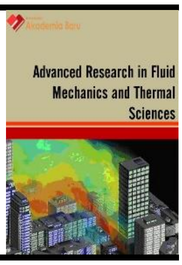
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Experimental Analysis of the Effect of Tube Pass Length on The Characteristics of Heat Transfer in U-Sharp Turned Channels

Open Access

Ahmad Syuhada[1,*], Muhammad Ilham Maulana[1,♣], Agus Adria[2]

[1] Department of Mechanical Engineering of Syiah Kuala University, Darussalam–Banda Aceh 23111, Indonesia
[2] Department of Electrical Engineering of Syiah Kuala University, Darussalam–Banda Aceh 23111, Indonesia

ARTICLE INFO

Article history:
Received 1 December 2018
Received in revised form 30 March 2019
Accepted 2 June 2019
Available online 15 August 2019

ABSTRACT

Pipe-shaped ducts with sharp turns are commonly used for flow passages on heat exchangers. Heat transfer characteristics on a sharp turn channel are very complicated; this is due to the emergence of secondary flow caused by turns. In response to this need some researchers have carried out both experimental and numerical studies of heat / mass transfer and fluid flow in rectangular channels with sharp turns of 180 ° for high-speed flow with forced convection heat transfer. Research on the characteristics of heat transfer in tubes with sharp turn technology is still very minimal. Therefore, experimental studies have been conducted to understand the characteristics of convection heat transfer on a U-shaped tube channel, which has a total length of 6 meters, with four variations in the length of the pass and different number of turns. Four combinations of Heat Exchanger (HE) with tube length of 60 cm, 45 cm, 30 cm and 15 cm were drained by hot water into a tube with a temperature of 60^0C, 700C and 80^0C, while cooling water with a constant temperature of 26^0C. The test is carried out until the temperature of the hot water comes out stable with the interval of taking data for 5 minutes. Turbulence flow will increase the heat transfer coefficient on the tube surface. The highest heat transfer coefficient occurs in HE with a length of 30 cm and has 30 turns, while the lowest heat transfer coefficient occurs in HE with a 60 cm pass length and 14 turns.

Keywords:
Heat exchanger; pass length; sharp turn; turbulence flow

1. Introduction

Pipe-shaped channels (tubes) with sharp turns are commonly used for flow passes on thermal equipment, especially in heat exchangers. The characteristics of heat transfer on a sharp turn channel are very complicated, this is due to the emergence of a secondary flow in the flow caused by a turn from the direction of flow, separation and recovery of the flow pattern that occurs around sharp turns [1,2]. A large difference in the rate of heat transfer between the surfaces will increase the thermal stress on the equipment component [3,4]. Data from a complete study of the problem of

* *Corresponding author.*
E-mail address: syuhada_mech@yahoo.com (Ahmad Syuhada)
♣ *Corresponding author.*
E-mail address: ilhammaulana@unsyiah.ac.id (Muhammad Ilham Maulana)

local heat transfer in this channel is needed for the design of critical heating or cooling of high temperature heat exchanger equipment.

In order to respond to this, need some researchers have carried out both experimental and numerical studies of heat/mass transfer and fluid flow in rectangular channels with sharp turns of 180° for high-speed flow with forced convection heat transfer. Applications from the research are usually used for high technology that is often designed to build in developed countries such as for internal cooling channels from gas turbine blades. For forced convection heat/mass transfer, the problem studied is Reynold (Re) number to get Nusselt number (Nu) for determining heat transfer characteristics, and Sherwood (Sh) number for assessment of mass transfer characteristics [5-11].

The study seeks to characterise the breakdown of the steady two-dimensional solution in the flow around a 180-degree sharp bend to infinitesimal three-dimensional disturbances using a linear stability analysis. The stability analysis predicts that three-dimensional transition is via a synchronous instability of the steady flows. A highly accurate global linear stability analysis of the flow was conducted with Reynolds number $Re < 1150$ and bend opening ratio (ratio of bend width to inlet height) $0.2 \leq \beta \leq 5$. This range of Re and β captures both steady-state two-dimensional flow solutions and the inception of unsteady two-dimensional flow [12].

