CFD Letters 12, Issue 3 (2020) 98-111

Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Investigation of the Effect of Flow Rate on Fluid Heat Transfer in Counter-Flow Helical Heat Exchanger Using CFD Method

Open Access

Pedram Ghiasi[1], Amar Salehi[2], Seyed Salar Hoseini[1], Gholamhassan Najafi[1,*], Rizalman Mamat[3], Balkhaya[4], Fitri Khoerunnisa[5]

[1] Department of Biosystems Engineering, Tarbiat Modares University, 111-14115, Tehran, Iran
[2] Department of Agricultural Machinery, University of Tehran, 6619-14155, Karaj, Iran
[3] Mechanical Engineering Department, University Malaysia Pahang, 26600, Pekan, Malaysia
[4] Politeknik Aceh Selatan, Tapaktuan, Aceh Selatan, Indonesia
[5] Department of Chemistry, Universitas Pendidikan Indonesia, Indonesia

ARTICLE INFO

Article history:
Received 23 January 2020
Received in revised form 18 March 2020
Accepted 23 March 2020
Available online 29 March 2020

ABSTRACT

Heat exchangers are generally used in the process of heat transfer between two different fluids separated from each other by a solid wall in order to save time and reduce expenses. Fluids behavior change by adding a wire-insert in its path. To investigate heat transfer parameters, we need to simulate the whole system. In this study, heat transfer of counter-flow helical double pipe heat exchanger was modelled by using Computational Fluid Dynamics (CFD) in "Ansys CFX". The cold and hot fluids temperature were in the ranges of 10-20C° and 30-50C° respectively. The Reynolds number of flows were in the range of 4×10^3 to 42×10^3 and the process was single-phase. The model was eventually evaluated by experimental data after simulation. The results indicated that the model was able to interpret the experimental results with correlation coefficients of 0.98 and 0.97 for hot and cold streams respectively. Furthermore, the wire-insert installed to the cold flow path caused more fluid turbulence and increased the temperature difference of the cold fluid inlet and outlet proportional to the hot fluid.

Keywords:
Simulation; Heat Exchanger with wire-insert; Turbulent Flow; Single-Phase flow; Helical Heat Transfer

1. Introduction

In recent years, there have been many researches showing that the rate of heat transfer in helical pipes is much higher than in straight pipes [1]. Helical-pipes heat exchangers, they are mostly used in various industries such as ventilation and heat recovery systems, nuclear industries, power plants, chemical reactors and food industry due to their dense structure and high heat transfer rate. The circular shape of pipes results in secondary flow with rotational motion that causes the fluid particles to move to the central region of the helical section [2]. The secondary flow increases the heat transfer

* *Corresponding author.*
E-mail address: g.najafi@modares.ac.ir (Gholamhassan Najafi)

https://doi.org/10.37934/cfdl.12.3.98111

rate by decreasing the temperature gradient at the cross-section of pipe. Therefore, another convective heat transfer mechanism perpendicular to the main flow that doesn't exist in conventional heat exchangers is produced [3]. Since the flow behavior and convective heat transfer behavior of the helical pipes largely depends on the secondary flow behavior, they are usually more complicated than straight pipes. Nevertheless, many studies have been conducted on fluid flow and heat transfer in helical pipes.

Shell and tubes heat exchanger modelling using CFD technique dates back to the early days of this knowledge, so it's quite difficult to trace its origin in a primitive research. However, the technique presented a study by Patankar & Spalding [4], was most probably the first to be able to calculate fluid heat fields and tube metals simultaneously and could be considered as a framework from which sequential distributions is developed. In another study, heat transfer in an annulus double-pipe helical heat exchanger was experimentally investigated. The results revealed that secondary flow occurred in both inner tube and the annulus. It was also reported that with increasing Dean number the heat transfer increased [5]. Effect of helical wire inserts on heat transfer and pressure drop in a double pipe heat exchanger were investigated with CFD techniques. Larger pitch length of coiled wire insert, led to increasing in efficiency of heat exchanger [6]. Seban & McLaughlin [7] experimentally studied heat transfer in helical pipes for the laminar flow of oil and the turbulent flow of water. Reynolds number range from 6000 to 6500 and Prandtl number range from 2.9 to 5.7 were used in their study. In the relationship presented in their study, it was assumed that the fluid properties were constant, which can cause a heat transfer coefficient error of 10 percent in the calculated values. Further they found that with increasing the Reynolds number heat transfer was increased also this result achieved by Ham *et al.,* [8]. Rehman *et al.,* [9] discussed flow distribution and heat transfer in shell-and-tube heat exchangers and compared their proposed model with the experimental results. Their model had an average error of 20% in prediction of the heat transfer and pressure drop. The results indicated that the assumption of plane symmetry was well applicable to heat exchanger lengths but not for outlet and inlet. Although, this model could be improved by using Reynold model instead of k-ε model. Ghazikhani *et al.,* [10] put effort on experiments to investigate the effect of different kinds of vortex generator on the performance of a heat exchanger. The results showed that the use of wire-insert in the air flow path not only improved heat transfer but also caused overall irreversibility of heat exchanger. Baruah *et al.,* [11] carried out an investigation of the performance of elliptical pin fin heat exchangers computationally. They found that the fluid temperature increased as it passed through the heat exchanger and the temperature difference between the fins and the surrounding fluid decreased over the computational domain. The flow structure and heat distribution inside the heat exchanger shell can be obtained by geometric modelling with as much accuracy as possible. These accurate data can be used to calculate general parameters such as heat exchanger coefficient and pressure drop, that can be compared with computational and experimental results as well [12]. Comparison of three heat exchangers in different heat flux showed that the dimpled helical coiled tube had a higher heat transfer coefficient. The helix tube and dimpled tube had a positive effect on heat transfer [13-14].

In some heat exchanger applications it is very important to know the temperature and pressure of all fluid elements. In experimental tests we are just able to see fluids input and output elements but in CFD simulation all elements are available. So a simulation process was conducted to investigate the fluid behavior. The interaction of pressure, velocity and temperature fields inside the heat exchanger were also studied precisely. In this study, a double-pipe heat exchanger with counter-flow wire-insert was simulated. The effect of wire-insert installed on inner pipe on heat transfer was also investigated. Additionally, the computational heat transfer data for the heat exchanger were compared to the results of the simulation.

