



CFD SIMULATION ON EFFECT OF POROUS MEDIUM IN TRIPLE CONCENTRIC TUBE HEAT EXCHANGER

Nadeem Akhtar^{1*}, Inder Singh Nagar² and Shashikant Sharma³

¹Department of Mechanical Engineering, School of Research and Technology, People's University, Bhopal, India

²Assistant Professor, Department of Mechanical Engineering, School of Research and Technology, People's University, Bhopal, India

³Associate Professor, Department of Mechanical Engineering, School of Research and Technology, People's University, Bhopal, India

*Author for Correspondence: E-mail: nadeem.akhtar692@gmail.com

Abstract

Heat exchanger is a system which is used to transfer heat between two or more fluids. They are used in both cooling and heating processes. Usually the fluids which are processed are heated as well as cooled before it undergoes phase change and the process. As per their applications, the heat exchangers are named differently. One of the main functions of third pipe in triple concentric pipe heat exchanger is to enhance the transfer rate of heat with the help of larger transfer per unit exchanger length and an additional flow passage. These pipes follow rule of CHN, i.e. Cold-Hot-Normal. The innermost pipe carries cold fluid, whereas middle pipe carries hot fluid and the outermost carries normal fluid. These three fluids in these pipes exchanges heat and thus come at equilibrium. Porosity that increases heat transfer is added and results are observed. As porosity is increased in the material of pipes, better results have started to show up. CFD is used to simulate and obtain the results of the pipe exchanging heat with different properties of porosity. Three cases were simulated on ANSYS software, one without porosity, another case with 25% of porosity added and final case with 50% of porosity added. In case of simple pipe, cold liquid gives temperature of 293.419 k on outlet, hot liquid shows 307.366 k and

normal temperature as 301.543 k, whereas after addition of 25% porosity, then the outlet of cold liquid is obtained as 301.414 k, hot liquid is obtained as 301.446 k and for normal liquid; it is obtained as 301.499 k. When porosity is further increased to 50%, cold liquid shows temperature at outlet as 301.466, hot liquid as 301.482 and for normal liquid; it is obtained as 301.497 k.

Keywords: Heat Exchanger, CHN (Cold-Hot-Normal), CFD (Computational Fluid Dynamics), concentric pipes, porosity.

INTRODUCTION

In many configurations, heat exchangers are available now a day. As per their process fluids, as well as application, along with mode of heat transfer including flow, heat exchangers might be classified. A simplified heat exchanger can be seen in figure 1. Through indirect ways with fluids as well as direct contact ways, heat can be transfer in heat exchangers. Classification can also be done on the basis of kinds of baffles along with shell as well as tube passes including smooth and also baffles surfaces in addition with the arrangement of the tube such as square and triangular and so on. As per their applications, the heat exchangers are named differently. An insight has been provided about the power requirements as well as capital cost through certain area that is needed along with Pressure drop of a heat exchanger [1].

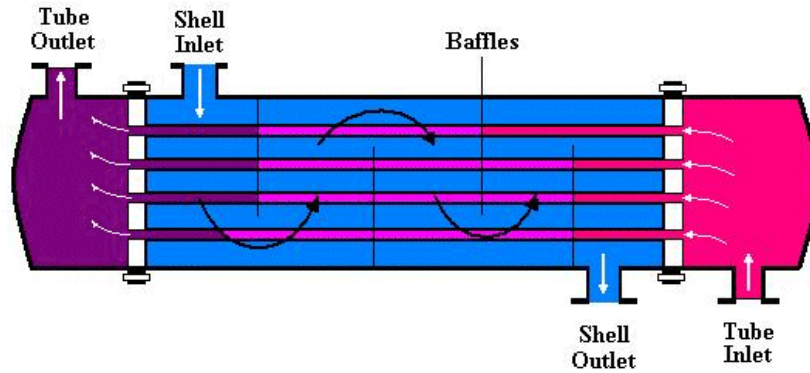


Figure 1: Counter-current Heat Exchanger Arrangement

Types of Heat Exchanger

As per their process fluids, as well as application, along with mode of heat transfer including flow, heat exchangers might be

classified. Through indirect ways with fluids as well as direct contact ways, heat can be transfer in heat exchangers. Heat pipe heat exchanger can be seen in figure 2.