One of the most important concerns of hydraulics engineers is predicting erosion at the outer banks of rivers by studying the flow pattern along the bend. Not only are the streamlines in meanders non parallel curved lines, but they are also twisted. To study the streamlines flowing a sharp bend, a 180° sharp bend was constructed at Persian Gulf University in Iran. Three dimensional flow components at different locations of the bend were measured using Vectrinovelocim [13].

The effects of air-induced duct diameter on flow and heat transfer of multiple impingement jet are investigate experimentally and numerically in 5 columns x 5 rows of pipe nozzles, which have diameter of d=17.2 mm, was arranged with inline configuration. The diameter of air-induced duct (D) were varied at D/d=2, 4 and 6, and Reynolds numbers was fixed at Re=30,000 [14]. In another study, the recycled concentric tube heat exchanger was designed with the principle of recycled heat transfer technique over a short concentric tube, has been successfully constrained the heat transfer area of the conventionally long arranged or floating tubes to a reduced size in terms of cycles/minute [15].

A mathematical model has been derived and used to develop a three-dimensional concentrating solar collector as presented in this article. The developed solar collector gives the required flux distribution along the longitudinal direction of tubular absorber. The model requires inputs like the profile of required flux distribution, local solar flux, dimensions of the absorber and the distance of absorber from the reflector [16]. Outlet temperature of the collector for efficiency of water-in-glass evacuated tube solar domestic hot water system with natural circulation is generally more than the flat plate collector systems. Additionally, this type of evacuated tube solar collectors with the free circulation is economically cheaper than heat pipes collector systems, this system is widely used in the world [17]. Heat losses of absorber tubes of parabolic trough collectors for the collector field of 250 kW Shiraz (Iran) solar thermal power plant is evaluated for various conditions. For analysis, both experimental measurements and numerical modelling are made to find the impact of failure of heat collecting tubes. The amounts of heat losses are compared numerically for 3 different types of tubes; vacuum, lost vacuum (air) and broken glass (bare) tube. The experimental measured data are used to validate the numerical simulation [18].

Other numerical studies have also examined the heat exchange of mixed convection in laminar (3D) three-dimensional cubic cavities. This study predicted the behaviour of the flow structure between Multi clear structure dominated by natural convection when the Reynolds number is small, and a Multi clear structure by forced convection when the Reynolds number is high [19]. Computational Fluid Dynamics (CFD) is widely used to investigate heat transfer, fluid flow, chemical reaction and mass transfer phenomenon. While solving the Navier-Stokes equations, the convection term is always prone to numerical instability and therefore the discretisation of the convection term requires special attention. The performance of various convection schemes had been previously performed on one-dimensional convection-diffusion problem [20].

Recent development of nanotechnology has led to the concept of using suspended nanoparticles in heat transfer fluids to improve the heat transfer properties of the base fluids. The heat transfer enhancement by nanofluids is the significant concern in the efficiency of domestic water heat exchanger system. A computational investigation of the heat transfer in a domestic water heat exchanger was conducted on the water and water-based nanofluids. Copper (Cu) nanoparticle and alumina (Al2O3) nanoparticles were selected in the water-based nanofluids [21].

A CFD simulation analysis about enhancement of turbulent flow heat transfer in a horizontal circular pipe by convenient software FLUENT was used to predict the heat transfer coefficient and Nusselt number for forced convection heat transfer of Ag/HEG+water nanofluid. The range of Reynolds number selected were 20,000 and 40,000 in a horizontal straight tube of diameter 0.01 m with heat flux of 1000 W/m^2 [22]. Considering that the feasibility of increasing Ag / Heg heat transfer is still controversial. The study of forced convective heat transfer in nanofluid silver / graphene (Ag / Heg) turbulent pipe flow in stainless steel straight circular pipes with 0.01 geometry in hydraulic diameter and 0.8 m length was also carried out using numerical studies (Ansys Fluent) [23]. Numerical studies also conducted on turbulent incompressible flow over 2D backwardfacing step in order to investigate the performance of three different turbulence models (standard k-e, realizable k-e and SST k-ω) in predicting the region of separation and reattachment behind the edge of the step [24].

