IJSRD - International Journal for Scientific Research & Development| Vol. 7, Issue 03, 2019 | ISSN (online): 2321-0613

Experimental, Analytical and Numerical Investigation of Double Pipe Parallel Flow Heat Exchanger

Nasim Hasan[1] Fikiru Tafase Mosisa[2] Muse Degefe[3] Ayantu Daniel[4] Juhar Adeba[5]
[1,2,3,4,5]Department of Mechanical Engineering
[1,2,3,4,5]Mettu University, India

Abstract— The computational fluid dynamic are a powerful method to study heat and mass transfer from the many years. Applications of heat exchanger include cogeneration, condensation in power plants, waste heat recovery etc. Heat exchanger is a process of decreasing the energy consumption by reducing the thermal energy lost to the sink. Due to reducing the thermal losses the performance of the overall system will be increase. In this work an experimental, analytical and numerical model has been developed for analysis of heat transfer co-efficient foe parallel flow heat exchanger. The numerical model has been validated to the experimental data. The double pipe parallel flow heat exchanger length, inner pipe diameter and outer diameter of pipe are 2.5 m, 0.0107 m and 0.0254 m respectively. Hot water is flowing in inner pipe and temperature hot water is 40^0C, cold water is flowing in outer pipe and temperature cold water is 15^0C. The cold water mass flow rat has been maintain constant and hot water mass flow rate has been change at different Reynolds (2000 to 9000) is used for find out the heat transfer co-efficient at fixed parameter like length of pipe, diameter of pipe, fluid (water), material of pipe (cupper) and inlet temperature.

Keywords: Thermal conductivity, CFD, Temperature, Velocity

I. INTRODUCTION

Heat exchanger is a device used for the conversion, utilization, and recovery of thermal energy in various industrial, commercial, and domestic applications. Currently, heat exchangers have a wide range of industry applications. They are widely used in space heating, refrigeration, power plants, petrochemical plants, petroleum refineries and sewage treatment [1]. There are many types of heat exchanger designs for various applications. The major types of heat exchanger include double pipe, shell-tube, plate and shell, plate fin, and phase change heat exchangers. The flow in a heat exchanger can be arranged as parallel flow, counter flow, and cross flow. New heat exchangers have been designed for emerging thermal engineering fields, such as miniaturized heat exchanger for cooling electronics components and systems, miniaturized heterogeneously catalyzed gas-phase reactions, thermoelectric generators, etc. [2-5] New materials, such as polymers, have been explored to develop polymer heat exchangers for better fouling and corrosion resistance [6]. Parallel-plated heat exchangers have been studied analytically and experimentally to provide formulations for heat exchanger design. Vera and Linan [3] analyzed multilayered, counter flow, parallel-plate heat exchangers numerically and theoretically. They developed a two dimensional model to find analytical expressions and their approximations for the fully developed laminar counter flow in long parallel-plate heat exchangers. Kragh et al. [7] developed a new counter flow heat exchange for ventilation systems in cold climates. The efficiency of the new heat exchanger was calculated theoretically and measured experimentally. Zhan et al. [8] used an experimentally validated model to understand the influence of operational and geometric parameters of the cross-flow and counter-flow exchangers on the different metrics of cooling performance. Overall the counter-flow exchanger demonstrated better cooling effectiveness and higher cooling capacity than the cross-flow system. However, the energy efficiency of the counter-flow system is often seen to be lower than that of the more conventional cross-flow dew point system [8]. The shape of the cross section of the heat exchanger also has a significant effect on efficiency. Kumaresan et al. this paper presents the heat transfer characteristics of water and nanofluids in a tubular heat exchanger of various lengths for cooling/heating applications [9]. The performance of EATHE at 23 m length and 3m/s velocity of spiral pipe have performed. The temperature fall from 320.6 K to 305.11 K [10]. Optimizing the Design of Earth Air Tunnel Heat Exchanger has been analysis for cooling mode .The temperature fall in straight, spiral and helical pipe of earth air tunnel heat exchanger is 13.360 C, 14.000 C, and 14.460 C [11]. Different diameter of earth air tunnel heat exchanger like 0.07m, 0.09m, 0.11m, 0.13m and 0.15m has been analysis for cooling mode. The temperature fall at the pipe diameter of 0.07m, 0.09m, 0.11m, 0.13m and 0.15m is 20.03 0C, 18.690C, 16.450C, 15.310C and14.460C [12]. CFD analysis earth air tunnel heat exchanger has been done for validating of CFD model to experimental data. The variation in simulation and experimental results are 6.07% at outlet of earth air tunnel heat exchanger [13].