2. Methodology
2.1 Structure and Operation of Heat Exchanger

The setup of this research is a double-pipe helical heat exchanger whose dimensions are shown in Table 1. The experimental experiments consisted of a double-pipe helical heat exchanger, the cooling and heating loops and the relevant instrumentation. The temperature of cold and hot streams were in the ranges of 10-20 C° and 30-50 C° respectively. Mass flow rate of both streams were in the range of 1-8 $kg.h^{-1}$. A cooper wire of 1 mm diameter was coiled and soldered as spiral wire-insert around the outer wall of the inner pipe inside the annulus. The pitch of wire-insert was 40 cm and had a 1 mm gap for air flow in the remaining free space of the annulus. In order to investigate the effect of the hot and the cold mass flow rates on heat transfer, the mass flow rate of the hot stream was kept constant in 3.9 $kg.h^{-1}$, while the cold mass flow rate was increased from 4 to 5.6 $kg.h^{-1}$ with step of 0.2 $kg.h^{-1}$. Similarly, the mass flow rate of the cold stream was kept constant 4 kg h^{-1}, and the mass flow rate of the hot stream was changed from 4.1 to 5.7 kg h^{-1} with step of 0.2 kg h^{-1}. Altogether, the model has been run for 18 times; 9 times for the hot flow changes and 9 times for the cold flow changes.

Table 1
Dimensions of the heat exchanger

Thickness (mm)	Dimensions (mm)	Components of the heat exchanger
1.25	12	Inner diameter of outer pipe
0.15	7.7	Inner diameter of inner pipe
	3770	The length of the heat exchanger
	100	The height of the heat exchanger
	300	Spiral diameter
Pitch of 40 mm	1	Diameter of wire-insert

2.2 Geometry Model
2.2.1 Physical model

According to the circular cross section of heat exchanger, the mesh was generated by division of cylinders edge. Moreover, bias type was selected; so that, small and large mesh size were applied to short distances from the pipe wall and far distances from the pipe respectively, to investigate the boundary layer effect. First, the meshing process was applied by choosing fewer amount of division number for the edge of spheres and thus the average skewness index, aspect ratio and the number of elements were studied. The parameters of 4 types of mesh with different division numbers for hot fluid and cold fluid are shown in Figure 1 and Figure 2. D mesh type with aspect ratio of 1:2 and also better feature and more compatibility was selected and eventually applied. Figure 3 shows the mesh design for D mesh type. The geometry of heat exchanger was created in SolidWorks 2018 and then was recalled in geometry section of Ansys environment (Figure 4).

Input, output and interaction between fluid and object were defined after the meshing process for hot and cold fluid, inner pipes and outer pipes. The zero velocity condition was compared for the analysis of the interaction between hot fluid and inner surface of inner pipe, cold fluid and outer surface of inner pipe, and cold fluid and inner surface of outer pipe. The heat exchanger were surrounded by glass wool when experimental test was being performed, so the outer surface of outer pipe was defined as zero thermal flux.

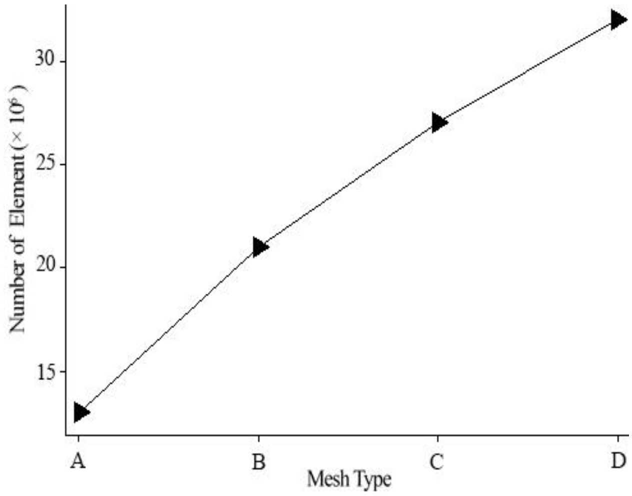


Fig. 1. Number of different mesh type

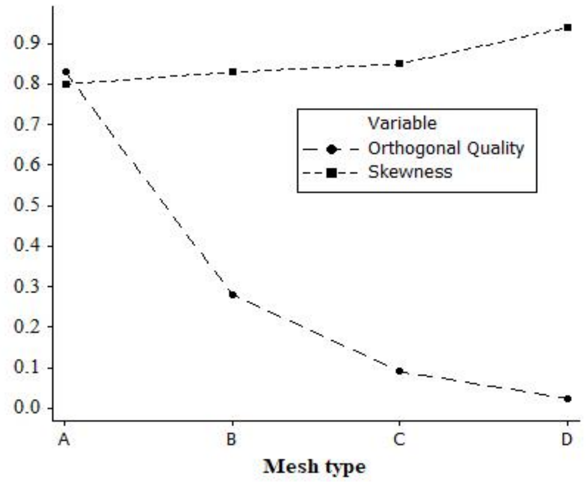


Fig. 2. Quality parameters of different mesh type

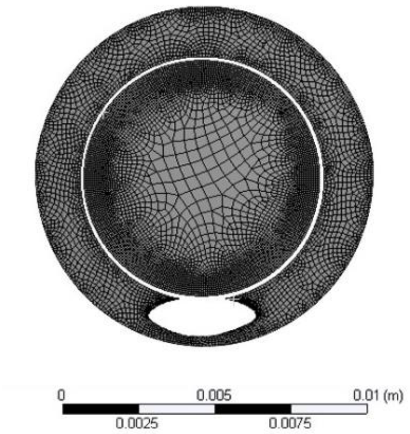


Fig. 4. Mesh design of heat exchanger

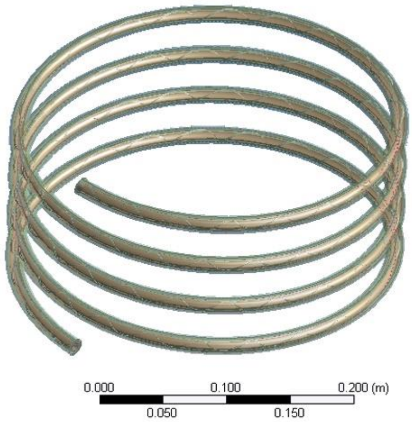


Fig. 3. Heat exchanger geometry

2.2.2 Description of numerical model

The main models were simply estimated by Navier-Stokes Equations. The relationship among flow rate, area and velocity was used to calculate Reynolds number theoretically, thereupon, the velocity of hot and cold fluid were specifically estimated. For the test condition, Reynolds number was calculated higher than 4000 for the both fluids so the flows are turbulent. The study of Computational Fluid Dynamics (CFD) in a system is normally initiated by developing an optimal geometry and a mesh for domain modelling. The modelling was started with definition of the boundary and initial conditions for the domain and then it resulted in the modelling of total system domain. Afterwards, it was proceeded with analyzing the results. CFD methods consist of numerical solutions of mass, momentum and energy conservation with other equations like species transfer the solution of these equations was done by the numerical algorithm and methods [15]. Conservation equations for compressible flow are:
Continuity equation:

$$\frac{\partial \rho}{\partial t} + div(\rho U) = 0 \tag{1}$$

In CFD programs, the continuity equation is often combined with momentum equation to form Poisson Eq. (2). For constant density and viscosity, the new equation can be written as below.

