Umudike Journal of Engineering and Technology (UJET); Vol. 9, No. 1, June 2023, pp. 37 – 45; Michael Okpara University of Agriculture, Umudike, Print ISSN: 2536-7404,Electronic ISSN:2545-5257; http://ujetmoua.com; https://doi.org/10.33922/j.ujet_v9i1_5

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF A CROSS FLOW HEAT EXCHANGER

[1]Ntunde, D. I.

[1]Department of Mechanical Engineering,
Michael Okpara University of Agriculture,
Umudike, Nigeria
(Email: ntunde.dilibe@mouau.edu.ng)

ABSTRACT

The paper presents the computational fluid dynamics analysis of a cross flow heat exchanger of hot Freon Fluid tube submerged in cold water domain. The simulation procedures were done for the two inlet velocities of the hot fluid at 0.015m/s and 0.035m/s; for a constant flow velocity of 0.02m/s for the cold water. The result of the temperature distribution showed a decrease from 305 to 301.69K and 305to302.44; and an increase from 300 to 301.75 and 300to302.45 for the hot Freon and cold water fluids respectively. The logarithmic mean temperature difference (LMTD) value for the two inlet velocities of hot Freon fluid were evaluated to be 2.71K and 2.68K; consequently, the required operating average temperature difference of the flow system for the two fluids was established to be 2.70K; while the effectiveness was determined as 89%.Further, the analysis revealed a constant pressure distribution across the water plate, while the Freon fluid increased from 1 to 1.77atm across the tube length. The velocity distribution showed that the cold water velocity increased rapidly around the horizontal and angled orientations of the Freon tube from 0.02 to 0.15m/s; while that of the hot Freon fluid remained constant across the tube length. This change in velocity was attributed to the high rate of heat exchange rate at those sections of the Freon tube. The findings from this study will be used to enhance the design and performance evaluation of heat exchangers to meet future technological demands.

Keywords: *Cross-flow, Fluids, Heat exchanger, temperature distribution*

1.0 INTRODUCTION

In many engineering applications, there is a need to drop the temperature of the flow system to meet its designed requirement, prevent damages and enhance efficient performance. The high temperature in the system can lead to an undesirable high thermal stress; therefore, a system is put in place to continuously drop and keep the system operating within the required temperature range. In most cases temperature reduction is usually achieved with a device known as heat exchanger. Heat exchangers are convective devices that exchange heat between two or more processes fluid streams at different temperature (Promoppatum, 2018). Based on the flow arrangement, most heat exchangers can be classified as parallel flow or concurrent, counter flow or counter current, single pass cross flow or multi pass cross flow heat exchangers. In the parallel flow configuration the hot and cold fluid enter the heat exchanger domain and flow in the same direction within the domain; while the hot and cold fluid enters the flow domain and flow in opposite directions for a counter flow heat exchanger configuration (Zohuri, 2017). In single-pass cross flow configuration, one of the two fluid moves through the heat transfer matrix at right angles to the flow path of the second fluid. In multi-pass cross flow heat exchanger configuration however, the arrangement is such that the single pass flow configuration is repeated several times, making one of the two-fluid stream shuttle back and forth across the flow path of the other fluid stream. Multi pass flow heat exchanger is usually adopted to increase the surface area needed for heat transfer to take place. Hybrids of the above flow types are often found in industrial heat exchangers application; for instance, a condenser for an air condition and refrigeration arrangement can have a multi pass cross flow arrangement.

The design and analysis of these heat exchangers can be done experimentally, numerically or using Computational Fluid Dynamic, CFD software tools. CFD is a science that exploits digital computers to produces quantitative predictions of fluid-flow phenomena based on the law conservation of mass, momentum, and energy governing fluid motion(Hu, 2012). He applied finite element, finite differences and finite volume methods and other numerical methods to solve the governing equations across a specified domain. CFD software tools were developed to help in providing quick solution to heat exchangers, most of which have the capabilities to predict the heat distributions and increase the efficiency of the flow system. There have been several of studies on the performance of cross flow heat exchangers using CFD approaches. The study by Kumar et al., (2013)predicted airflow and temperature distribution in the tube style heat exchanger associated with a large electrical motor, a three-dimensional numerical simulation study. His work was based on using Phoenics software to perform a CFD analysis on the heat exchanger. According to them, the simulated result predicted the temperature distribution reasonably at different locations of the heat exchanger. Consequently, Wasewar et al., (2007)performed an experimental analysis for flow distribution in the header of plate-fin heat exchangers. A preliminary investigation of the flow distribution in the header of the plate fin heat exchanger showed a very serious uniform distribution in the y-direction of the header. The team proposed a modified header configuration, which was again simulated using CFD. The result shows that the modified header configuration had a more uniform flow distribution than the experimental header configuration.