A. *Objective of the Paper*

- To validation of CFD model to experimental parallel flows heat exchanger.
- To perform CFD analysis of parallel flow heat exchanger at the Reynolds number of fluid 2000, 3000, 4000, 5000, 6000, 7000, 8000 and 9000.
- To complier the experimental, analytical and numerical analysis of double pipe parallel flow heat exchanger.

II. MATERIAL AND METHODOLOGY

A. *Description of CFD Model*

The computational fluid dynamic (CFD) are a powerful method to study heat and mass transfer from many years. CFD codes are structured around numerical algorithms that can tackle fluid flow problems. CFD provides the numerical solutions of partial differential equations witch governing airflow and heat transfer in a discretised form. The complicated fluid flow and the heat transfer processes involved in any heat exchanger can be examined by CFD software, ANSYS fluent. The CFD codes in fluent contain three main elements as shown in fig.1

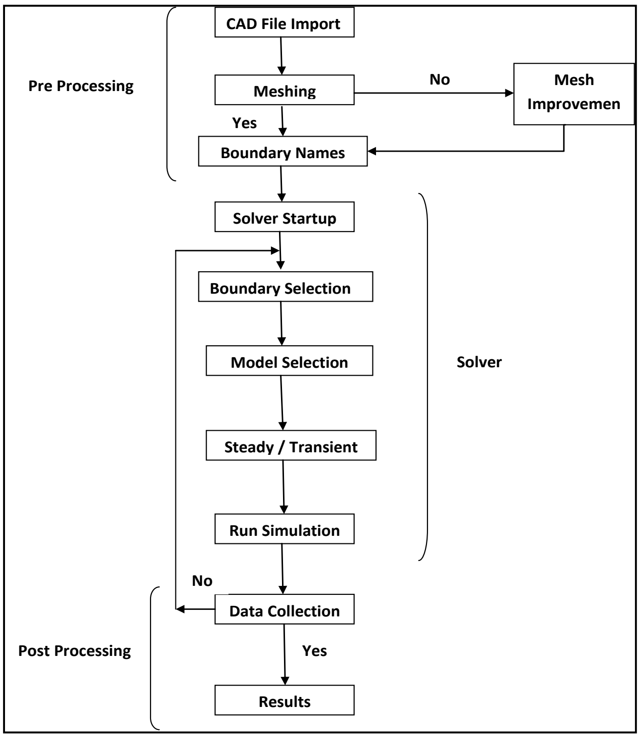


Fig. 1: Flow Chart of CFD Methodology

1) Pre-processor
2) Solver
3) Post-processor.

1) Pre-processor are consists of input of a flow problem to a CFD program by means of definition of geometry of the region of interest. The CFD domain generating grid to subdivision of fluid domain. The domain is dividing into a number of sub domains. The sub-domains are a grid (or mesh) of cells (or control volumes or elements), with or touch the domain boundary.
2) Solver uses finite control volume method for solving the governing equations of the fluid flow and heat transfer.
3) Post-processor shows results of the simulations using vector plots, contour plots, graphs, animations, etc. [14-16]

B. Geometry

After create EATHE model in CATIA the file has been save in stp file and this file has been imported in ANSYS work bench which is shown in figure 2.

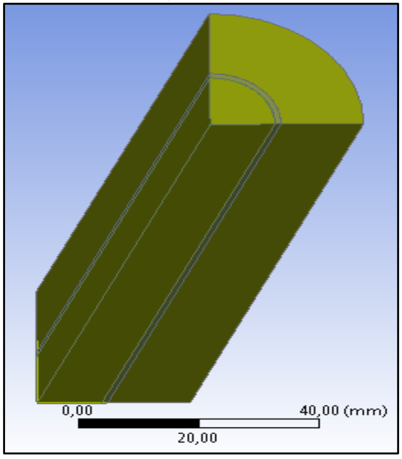


Fig. 2: Model of heat exchanger

C. Meshing

The next step in pre-processing stage is generation of mesh to be used in the ANSYS fluent. ANSYS ICEM is used for generating the mesh of the heat exchanger model which is shown in fig. 3.