Optimum design of HEN can cause significant reduction in the total cost of the plant. Targeting method using pinch analysis diagrams was presented to find out investment cost required and the period of return of the investment of optimization of the refinery system. This method can be done by knowing the amount of ΔTmin and by pointing the composite curve of saving vs investment (S-I curve). The targeting method is the modification of the system that need to be done to avoid movement of heat exchangers in order to minimize the return of the investment [25].

From previous studies, it appears that more research is done by numerical methods. While research on the characteristics of heat exchangers experimentally especially those that study on the characteristics of heat transfer in tubes with sharp turn technology is still very minimal. That is why the author has conducted an experimental study of the influence of the length of the tube pass to the characteristics of heat transfer on a sharp U-channel. In this study, it will be analysed how far the influence of the tube pass length on the application of U-shaped sharp turn technology to the characteristics of heat transfer. The total length of the pipe used is 6 meters. From the length of the pipe each will be formed 4 Heat Exchanger (HE) units with different lengths of passes. HE with a tube length of 60 cm have 14 sharp U turns HE with a tube length of 45 cm has 22 sharp turns U, HE with a 30 cm tube length has 30 sharp turns U and HE with a 15 cm tube length has 55 sharp U turns.

2. Methodology

This study tested experimentally the effect of tube pass length on the characteristics of heat transfer on U-sharp turns. For this purpose, four units of tube-type with the same length of pipe are 6 m, but have different lengths of passage. In this case, HE has a number of U turns different from one another. The four HE units as shown in Figure 1 and the HE dimensions are shown in Table 1.

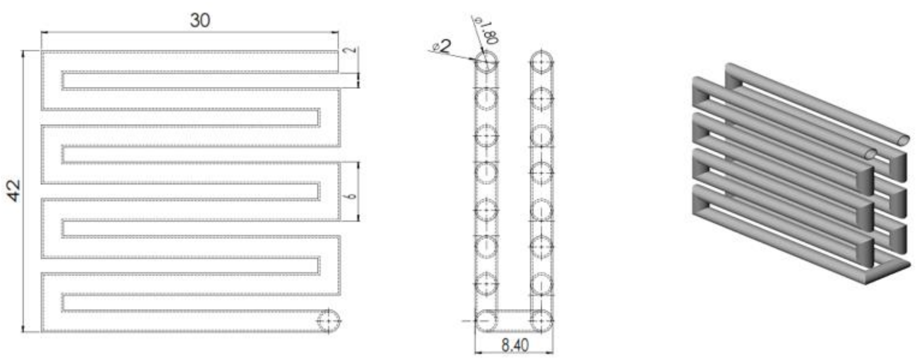


Fig. 1. Heat Exchanger Scheme tested

Table 1
Dimensions of the Heat Exchanger

Type	N Turn	Tube Length	HE Length
A	14	60 cm	18 Cm
B	22	45 cm	30 Cm
C	30	30 cm	42 Cm
D	55	15 cm	84 Cm

Testing support equipment used in this study is a heater and pump. Heater is a heating element that functions to heat water for the needs of hot water as a medium that will be cooled at a certain temperature, and the pump as a distributor of hot water that will be distributed into the tube. The required temperature was measured using a digital thermometer using a thermocouple cable K-type. The material used is galvanized iron pipe with an overall length of each 6 m heat exchanger, with a tube outer diameter (Do) of 22 mm and an inside tube diameter (Di) of 20 mm.

3. Results

3.1 Experimental Results

In this test, hot water is fed into the HE tube with three different temperature conditions, namely 60°C, 70°C and 80°C.While the cooling water fluid has a constant temperature of 260°C. After measuring the temperature distribution in the four HEs, it was obtained as shown in Table 2 for hot water entering HE at 80°C.

