Analytical and comparative investigations on counter flow heat exchanger using computational fluid dynamics

Published in:
Journal of Computational and Applied Research in Mechanical Engineering

Publication Status:
Published (in print/issue): 30/09/2020

DOI:
10.22061/jcarme.2019.4665.1564

Document Version
Publisher's PDF, also known as Version of record

SRTTU

JCARME

ISSN: 2228-7922

Journal of Computational and Applied Research in Mechanical Engineering

Vol. 10, No. 1, pp. 139-152

jcarme.sru.ac.ir

Analytical and comparative investigations on counter flow heat exchanger using computational fluid dynamics

Shuvam Mohanty*, Om Prakash and Rajesh Arora

Department of Mechanical Engineering Amity University Haryana, Gurgaon, India

Article info:

Type:	Research
Received:	10/01/2019
Revised:	10/08/2019
Accepted:	23/08/2019
Online:	25/08/2019

Keywords:

Counter flow heat exchanger,
k-epsilon,
RNG turbulence model,
Temperature contours

Abstract

This paper presents a comprehensive and exclusive thermodynamic analysis of counter flow heat exchanger under various operating and geometrical conditions. Analysis system workbench 14.0 is used for computational analysis, and comparison with previous literature is carried out in view of variable temperature and mass flow rate of hot and cold fluids. Analytical and statistical methods of computational fluid dynamics analysis are used for simulation and validation of the heat exchanger under steady and dynamic operating conditions. A 3-D model of a heat exchanger having 1000 mm and 1200 mm outside and inside tube lengths with diameter 12.7 mm is designed in the analysis system environment using the Renormalization Group k-ε approach in order to get better effectiveness of the system. The variable effects of the steady-state temperature and mass flow rate are investigated. The influence of turbulence over temperature and pressure profiles is also studied. Moreover, the analytical outcome of the present investigation is compared with that of previous existing literature and found to be in agreement with the previous one. The proposed analysis presents an in-depth perspective and simulation of the temperature gradient profile through the length of the heat exchanger. The proposed modified design of the heat exchanger along with changing flow direction yields much better results with small computational error 0.66% to 1.004% and 0.83% to 1.05% with respect to change in temperature and mass flow rate, respectively.

1. Introduction

Today's demand of higher energy consumption and reduced availability of fossil fuel resources increase the impact of thermal performance of heat exchanger day by day. Heat exchangers are very effective for the transfer of heat from one medium to another without even intermixing one fluid with another. One of the most promising devices for heat transfer is the counter flow heat exchanger mostly adapted by the chemical plants, petrochemical plants, oil refineries etc. Reducing the temperature of hot outlet fluid without affecting the cost is a big task for various industries that could be only possible by the proper selection of input. Typically, in a heat exchanger two segregated fluids at different temperature with a solid boundary, exchange thermal energy from one fluid to another via surface without even intermixing. There are numerous configurations of classifying heat exchanger. In context with the flow configuration, there exists three primary types for heat transfer:parallel flow, counter flow and

*Corresponding author
email address: shuvammohanty724@gmail.com

cross flow. According to Fourier for the conduction states the more the area of heat exchanger, the more will be the heat transfer rate. By second law of thermodynamics only transfer of sensible heat occurs in the heat exchanger. One of the greatest advantages of the counter flow heat exchanger is higher uniform temperature difference as well as that the mass flow rate and time for the interaction of one fluid with other increases, the heat transfer also goes up as compared to parallel flow heat exchanger. Maximization of surface area and minimization of flow resistance lead to better effectiveness of heat exchanger, which is the main focus for designing. On the contrary, the increase in area increases the space for the installation and correspondingly manufacturing cost will get increased. On the other hand, reduction in flow resistance can be achieved by improving the surface finishing of the heat exchanger. Many experiments have been carried out on the counter flow heat exchanger citing the flow in either laminar or turbulent manner, for achieving its better configuration. However, very limited CFD simulation has been done on the counter flow heat exchanger at different flow configuration to verify the thermo-hydraulic performance or to check the heat transfer and velocity distribution inside the flow domain. A prominent computational model for enhancement in heat transfer and effectiveness is needed to crave the variation in inlet temperatures and mass flow rates. In this paper a computational model of counter flow heat exchanger has been designed to perform comparative analysis amongst the designing outcomes of heat exchanger existing in previous literature related to the present study. With the use of aluminum as a material of construction for heat exchanger and making enhanced wall treatment, the heat transfer rate increases. Time and cost being the predominant factors for conducting experiments on heat exchanger, CFD simulation studies yield thus nearly practical and better behavior profile in less computation time. The objective of this study is further amplification of a computational model for predicting the response while studying various effects of heat exchanger parameters on heat transfer rate. The crucial part is to find out the most optimum and influencing parameters for the CFD simulation.