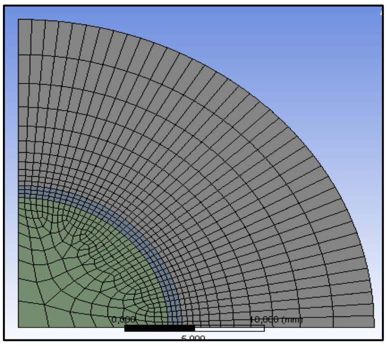


Fig. 3: Meshing of model

Equations used in heat exchanger

1) *Heat lost by the Hot Water,*

$$Q_h = m_h c_{ph}(T_{hi} - T_{ho}) \quad \text{(W)}$$

where, m_h – Mass flow rate of hot water, (kg s^{-1})
c_{ph} – Specific heat of hot water,(J kg^{-1}, k^{-1})
T_{hi} – Temperature of hot water at the inlet, (ºC)
T_{ho} –Temperature of hot water atthe outlet, (ºC)

2) *Heat Gained by the Cold Water,*

$$Q_c = m_c c_{pc}(T_{co} - T_{ci}) \quad \text{(W)}$$

where, m_c-Mass flow rate of cold water,(kg s^{-1})
c_{pc}- Specific heat of cold water,(J kg^{-1}, k^{-1})
T_{co}- Temperature of cold water atthe outlet, (ºC)
T_{ci} - Temperature of cold water atthe inlet, (ºC)

3) *Logarithmic Mean Temperature Difference,*

$$LMTD = \frac{(dT_1 - dT_2)}{1n\dfrac{dT_1}{dT_2}}$$

Overall Heat Transfer Coefficient based on Inner Diameter of the Inner Tube

4)
$$U_i = \frac{Qh}{A_i(LMTD)}$$

$$A_i = \pi\, d_i l \quad , m^2$$

Where, A_i= Area of inner tube ($m^{2)}$
d_i = inner diameter of the inner tube (m)
l = length of the inner tube (m)

5) *The value Nusselt Number of hot water flowing through the annulus under turbulent conditions is calculated by using Gnielinski's equation,*

$$Nu_0 = \frac{(f/8)(Re_w - 1000)Pr_w[1+(D_i/L)^{2/3}]}{1+12.7(f/8)^{0.5}(Pr_0^{2/3} - 1)}$$

where, Re – Reynolds number
Pr – Prandtl number
D_i – Inner diameter of outer tube, (m)
L – Length of the heat exchanger, (m)
f – Friction factor.
f = $[0.794 \text{ in } (Re_0) - 1.64]^{-2}$

6) The value of Convective Heat Transfer Co-Efficient of Hot Water in the Annulus Is Calculated From,

$$h_o = \frac{Nu_o k_w}{D_h}$$

where, Nu_0 – Nusselt number of hot water
k_w – Thermal conductivity of hot water at bulk mean temperature, $Wm^{-1}K^{-1}$.
D_h - D_i-d_o- Hydraulic diameter, (m)
D_i – Inner diameter of outer tube, (m)
D_0 – Outer diameter of inner tube, (m)

7) The Value of h_i is Calculated from

$$\frac{1}{UA_i} = \frac{1}{h_i A_i} + \frac{\ln\left(\frac{D_o}{D_i}\right)}{2\pi kl} + \frac{1}{h_o A_o}$$

III. RESULT AND DISCUTION

A. *Grid Independence Study*

The grid independence study was done with different number of mesh element. The outlet temperature of heat exchanger is constant at the no. of mesh element 145533, 1262400 and 1354530.