Table 2
Temperature distribution in HE with the incoming temperature of 800°C obtained from the measurement results

			Hot Water		Cool Water
HE Type	Tube length	N Turn	T_{in}(^{0}C)	T_{out}(^{0}C)	T∞(^{0}C)
A	60 Cm	14	80	71.3	26
B	45 Cm	22	80	70.3	26
C	30 CM	30	80	65.4	26
D	15 Cm	55	80	67.3	26

3.2 Data Analysis

The measurement results as shown in Table 2 after being changed to graphical temperature distribution as shown in Figure 2 shows a decrease in temperature of hot water in the tube after cooling for 10 minutes. The flow of hot fluid entering with a temperature of 80°C is seen only to experience a transient temperature decrease in the initial 5 minutes. In a heat exchanger that has 14 turns, a temperature drops of 8.7°C occurs. In a heat exchanger that has 22 turns there is a temperature decrease of 9.7°C. The heat exchanger with 30 turns has a temperature decrease of 14.60°C and the heat exchanger 55 turns a temperature decrease of 12.70°C. Then in the 10th to 50th minute the temperature of the hot water has a very small regular decrease until the steady state is reached.

From Figure 2 it is clear that the largest temperature gradient between the inlet and the outlet occurs at a 30 Cm pass length with a total of 30 U-turn.

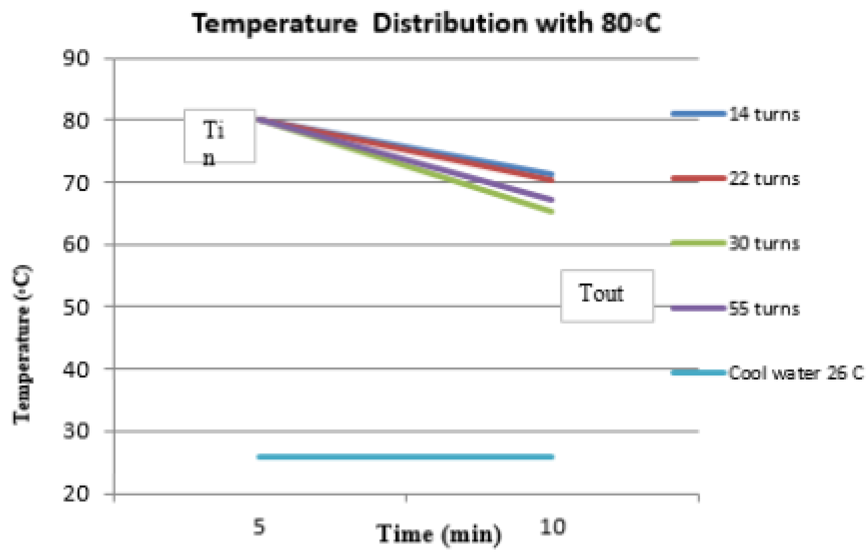


Fig. 2. Temperature distribution of tube cooling with 80^0C inlet temperature for U-HE Turning Variations

Figure 3 shows the temperature distribution of the 30 Cm long pass with 30 turns at the temperature in the tube 60°C, 70°C and 80°C. This condition clarifies the difference in gradient temperature between the tube inlet temperature and the tube out temperature which is very real, showing the same heat transfer characteristics for each temperature in the tube 60°C, 70°C and 80°C.