Subsequently, Bhanuchandrarao et. al., (2013)investigated CFD analysis and parallel and counter flow performance in concentric tube heat exchangers. The results were compared between each model and a parallel and counter flow with fouled piping. The Turbulent flow was also analysed during the development of the heat exchangers to determine its effect on the heat transfer. As expected, the heat exchanger had a lower performance and therefore cooled the working fluid less. The investigation by Thomachan et al (2016)for a tube in tube heat exchanger with fins revealed that using fins increased the systems effectiveness. The reason behind maximum effectiveness was due to use of fins, the turbulence was increased as they allowed more mixing of fluid layers that resulted to an increase of heat transfer through the tube. Also, Kadhim et al., (2016) carried out a CFD investigation using ANSYS to study the temperature difference for cross flow heat exchangers with smooth tube and low integral finned tube. Their team used a copper tube with eight passes for air as the cooling fluid. The result of the analysis showed that the temperature differences and heat transfer coefficient for heat exchangers with finned tube is higher than with smooth tube. Kanti et al (2016) investigated CFD analysis of shell and tube heat exchanger. Their calculations and simulations were done for counter flow heat exchanger; while Digvendra et al (2016) investigated CFD analysis was for shell and tube heat exchanger using different headers. These CFD simulations were used for the optimum positioning of the inlet nozzle, which was proposed for a uniform distribution of the liquid methanol and the uniform velocity distribution though each tube. Further Mangrulkar et al., (2017) performed a three-dimensional CFD simulation of cross flow tube bank in staggered arrangements with and without the splitter plate for Reynolds number between 5500 and 14,500. The splitter plate length to tube diameter (L/D) ratio was set as 1 and the longitudinal and transverse tube pitch ratio SL/D and ST/D were maintained constant as 1.75 and 2.0 respectively. Their result of their study showed that provision of the splitter plate increased the Nusselt number for the fluid flow which increase the heat transfer. Other studies such as Rajput & Arya, (2019)work on the heat transfer and temperature gradient of a cross flow heat exchanger using different fin thickness as a case study using CFD simulation showed that the heat transfer rate increased by 15% by using 3 mm fin when compared with the 1.5 mm fin. It was also discovered that the temperature differences increased with increase cooling air velocity. The temperature differences also increased when the hot air velocity inside the tubes are reduced. They recommended further works on CFD ANSYS Fluent computer software tool for future analysis of flow parameters and enhancement of heat exchanger.

CFD simulation can also be used to increase the performance of an existing heat exchanger, even when equipped with heat pipes, as has been proven by Selma et.al, (2020). The improvement in performance resulting from changes in the pipe diameter and the angle between the pipes was investigated within the CFD simulation and then applied to the heat exchanger under investigation. The advantage of this method is a lower simulation time and high adaptability, possibly being used in other heat exchanger designs equipped with heat pipes. Additionally, Soleimanikutanaei et al., (2018) presented a research work titled modelling and simulation of cross-flow transport membrane condenser heat exchangers; where their work targeted at obtaining the optimal transport membrane condenser (TMC), studied the effect of different tube spacing and the inlet water vapour mass fraction on the overall performance of membrane- based heat exchanger using numerical methods. A combined condensation model based on the capillary condensation and condensation on solid was used and the results were obtained in terms of Euler number, dimensionless volumetric heat transfer density, the contour of the water mass fraction and the temperature distribution inside the TMC heat exchanger.

Studies have revealed that the CFD simulation tool is an effective tool to understand, analyse and perform a process operating flow parameters of a system. Also, heat transfer enhancement in heat exchangers is very important in many engineering applications where the main aim of a heat exchanger isto increase the heat transfer rate. Hence, this paper present the application of CFD tool to model, simulate and analyse the basic flow parameters of a cross flow heat exchanger of Hot Freon copper tube in Cold water plate; to calculate the effectiveness of the flow system.

2.0 METHODS

2.1 Model Design Procedures

The basic materials used in this study were Freon12, water and copper tube. The materials were applied for an insulated flow system model; where the Freon cold fluid will flow in a copper tube with an inner diameter and length of 0.5mm and 2500mm length, submerged across a domain of hot water fluid of 650mm length and 1700mm width that flow in a reverse direction across the tube to generate a cross flow heat exchanger. The design was done using the Comsol Multi-physics Geometry Module shown in Fig. 2.1(a), and imported into a Computational Fluid Dynamics 19.2simulation software tool; where further simulation analysis of the working operations flow parameters and fluid properties were done. Table 1, and Fig. 1(b) present the meshing statistics and discretization used for the CFD simulations.