S.no.	Number of element	Inlet Temperature of Water (C)		Outlet Temperature of Water (C)	
		Inner Fluid	Outer Fluid	Inner Fluid	Outer Fluid
1	1,45,533	40	15	27.044	16.973
2	6,65,312	40	15	26.851	17.013
3	11,39,922	40	15	26.641	17.1215
4	12,62,400	40	15	26.646	17.127
5	13,54,530	40	15	26.645	17.128

Table 1: Grid independence Study with Different Number of Mesh Element

B. *Validation of Model to Experimental Data*

The CFD based heat exchanger model is validated by taking observations of an actual heat exchanger data [9] which is shown in fig.4. The properties which is used in simulation for validation of double pipe parallel flow heat exchanger is shown in table 2. Figure 5 is shown temperature contour of heat exchanger, read cooler show the maximum temperature and blue cooler show minimum temperature.

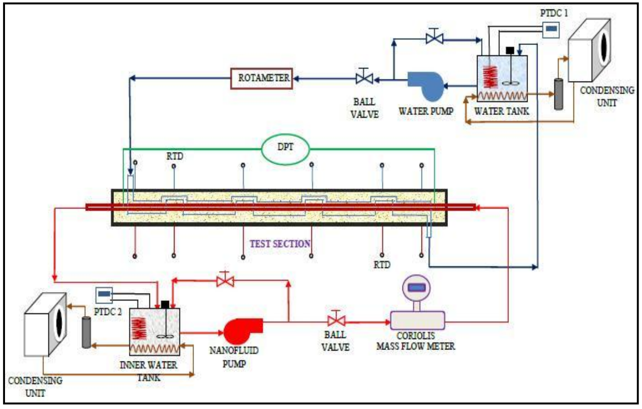


Fig. 4: Experimental setup of heat exchanger

S.no.	Material	Thermal conductivity (w/m K)	Density (kg/m3)	Specific heat capacity (j/kg K)
1	Water	0.6	998.2	4182
2	Copper	387.6	8978	381

Table 2: Physical and Thermal Parameters Used in Validation

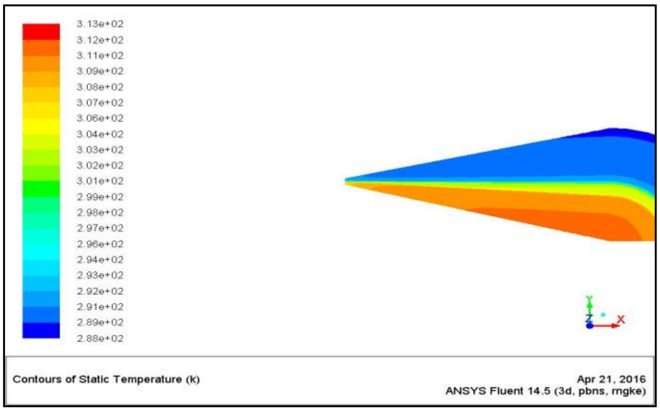


Fig. 5: Temperature contour of heat exchanger

Experimental/ Simulation	Inleat Temperature of Water (C)		Outlet Temperature of Water (C)	
	Inner Fluid	Outer Fluid	Inner Fluid	Outer Fluid
Experimental	40	15	31.044	17.89
Simulation	40	15	30.646	17.117
% difference	0.00	0.00	1.28	4.32

Table 3: Comparison of Experimental and Simulation Temperature

Table 3 shows the validation of simulated temperatures with experimental results. The variation in simulation and experimental results are 1.28 and 4.32 which is due to variation in the coefficient of friction of the engineering material which is used in simulation and experiment, irregularities such as joints in experimental set-up and improper insulation at the risers of experimental set-up.

C. *Analysis of Heat Exchanger at Different Remolds Number*

The Computational fluid dynamics analysis of parallel flow heat exchanger has been perform at the Reynolds number of fluid 2000, 3000, 4000, 5000, 6000, 7000, 8000 and 9000,. The nature of flow according to Remolds number for pipe flow and open channel flow is given in table.