Jayakumar et al. [1] proposed the features of double pipe helical heat exchanger where the transfer of heat from fluid to fluid is undertaken. An experimental set up has been established and contrived and the results were taken for CFD validation. Mandal, & Nigam [2] presented the experimental results conducted over a helical counter flow heat exchanger for the hot compressed air in the inner tube and cold fluid over the outer tube. The flow rate of compressed air and pressure drop variations were observed for analyzing the friction factor and the Nusselt number which was later compared with the experimental results given in the literature. Stevic [3] considered the dynamics to be an integral part of the heat exchanger, a step change in the inlet temperatures mass flow rates, and optimized the cross flow heat exchanger. According to distinctive approach of Laplace, transformation was applied to overcome the geometric optimization along with some finite difference of partial differential equation, also to avoid the solution difficulties. Bansal et al. [4] came with a new idea of earth pipe air heat exchanger to reduce the cooling load and to predict the thermal performance of heat exchanger. An experimental set up was placed in Ajmer, and the values are compared with the simulation results changing the effects of different input parameters to study the behavior of heat exchanger. Kee et al. [5] carried out a study to achieve the effectiveness of heat exchanger using ceramics instead of changing the fluid to see the temperature difference. However ceramics withstands the high temperature and enables the higher temperatures in chemical reactors. A numerical investigation of double tube heat exchanger by Demir et al. [6] in a horizontal tube has been carried out by studying Nano fluids particles such as Tio_2 and Al_2O_3. Different thermal and hydrodynamic properties are analyzed through a correlation-based approach. Following the above numerical investigations, many researchers such as Huminic and Huminic, Mohammed and Narrein, Kamyar et al., and Xu et al. [7-10] addressed the heat transfer characteristics using different nano particles in the water. Porous baffles and flow pulsation inside a double pipe heat exchanger was investigated by Targui and Kahalerras [11] to check the performance and effectiveness. The study focused on flow structure after the liquid strikes the baffles; and it concludes that heat exchanger performance was directly proportional to hot fluid pulsation. Similar experimental work has been done upon the heat exchanger by Tiwari et al. [12] focusing on thermo-fluid properties on nano fluidic particles. Oon et al. [13] have done a CFD simulation in fluent to avoid the backflow in an annular passage from the sudden expansion of pipe. The Nusselt number and the Reynolds numbers have been considered for the investigation and to show the uniform heat transfer from the sudden expansion. Then the heat flux approach was used in the fluent set up to show the heat transfer in the tubular pipe. Nagarsheth et al. [14] used a systematic approach on a cross flow water tube in tube heat exchanger with a string of basic instrument like PID controller to regulate the temperature. PID controller is a kind of sensor which works in a closed loop feedback mechanism in industrial control system. Yaici et al. [15] presents a crucial part for implementing 3D CFD investigation of inlet air flow misdistribution in the interior pipe to check both the thermal and hydraulic performance; and the validation is done with the literature experimental data. Nagrajan et al. [16] carried out a dynamic analysis of changing the design of fins operating at a higher temperature. With using four different fins and at different configurations inside the heat exchanger, the results obtained were compared with each other. Among the four different fins, e.g. rectangular, triangular, inverted bolt and ripsaw, the inverted bolt fins gives the better heat transfer but due to its own configuration it shows a higher pressure drop. Safaei et al. [17] examined different thermo physical properties of Nano fluid coolant along with pressure drop and power loss of a coolant flow through the rectangular duct. They made a series of calculations with Nano fluid to achieve higher heat transfer rate under turbulent flow scheme and experimented over different Nano sheet materials. However, it was observed that using the Nano fluid helps to carry out more heat and pressure drops which leads to an increase in the pumping power. Sabaghan et al. [18] give a new comprehensive approach of using two phases Eulerian- Eulerian model in a rectangular micro channel along with a Nano fluid which helps to remove the heat generated in the micro channel. Further it presents the distinction between the longitudinal vortex generators (LVGs) and plain channel to show the better site efficiency. Pal et al. [19] using the CFD code open FOAM 2.2.0 analysed the recirculation zone and leakage flow during the heat transfer in a shell and tube heat exchanger with a considerable change of using baffles and without baffles which later comes out to be unproductive and later analyses using $k - \varepsilon$ model which provides a better profile for velocity and heat transfer. Andhare et al. [20] demonstrated a single phase, flat plate and manifold micro channel heat exchanger for performance enhancement shaping up with the counter flow technique. Correlation developed between the experimental and numerical results for heat transfer analysis in counter flow heat exchanger which is to be implemented in the industrial applications such as refrigeration/air conditioning and some power producing sectors. To analyze the heat transfer coefficient and heat exchanger efficiency, Mazidi et al. [21] proposed an inverse method. For this analysis function estimation approach is utilized to predict the heat transfer inside a double pipe heat exchanger. A sensitive analysis on double pipe heat exchanger has been carried out by Shirvan et al. [22] using response surface methodology and multiphase model. The Reynolds number and the Nusselt number are studied for an effective analysis. Mello et al. [23] explains the structural damage due to thermal stress in a heat exchanger for using the ceramics. But it has a tendency to resist oxidation. So to avoid the damage, alumina was used that was produced from a gel casting technique in a counter flow heat exchanger; and the model was predicted from CFD technique. Owen & Kröger [24] determined flow distribution by the tube inlet loss co-efficient of a large air cooled condenser using a mixture of CFD, numerical and analytical methods. For performance enhancement simulation using the CFD design phase of an air cooled condenser (ACC). Shahril et al. [25] gave an effective comparison between shell and double tube concentric heat exchanger and shell and tube heat exchanger using analytical system (ANSYS) fluent 14. The factor has been analyzed is the overall heat transfer rate per overall pressure drop which also specifies that the lesser will be the inner tube diameter, the higher will be the heat exchanger effectiveness. Nagarseth et al. [26] have explored the dynamic behavior of an operational disturbance and also done a discretization along the length of the tube to study temperature profile at different fluid inlet temperatures with a step change. They adapted two software named MATLAB and ANSYS for their analysis and validation. They carried out the main focus on MATLAB for simulation and CFD Gambit/Fluent for the validation. Behnampour et al. [27] experimented to see the heat and hydrodynamic behavior of water-argon Nano fluid flow in a micro channel using finite volume method (FVM) under constant heat flux. They also inspected different shapes of the micro channel using the Reynolds number and the Nusselt number; they later described that the rectangular ribs are suitable for Reynolds number and trapezoidal ribs for Nusselt number analysis. Lin et al. [28] described the effectiveness of the dew point evaporative cooler introducing the counter flow technique under different tests such as log mean temperature difference and humidity method. Ahmadi et al. [29] changed the approach of simulating the heat exchanger with a two phase Euler-Lagrange approach through a straight cylinder altering different parameters. Making some remarkable changes at the boundary by providing the constant wall heat flux, the increase in the Reynolds number and volume concentration was studied. Presence of Nano particle in the fluid leads to a rise in the pressure drop. Mahfouz et al. [30] came with an innovation of inserting a twisted rod in a tube for performance enhancement of the heat exchanger. The commercial CFD software is used and different types of twisted tapes are analyzed by CFD simulation. Performance evaluation criteria had been chosen using the dimensionless Nusselt number and friction factor. Kumar and Chandrasekhar [31] considering the Dean number of the range (1300-2200) carried out a computational analysis on helically coiled double pipe heat exchanger on ANSYS 14.5. Tusar et al. [32] investigated the performance and fluid flow characteristics on a helical screw tape inserted on ANSYS Fluent that showed that as the Nusselt number increases, the rate of heat transfer is affected.

1.2. State of the art

A few CFD simulations have been done on the counter flow heat exchanger at different flow configurations to verify the thermo-hydraulic performance or to check the heat transfer and velocity distribution inside the flow domain. The main motive of the present research is to find out the optimum value of outlet temperatures and effectiveness for different operating parameters. In this context, we have designed a thermodynamic model for the experimentation and investigated the effects of various design parameters on the performance of counter flow heat exchanger using CFD. Thereafter, different simulations have been conducted in the fluent set-up keeping the input parameters at different level. The design modular is associated with ANSYS workbench 14.0. The DELL workstation of 64 bit with parallel processing and 500 GB hard disk and Intel(R) core(TM) i3-4010U CPU @ 1.70 GHz 1.70 GHz with 4.00 GB RAM has been used for the simulation in FLUENT.