Table1: Mesh Statistics

Properties	Values
Average element quality	0.5469
Minimum element quality	0.002924
Total elements	39809

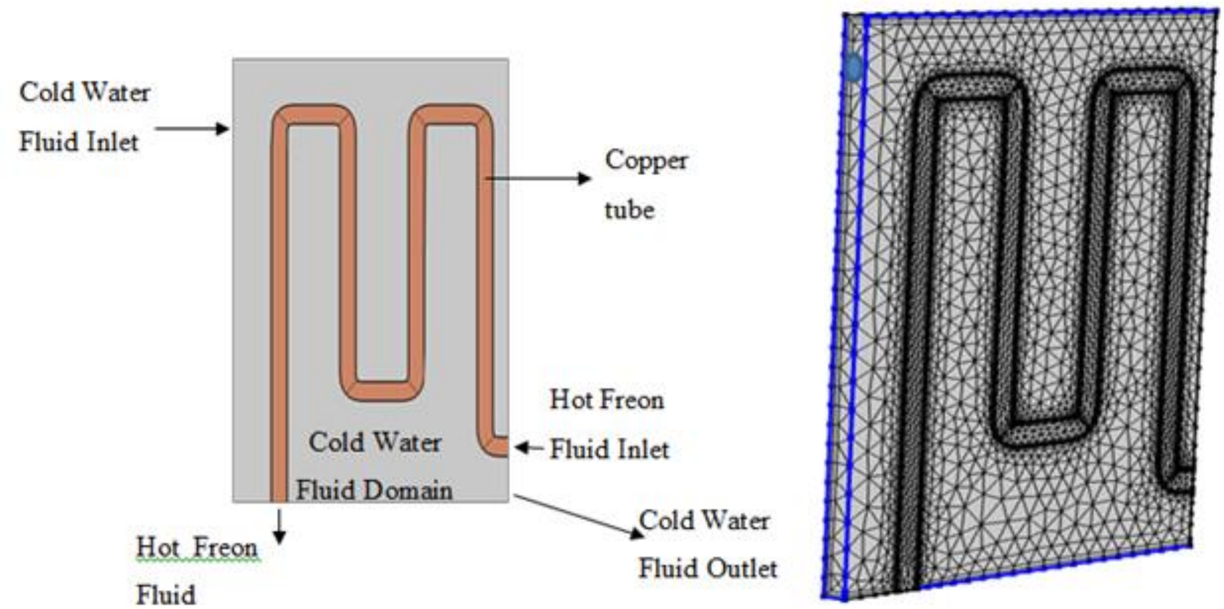

Fig. 1: Cross Flow Heat Exchanger Model

(a) Design **(b) CFD Meshing and Discretization**

2.2 Model Simulation Procedures

The CFD simulation of the flow system was set to assume that the fluids are incompressible; to flow in a laminar flow regime with the cold water velocity constant throughout the study period. The simulation applied the Navier-Stokes governing equations to analyse the flow parameters for the single-phase laminar flow of the cold Freon fluid across the hot water domain, flowing across each other. The general form of the Navier-Stoke's equation is given in equations 1 to 3; where equation 1represent the continuity equation for the conservation of mass flow, equations2 and 3 described the vector equation that represents conservation of momentum and the conservation of energy formulated in terms of temperature of the flow system respectively.

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = 0 \qquad 1$$

$$\rho \frac{\partial u}{\partial t} + \rho(u.\nabla)u = \nabla.[-\rho I + \tau] + F \qquad 2$$

$$\rho C_p \left(\frac{\partial T}{\partial t} + (u.\nabla)T\right) = -(\nabla.q) + \tau{:}S - \frac{T}{\rho}\frac{\partial \rho}{\partial T}\Big|_P \left(\frac{\partial P}{\partial t} + (u.\nabla)p\right) + Q \qquad 3$$

where:
$\rho, u, P, \tau, F, C_p\ T,$ q, Q and S represent the density, velocity, pressure, viscous stress tensor, Volume force vector, Specific heat capacity at constant pressure, Absolute temperature, Heat flux vector, Heat sources and Strain-rate

tensor. Hence, the steady-state form of the heat transfer in fluids was determined equation 4 (Bird et al., 2006).

$$\rho C_p(u.\nabla T) + (\nabla.q) = -\propto_p T(u.\nabla_P) + (t:\nabla u) + Q \quad 4$$

Subsequently, the model was simulated by using the following boundary conditions presented in Table.2

Table 2: Model Boundary conditions

Boundary Condition	Value
Cold water inlet temperature	300K
Hot Freon inlet temperature	305K
Hot Freon inlet velocity 1	0.015m/s
Hot Freon inlet velocity 2	0.035m/s

2.3 Temperature Difference and Effectiveness Analysis Procedures of the Heat Exchanger

The Logarithmic Mean Temperature Difference (LMTD) is used to determine the required operating temperature of the heat transfer in flow systems. It is the logarithmic average of the temperature difference between the hot and cold feeds at each end of the double pipe exchanger.