S.No.	Flow Condition	Pipe Flow	Open Channel Flow
1	Laminar Flow	$Re \leq 2,000$	$Re \leq 5,000$
2	Transition Flow	$2000 < Re < 4,000$	$5000 < Re < 10,000$
3	Turbulent Flow	$Re > 4,000$	$Re > 10,000$

Table 4: Nature of flow according to Remolds number for pipe and open channel

1) *Case I Laminar Flow*

If the water flow in pipe at Reynolds number of 2000 is known as laminar flow. The computational fluid dynamics analysis of double pipe parallel flow heat exchanger has been analysis for calculating the heat exchange in laminar flow condition. The contour of static temperature is shown in fig.6.

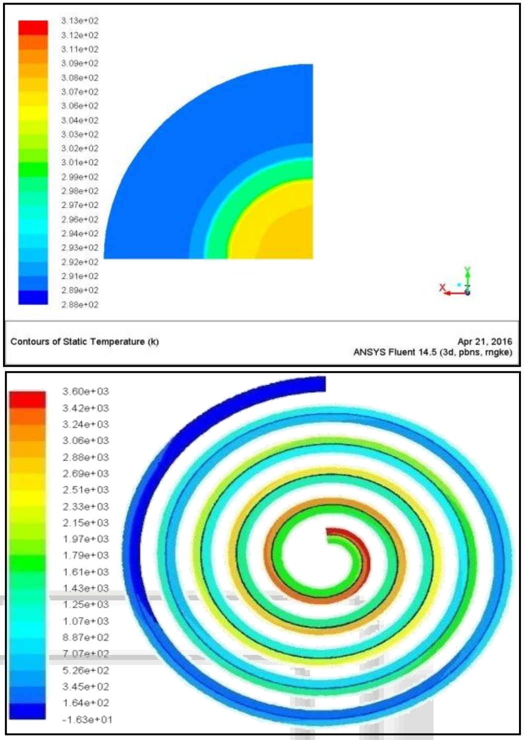
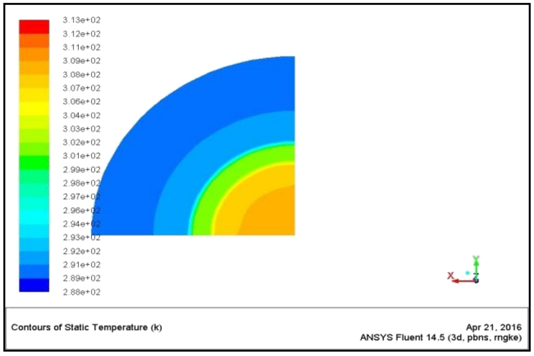


Fig. 6: Temperature contour of heat exchanger for laminar flow in straight and spiral pipe

2) *Case II Transition Flow*

If the water flow in pipe at Reynolds number of 4000 is known as transition flow. The computational fluid dynamics analysis of double pipe parallel flow heat exchanger has been analysis for calculating the heat exchange in transition flow condition. The contour of static temperature is shown in fig.7.



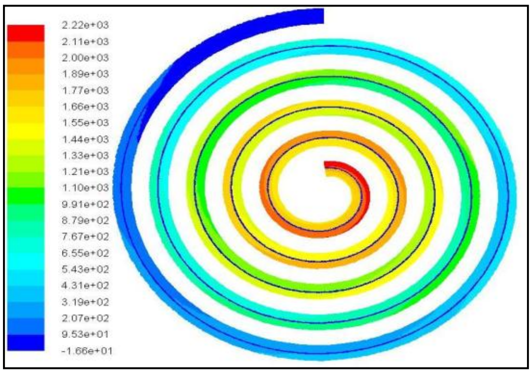


Fig. 7: Temperature contour of heat exchanger for transition flow in straight and spiral pipe

3) *Case III Turbulent Flow*

If the water flow in pipe at Reynolds number of 4000 is known as turbulent flow. The computational fluid dynamics analysis of double pipe parallel flow heat exchanger has been analysis for calculating the heat exchange in turbulent flow condition. The contour of static temperature is shown in fig.7.