$$\frac{\partial}{\partial x_i}\left(\frac{\partial P}{\partial x_i}\right) = -\frac{\partial}{\partial x_i}\left(\frac{\partial(\rho U_i U_j)}{\partial x_j}\right) \tag{2}$$

Reynolds equations:

$$\begin{aligned}
&\frac{\partial(\rho u)}{\partial t} + div(\rho u U) = -\frac{\partial p}{\partial x} + div(\mu.grad.u) + \left[-\frac{\partial(\rho u^2)}{\partial x} - \frac{\partial(\rho uv)}{\partial y} - \frac{\partial(\rho uw)}{\partial z}\right] + S_{mx}\\
&\frac{\partial(\rho v)}{\partial t} + div(\rho v U) = -\frac{\partial p}{\partial y} + div(\mu.grad.v) + \left[-\frac{\partial(\rho uv)}{\partial x} - \frac{\partial(\rho v^2)}{\partial y} - \frac{\partial(\rho vw)}{\partial z}\right] + S_{my}\\
&\frac{\partial(\rho w)}{\partial t} + div(\rho w U) = -\frac{\partial p}{\partial z} + div(\mu.grad.w) + \left[-\frac{\partial(\rho uw)}{\partial x} - \frac{\partial(\rho vw)}{\partial y} - \frac{\partial(\rho w^2)}{\partial z}\right] + S_{mz}
\end{aligned} \tag{3}$$

It is needed a model of the turbulence to solve these equations. There are many equations to model the turbulence. The k-ε model is one of the most reliable models that exists. This model is applied following turbulence parameters:

$$\begin{aligned}
&\mu_{eff} = \mu + \mu_T\\
&\mu_T = \rho C_\mu \frac{K^2}{\epsilon}\\
&\frac{\partial(\rho K)}{\partial t} + div(\rho K U) = div\left(\frac{\mu_{eff}}{\sigma_L} grad.K\right) + G - \rho\epsilon\\
&\frac{\partial(\rho\epsilon)}{\partial t} + div(\rho\epsilon U) = div\left(\frac{\mu_{eff}}{\sigma_L} grad.\epsilon\right) + C_{1\varepsilon}\frac{\epsilon}{K} 2\mu_{eff}.E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\epsilon^2}{K}
\end{aligned} \tag{4}$$

2.2.3 Boundary condition

Boundary conditions in the model are summarized as follow:

i. Inner and outer tube side mass flow rate: 1-8 $kg.h^{-1}$
ii. The temperature of cold stream: 10-20 C° and The temperature of hot stream: 30-50 C°
iii. Pressure outlet in both tube sides: 1 bar
iv. No-slip boundary condition on the walls
v. Adiabatic wall boundary condition on the outer wall
vi. The coupled temperature on the interface wall

2.3 Basic Equations for Heat Exchanger

The energy balance between the hot and cold streams (Q) was calculated using the following equation:

$$Q = F_c \rho_c C_{p_c}(T_{c,o} - T_{c,i}) = F_h \rho_h C_{p_h}(T_{h,i} - T_{h,o}) \tag{5}$$

Where F_c, ρ_c, C_{pc}, $T_{c,i}$ and $T_{c,o}$ are volumetric flow rate, density, specific temperature, the inlet temperature of cold stream and the outlet temperature of cold stream respectively; and h subscript in all parameters refer to the hot stream.

$$Q = U_i A_i LMTD = F_h \rho_h Cp_h(T_{h,i} - T_{h,o}) \tag{6}$$

Then;

$$U_iA_iExp = \frac{F_h\rho_hCp_h(T_{h,i}-T_{h,o})}{LMTD} \tag{7}$$

Where U_i is heat transfer; and A_i is the inside surface area of internal pipe.

Heat transfer for the heat exchanger were modelled using experimental tests. This model can be different by changing the structure of heat exchanger. There are various methods used to predict the amount of heat transfer of heat exchanger in which not only the heat transfer of inner pipe and outer pipe are not separated from each other but also a same equation is used to calculate the heat transfer of the inner and the outer pipes. Eq. (10) presented by Majidi *et al.,* [16] is particularly for heat transfer of heat exchanger with the separation feature for cold and hot streams (r^2 = 0.98). Their proposed equation was based on the equations presented by Rohsenow *et al.,* [17] and Garimella *et al.,* [18] which after modifying and applying wire-insert effects, was accordingly set for the heat exchanger whose information was already mentioned.

Eq. (8) is used for inner pipe of heat exchanger and Eq. (9) is presented for heat transfer of outer pipe;

$$h_{in} = \frac{k(0.023Re^{0.8}pr^{0.3})(1+3.4\frac{di,o-2t}{D})}{di,o-2t} \tag{8}$$

$$h_{out} = \frac{k(0.027De^{0.94}pr^{0.69})(\frac{do,i-di,o}{D})^{0.01}}{D_{eq}} \tag{9}$$

The amount of U_iA_i is calculated by substituting the results of the previous equations in the following equation;

$$U_iA_iCal. = \left[\frac{1}{h_{in}} + \frac{\ln(\frac{di,o}{di,o-2t})}{2\pi kL} + \frac{di,o-2t}{h_{out}d_{i,o}}\right]^{-1} A_i \tag{10}$$

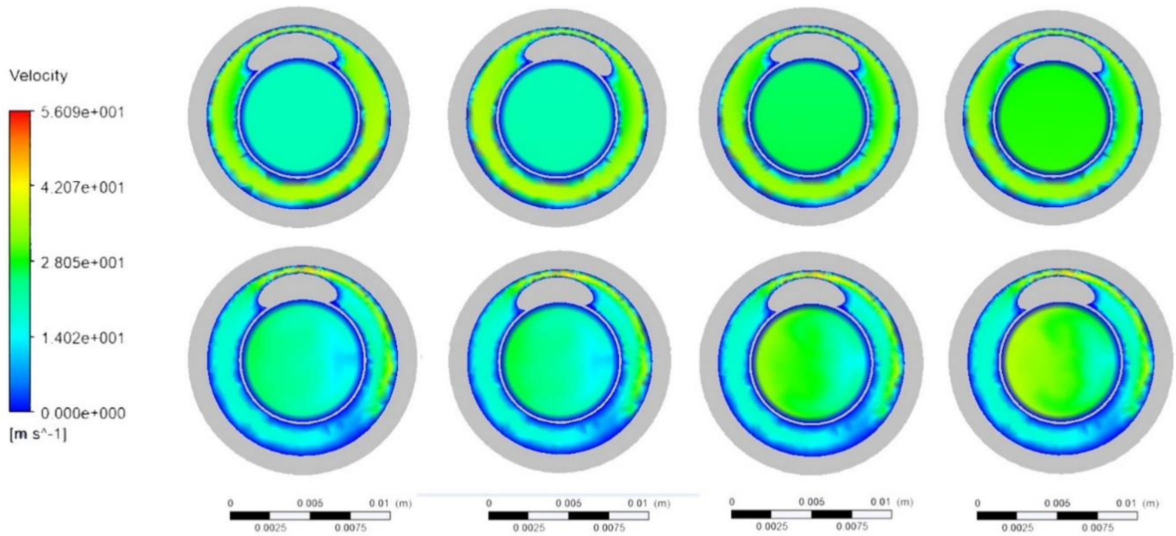
Eq. (10) is used to obtain the amount of heat transfer in order to be compared with the modelling results.