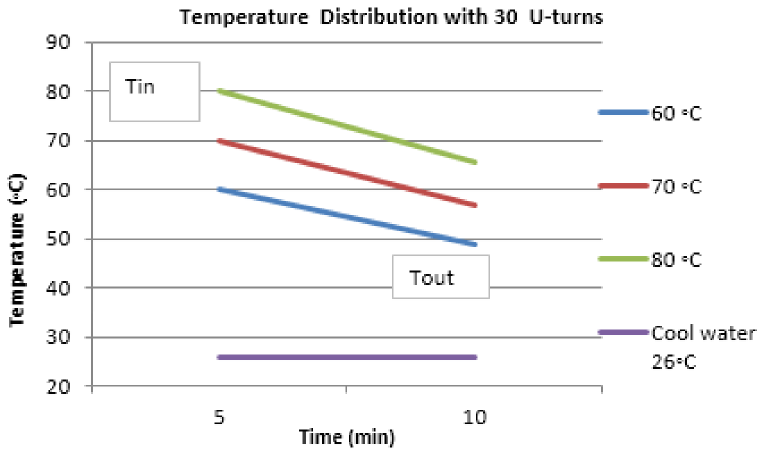


Fig. 3. HE Temperature Distribution of 30 Cm long pass and 30 turns with temperature entering tube 60°C, 70°C and 80°C

4. Discussion

From the results of the measurement of temperature distribution in the HE system that is tested as obtained from Table 1 and Figure 2, after the analysis is obtained, coefficient of convective heat transfer hc is obtained as shown in Figure 4.

Figure 4 shows that HE with a 60 cm pass length has the lowest convection heat transfer coefficient, because of the lack of flow turbulence fields along the pipe. The effect of turbulence is what causes the convection heat transfer coefficient (hc) along the tube pass. On a tube that is 45 cm long the convection heat transfer coefficient is higher than the 60 cm tube length. This is because the number of turn U is more than 22, while the length of the tube 60 cm is only 14 turns. Tube is the highest coefficient of heat transfer convection which has a tube length of 30 cm with a number of 30 turns. But on a tube that is shorter than 30 cm which is 15 cm long with a number of turn 55 coefficient the convection heat transfer is lower than the 30 cm tube. But this value is still higher than the convection heat transfer coefficient of a tube that is 45 cm long.

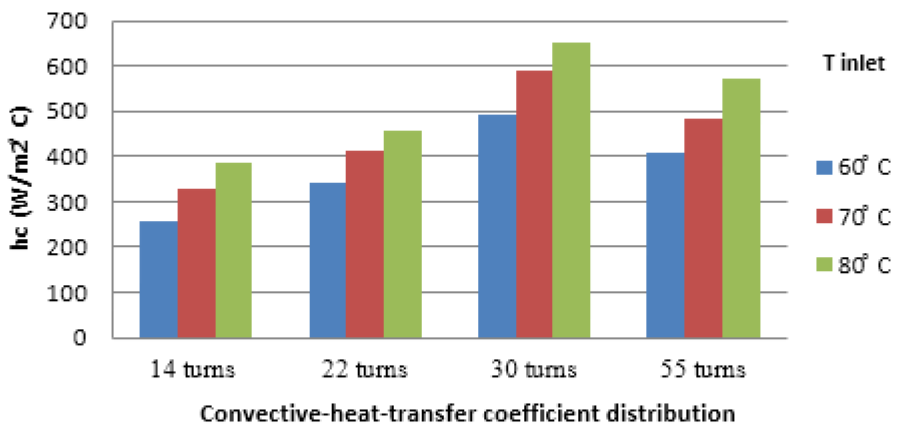


Fig. 4. The coefficient distribution of convective heat transfer hc to the length of the HE tube pass

Thus, the results of this study indicate that the highest convection heat transfer coefficient occurs at 30 cm pass length with a number of 30 turns. Followed by the value of convection heat transfer coefficient on 16 cm tube length with the number of turns 55 pieces. The lowest convection heat transfer coefficient is the one on the tube length of 60 cm with the number of 14 turns followed by the convection heat transfer coefficient of the tube length of 45 cm with the number of turns 22.

A large mass / heat transfer rate occurs in a sharp turn 180 degree square rectangular and is caused by high turbulence at bends and flows after bends (1). Turbulence occurs in the flow entering the turn and after the turn along a distance of 10-14 Dh depending on the turning distance (10). The value of convection heat transfer coefficient increases slowly in the direction of flow at the turn. A sudden change in flow direction causes a low hc area at the turn and after the turn. The heat transfer coefficient soars very high reaching the local maximum near the meeting on the final wall and outside wall.