Hence, for the studied heat exchanger at which the hot and cold streams enter or exit on either side; the LMTD is defined mathematically as:

$$LMTD = \frac{\Delta T_{hot} - \Delta T_{cold}}{ln\left(\frac{\Delta T_{hot}}{\Delta T_{cold}}\right)} = \frac{\Delta T_{(o)} - \Delta T_{(i)}}{ln\Delta T_{hot} - ln\Delta T_{cold}} \quad 5$$

where $\Delta T_{(o)}$ and $\Delta T_{(i)}$ is the temperature difference between the two streams at inlet and outlet; and $\Delta T_{hot}$and ΔT_{cold} is the temperature difference between the two streams.

Further, the effectiveness of heat exchanger was defined in equation 6 as:

$$\varepsilon = \frac{\dot{m}_{hot} c_{hot} (T_{hot} - T_{cold})}{\dot{m}_{cold} c_{cold} (T_{hot} - T_{cold})} \quad 6$$

Where m_{hot} and $m_{cold}$are the mass flow rate; and c_{hot} and c_{cold} are the specific heat capacities of the hot and cold Freon and water fluids respectively.

3.0 RESULTS AND DISCUSSION

3.1 Temperature distribution in heat exchanger

The results of the simulation presented in Table 3 and Fig. 2 revealed that the hot Freon fluid flowing across the tube at 0.015m/s velocity was cooled from a temperature of 305 to 301.69K; while the cold Water was heated from 300 to 301.75K respectively. Subsequently, as the velocity of the hot Freon flow was increased to 0.035m/s, the temperature was reduced from 305 to 302.44; and the cold water was heated from 300 to 302.45K respectively. This result showed that at a constant velocity of the cold water in the plate, the rate of heat exchange of the fluids was reduced due to increased Freon flow across the tube.

Table. 3: Temperature magnitude and distribution across the domain for heat exchanger configuration.

	Inlet Temperature (K)		Outlet Temperature (K)	
Inlet Velocity (Freon)	**Cold Fluid (Water)**	**Hot Fluid (Freon)**	**Cold Fluid (Water)**	**Hot Fluid (Freon)**
0.015m/s	300	305	301.75	301.69
0.035m/s	300	305	302.45	302.44

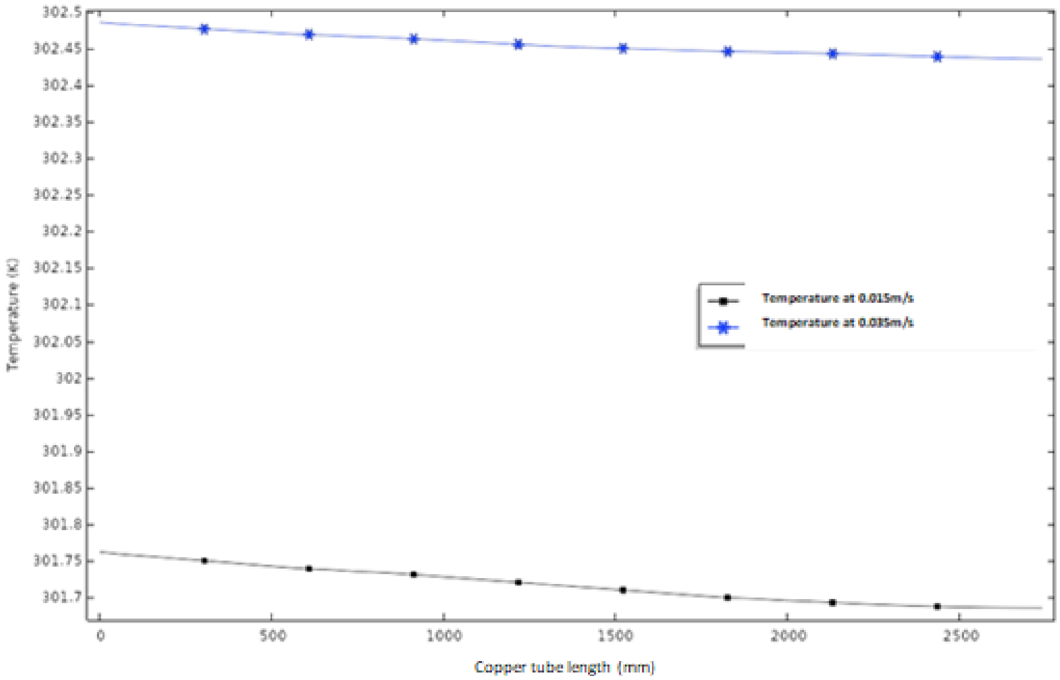

Fig. 2: Temperature Distribution of the Hot Freon and old water in the heat exchanger