3. Results

3.1 Fluid Velocity

Fluid velocity at the cross section of the inlet and outlet of the heat exchanger are shown in Figure 5. The first row represents the hot air inlet and the cold air outlet. The second row shows the hot air outlet and cold air inlet in various mass flow rate for hot air. In this study, nine simulations (models) in various rates for the cold and the hot streams were conducted. Four inlet and outlet cross sections of the heat exchanger were used to show the simulated cross sections. Due to the friction between the fluid and the pipe, fluid velocity around the pipe was zero, however, it reached to 60 $m.s^{-1}$ in the farther distances from the pipe wall. The linear velocity of the hot air inside the pipe was increased in farther distances from the center of the circle around which the whole exchanger was coiled, because of the exchanger torsion. This can be seen from the color difference in the cross section of the hot air inlet of the exchanger, as the increase in flow rate let to more velocity difference. The spiral radius effect of the heat exchanger couldn't be observed in the first moment of the hot air influx inside the pipe, and the entire cross section of hot air fluid moved with approximately constant velocity except for walls; because the fluid didn't pass along the circular path in the exchanger. Due to Venturi effect, the fluid velocity increased in the space between the coil and the outer pipe, however, it reached to average value in farther distances of the pipes wall. Part of hot fluid particles were forced to rotate in the coil direction due to collision with the coil. The other part of hot fluid particles passed through the space between coil and wall. So the Kinetic energy of those cold fluid particles that were forced to rotate in the coil direction decreased slightly. This decrease in the particles velocity was significant in some parts of cross section of the cold fluid. The air particles transfer occurred rarely in the space between the coil and the outer pipe after that the cold fluid passed across the direction and also after that the pressure was stabilized in the both sides of the coil. Accordingly, the increase in velocity that occurred in the cold air inlet, wouldn't be observed in the space between the coil and outer pipe of the cold air outlet.

By comparing the heat exchanger cross sections in various flow rates shown in Figure 5 as a, b, c and d, it can be seen that the increase in the hot mass flow rate has no significant influence on the cold fluid velocity and the increase in flow rate just results in a change in kinetic energy of hot fluid particles. For this reason, the velocity spectrum in the cross section of cold fluid in the heat exchanger inlets is the same in various flow rates. The same result is obtained from the cold fluids outlets.



a b c d

Fig. 5. First row shows the inlet hot and outlet cold air cross sections and second row shows the outlet hot and inlet cold air cross sections of flow velocity of heat exchanger in various mass flow rate that cold mass flow rate and hot mass flow rate for a, b, c and d is (4 $kg.h^{-1}$, 4.1 $kg.h^{-1}$), (4 $kg.h^{-1}$, 4.7 $kg.h^{-1}$), (4 $kg.h^{-1}$, 5.3 $kg.h^{-1}$) and (4 $kg.h^{-1}$, 5.7 $kg.h^{-1}$) respectively.

Figure 6 shows the effect of changing the cold mass flow rate on the velocity field in the cross section of the exchanger pipes. The exchanger spiral effect on the hot fluid velocity is observed in the fluid outlet. By the same token, the velocity in the walls is zero for the fluid.

In higher flow rates in the cold fluid inlet, the velocity increase was observed much more in the space between the wire-insert and the pipe, and Venturi effect was more considerable. As can be seen in Figure 6, the installed wire-insert in the path of cold fluid flow resulted in turbulence and improvement in the heat transfer. By comparing a, b, c and d it can be found that mass flow rate directly let to increasing in flow velocity.

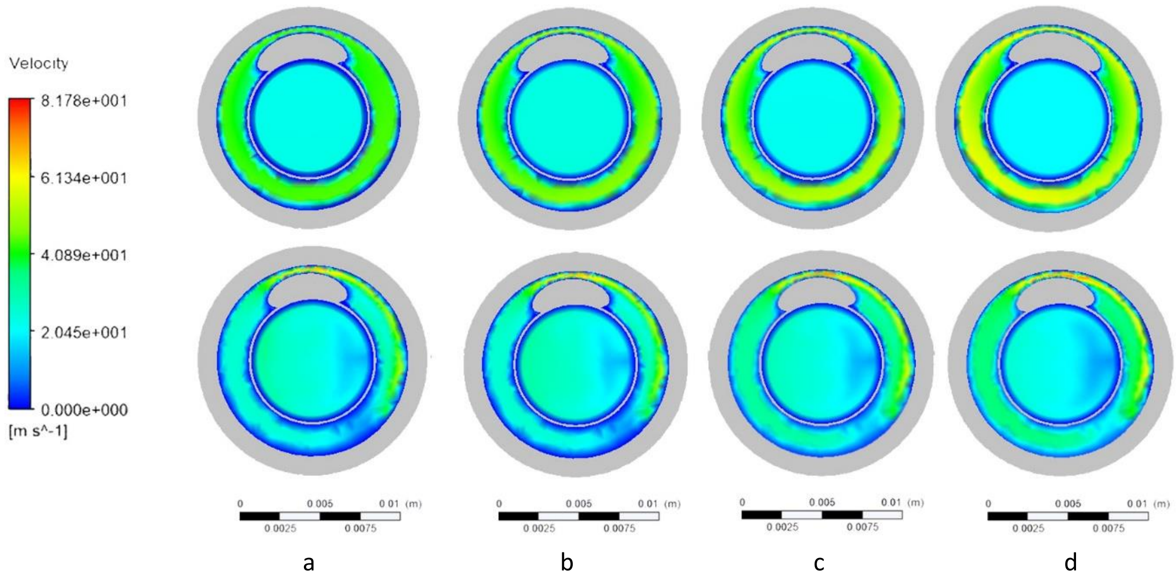


Fig. 6. First row shows the inlet hot and outlet cold air cross sections and second row shows the outlet hot and inlet cold air cross sections of flow velocity of heat exchanger in various mass flow rate which cold mass flow rate and hot mass flow rate for a,b,c and d is (4 $kg.h^{-1}$, 3.9 $kg.h^{-1}$), (4.6 $kg.h^{-1}$, 3.9 $kg.h^{-1}$), (5.2 $kg.h^{-1}$, 3.9 $kg.h^{-1}$) and (5.6 $kg.h^{-1}$, 3.9 $kg.h^{-1}$) respectively.