The characteristic phenomenon of rectangular channel convection heat transfer with sharp turns applies also to the heat transfer characteristics of U-turn flow in a pipe-shaped channel. Tubes that have a length of 60 cm are 33 times the size of the tube diameter, while those that are 45 cm long are 25 times the diameter, which are 30 cm long, 16 times the diameter and 15 cm long, 8 times the diameter.

From this study it can be stated that the influence of the length of the pass on the U turn is very real if the length of the tube is about 10-16 times the diameter of the tube. With another statement stated that the highest convection heat transfer coefficient occurs in the tube length between 10-16 times the tube diameter.

So that in the design of a heat exchanger it needs to be optimized, where we must be able to design a heat exchanger with a high convex heat transfer coefficient so that we can use relatively small HE dimensions.

5. Conclusions

The results of the current study can be summarised as follow;

i. The number of turns or pass length tube influences the convection heat transfer coefficient value.
ii. The value of local convection heat transfer coefficient increases due to turbulence of flow in certain regions of the flow.
iii. Flow turbulence after turns still occurs from the turn area up to 10-16 times tube diameter.

Acknowledgement
This research was funded by a grant from Syiah Kuala University.

References
[1] Hirota, M., H. Fujita, A. Syuhada, S. Araki, T. Yoshida, and T. Tanaka. "Heat/mass transfer characteristics in two-pass smooth channels with a sharp 180-deg turn." *International Journal of Heat and Mass Transfer* 42, no. 20 (1999): 3757-3770.
[2] Syuhada, A., M. Hirota, H. Fujita, S. Araki, M. Yanagida, and T. Tanaka. "Heat (mass) transfer in serpentine flow passage with rectangular cross-section." *Energy conversion and management* 42, no. 15-17 (2001): 1867-1885.
[3] Sahm, M. K. "Heat Transfer Around Sharp 180-deg Turns in Smooth Rectangular Channels." *Journal Heat Transfer* 108, no. 3 (1986): 500-506.
[4] Chyu, M. K. "Regional heat transfer in two-pass and three-pass passages with 180-deg sharp turns." *Journal of Heat Transfer* 113, no. 1 (1991): 63-70.
[5] Syuhada, A., M. Hirota, H. Fujita, S. Araki, M. Yanagida, and T. Tanaka. "Heat (mass) transfer in serpentine flow passage with rectangular cross-section." *Energy conversion and management* 42, no. 15-17 (2001): 1867-1885.
[6] Chandra, P. R., J. C. Han, and S. C. Lau. "Effect of rib angle on local heat/mass transfer distribution in a two-pass rib-roughened channel." *Journal of Turbomachinery* 110, no. 2 (1988): 233-241.
[7] Chang, S. M. "Turbulent flow in a strongly curved U-bend and downstream tangent of square cross-sections." *Phys. Chem. Hydrofyn.* 3 (1983): 243-263.