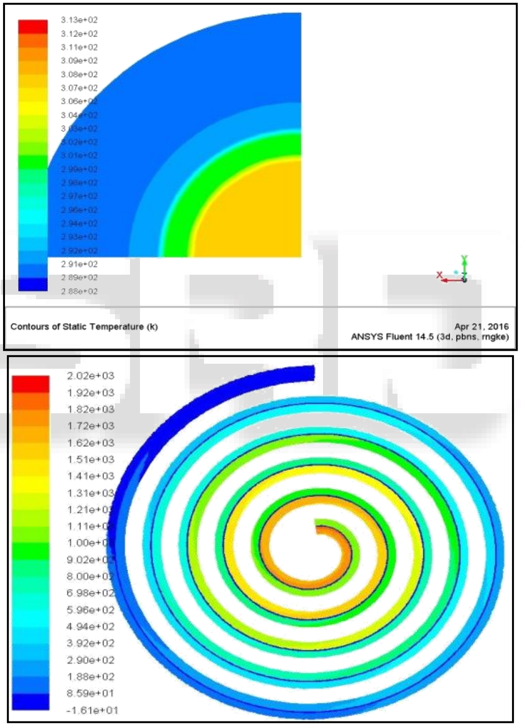


Fig. 8: Temperature contour of heat exchanger turbulent flow in straight and spiral pipe

The computational fluid dynamics analysis is perform for parallel flow heat exchanger at different Reynolds number for finding the heat transfer coefficient shown in table 5.

Re. No.	Mass (kg/s)	t_i (C)	t_o (C)	T_i (C)	T_o (C)	Q (kW)	LMTD	U (W/m2K)	h (Numerical Method)
2000	0.011037	40	26.4	15	17.1	0.630639	15.82671	474.3629	950.014414
3000	0.016555	40	28.8	15	17.8	0.776171	17.07957	541.0048	1261.13332
4000	0.022074	40	30.4	15	18.2	0.8889	17.82187	593.773	1590.65925
5000	0.027592	40	31.4	15	18.6	0.997934	18.21811	652.1074	2091.98623
6000	0.033111	40	32.2	15	18.9	1.083867	18.54495	695.7785	2619.41949
7000	0.038629	40	32.8	15	19.1	1.157788	18.79316	733.4152	3246.65597
8000	0.044148	40	33.4	15	19.4	1.219697	18.99594	764.3845	3956.21205
9000	0.049666	40	33.9	15	19.6	1.272366	19.1727	790.0407	4755.50714

Table 5: Heat transfer coefficient (h) at different Reynolds number

The heat transfer co-efficient of double pipe parallel flow heat exchanger is compare to experimental, theoretical and simulation which is shown in table 6.

0	Mass (kg/s)	h (Experimental)	h (A)	h (Numerical Method)
2000	0.011037	1000	900	950.014414
3000	0.016555	1250	1200	1261.13332
4000	0.022074	1800	1850	1590.65925
5000	0.027592	2300	2250	2091.98623
6000	0.033111	2800	2825	2619.41949
7000	0.038629	3300	3500	3246.65597
8000	0.044148	3700	3750	3956.21205
9000	0.049666	4250	4400	4755.50714

Table 6: Heat transfer coefficient at different Reynolds number by experimental, analytical and numerical method

From fig.9 it is clear that increasing the Reynolds number the heat transfer co-efficient is increase and difference of experimental, theoretical and simulation result is very little.

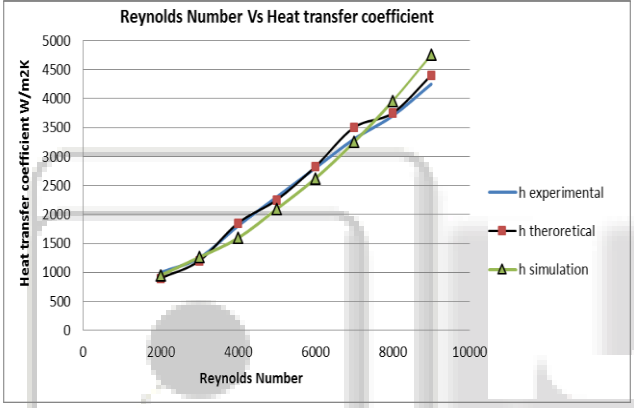


Fig. 9: Graph of heat exchanger turbulent flow

IV. CONCLUSION

A CFD package (ANSYS FLUENT 14.5) was used for the numerical study of heat transfer characteristics of parallel flow heat exchanger for laminar, transient and turbulent flow. The CFD results when compared with the experimental results from different studies were well within the error limits. This variation in simulation and experimental results are due to variation in the coefficient of friction of the engineering material which is used in simulation and experiment, irregularities such as joints in experimental set-up and improper insulation at the risers of experimental set-up.