3.2 The Fluid Temperature Contour

The temperature of the pipes with copper coiled wire with thermal conductivity coefficient of 390 (W/m ˚C) increased as a result of hot fluid passing through them. In addition, the cold fluid temperature increased owing to cold air collision with copper components, so this led to temperature change in both fluids conclusively due to passing through the pipes. The temperature field of fluid in the cross section of the heat exchanger inlet and outlet are shown in Figure 7. The first row represents the hot air inlet and the cold air outlet, and the second row shows the hot air outlet and the cold air inlet in various mass flow rates of hot air. The temperature difference between the wall and the center of the fluid at the hot fluid inlet shows that there was no time for the fluid and the inner pipe to transfer heat. As the hot fluid passed along the heat exchanger, and heat transfer occurred, the temperature differences observed in the inlet would be dwindled and the fluid would go out of the exchanger with a temperature less than the inlet temperature as it can be seen as "D" in the second row of Figure 7 in the hot fluid outlet. By increasing the flow rate, the hot fluid velocity increased, and consequently it increased heat transfer. According to the cross section of cold fluid outlet shown in Figure 8, the increase in the cold flow rate let to increase in cold fluid temperature, and considering the temperature difference of the both fluids, the heat transfer was improved. As can be seen in Figure 8, increasing cold flow rate leads to increase in the fluid temperature in the outlet. By comparing Figure 7 (a, b, c and d) and Figure 8 (a, b, c and d), it can be inferred that the temperature difference of the cold fluid in Figure 8 is more than Figure 7. Although an increase in the cold fluid temperature was observed in the circumstances that hot

The temperature in the space between the coiled wire and the wall of the outer pipe increased simply because of more efficient surface for heat transfer, and also more fluid turbulence. The coiled wire in the path of cold flow let to an increase in the area of heat exchange between the pipe and fluid. On the other hand, more turbulence in the fluid caused an increase the heat transfer rate. This results was also observed in the study of Maakoul *et al.,* [19].

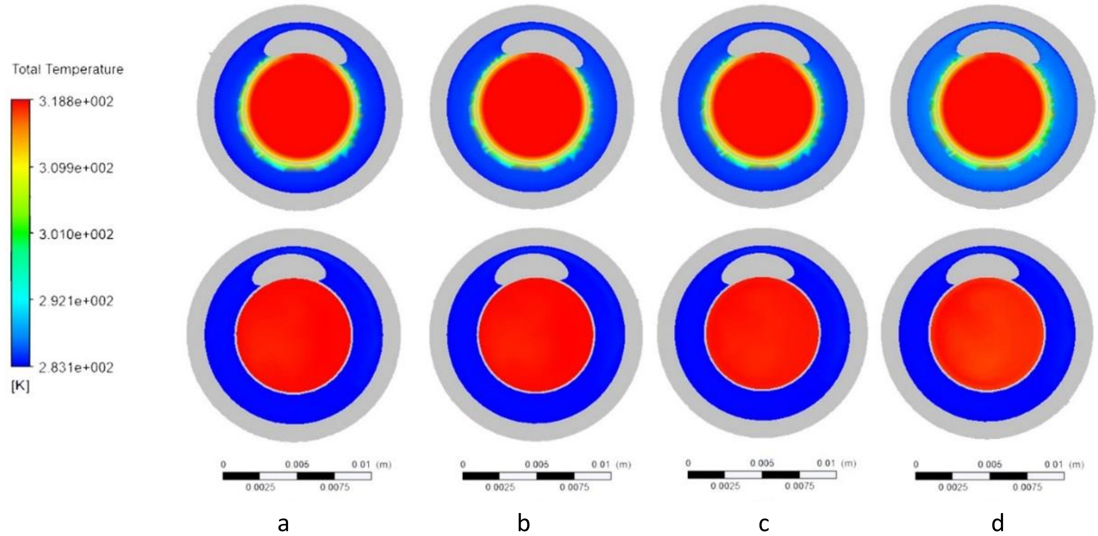


Fig. 7. First row show the inlet hot and outlet cold air cross sections and second row show the outlet hot and inlet cold air cross sections of fluid temprature of heat exchanger in various mass flow rate that cold mass flow rate and hot mass flow rate for a,b,c and d is (4 kg.h^{-1}, 4.1 kg.h^{-1}), (4 kg.h^{-1}, 4.7 kg.h^{-1}), (4 kg.h^{-1}, 5.3 kg.h^{-1}) and (4 kg.h^{-1}, 5.7 kg.h^{-1}) respectively.

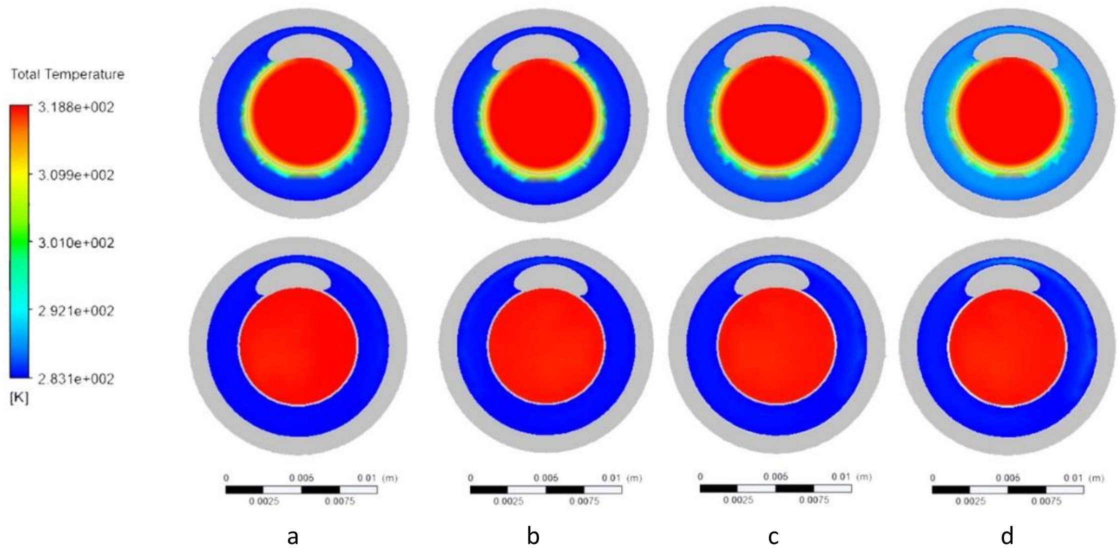
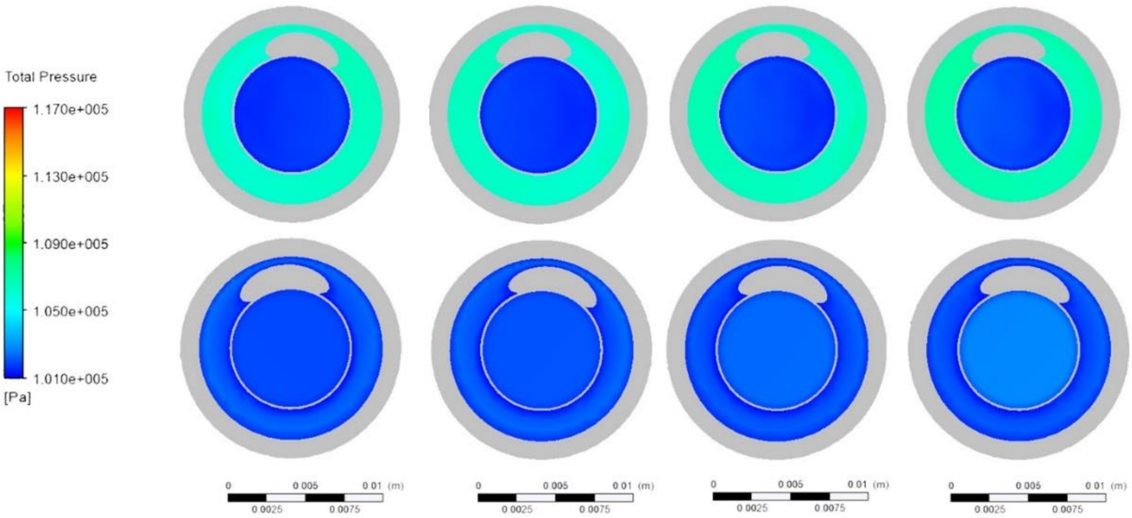