[8] Han, J. C., P. R. Chandra, and S. C. Lau. "Local heat/mass transfer distributions around sharp 180 deg turns in two-pass smooth and rib-roughened channels." *Journal of Heat Transfer*110, no. 1 (1988): 91-98.
[9] Hirota, M., H. Fujita, A. Syuhada, S. Araki, M. Yanagida, and T. Tanaka. "Heat/Mass Transfer Characteristics in Serpentine Flow-Passage with a Sharp Turn, (Influence of Entrance Configuration)." *Proc. Compact Heat Exchangers and Enhancement Technology for Proces Industries, Banff* (1999): 159-166.
[10] Ahmad Syuhada and Muhammad Ilham Maulana. "Study of Heat Transfer Characteristics on Sharp Turn Channels for Solar Collectors." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 45, no.1 (2018): 82-91.
[11] Sapardi, Azan M., Wisam K. Hussam, Alban Pothérat, and Gregory J. Sheard. "Linear stability of confined flow around a 180-degree sharp bend." *Journal of Fluid Mechanics* 822 (2017): 813-847.
[12] Akbari, Maryam, and Mohammad Vaghefi. "Experimental investigation on streamlines in a 180º sharp bend." *Acta Scientiarum. Technology* 39, no. 4 (2017): 425-432.
[13] Hussain, Muhammad Ibrar, Esmail MA Mokheimer, and Shakeel Ahmed. "Optimal Design of a Solar Collector for Required Flux Distribution on a Tubular Receiver." *Journal of Energy Resources Technology* 139, no. 1 (2017): 012006.
[14] Kirttayoth Yeranee, Makatar Wae-hayee, Yu Rao, Chayut Nuntadusit. "Effects of Air-Induced Duct Diameter on Flow and Heat Transfer Characteristics of Multiple Impinging Jets. " *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 51, no.2 (2018): 247-258.
[15] Bako, M. D., A. Nasir, M. S. Abolarin, N. A. Musa, and Muhammadu Masin Muhammadu. "Design and Perfomance Evaluation of a Short Recycled Concentric Tube Heat Exchanger for Raw Milk Pastuerization." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 54, no. 2 (2019): 204-220.
[16] Mohammadkarim, Ahmadreza, Alibakhsh Kasaeian, and Abdolrazagh Kaabinejadian. "Performance investigation of solar evacuated tube collector using TRNSYS in Tehran." *International Journal of Renewable Energy Research (IJRER)*4, no. 2 (2014): 497-503.
[17] Yaghoubi, M., F. Ahmadi, and M. Bandehee. "Analysis of heat losses of absorber tubes of parabolic through collector of Shiraz (Iran) solar power plant." *Journal of Clean Energy Technologies* 1, no. 1 (2013): 33-37.
[18] Syuhada, Ahmad, and Muhammad Ilham Maulana. "Characteristics of heat transfer on solar collector channel by using a sharp turn." In *AIP Conference Proceedings*, vol. 1984, no. 1, p. 020003. AIP Publishing, 2018.
[19] Benderradji, Razik, and DjedidTaloub Hamza Gouidmi. "Numerical study three-dimensional of mixed convection in a cavity: Influence of Reynolds and Grashof numbers." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 51, no. 1 (2018): 42-52.
[20] Wah Yen Tey, Ryan Yuen Wye Hong, Yutaka Asako, Hooi Siang Kang, Khai Ching Ng. “Analysis on Computational Efficiency of Convection Discretisation Schemes in SIMPLE Algorithm.” *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 58, no. 1 (2019): 100-117.
[21] Y. K. Lee. “The Use of Nanofluids in Domestic Water Heat Exchanger.”2014, *Journal of Advanced Research in Applied Mechanics* 3, no. 1. (2014): 9-24.
[22] S. Zainal, C. Tan, C. J. Sian, T. J. Siang. “ANSYS simulation for Ag/HEG Hybrid Nanofluid in Turbulent Circular Pipe.” *Journal of Advanced Research in Applied Mechanics* 23, no. 1 (2016): 20-35.
[23] G. Y. Ny, N. H. Barom, S. M. Noraziman, S. T. Yeow. “Numerical Study on Turbulent-Forced Convective Heat Transfer of Ag/Heg Water Nanofluid in Pipe.” *Journal of Advanced Research in Materials Science* 22, no. 1. (2016): 11-27.
[24] D. G. Jehad, G. A. Hashim, A. K. Zarzoor, C. S. Nor Azwadi. “Numerical Study of Turbulent Flow over Backward-Facing Step with Different Turbulence Models.” *Journal of Advanced Research Design* 4, no. 1. (2015): 20-27.
[25] R. Mohammad , F. Sharifi, D. Sharifi, N. Othman, Z. A. Kadir. “Targeting Method of Heat Exchanger Network in Olefin Unit of Oil Refinery.” *Journal of Advanced Research Design* 13, no. 1. (2015): 1-15.