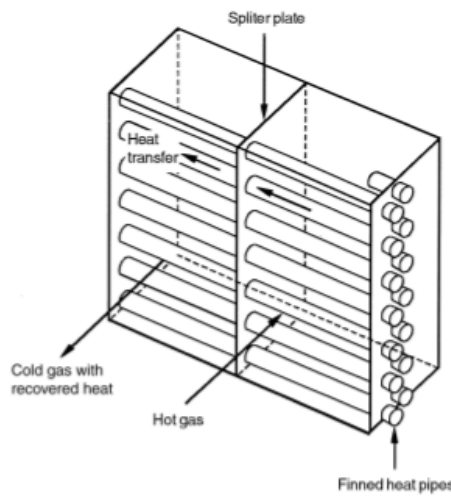


Figure 2: Heat pipe heat exchanger

Shell and Tube Heat Exchanger : A Shell as well as tube heat exchanger is a designed class of heat exchanger. In power plants as well as in large chemical processes along with oil refineries, heat exchanger are the most common which are used in it as shown in figure 3. Such kinds of heat exchangers are consisted of large vessel known as shell along with the bundle of tubes which are inside it. Huge amount of area is required of heat transfer for transferring the heat efficiently along with many tubes. Waste heat can also be taken in usage through this way.

According to the changes in condition that occurs in the pipes: flow rates, as well as inlet

temperatures, along with fluid composition, including the fluid properties and so on, changes also occur in the amount of heat transferred. A Shell as well as tube heat exchanger is a designed class of heat exchanger. In power plants as well as in large chemical processes along with oil refineries, heat exchanger are the most common which are used in it. Such kinds of heat exchangers are consisted of large vessel known as shell along with the bundle of tubes which are inside it. Heat exchanger is used for transferring the heat from one fluid to another fluid.

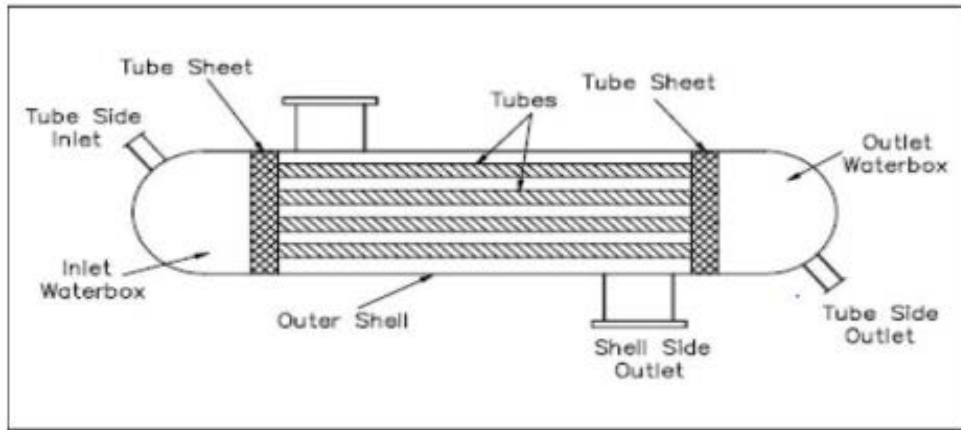


Figure 3: Shell and Tube heat exchanger

One of the basic components of heat exchanger is pipe which runs the fluid in the inner pipe as well as outer pipe. Heat as well as cold water flow in opposite along with same direction in heat exchanger's simple context of a concentric tube construction.

Triple Concentric Tubes: Heat exchanger is used for transferring the heat from one fluid to another fluid. One of the basic components of

heat exchanger is pipe which runs the fluid in the inner pipe as well as outer pipe. Heat as well as cold water flow in opposite along with same direction in heat exchanger's simple context of a concentric tube construction. When arrangement of flow is parallel then at the same end fluid starts entering that is cold as well as hot, along with leave at the same end including flow in same direction shown in figure 4.