Fig. 8. First row show the inlet hot and outlet cold air cross sections and second row show the outlet hot and inlet cold air cross sections of fluid temprature of heat exchanger in various mass flow rate that cold mass flow rate and hot mass flow rate for a,b,c and d is (4 kg.h^{-1}, 3.9 kg.h^{-1}), (4.6 kg.h^{-1}, 3.9 kg.h^{-1}), (5.2 kg.h^{-1}, 3.9 kg.h^{-1}) and (5.6 kg.h^{-1}, 3.9 kg.h^{-1}) respectively.

3.3 The Fluid Pressure Contour

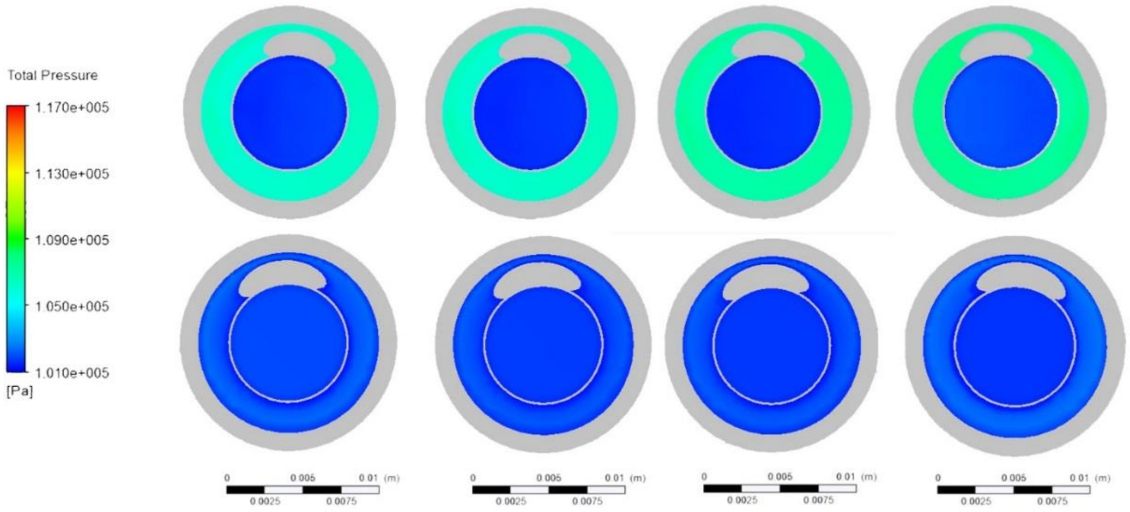
The first and the second rows of Figure 9 shows the cross sections of the inlet and the outlet in various flow rates for hot fluid respectively. As a matter of fact, the fluid pressure variation are caused by the temperature variation during the movement from the inlet to the outlet. After the hot fluid passed the pipes, a temperature drop occurred in the hot fluid and the fluid pressure decreased in the outlet. Moreover, increasing the flow rate for the hot fluid resulted in a reduction in the temperature and the pressure evidently.

As the cold fluid passed through the pipe, the cold fluid temperature increased and the heat was eventually transferred from the hot fluid to the cold. The heat transfer was observed far more in the cold fluid; because of the wire coil, so as it can be seen in Figure 10, the input and output temperature variations of the cold fluid were more than the hot fluid, therefore, as a result of the temperature variations, the input and the output pressure differences in the cold fluid were notably higher. Part a, b, c and d in Figure 9 and Figure 10 show the effect of mass flow rate on pressure in the pipes.



a b c d

Fig. 9. First row show the inlet hot and outlet cold air cross sections and second row show the outlet hot and inlet cold air cross sections of fluid pressure of heat exchanger in various mass flow rate that cold mass flow rate and hot mass flow rate for a,b,c and d is (4 $kg.h^{-1}$, 4.1 $kg.h^{-1}$), (4 $kg.h^{-1}$, 4.7 $kg.h^{-1}$), (4 $kg.h^{-1}$, 5.3 $kg.h^{-1}$) and (4 $kg.h^{-1}$, 5.7 $kg.h^{-1}$) respectively.



a b c d

Fig. 10. First row show the inlet hot and outlet cold air cross sections and second row show the outlet hot and inlet cold air cross sections of fluid pressure of heat exchanger in various mass flow rate that cold mass flow rate and hot mass flow rate for a,b,c and d is (4 $kg.h^{-1}$, 3.9 $kg.h^{-1}$), (4.6 $kg.h^{-1}$, 3.9 $kg.h^{-1}$), (5.2 $kg.h^{-1}$, 3.9 $kg.h^{-1}$) and (5.6 $kg.h^{-1}$, 3.9 $kg.h^{-1}$) respectively.

3.4 The Effect of Hot Mass Flow Rate on UiAi

Mass flow rate of the cold flow was kept constant in order to detect the effect of hot flow on heat transfer rate, however, mass flow rate of the hot flow was changed. Figure 11 shows the results of the test for both mass flow rate of cold flow which was kept constant in 4 kg h^{-1} and mass flow rate of hot flow which was changed from 4.1 kg h^{-1} to 5.7 kg h^{-1} with step of 0.2 kg h^{-1}. The temperature of cold input flow and hot flow were set on 10±0.1 °C and 45±0.1 °C respectively. The temperature of the water that was warmed up electrically, was decreased in order to keep the hot flow temperature constant; because the increase in flow rate in the hot flow let to increase in convective hot transfer of the hot flow in heat cycle and therefore the increase in hot flow temperature. Temperature contorl of cold flow in the outlined level was basically easy; because its flow rate was remained constant.