2. Mathematical modelling

2.1. Governing equation

The following continuity and momentum equations are used in the present study (Stevic, 2010):

Conservation of mass:

$$\nabla.(\rho V)=0 \tag{1}$$

Momentum equation:

In X-direction;

$$\nabla.(\rho u V)= -\frac{\partial \rho}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z} \tag{2}$$

In Y-direction;

$$\nabla.(\rho\upsilon V)= -\frac{\partial \rho}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho g \quad (3)$$

In Z-direction;

$$\nabla.(\rho w V)= -\frac{\partial \rho}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z} \quad (4)$$

Energy equation:

$$\nabla.(\rho e V)= -p\nabla.V+\nabla.(k\nabla T)+q+\varphi \quad (5)$$

φ is the heat dissipation calculated from;

$$\varphi = \mu\left[2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2+\lambda(\nabla.V)^2 \quad (6)$$

For the purpose of modeling, some of the factors are needed to be taken care of; therefore, the following assumptions are undertaken in the present study.

a) There is no interaction between the system and the surrounding.
b) Heat transfer due to conduction is not taken into account.
c) The flow velocities of the process fluids are assumed to be constant.

The energy balance equation applied at the hot and cold inlets are given as

$$(m_c\times C_{pc})\frac{dT_{cout}}{dt}=\left(T_{cin}\times C_{pc}\times \dot{m}_h\right)-\left(T_{cout}\times C_{pc}\times \dot{m}_c\right)+(h\times A\times(T_{hout}-T_{cout})) \quad (7)$$

$$(m_h\times c_{ph})\frac{dT_{hout}}{dt}=\left(T_{hin}\times C_{ph}\times \dot{m}_h\right)-\left(T_{hout}\times C_{ph}\times \dot{m}_h\right)+(h\times A\times(T_{hout}-T_{cout}) \quad (8)$$

The energy equation turned on for the heat exchanger and "k-ε model of RNG with enhanced wall treatment viscous model is selected.

The turbulent quantities are well described by RNG k-ε model.

$$\tau_{t,f} = -\frac{2}{3}\left(\rho_f K_f+\mu_{t,f}\nabla\upsilon_f\right)I + \mu_{t,f}(\nabla\upsilon_f+\nabla\upsilon_f^{tr}) \quad (9)$$

RNG model solves the epsilon by the help of Navier-Stoke equation and contributes a differential correlation for turbulent viscosity. The equation is given below:

$$\mu_{t,f} = \rho_f C_\mu \frac{k_f^2}{\epsilon_f} \quad (10)$$

The RNG model constants are

C_μ = 0.0845;$C_{1\varepsilon}$ =1.42;$C_{2\varepsilon}$= 1.68 and Wall Prandtl number = 0.85

2. Geometrical considerations

Commercial CFD software has been used for the simulation and validation study. The ANSYS proposes variable characteristics as design modeling, meshing and complicated flow simulation under a single platform. A 3-D geometry is generated in the ANSYS Fluent v14 design modular and modified into two computational zones for advance meshing. The governing energy equation and temperature fields are discretized into two different mesh zones and then the subsequent equations are solved with the application of iterative method. Tecplot software and CFD post are utilized for graph plotting.

The 3-D computational model is illustrated in Fig. 1, working on the basis of first principle which shows that the fluid flow inside the two concentric pipes is established on the laws of mass and momentum conservation. The inlet mass flow rate and pressure outlet are defined as the boundary conditions. The temperature and flow rates are kept on changing as mentioned in the Tables 2 and 3.

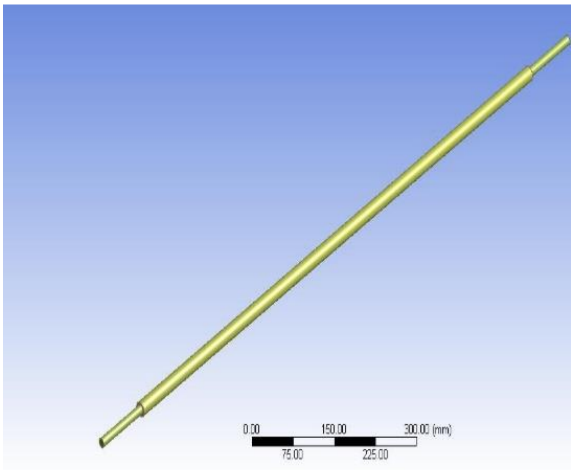

Fig. 1. Counter flow heat exchanger computational model.

2.1. Experimental set-up

Nagarseth et al. [26] conducted an experiment on counter flow heat exchanger of dimension of 1000mm length for the outer pipe and 12.7mm diameter for inner pipe, flowing water as a fluid in both hot and cold channel. A schematic diagram of the counter flow heat exchanger is shown in Fig. 2. There is a wall between the inner and outer pipe acting as a medium in order to transfer the heat from one fluid to another. With an aim of developing heat exchanger model in order to transfer large amount of heat from one medium to another, different temperatures at both the inlets are assigned. Temperatures are altered between 293-308K for cold fluid inlet and 343-358K for hot fluid inlet to see the heat transfer effects.

Heat removed from cold liquid by hot liquid:

$$(Q_h) = m_h \times C_{ph} \times (T_{hi} - T_{ho}) \quad (11)$$

Heat absorbed from hot liquid by cold liquid

$$(Q_c) = m_c \times C_{pc} \times (T_{co} - T_{ci}) \quad (12)$$

Computational simulation is done in ANSYS Fluent. Fig. 3 illustrates that the channel floors that separate the hot and cold fluid are approximately 1200 microns thick. The separation between the two channels describes three things, i.e. there should be pressure equalization.

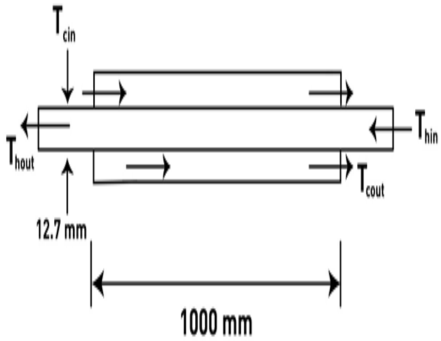

Fig. 2. Schematic diagram for counter flow heat exchanger.

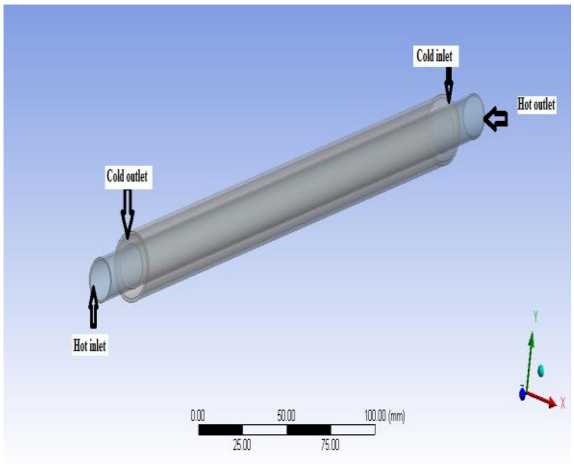

Fig. 3. Flow domain of counter flow heat exchanger.