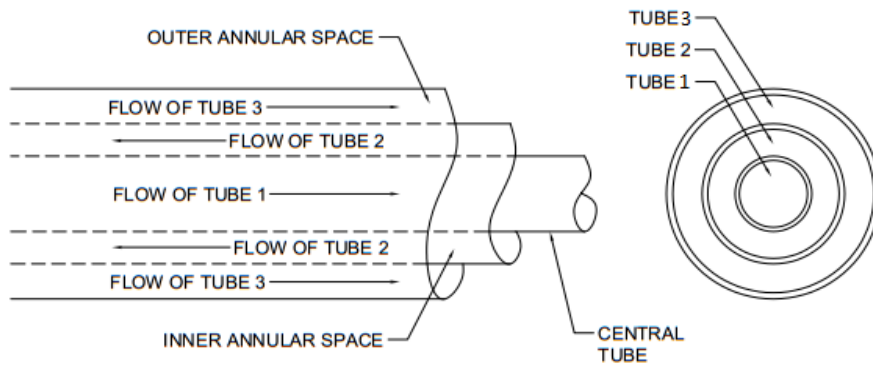


Figure 4 Triple concentric tubes

When thermal fluids get in contact with Heat transfer fluids which are hot in TCTHE thermal fluid gets in contacts with hot fluid that is heat transfer fluid from both sides, thus, greater heat transfer area has been provided by it along with the ascending heat transfer rate. Usually, hot as

well as cold are two kinds of fluid are used here. For evaporation as well as condense the fluids, heat exchangers are being installed in refrigerators along with Air conditioners. Triple concentric tube heat exchanger can be seen in figure 5.

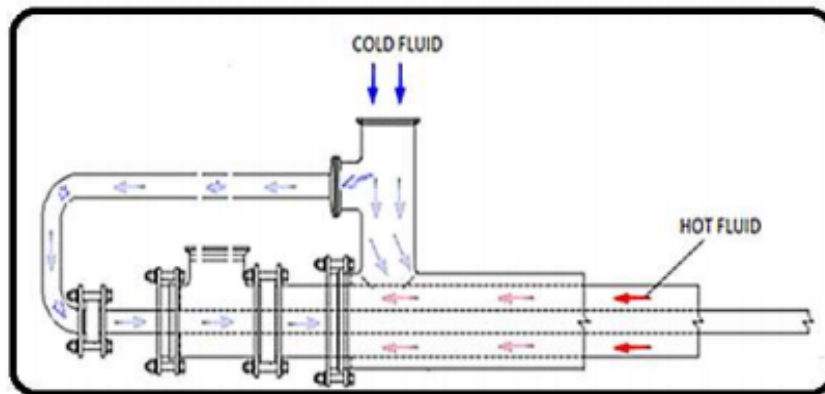


Figure 5: Triple concentric tube heat exchanger

LITERATURE REVIEW

(Subramanian, Justin, & Sebastin, 2018) The work is investigating the convective transfer of heat along with the characteristics of friction factors in a plane plus ellipsoidal that is concentrating on the both side of tubes namely zigzag dimple tubes along with dimples tubes with constant heat flux under the turbulent flow where water has been used as a working fluid. Without causing any effect on the overall workability of the system, different methods have been taken in usage to increase the heat transfer rate that has been referred by heat transfer techniques. In past few decades, on passive techniques of heat transfer, numerous studies have been reported. With the usage of dimple tube the inner tube has been modified in the present study of concentric tube heat exchanger. For different Reynolds number with a fully developed turbulent flow, thermal performance as well as the friction factor of the effects on dimples has been determined. The results that are brought out from this analysis and experiment states that the utilization of water in ellipsoidal along with the both tubes namely dimpled tubes are showing an increment in the transfer of heat but that rate of increment is very low in comparison to the plain tube under friction factors.

(kumar Hotta, Nanda Sajja, Teja Sigireddy, & Sai Mugala, 2018) [3] pivotal parts is an SI engine. For the combustion of the fuel, on such principle SI engines have been operated. For the production of such useful work, this phenomenon has been partly utilized due to the production of heat energy. In the form of convection as well as conduction in atmosphere a part of the thermal energy has been dissipated. Changes of sharp temperature have been experienced by engine materials because of internal combustion. Efficiently as well as quickly heat dissipation needs to be carried out for the improvement of efficiency along with sustainability of the engine. For heat dissipation, exposure in the atmosphere of the engine's surface area was critical. Fins are known extended surfaces that help for the increase in surface area along with taking away the produced heat to the atmosphere of the combustion chamber. Characteristics of heat transfer of the fins have been vastly improved due to the advent of new alloys without any compromise on the other structural parameters.