Experimental tests were undertaken in software model to investigate the effect of flow rate on heat transfer of heat exchanger. Figure 11 presents the results of heat tranfer simulation and experimental results of hot fluid in various rates. Accordingly, the increase in hot flow rate in inner pipe of heat exchanger was associated with increase in heat transfer of hot flow. Additionally, the temperature difference between the input and output in the inner pipe increased with increasing flow rate. Furthermore, the increase in flow rate from 0.00145 kg.s^{-1}, reduced the upward trend of heat transfer. The conductivity between the fluid and inner wall of inner pipe was augmented as the flow rate increased and then resulted in an increase in heat transfer from the hot fluid to inner pipe and also from inner pipe to cold fluid. The temperature difference between the cold and hot fluid was decreased after the increase in flow rate from 0.00145 kg.s^{-1} and therefore caused a decrease in heat transfer gradient (Figure 11). Differences between experimental and CFD results increased in higher flow rates; because the increase in fluid flow resulted in an increase in the flow turbulence. It also complicated the computation. Regression relationship between the simulation and experimental test is shown in Figure 12. The regression coefficient indicates that the experimental results fit well with the simulation.

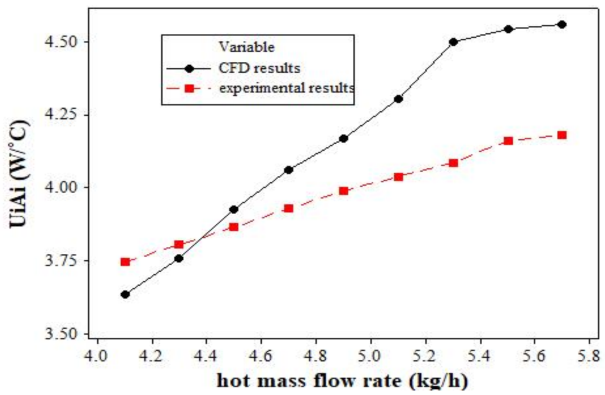


Fig. 11. (UiAi) versus hot mass flow rate for the experimental results and CFD results

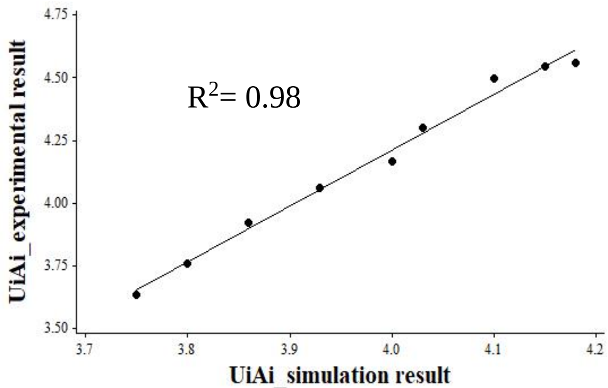


Fig. 12. Regression relationship between experimental results and simulation results for various hot mass flow rate

3.5 The Effect of Mass Flow Rate on UiAi

To investigate the effect of the mass flow rate of cold flow on OHTC, the mass flow rate of hot flow was kept constant. Figure 13 shows the comparison between the suggested coefficient for the time that mass flow rate of hot flow was kept constante in 3.9 kg h^{-1} and the time that the mass flow rate of cold flow was changed from 4 kg h^{-1} to 5.6 kg h^{-1} with step of 0.2 kg h^{-1}. The temperature of input flows were the same; 10 ± 0.1 °C for cold flows and 45 ± 0.1 °C for the hot flow.

To study the effect of flow rate on heat transfer of heat exchanger, experimental tests were undertaken in software model. According to the results of the heat tranfer simulation of cold fluid in various rates that are shown in Figure 10, the increase in cold flow rate in outer pipe of heat exchanger was associated with increase in heat transfer of cold flow. The temperature difference between input and output in the inner pipe increased with increasing flow rate. Furthermore, the increase in flow rate from 0.0015 kg.s^{-1}, reduced the upward trend of heat transfer; because the temperature difference between the both fluid decreased due to the more heat transfer. Figure 14 shows the regression relationship between the simulation and the experimental test. The regression coefficient indicates that the experimental results fit well with the simulation.

By comparing Figure 11 and Figure 13 for various flow rates and the heat transfer amounts, it can be inferred that the coil around the inner pipe wall was effectively useful and resulted in an increase in heat transfer of the cold fluid in comparison with the hot fluid. The Positive effect of coiled wire in tha path of fluid on the rate of heat transfer was observed in several studies. Affirmatively, in the same inflow due to the existence of coiled wire, more increase was observed in the heat transfer rate of cold fluid proportional to the cold fliud [13-19-20-21]. The amount of differences between experimental and CFD results in the heat transfer of cold stream were more than hot flow. It can be inferred from the regression coefficient of both flows. The reason of this low accuracy in CFD results for the cold flow is the existence of the coiled wire which made the geometry of model very complicated in the program.

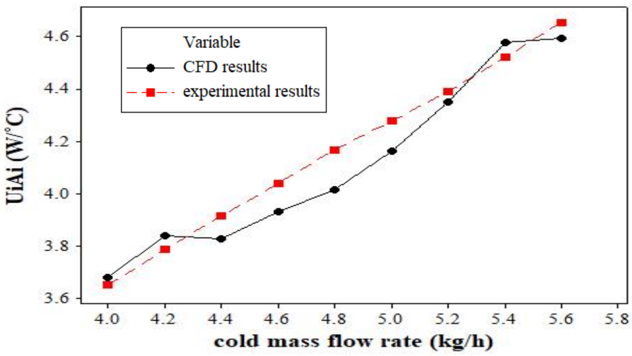


Fig. 13. (UiAi) versus cold mass flow rate for the experimental results and CFD results

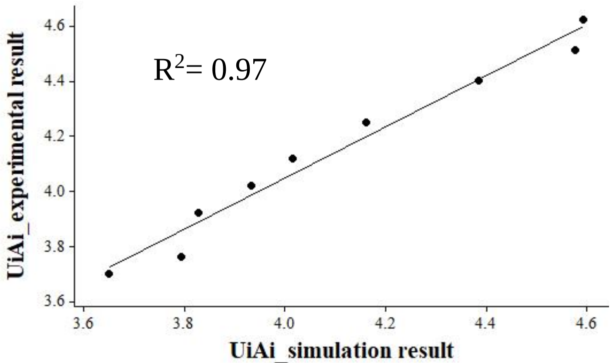


Fig. 14. Regression relationship between experimental results and simulation results for various cold mass flow rate

4. Conclusions

In this study, the performance of a heat exchanger with a wire-insert installed in the cold flow path was investigated using Computational Fluid Dynamics (CFD). The regression coefficient of the model indicates that the CFD model fits well with the presented equation by Majidi et.al. The accuracy of obtained results for the hot fluid is slightly higher than the cold fluid, which is due to the simper geometry of the hot flow motion path. As the mixing and the turbulence of fluid particles in heat exchangers increase, the rate of heat transfer increases. However, the heat transfer rate in the cold fluid was observed to be more compared to the cold fluid; because of the coiled wire in the path of cold fluid. Increasing the inlet flow rate of both hot and cold fluids resulted in an increase in the heat transfer rate of both fluids. However, increasing the flow rate more than a specific amount, the diagram slope of the heat transfer decreased because of a less temperature difference between both fluids and high velocity of the fluid in higher rates. Due to high costs and short times, some different adaptations, tests, and experiments have been assigned for the future works. Future works will be focused in comparison between helical tube and hairpin double pipe heat exchanger. The effect of flow rate on fluid heat transfer in cross flow and multi-pass split flow heat exchangers also will be studied in future works.