Subsequently, Fig. 3(a) and (b) showed that the temperature distribution simulation result of the studied heat exchanger across the 2500mm length of the copper tube for the velocities of 0.015 and 0.035m/s. The temperature of the Freon velocities at 0.015m/s and 0.035m/s decreased from 305.76 - 301.65K and 305.00 - 302.45K across the length of the copper coil respectively. Further, Fig 4 and 5 showed that an increased temperature across the 600mm air flow heat exchanger domain was observed when the temperature of the cold air increased from 300K to 301.65K; and from 300K to 302.45K across the length heat exchanger domain for the Freon refrigerant input velocities 0.015m/s and 0.035m/s respectively. The Logarithmic Mean Temperature Difference value of the studied system from the presented results was evaluated to 2.71K and 2.68K for the 0.015 and 0.035m/s inlet velocity of the Hot Freon fluid in the tube. Hence, the required operating average temperature difference of the two fluids for the studied flow system was determined as 2.70K with an effectiveness of 89%.

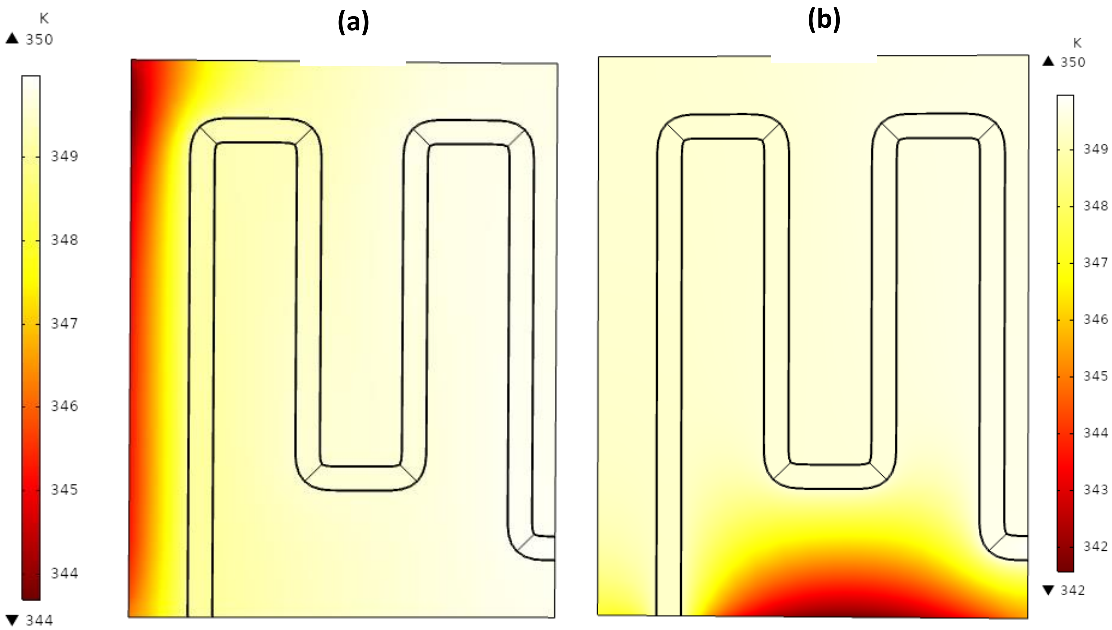

Fig. 3: **Temperature distribution of the cross flow heat exchanger**

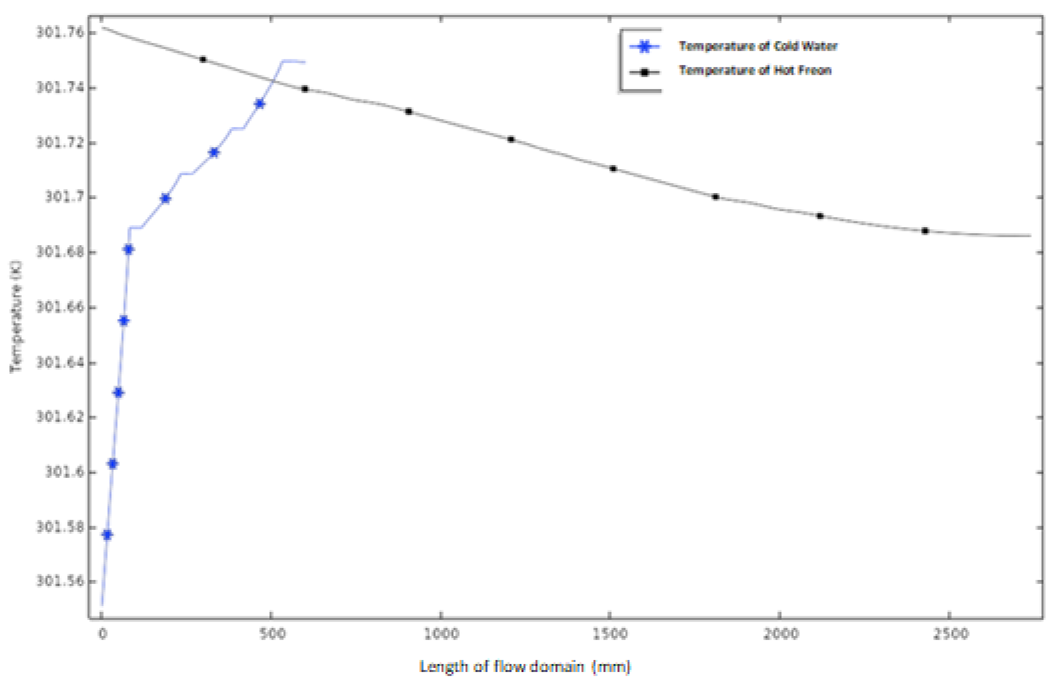

Fig. 4: Temperature variation of the cold water and hot Freon fluidsat 0.015m/s

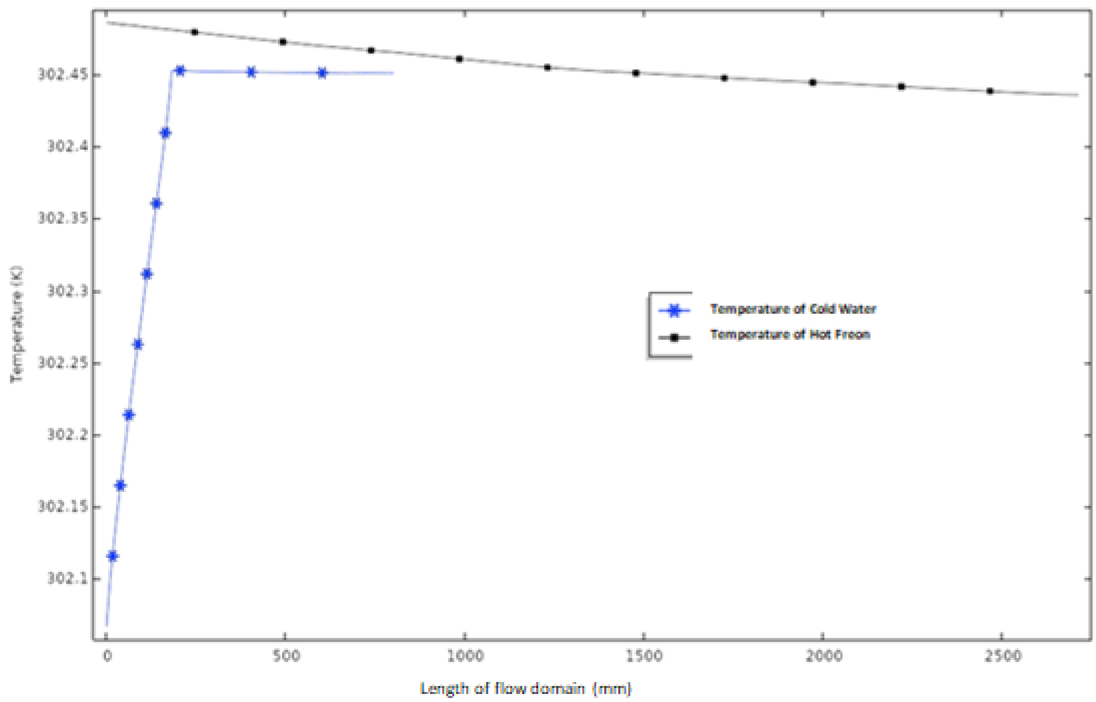

Fig. 5: Temperature variation of the cold water and hot Freon fluids at 0.035m/s

3.2 Pressure and Velocity Distribution of the Heat Exchanger