The study showed that there is not much difference in the heat transfer performances of the parallel-flow configuration. The simulation was carried out for water to water heat transfer characteristics and different inlet temperatures were studied.

REFERENCES

[1] S. Kakac and H. Liu, Heat Exchangers: Selection, Rating and Thermal Design, 2nd ed, CRC Press, 2002.

[2] A.E. Quintero, M. Vera and B. Rivero-de Aguilar, Wall conduction effects in laminar counterflow parallel-plate heat exchangers, International Journal of Heat and Mass Transfer, 70, 2014, pp. 939-953.

[3] M. Vera and A. Linan, Laminar counterflow parallel-plate heat exchangers:exact and approximate solutions, International Journal of Heat and Mass Transfer, 53, 2010, pp. 4885-4898.

[4] J. Esarte, G. Min, D.M. Rowe, Modelling heat exchangers for thermoelectric generators, J. Power Sources, 93, 2001, pp. 72-76.

[5] D.B. Tuckerman, R.F.W. Pease High-performance heat sinking for VLSI IEEE Elect. Device Lett., 2 (5) (1981), pp. 126–129.

[6] J. Yu, and H.Zhao, A numerical model for thermoelectric generator with the parallel-plate heat exchanger, J. Power sources 172, 2007, pp. 428-434.

[7] J. Kragh, J. Rose, T.R. Nielsen, and S. Svendsen, New counter flow heat exchanger designed for ventilation systems in cold climates, Energy and Buildings, 39, 2007, pp. 1151-1158.

[8] Changhong Zhan, Zhiyin Duan, Xudong Zhao Energy Volume 36, Issue 12, December 2011, Pages 6790–6805.

[9] Kumaresan, S. Mohaideen Abdul Khader, S. Karthikeyan, R. Velraj, Convective heat transfer characteristics of CNT nanofluids in a tubular heat exchanger of various lengths for energy efficient cooling/heating system, International Journal of Heat and Mass Transfer, 60 (2013) 413-42.

[10] Hasan, N., & Mathur, Y. B. Computational Fluid Dynamics Analysis of Earth Air Tunnel Heat Exchanger for Cooling in Summer Season, Rex Journal Volume 4 Issue 2, 2017 Page 641-646.

[11] Hasan, N., & Mathur, Y. B. Optimizing the Design of Earth Air Tunnel Heat Exchanger for Cooling in Summer Season, International Multidisciplinary Mulitilingual E-Journal, Volume 5 Issue 1 Page 19-29.

[12] Hasan, N., & Mathur, Y. B. Optimizing the Diameter of Earth Air Tunnel Heat Exchanger for Cooling in Summer Season. ACHIEVING SUSTAINABLE STRATEGIC ADVANTAGE, 98, ISBN: 978-93-86608-37-6.

[13] Hasan N, Mathur Y. B., Khader M. K., Validation of Earth Air Tunnel Heat Exchanger CFD Model To Experimental Setup, IOSR Journal of Engineering, Vol. 08, Issue 01 (January. 2018), V2, PP 20-25.

[14] Saifi, M. S., Jakhar, O. P., & Hasan, N. (2016). CFD Parametric Investigation for Two Phase Flow of Refrigerant 134a In an Adiabatic Capillary tube. International Journal of Civi, Mechanical and Energy Science, 2(2), 94-98.

[15] Khader, M. A., Hasan, N., & Degefe, M. Optimization of Bajaj three wheeler carburetor fuel tube for better performance, International Digital Library of Technology & Research Volume 1, Issue 6, June 2017.

[16] Daniel A., Yilma T. B. & Hasan N., “Experimental Analysis of Cook Stove for Efficient Utilization of Biomass Energy in Ethiopia”, International Journal of Multidisciplinary Educational Research, Volume 6, Issue12(3),December2017.