The axial pressure drop and channel height both are inversely related, which comes to a very important point of discussion that small variation has a greater disturbance in flow domain. Reversing the direction of flow means that making counter flow configuration gives us a better performance. The flow configuration can be parallel and counter flow. For different heat transfer requirements the configuration can be possibly altered.

2.2. Grid independency test

Grid independent test is a dynamic approach for the improvement of the results by employing sequentially smaller cell size for the calculation. In the present study, we run the same simulation on progressively finer grids by changing the grid parameters rather than changing the local refinement. Three computational grids with total cell amounts of 0.97 Lakhs, 1.52 Lakhs, 2 Lakhs are computed for the coarse, medium and fine mesh sizes, respectively (Table 1). Choosing a fine grid over a medium grid, the simulation results are almost close as shown in Fig. 4. Hence the medium size mesh was chosen among the three.

2.3. Meshing and convergence

Meshing plays a vital role for achieving accurate and detailed flow in the tube. A high quality simulation is required for successful numerical simulation. The size of the mesh element near the wall is taken as small in order to find out the close results near the boundary walls of the heat exchanger. Better mesh compatibility near the contact region should be structured. Initially, relatively coarser mesh and mixed cells of triangular and quadrilaterals faces are created (Fig. 5) but the aim is to create structured hexahedral cells as much as possible.

4. Results and discussion

4.1. Steady state analysis

By fixing the cold inlet temperature at 293K and hot inlet temperature at 343K the open loop temperature profile is obtained. The graph below in Fig. 6 illustrates open loop temperature profile where the net average temperature obtained after steady state condition is 323.8K.
Specific operating conditions are as follows [26]:

Inlet temperature of cold fluid = 293K
Inlet temperature of hot fluid = 343K
Mass flow rate of cold fluid = 0.0041667kg/s
Mass flow rate of hot fluid = 0.0075kg/s
The temperature contours obtained after the iterations is shown in Fig. 7.
The change in the operating conditions leads to performing the steady state analysis for 3-D CFD model. So, there should be some significant alteration to typically compare or persuade the change. First of all, the cold inlet temperature was customized leading by 293 to 308K one after other and keeping hot inlet temperatures at a fixed value of 343K.

Table 1. Number of cells in three different levels of mesh.

Size of mesh	Elements	Nodes
Coarse	0.97 Lakhs	0.90 Lakhs
Medium	1.52 Lakhs	1.40 Lakhs
Fine	2.00 Lakhs	1.89 Lakhs

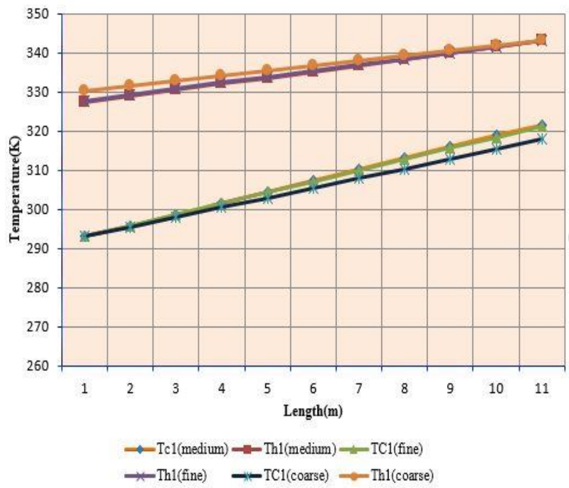

Fig. 4. Grid independency assessment.

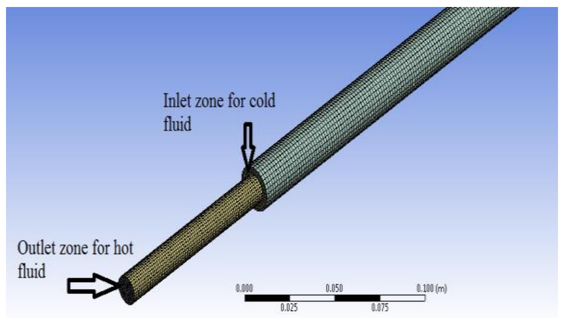

Fig. 5. Mesh structure of computational domain.

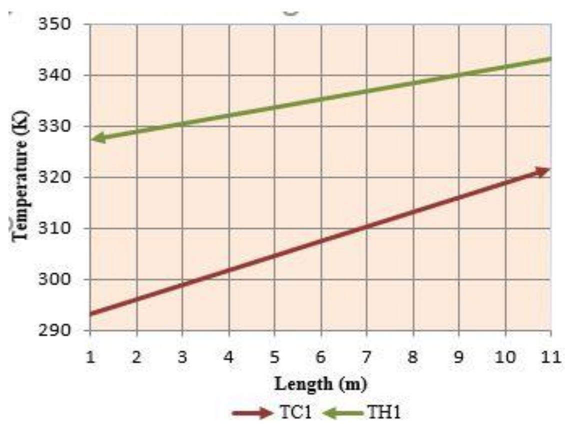

Fig. 6. Open loop response for single iteration.

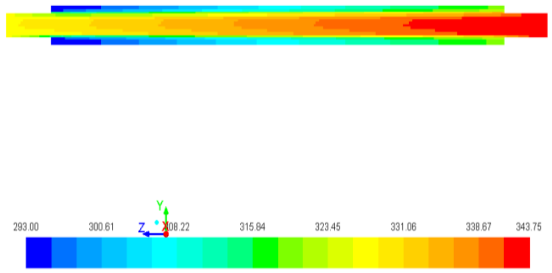

Fig. 7. Open loop response in From Fluent simulation.

Similarly, the change takes place keeping cold inlet at a fixed temperature and varying the hot inlet; the same will be followed for cold and hot mass flow inlet. This only can be done after successfully performing the steady-state analysis portrayed in Fig. 8.