The cross flow heat exchanger velocity contour shown in Fig. 6(a) and (b) revealed the effect of water fluid velocity for the different input velocities of Freon fluid at 0.015m/s and 0.035m/. As shown in Fig 6(a), at constant flow of refrigerant velocity of 0.015/s, the water flow velocity rapidly increased at the horizontal paths from 0.08 - 0.12m/s. It decreased to 0.04m/s at the vertical paths of the copper coil. The difference velocity profiles observed across the dimensions of the water plate resulted from the Freon coil bends. The increase in temperature observed at the short horizontal part was due to rise in temperature due to flow across bend and the reverse was observed at the vertical part. Subsequently, a higher constant refrigerant velocity of 0.035m/s shown in Fig 6(b) where the Freon flow velocity gradually increased at the vertical paths from 0.04-0.07m/s; and decreased to 0.02m/s at the horizontal paths of the copper coil. This time the flow a higher flow was observed in the water plate at the vertical paths of the Freon tube; and the reverse case at the horizontal paths. This temperature change was due to the rate of heat exchange at different velocities across the dimensions of the Freon tube and the water plate. Additionally, the CFD simulation results in Fig. 7revealed a laminar flow for the pressure contour of the heat exchanger plane along the X-Z plane. The pressure was 1atm pressure at the inlet and increased to 1.77atm at the outlet of the copper coil. This increase in pressure resulted from the increase in temperature of the Freon fluid across the length. The pressure difference across the tube length was 1.77 x 10^3Pa. However, the pressure across the Freon fluid in the tube remained constant.