Acknowledgement
This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

References

[1] Berger, S. A., L. Talbot, and L. S. Yao. "Flow in curved pipes." *Annual review of fluid mechanics* 15, no. 1 (1983): 461-512.
https://doi.org/10.1146/annurev.fl.15.010183.002333
[2] Naphon, Paisarn, and Somchai Wongwises. "A review of flow and heat transfer characteristics in curved tubes." *Renewable and sustainable energy reviews* 10, no. 5 (2006): 463-490.
https://doi.org/10.1016/j.rser.2004.09.014
[3] Ohadi, Michael, Xiang Zhang, Hadi Keramati, Martinus Arie, Farah Singer, Ratnesh Tiwari, and Amir Shooshtari. "Recent developments in high temperature heat exchangers: A review." *Frontiers in Heat and Mass Transfer (FHMT)* 11 (2018).
https://doi.org/10.5098/hmt.11.18
[4] Patankar, S. V., and D. B. Spalding. "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows." *International Journal of Heat and Mass Transfer* 15, no. 10 (1972): 1787-1806.
https://doi.org/10.1016/0017-9310(72)90054-3
[5] Rennie, Timothy J., and Vijaya GS Raghavan. "Experimental studies of a double-pipe helical heat exchanger." *Experimental Thermal and Fluid Science* 29, no. 8 (2005): 919-924.
https://doi.org/10.1016/j.expthermflusci.2005.02.001
[6] Sharifi, Khashayar, Morteza Sabeti, Mehdi Rafiei, Amir H. Mohammadi, and Laleh Shirazi. "Computational fluid dynamics (CFD) technique to study the effects of helical wire inserts on heat transfer and pressure drop in a double pipe heat exchanger." *Applied Thermal Engineering* 128 (2018): 898-910.
https://doi.org/10.1016/j.applthermaleng.2017.08.146
[7] Seban, R. A., and E. F. McLaughlin. "Heat transfer in tube coils with laminar and turbulent flow." *International journal of heat and mass transfer* 6, no. 5 (1963): 387-395.
https://doi.org/10.1016/0017-9310(63)90100-5
[8] Ham, Jeonggyun, Yunchan Shin, and Honghyun Cho. "Theoretical investigation of the influence of pipe diameter and exit channel width in welded plate heat exchanger on heat exchanger performance." *Heat and Mass Transfer* (2019): 1-13.
https://doi.org/10.1007/s00231-019-02733-8
[9] U. U. Rehman, "Heat transfer optimization of shell-and-tube heat exchanger through CFD studies" (2011): 25–26.
[10] M. Ghazikhani, E. Noorifar, and A. Sharafi, "Experimental investigation of different kinds of vortex generator on a gas liquid finned-tube heat exchanger using exergy analysis," *Journal of Solid and Fluid Mechanics* 2, no. 4 (2013): 61–75.
[11] Baruah, Monoj, Anupam Dewan, and Pinakeswar Mahanta. "Performance of Elliptical Pin Fin Heat Exchanger with Three Elliptical Perforations." *CFD letters* 3, no. 2 (2011): 65 – 73.
[12] Heydari, Ali, Mostafa Shateri, and Sina Sanjari. "Numerical analysis of a small size baffled shell-and-tube heat exchanger using different nano-fluids." *Heat Transfer Engineering* 39, no. 2 (2018): 141-153.
https://doi.org/10.1080/01457632.2017.1288052
[13] Solanki, Anand Kumar, and Ravi Kumar. "Condensation of R-134a inside dimpled helically coiled tube-in-shell type heat exchanger." *Applied Thermal Engineering* 129 (2018): 535-548.
https://doi.org/10.1016/j.applthermaleng.2017.10.026
[14] Solanki, Anand Kumar, and Ravi Kumar. "Two-phase flow condensation heat transfer characteristic of R-600a inside the horizontal smooth and dimpled helical coiled tube in shell type heat exchanger." *International Journal of Refrigeration* 107 (2019): 155-164.
https://doi.org/10.1016/j.ijrefrig.2019.07.017

[15] Zuo, Jia-Wei, Kok-Cheong Wong, and Hoon Kiat Ng. "The Thermal Performance of Three-Layered Microchannel Heat Sink with Tapered Channel Profile." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 56, no. 1 (2019): 147-156.
[16] Majidi, Davood, Hashem Alighardashi, and Fatola Farhadi. "Experimental studies of heat transfer of air in a double-pipe helical heat exchanger." *Applied Thermal Engineering* 133 (2018): 276-282. https://doi.org/10.1016/j.applthermaleng.2018.01.057
[17] Rohsenow, Warren M., James P. Hartnett, and Young I. Cho. *Handbook of heat transfer*. Vol. 3. New York: McGraw-Hill, 1998.
[18] GARIMELLA, S., DE RICHARDS, and RN CHRISTENSEN. "Experimental investigation of heat transfer in coiled annular ducts." *Journal of heat transfer* 110, no. 2 (1988): 329-336. https://doi.org/10.1115/1.3250488
[19] El Maakoul, Anas, Mustapha El Metoui, Abdellatif Ben Abdellah, Said Saadeddine, and Mohamed Meziane. "Numerical investigation of thermohydraulic performance of air to water double-pipe heat exchanger with helical fins." *Applied Thermal Engineering* 127 (2017): 127-139. https://doi.org/10.1016/j.applthermaleng.2017.08.024
[20] Mule, Bandu A., D. N. Hatkar, and M. S. Bembde. "Analysis of Double Pipe heat exchanger with Helical fins." *International Research Journal of Engineering and Technology* 4 (2017): 961-966.
[21] Ahmad, Waseem, Khalid Saifullah Syed, Muhammad Ishaq, Ahmad Hassan, and Zafar Iqbal. "Numerical study of conjugate heat transfer in a double-pipe with exponential fins using DGFEM." *Applied Thermal Engineering* 111 (2017): 1184-1201. https://doi.org/10.1016/j.applthermaleng.2016.09.171