So, starting from customizing the cold inlet flow temperature from 293 to 308, the flow achieves a new steady state at the cold outlet. However, it is seen that changing the cold fluid inlet temperature and fixing the hot fluid temperature lead to difference in heat transfer, which shows that by decreasing the cold fluid temperature the heat transfer increases, as a large amount of heat is carried out by the cold fluid and significantly decreases the hot fluid temperature as explained in Fig. 9. The graph also depicts that hot fluid curves shift more towards down if we decrease the cold fluid temperature. Therefore, a steady-state is accomplished at a higher temperature drop for hot and cold fluids. Again keeping the cold fluid temperature fixed and altering the temperature values of hot fluid temperature, a difference of heat transfer takes place shown in the graph shown in Fig. 10. As can be seen, the hot fluid temperature profiles shifted in the inlet as well as the outlet. There is a drastic change, if we keep increasing the hot fluid temperature from 343K to 358K, the heat transfer increases in both hot side and cold side in comparison with the previous one that causes a large amount of heat being taken by the cold fluid.

The summary of the above results is outlined below in Table 2 which shows the comparison of results obtained from the CFD fluent simulation and the previous simulation. Similarly, changing the mass flow rate of the cold fluid and hot fluid gives us better insight to study the heat transfer.

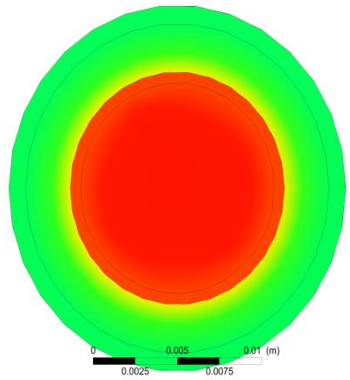

Fig. 8. Contours of temperature distribution.

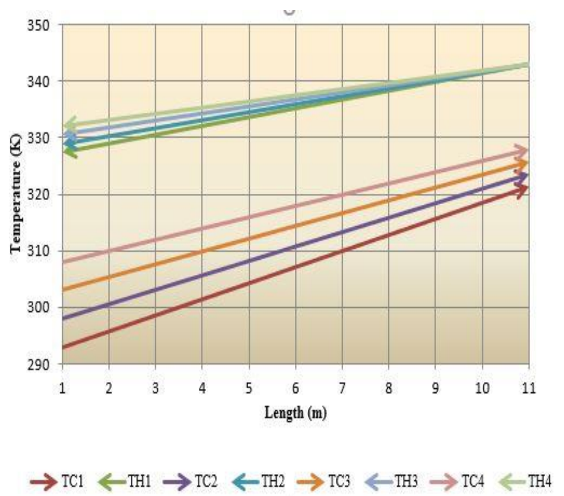

Fig. 9. Temperature variation of cold fluid from 293-343K.

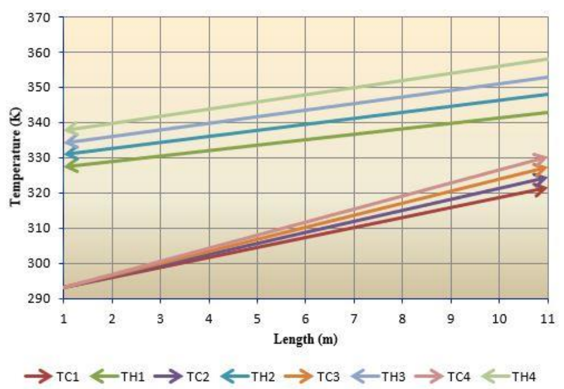

Fig. 10. Temperature variation of hot fluid from 343-358K.

As following the previous steps, the cold flow rate has gone through the step change fixing both the hot inlet and cold inlet temperatures, which will show the steady state values at the outlet.

Table 2. Comparison of fluent results for cold and hot inlet temperature step change.

T_v(K)	Cold temperature T_{Cout} (K)		Hot temperature T_{Hout} (K)		Result comparison
	Rs	(P.R)s	Rs	(P.R)s	% Error
T_{C1}=293	321.6	325.1	327.3	329.5	0.66
T_{C2}=298	323.8	327.4	328.9	331.4	0.78
T_{C3}=303	325.9	329.7	330.4	333.4	0.89
T_{C4}=308	328.0	332.1	332.0	335.4	1.004
T_{H1}-343	321.6	322.1	325.3	325.4	0.021
T_{H2}-348	324.5	324.7	327.3	328.4	0.33
T_{H3}-353	327.4	327.4	329.2	331.4	0.69
T_{H4}-358	330.2	330.1	332.6	334.5	0.56

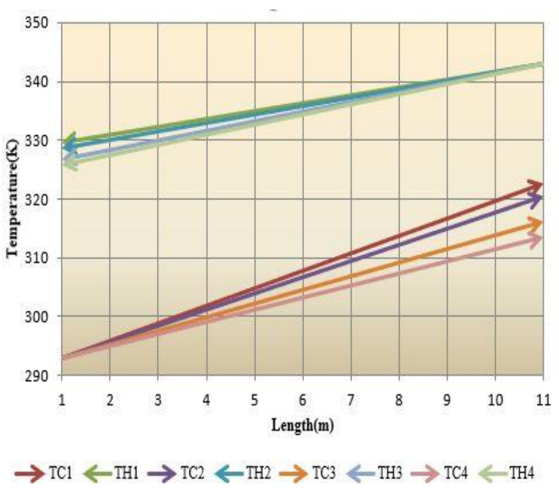

Fig. 11. Variation of cold fluid flow rates from (0.0041667 to 0.0077778)kg/s.

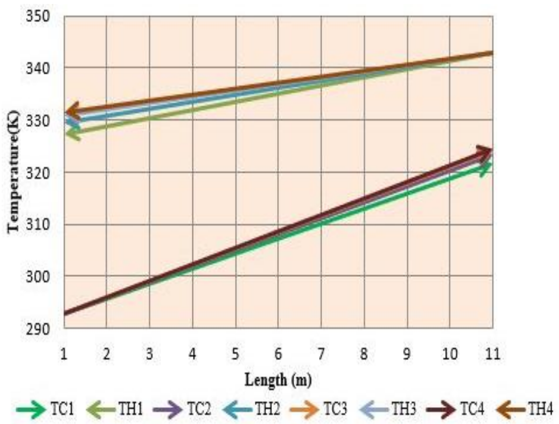

Fig. 12. Variation of hot fluid flow rate form (0.0075 to 0.00111)kg/s.

But, it has been seen that the heat transfer decreased by step changing the cold flow rate. By increasing the flow rate of the cold fluid, the interaction time decreases for the hot fluid with respect to the area and time. Both area and time being one of the key factors for the heat transfer, or we can say that directly proportional to the heat transfer, so the heat transfer lags and the steady state temperature lowers, so as heat transfer decreases. While increasing the hot fluid rate and fixing the inlet temperatures, it is observed that the heat transfer enhances and steady state temperature at the outlet of the cold fluid increases (Figs. 11 and 12).