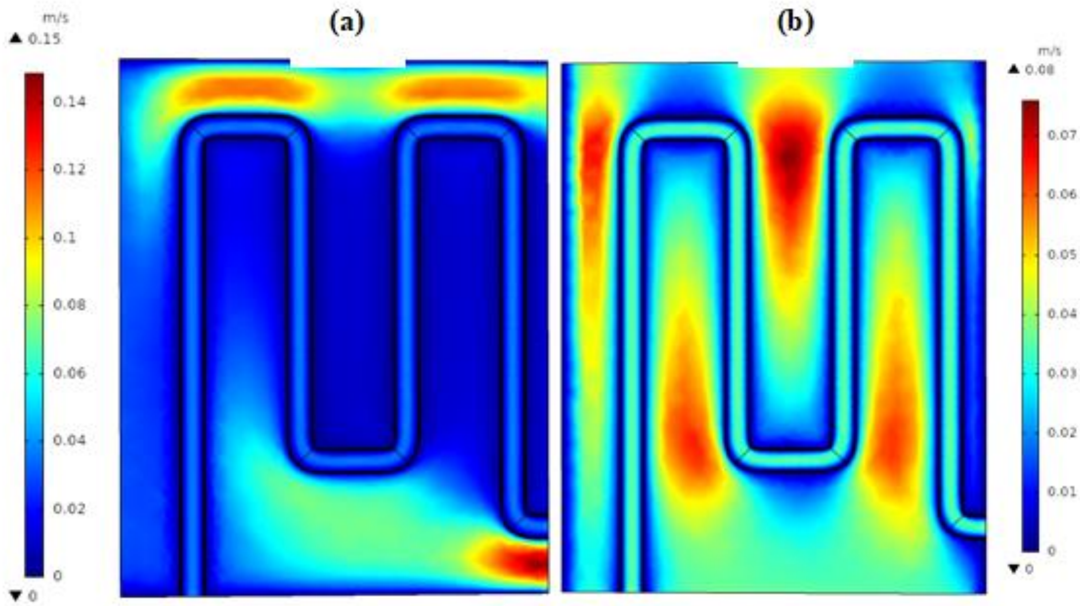

Fig. 6: Velocity Distributionfor Cross flow Heat Exchanger

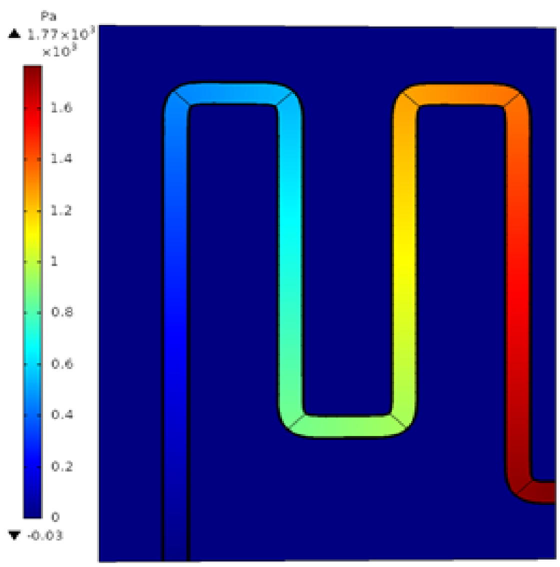

Fig. 7: Pressure Distribution for Cross flow Heat Exchanger

4. CONCLUSION

The study CFD analysis of a cross flow heat exchanger revealed that the rate of heat exchange of the fluids was reduced due to the increase in the velocity of the hot Freon flow across the tube at a constant velocity of the cold water in the plate. Also the studied system's Logarithmic Mean Temperature Difference value was evaluated to 2.71K and 2.68K for the 0.015 and 0.035m/s inlet velocity of the Hot Freon fluid in the tube. Consequently, the required operating average temperature difference of the two fluids of the studied flow system was determined as 2.70K with an effectiveness of 89%. Further analysis of the studied flow system revealed that the pressure and temperature of the cold water fluid increased when the velocity of the hot Freon fluid increased. This velocity distribution of the heat exchanger flow materials can be attributed to the high heat exchange rate that occurred at the bend sections of the hot Freon tube. The analysed results of the application of CFD simulation can be used to enhance the design and performance evaluation that meets future technological demands of heat exchangers.

