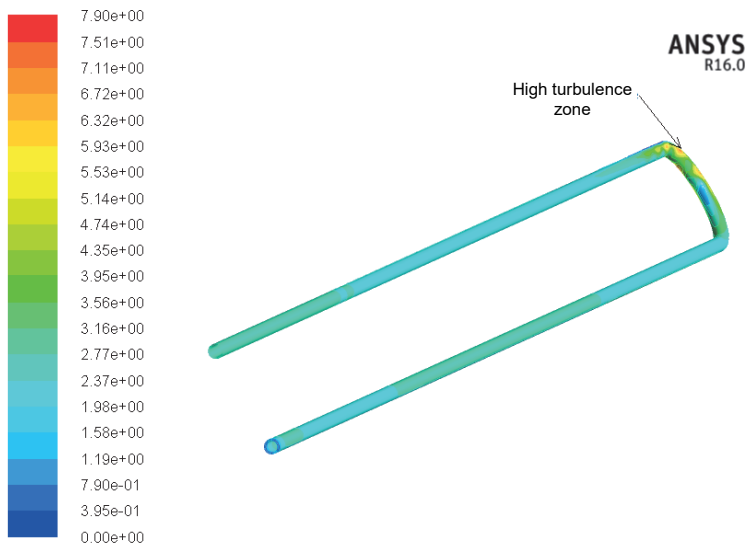

Fig. 13. Flow turbulence inside the heat exchanger

bend section, whereas the uniform flow behavior is observed at the straight inlet and outlet section. Few of the fluid particles get deviated from the main flowing stream and thus collide with the pipe wall. This collision causes significant loss of kinetic energy at that section, thus randomness in the flow is greater at the bend section.

6. Conclusions

A performance analysis of a double pipe heat exchanger for different flow configuration has been performed by using experimental and computational techniques. A heat exchanger test rig with suitable flow valve regulator to obtain different flow configuration was used for this work. The computational analysis was performed by using CFD code FLUENT with Standard k-ε turbulence modeling scheme. The computational results are in good agreement with the experimental findings. Some major outlines from this work can be summarized as:

1. The effectiveness for both the heat exchangers decreases with the increase of mass flow rate. The counter-flow configuration showed complete dominance with 29.4% greater effectiveness over the parallel one at low mass flow rate of 0.02 kg/s. However, at higher flow rate of 0.10 kg/s this difference dropped to 6.3%.
2. The LMTD value encountered a decline trend with the increase of mass flow rate, however, the LMTD for parallel flow configuration lags behind the counter one by 11.5% at a mass flow rate of 0.10 kg/s.
3. Direct relationship between mass flow rate and heat exchange rate was observed. At higher flow rates (0.10 kg/s), the heat exchanging ability for counter-flow heat exchanger leads parallel by only 9.3%.
4. The hot water inlet temperature directly influences the LMTD and the heat transfer rate. Approximately 19% higher heat transfer rate was found when the hot fluid was at 80°C temperature, compared to that obtained at 65°C.
5. The NTU and effectiveness experiences drop in its initial value with the rise in cold water inlet temperature. The effectiveness when the fluid is at 23°C temperature is 4.5% higher than that of when the fluid is at 33°C.

References

[1] A. Singh and S.S. Sehgal. Thermohydraulic analysis of shell-and-tube heat exchanger with segmental baffles. *ISRN Chemical Engineering*, 1–5, 2013. doi: 10.1155/2013/548676.

[2] P. Gunnasegaran, N.H. Shuaib, and M.F. Abdul Jalal. The effect of geometrical parameters on heat transfer characteristics of compact heat exchanger with louvered fins. *ISRN Thermodynamics*, 1–10, 2010. doi: 10.5402/2012/832708.

[3] P. Gupta and M.D. Atrey. Performance evaluation of counter flow heat exchangers considering the effect of heat in leak and longitudinal conduction for low-temperature applications. *Cryogenics,* 40(7):469–474, 2000. doi: 10.1016/S0011-2275(00)00069-2.

[4] R. Manimaran, K. Palaniradja, N. Alagumurthi, and J. Hussain. Factors affecting the thermal performance of heat pipe – a review. *Journal of Engineering Research and Studies*, 3(2):20–4, 2012.

[5] S.P. Narayanan and G. Venkatarathnam. Performance of a counterflow heat exchanger with heat loss through the wall at the cold end. *Cryogenics,* 39(1):43–52, 1999. doi: 10.1016/S0011-2275(98)00123-4.

[6] J.C. Chato, R.J. Laverman, and J.M. Shah. Analyses of parallel flow, multi-stream heat exchangers. *International Journal of Heat and Mass Transfer*, 14(1):1691–1703, 1971. doi: 10.1016/0017-9310(71)90077-9.

[7] [7] M. Khaled, M. Ramadan, A. Shaito, H.E. Hage, F. Harambat, and H. Peerhossaini. Parametric analysis of heat exchanger thermal performance in complex geometries – effect of air velocity and water flow distributions. *Heat Transfer Engineering*, 37(12):1027–1037, 2016. doi: 10.1080/01457632.2015.1104166.

[8] B. Kalidasan and M. Ravikumar. Numerical analysis of compact heat exchanger for flow distribution. *Indian Journal of Science and Technoogy*, 9(6):1–9, 2016. doi: 10.17485/i-jst/2016/v9i6/74596.

[9] M. Tutar and A. Akkoca. Numerical analysis of fluid flow and heat transfer characteristics in three-dimensional plate fin-and-tube heat exchangers. *Numerical Heat Transfer, Part A: Applications*, 46(3):301–321, 2004. doi: 10.1080/10407780490474762.

[10] J.B.B. Rao, V.R. Raju. Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes. *International Journal of Mechanical and Materials Engineering*, 11(6):1–18, 2016. doi: 10.1186/s40712-016-0059-x.

[11] H. Ma, D.E. Oztekin, S. Bayratkar, S. Vayla and A. Oztekin. Computational fluid dynamics and heat transfer analysis for a novel heat Eechanger. *Journal of Heat Transfer*, 137(5):051801, 2015. doi: 10.1115/1.4029764.

[12] T. Dang, J. Teng, and J. Chu. Effect of flow arrangement on the heat transfer behaviors of a microchannel heat exchanger. In *Proceedings of International Multiconference of Engineers of Computer Scientists*, IMECS 2010, Vol. 3, Hong Kong, 17–19 March, 2010.

[13] R.L. Mohanty, S. Bashyam, and D. Das. Numerical analysis of double pipe heat exchanger using heat transfer augmentation techniques. *International Journal of Plastics Technology*, 18(3):337–348, 2014. doi: 10.1007/s12588-014-9093-9.

[14] Fluent: Fluent User's Guide, Lebanon, NH, Fluent Inc. 2006.

[15] M.N. Ozisik. *Heat Transfer: a Basic Approach*. McGraw-Hill, New York, 1985.

[16] F.P. Incropera and D.P. DeWitt. *Fundamentals of Heat and Mass Transfer*. 5th edition, J. Wiley & Sons, New York, 2002.

[17] S.C. Chapra and R.P. Canale. *Numerical Methods for Engineers*. 3rd edition. McGraw-Hill, New York, 1998.

[18] D.Q. Kern and A.D. Kraus. *Extended Surface Heat Transfer*. McGraw-Hill, 1972.