The dynamics analysis of the heat exchanger is observed by customizing the whole parameters including the inlet temperatures of both the fluid and the mass flow rates, which helps us to display the dynamics response along with the length of the heat exchanger.

To speculate the heat transfer, the temperature of both the flow inlets has to change simultaneously. From Fig. 13 it can be observed that changing the hot inlet temperature and cold inlet temperature simultaneously for four different times the heat transfer changes linearly. The temperature contour for cold inlet is (303)K and for the hot inlet is (353)K. Similarly, changing the hot and cold fluid flow rates with remarkable combination simultaneously and at a fix temperature of (293-343)K, it was perceived that no drastic change of heat transfer occurs.

To predict the change in heat transfer, the entire dynamics has to be changed. For that reason the mass flow rates changed along with the inlet temperature and the results are shown below.

Nominal values:

T_{Cin} = 293K, T_{Hin} = 343K,
F_{Cin} = 0.0041667kg/s, F_{Hin} = 0.0075kg/s

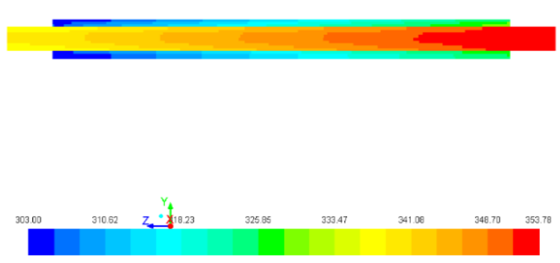

Fig. 13. Fluent simulation of temperature profile at specific operating condition.

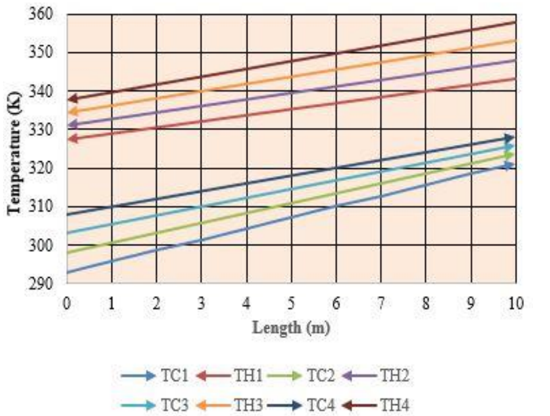

Fig. 14. Fluent simulation results for both side temperature change.

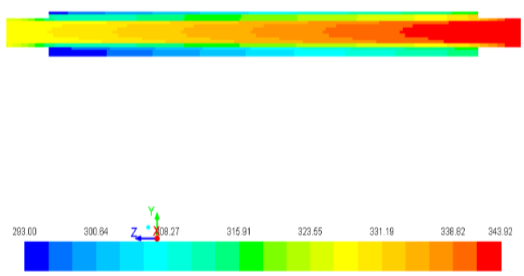

Fig. 15. Fluent simulation of temperature profile at flow rate (0.0063888-0.01055)kg/s.

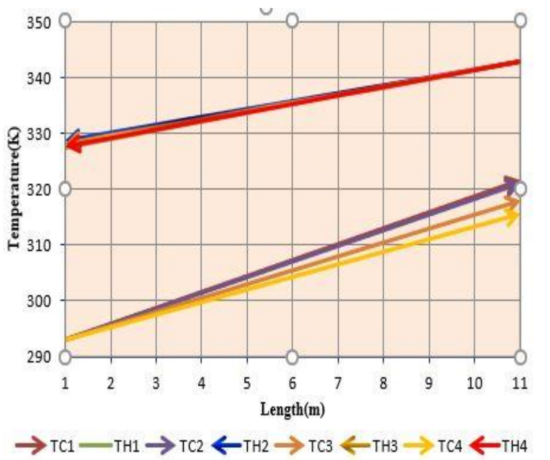

Fig. 16. Fluent results for change of flow rate on both sides.

From Fig.17 it is seen that only by considering the inlet temperatures and mass flow rate being the optimum parameters and by decreasing them, there would be chances for getting higher heat transfer. Fig.17 also shows that the periodic change of inlet temperatures leads to change in the temperature profile. Table 3 shows the results of dynamic response with the linear disturbances.

5. Validation of results

The main focus of this study is to validate the expected simulation results of counter flow heat exchanger showing the effects of heat transfer or temperature drop. For accomplishing the validation, the simulated results were compared with the experimental results of temperature drop available in the literature of Nagarsheth et al. [26]. Figs. 18-19 show the effects of heat transfer or temperature drop, over the variation of cold fluid temperature and mass flow rate. From the graph it can be observed that the variation of simulated results and comparing both the results, the numerical error is found to be at an average of 0.83% for step changing the fluid temperature; and at an average of 0.927% for step changing the mass flow rates.

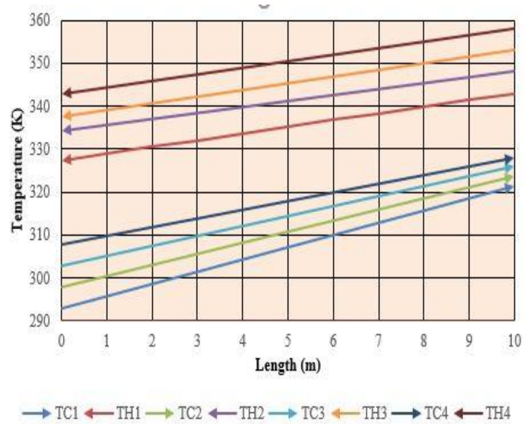

Fig. 17. Fluent simulation for change in both temperature and flow rate.

Table 3. Comparison of fluent results for hot and cold inlet flow rate step change.

Hot and cold fluid flow rate variation (kg/s)	Cold temperature T_{Cout} (K)		Hot temperature T_{Hout}(K)		Result comparison
	R_S	$(P.R)_S$	R_S	$(P.R)_S$	% Error
F_{C1}-0.0041667	323.2	329.8	329.5	332.2	0.83
F_{C2}-0.0047778	321.1	327.4	328.6	331.4	0.87
F_{C3}-0.0063888	316.9	321.932	326.6	329.8	0.96
F_{C4}-0.0077778	314.1	318.649	325.4	328.9	1.05
F_{H1}-0.0075	321.6	326.3	327.3	328.0	0.22
F_{H2}-0.00917	323.2	327.4	329.5	331.4	0.60
F_{H3}-0.01055	324.1	328.1	330.8	333.7	0.84
F_{H4}-0.01111	324.5	328.3	331.3	334.4	0.92

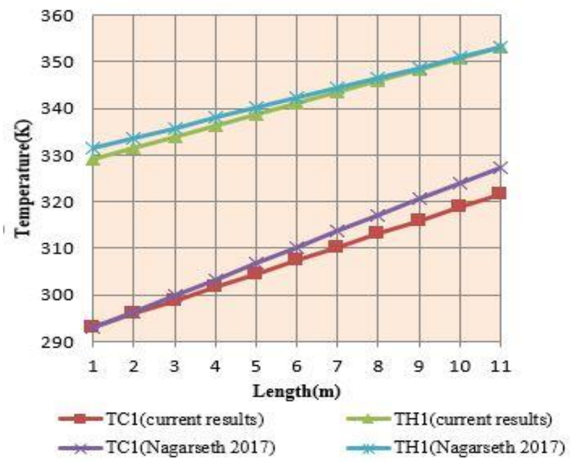

Fig. 18. Validation of result considering temperature.