REFERENCES

Chiasson, A. (2016). 15—Waste heat rejection methods in geothermal power generation. In R. DiPippo (Ed.), *Geothermal Power Generation* pp. 423–442.

Hu, H. H. (2012). Chapter 10—Computational Fluid Dynamics. In P. K. Kundu, I. M. Cohen, & D. R. Dowling (Eds.), *Fluid Mechanics (Fifth Edition)* pp. 421–472.

Kadhim, Z. K., Kassim, M. S., & Hassan, A. Y. A. (2016). CFD study for cross flow heat exchanger with integral finned tube. *International Journal of Scientific and Research Publications, 2016. 6 (6)*.

Mardiana-Idayu, A. and S. B. Riffat, (2013). Review on physical and performance parameters on heat recovery technologies for building applications. Renewable Sustainable Energy

Kumar, V., Gangacharyulu, D., Rao, P. M., & Barve, R. S. (2003). CFD analysis of cross flow air to air tube type heat exchanger. *PHOENICS 10th International User Conference*.

Mangrulkar, C. K., Dhoble, A. S., Chakrabarty, S. G., & Wankhede, U. S. (2017). Experimental and CFD prediction of heat transfer and friction factor characteristics in cross flow tube bank with integral splitter plate. *International Journal of Heat and Mass Transfer*, *104*, pp. 964–978.

Manivannan, S., Rengasamy, A., & Sudharsan, N. M. (2009). Design Optimization of Microprocessor Heatsink and its Impact on Processor Performance. *International Journal on Information Sciences and Computing*, *3*, pp. 8–14.

Navarro, H. A., & Cabezas-Gómez, L. C. (2007). Effectiveness-NTU computation with a mathematical model for cross-flow heat exchangers. *Brazilian Journal of Chemical Engineering*, *24*(4), pp. 509–521.

Promoppatum, P., Yao, S.-C., Hultz, T., & Agee, D. (2018). Experimental and numerical investigation of the cross-flow PCM heat exchanger for the energy saving of building HVAC. *Energy and Buildings*, *138*, pp. 468–478.

D.Bhanuchandrarao, M.Ashokchakravarthy, Dr. Y. Krishna, Dr. V .V. SubbaRao, T.HariKrishna (2013). CFD Analysis And Performance Of Parallel And Counter Flow In Concentric Tube Heat Exchangers,Vol. 2 Issue 11 pp. 25 -28.

Nice Thomachan, AnoopK.S, DeepakC. S, Eldhose P.Kuriakose, Habeeb Rahman K.K,Karthik K.V (2016). CFD analysis of tube in tube heat exchanger with fins, e-ISSN: Volume:03 Issue: 04 pp. 2395 - 0056

Kanti, Karthika U.P, Sabeer Ali, Sanath Kumar N, ShyamChandran C. (2016). CFD analysis of shell and tube heat exchanger, ISSN: 2319- 6890 (online),2347 5013(print) VolumeNo.5Issue:Special 6, pp: 1129 -1254

Digvendra Singh, Dr.JitendraPandey and Dr. Abhishek Tiwari (2016). CFD analysis of shell and tube heat exchanger using different header section.Vol.4 Issue 4 pp 2321-3051,

Maheshwari and K. M. Trivedi. A Review on Experimental Investigation of U-Tube Heat Exchanger using Plain Tube and Corrugated Tube. Volume 3, Issue 4 pp 56 - 60.

Pardhi,Dr.PrasantBaredar. Performanceimprovement of double pipe heat exchanger byusing turbulator,Volume2, Issue-4, pp. 881-885

Annamalai, A. & Ramalingam, V.(2011), Experimental Investigation and Computational FluidDynamics of a Air Cooled Condenser Heat Pipe, *Thermal Science*, pp.15.

Soleimanikutanaei, S., Lin, C. X., & Wang, D. (2018). Modeling and simulation of cross-flow transport membrane condenser heat exchangers. *International Communications in Heat and Mass Transfer*, *95*, 92–97.

Wasewar, K. L., Hargunani, S., Atluri, P., & Naveen, K. (2007). CFD Simulation of Flow Distribution in the Header of Plate Fin Heat Exchangers. *Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment-Process Engineering-Biotechnology*, *30*(10), pp. 1340–1346.

Zohuri, B. (2017). *Heat Exchanger Types and Classifications* pp.19–56.