The parameters are the geometry as well as pitch along with height including the materials of the fins that can be altered for the optimization of the heat transfer from the engine. Thermal properties has been analyzed in this project of the engine cylinder through varying geometry of fins as well as size of fins along with size of fins including the material of fins at different operational speeds has been done. For commercial engine fins, standard materials have been taken in usage that Aluminium alloys A204 with the thermal conductivity of 120 W/m-k. Analysis has been done with the usage of materials such as Aluminium 204 as well as Aluminium 356 along with 6061 for the heat transfer characteristics of engine fins. With the help of SOLIDWORKS software as well as thermal analysis along with thermal properties are utilized with ANSYS Fluent Version 16.0 , the parametric model of the engine along with dissimilar fins are designed also the data fit software shows the results in the form of graphs.

(Taamneh, 2017) [4] combination of conventional air cooling technique along with the rotating heat pipe might be taken in consideration as effective as well as emerging technique for cooling for gas turbine disk. Consequently, numerical investigation has been taken of heat pipes along with simplified turbines including the analysis of transient as well as the thermal steady. For several parameters such as for heat as well as air pipes, the convection heat transfer coefficient along with thermal conductivity including the number of heat pipes together with the dimension of the disk, investigation has been taken place in the absence of heat pipes along with presence of transient as well as steady temperature variations. With the usage of finite element (FE) modeling software ANSYS-17.2, the thermal analysis has been performed. More than 100 degree, the temperature can be decreased at disk edge from the maximum temperature when number of heat pipes were equal to 32 and the extensive numerical simulations has been shown. Furthermore, with the increase in the working fluid of convective heat transfer coefficient which is inside the heat pipes up to 10,000 W/m².°C, along with the maximum temperature at the disk rim could be decreased with more than 280 degree. The observation has

been taken that the required time for achieving the minimum steady-state temperature has been more sensitive to the air convective heat transfer coefficient.

(Naveen & Bhuvaneshwaran, 2017)[5] Three dimensional CFD modeling has been studied in this paper on friction factor along with heat transfer including the thermal workability of concentric tube heat exchanger with the usage of twisted tapes that are Jagged V-cut as well as V-cut including Plain, V-cut along with several twist ratios that are $y=4.0$, 2.0 have been taken in usage. By creating turbulence in the fluid flow, twisted tapes have been taken in usage to augment the heat transfer. For increasing the thermal performance several methods have been applied on heat transfer devices like rough surface as well as treated surface including coiled tubes together with surface tension devices and also swirling flow devices. For an increase in the thermal workability from many of these methods, twisted method has been taken in usage. For several Reynolds Number effectiveness has been analyzed of heat exchanger by inserting twisted tape. With the help of Jagged V-cut twisted tape that is $H=50$, maximum thermal performance factors have been obtained by inserting as compared to other twisted tapes. Consecutively analysis has been done on the friction factor.

(SRIDHAR & 1, 2017)[6] heat exchanger has been affected by the flow pattern through a heat exchanger. Lowest transfer surface area has been required by the heat exchanger for counter flow. Higher value has been given for the log mean TEMPERATURE difference as compared to the heat exchanger of parallel flow along with the heat exchanger of cross flow. Numerous different types of flow patterns could be provided from the heat exchangers. Some common types of heat exchangers are cross flow as well as parallel flow along with counter flow. The most efficient heat exchanger with flow pattern out of three is the counter flow heat exchanger. It has been indicated that the surface area with lowest needed heat exchanger due to the log mean TEMPERATURE drop has been the maximum for a counter flow heat exchanger. Analysis has been done in this research for the comparison of heat transfer rates among the basic two flow arrangements: (i) the unidirectional parallel flow also known as parallel flow as well as co-current flow. (ii) Bidirectional flow also known as the counter

flow as well as counter-current flow. For various fluids, thermal analysis as well as CFD analysis has been done on the heat exchanger by taking refrigerants R134A, as well as R22, along with R600A including hot water together with various materials of heat exchangers. In Pro/Engineer, 3D models have been done along with the Ansys done by the analysis.

(Dubey, Singh, Yadav, Pal, & Thakur, 2017) [7] Fins have been lengthy surfaces that help to dissipate the heat which is generated in the engine. On the other hand such types of lengthy surface length have been restricted that bound the rate of heat dissipation. Several industries of automobiles have worked on the rate of heat dissipation for increase through which the efficiency of the engine could also be increased. **(ISagar Jagtap, 2017)** [8] inner tube of the threaded pipe with dimple in the middle of the horizontal THE, the characteristics of heat transfer has been investigated in this study. The effect on heat transfer through triple pipe has been lead to finding of this research. Through this research it has been expected that outcome of experimental as well as application of CFD were equal to the TTHE with the providence of turbulence in inner tube along with the spiral movement of the fluid of triple heat exchanger intermediate tube.