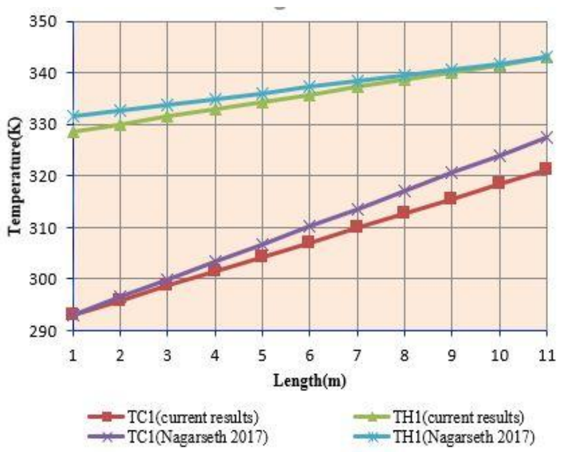

Fig. 19. Result validations considering the mass flow rate.

6. Conclusions

In the present work, the counter flow heat exchanger using CFD analysis has been investigated. The heat transfer and flow distribution of 3-D model counter flow heat exchanger is discussed in detail and compared with the previous literature in view of varying the inlet temperature and flow rate. The discretized 3-D model outcome is in agreement with the previous literature and hence are validated for proposed heat exchanger using fluent simulation tool of ANSYS 14.0. The present work observes better heat transfer and comprehensive understanding of a heat exchanger.
The major outcome can be summarized as below:

- The proposed modified design of the heat exchanger yields better outcome as compared to the design parameters available in previous literature.
- As the cold fluid temperature goes down, the steady state condition for heat transfer can be achieved at a faster rate with net average temperature of 323.80K; correspondingly the system achieves a low hot outlet temperature with a small marginal error of 0.66%.
- The mass flow rate is a dominant factor of designed heat exchanger and hence as it increases, the hot fluid temperature suffers a fall in temperature to about 325.45K with a marginal error of 1.05% compared to the present results.
- As the cold fluid rate goes up, the steady state condition for heat transfer gets lowered significantly. Moreover, with the change in boundary conditions, the hot outlet temperature also goes down up to a certain extent.
- The pressure and temperature contours show a higher velocity and pressure magnitudes along the outer pipes as compared to the inner pipe.

Moreover, new materials with better thermal properties for the construction of counter flow heat exchanger could lead to bright and improved changes in heat transfer. The present work can further be enhanced by considering the Nusselt and Reynolds numbers, stressing models to improve the flow characteristics in the system, and comparing the results with other computational methods.

Acknowledgement

The authors would like to thank Amity University Haryana, India for providing computational support in carrying out this valuable research work.

References

[1] J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, P. K. Vijayan and R. Bhoi, “Experimental and CFD estimation of heat transfer in helically coiled heat exchangers”, *Chem. Eng. Res. Des.*, Vol. 86, No. 3, pp. 221-232, (2008).

[2] M. M. Mandal and K. D. P. Nigam, "Experimental study on pressure drop and heat transfer of turbulent flow in tube in tube helical heat exchanger", *Ind. Eng. Chem. Res.,* Vol. 48, No. 20, pp. 9318-9324, (2009).

[3] D. Z. Stevic, "Mathematical Modelling of the Recuperative Heat Exchangers-The Comparative Analysis and Geometric Optimization", *Int. J. Info. Syst. Sci., Institute for Scientific Computing and Information*, Vol. 6, No. 4, pp. 435-455, (2010).

[4] V. Bansal, R. Misra, G. D. Agrawal and J. Mathur, "Performance analysis of earth–pipe–air heat exchanger for summer cooling", *Energ. Buildings*, Vol. 42, No. 5, pp. 645-648, (2010).

[5] R. J. Kee, B. B. Almnd, J. M. Blasi, B. L. Rosen, M. Hartmann, N. P. Sullivan, H. Zhu, A. R. Manerbino, S. Menzer, W. G. Coors and J. L. Martin, "The design, Fabrication and Evaluation of a Ceramic Counter-Flow Micro Channel Heat Exchanger", *Appl. Therm. Eng.*, Vol. 31, pp. 11-12, (2011).

[6] H. Demir, A. S. Dalkilic, N. A. Kürekci, W. Duangthongsuk and S. Wongwises, "Numerical investigation on the single phase forced convection heat transfer characteristics of TiO2 nanofluids in a double-tube counter flow heat exchanger", *Int. Commun. Heat Mass,* Vol. 38, No. 2, pp.218-228, (2011).

[7] G. Huminic and A. Huminic, "Heat transfer characteristics in double tube helical heat exchangers using nanofluids", *Int. J. Heat Mass Transf.*, Vol. 54, pp. 4280-4287, (2011).

[8] H. A. Mohammed and K. Narrein, "Thermal and hydraulic characteristics of nanofluid flow in a helically coiled tube heat exchanger", *Int. Commun. Heat Mass,* Vol. 39, No. 9, pp.1375-1383, (2012).

[9] A. Kamyar, R. Saidur and M. Hasanuzzaman, "Application of computational fluid dynamics (CFD) for nanofluids" *Int. J. Heat Mass Transf.*, Vol. 55, pp. 4104-4115, (2012).

[10] N. Targui and H. Kahalerras, "Analysis of a double pipe heat exchanger performance by use of porous baffles and pulsating flow", *Energy Convers. Manag.*, Vol. 76, pp. 43-54, (2013).

[11] H. J. Xu, Z. G. Qu and W. Q. Tao, "Numerical investigation on self-coupling heat transfer in a counter-flow double-pipe heat exchanger filled with metallic foams", *Appl. Therm. Eng.*, Vol. 66, No. 1-2, pp.43-54, (2014).

[12] A. K. Tiwari, P. Ghosh, J. Sarkar, H. Dahiya and J. Parekh, "Numerical investigation of heat transfer and fluid flow in plate heat exchanger using nanofluids", *Int. J. Therm. Sci.*, Vol. 85, pp.93-103, (2014).

[13] C. S. Oon, H. Togun, S. N. Kazi, A. Badarudin and E. Sadeghinezhad, "Computational simulation of heat transfer to separation fluid flow in an annular passage", *Int. Commun. Heat Mass*, Vol. 46, pp. 92-96, (2013).

[14] S. Nagarsheth, U. Pandya and H. Nagarsheth, "Control Analysis Using Tuning Methods for a Designed, Developed and Modeled Cross Flow Water Tube Heat Exchanger", *Int. J. Mech., Aero, Ind. Mechatron. Eng., World Academy of Science and Technology*, Vol. 8, No. 12, pp. 1889-1894, (2014).

[15] W. Yaïci, M. Ghorab and E. Entchev, "3D CFD analysis of the effect of inlet air flow maldistribution on the fluid flow and heat transfer performances of plate-fin-and-tube laminar heat exchangers", *Int. J. Heat Mass Transf.*, Vol. 74, pp. 490-500, (2014).

[16] V. Nagarajan, Y. Chen, Q. Wang and T. Ma, "CFD modeling and simulation of sulfur trioxide decomposition in ceramic plate-fin high temperature heat exchanger and decomposer", *Int. J. Heat Mass Transf.*, Vol. 80, pp. 329-343, (2015).

[17] M. R. Safaei, G. Ahmadi, M. S. Goodarzi, M. S. Safdari, H. R. Goshayeshi and M.

Dahari, "Heat transfer and pressure drop in fully developed turbulent flows of grapheme nanoplatelets–silver/water nanofluids", *Fluids*, Vol. 1, No. 3, pp. 20, (2016).

[18] A. Sabaghan, M. Edalatpour, M. C. Moghadam, E. Roohi and H. Niazmand, "Nanofluid flow and heat transfer in a microchannel with longitudinal vortex generators two-phase numerical simulation", *Appl. Therm. Eng.*, Vol. 100, pp. 179-189, (2016).

[19] E. Pal, I. Kumar, J. B. Joshi and N. K. Maheshwari, "CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles", *Chem. Eng. Sci.*, Vol. 143, pp. 314-340, (2016).

[20] R. S. Andhare, A. Shooshtari, S. V. Dessiatoun and M. M. Ohadi, "Heat transfer and pressure drop characteristics of a flat plate manifold microchannel heat exchanger in counter flow configuration", *Appl. Therm. Eng.*, Vol. 96, pp. 178-189, (2016).

[21] M. Mazidi, M. Alizadeh, L. Nourpour and S. V. Shojaeen, "Estimating the unknown heat flux on the wall of a heat exchanger internal tube using inverse method", *Journal of Computational & Applied Research in Mechanical Engineering*, Vol. 5, pp. 127-136, (2016).

[22] K. M. Shirvan, M. Mamourian, S. Mirzakhanlari and R. Ellahi, "Numerical investigation of heat exchanger effectiveness in a double pipe heat exchanger filled with nanofluid: a sensitivity analysis by response surface methodology", *Powder Technol.*, Vol. 313, pp.99-111, (2017).

[23] P. E. B. Mello, H. H. S. Villanueva, S. Scuotto, G. H. B. Donato and F. dos Santos Ortega, "Heat transfer, pressure drop and structural analysis of a finned plate ceramic heat exchanger", *Energy*, Vol. 120, pp. 597-607, (2017).

[24] M. Owen and D. G. Kröger, "A numerical investigation of vapor flow in large air-cooled condensers", *Appl. Therm. Eng.*, Vol. 127, pp. 157-164, (2017).

[25] S. M. Shahril, G. A. Quadir, N. A. M. Amin and I. A. Badruddin, "Thermo hydraulic performance analysis of a shell-and-double concentric tube heat exchanger using CFD", *Int. J. Heat Mass Transf.*, Vol. 105, pp. 781-798, (2017).

[26] S. H. Nagarsheth, D. S. Bhatt and J. J. Barve, "Temperature Profile Modelling, Simulation and Validation for a Counter Flow Water Tube in Tube Heat Exchanger", *Control Conference (ICC), Indian. IEEE*, pp. 1-6, (2017).

[27] A. Behnampour, O. A. Akbari, M. R. Safaei, M. Ghavami, A. Marzban, G. A. S. Shabani and R. Mashayekhi, "Analysis of heat transfer and nanofluid fluid flow in microchannels with trapezoidal, rectangular and triangular shaped ribs", *Physica E: Low-dimensional Systems and Nanostructures*, Vol. 91, pp. 15-31, (2017).

[28] J. Lin, D. T. Bui, R. Wang and K. J. Chua, "On the Fundamental Heat and Mass Transfer Analysis of the Counter Flow Dew Point Evaporative Cooler", *Appl. Energy*, Vol. 217, pp. 126-142, (2018).

[29] A. A. Ahmadi, E. Khodabandeh, H. Moghadasi, N. Malekian, O. A. Akbari and M. Bahiraei, "Numerical study of flow and heat transfer of water-Al_2O_3 nanofluid inside a channel with an inner cylinder using Eulerian–Lagrangian approach", *J. Therm. Anal. Calorim.*, Vol. 132, No. 1, pp. 651-665, (2018).

[30] A. E. Mahfouz, W. A. Abdelmaksoud and E. E. Khalil, "Heat Transfer and Fluid Flow Characteristics in a Heat Exchanger Tube Fitted with Inserts", *J. Therm. Sci. Eng. Appl.*, Vol. 10, No. 3, pp. 031012, (2018).

[31] P. M. Kumar and M. Chandrasekar, tube heat exchanger handling MWCNT/ water nanofluids", *Heliyon*, Vol. 5, No. 7, pp. 02030, (2019).

[32] M. Tusar, K. Ahmed, M. Bhuiya, P. Bhowmik, M. Rasul and N. Ashwath, "CFD study of heat transfer enhancement

and fluid flow characteristics of laminar flow through tube with helical screw tape insert", *Energy Procedia*, No. 160, pp.699-706, (2019).

How to cite this paper:

Shuvam Mohanty*, Om Prakash and Rajesh Arora, "Analytical and comparative investigations on counter flow heat exchanger using computational fluid dynamics", *Journal of Computational and Applied Research in Mechanical Engineering*, Vol. 10, No. 1, pp. 139-152, (2020).

DOI: 10.22061/jcarme.2019.4665.1564

URL: http://jcarme.sru.ac.ir/?_action=showPDF&article=1120