(Paul, Mathew, & Shripad, 2017)[9] Most popular illumination sources that are the advantage of LEDs are its higher efficiency as well as its compact size along with long life and so on. One of the highest energy saving source are the usage of lamp that are LED. Thermal analysis of honeycomb heat sink has been taken in usage in street lighting for high power LED application that has been presented in this paper. Three structure which are single cell, as well as two cell along with three cell honeycomb heat sink for high power LED lamps have been modelled in Solidworks. In ANSYS software, similar thermal analysis has been performed. One of the better configuration has been selected from the three models has been based on the outcomes from the thermal analysis along with the consideration amount and the cost of the heat sink materials. The reliability as well as the life of the heat sink would be increased of the properly designed heat sink of the LED lamp.

(P, Khond, & Lele, 2016) [10] concentric triple tube heat exchanger has been involved in the present study which includes heat transfer CFD

analysis. For evaluation as well as numerical simulation of heat transfer theoretical studies have been carried out. The workability of TCTHE at variable operating conditions has to be evaluated. For C-H-N as well as N-H-C configuration experimental along with studies has been carried out already in TTHE. For various operating conditions, numerical investigation has been also carried out already. In present study, for N-H-C configuration, CFD analysis has been taken in usage to validate the experimental data.

(Ali, Krishna, Reddy, & Ali, 2015) [11] exchange heat between two fluids a device has been taken in usage that is Heat Exchanger through which the efficiency of operating has been increased. A major role has been played by such efficiencies in process industries for cost effective operations. In the meantime when the fluids are flowing through the heat exchanger, temperature would start exchanging with the fluids. To deal with the double pipe heat exchanger workability rate is the main aim of this paper through exchanging the materials that are used to recover the waste through heat input in refinery process of steam. By GAMBIT as well as CATIA, the design of Double pipe heat exchangers has been implemented. With the usage of ANSYS, the analysis of CFD has been done. Through three different kinds of materials such as copper as well as aluminium along with steel the Final Results have been obtained.

(Quadir, Badruddin, & Salman Ahmed, 2014) [12] usage of finite element method, workability of TCPHE has been carried out numerically under the steady state condition for arrangement of different flow along with non-insulated as well as insulated conditions of the heat exchanger. Three fluids have been taken in considerations which are cold water, as well as normal tap water including hot water. The outcomes have been presented for three fluids including their length for their different flow rates in the form of the dimensionless temperature variations of the heat exchanger. It has been found through the three fluids of temperature variation with the help of numerical predictions by using FEM follow closely to those obtained from experiments both in magnitude as well as trend provided correct overall heat transfer coefficients have been taken in usage. Studies of Parametric have been also carried out to show the effect of the

individual design parameter on the workability of the heat exchanger.

(Peigné, Inard, & Druette, 2013) [13] energy-efficient dwellings a new wood based air heating system has been presented in this paper for conducting the experimental test. To evaluate the resulting outlet temperatures along with the recovered amount of heat by the ventilation air in order to assess feasibility including the workability of the coupling a mechanical ventilation heat-recovery unit along with the TCTHE which was integrated into the chimney of a room-sealed wood-pellet stove for providing heat to the entire house was the main objective of this paper.

(Quadir, Jarallah, Ahmed, & Anjum, 2013) [14] Experimental study has been performed on the TCPHE under the condition of steady state for NHC as well as CHN two different flow arrangements, along with insulated including non-insulated conditions of the heat exchanger. Normal tap water as well as cold water along with hot water has been considered as three fluids. In the innermost pipe normal water has been flown under the NHC arrangement and in the inner annulus hot water has been flown along with the outer annulus hot water has been flown. Parallel are fluids are being flown with each other. By keeping the water being unchanged in the CHN arrangement, normal as well as cold water has been interchanged. In the form temperature variation the results has been presented of three fluids including the different flow rates of the heat exchanger together with their length. For two arrangements, temperature variation has been found including the length of the pipe which substantially differs. In the non-insulated condition, the temperature of cold water has been rapidly increased of NHC arrangement. In NHC arrangement has been found for cross over points for three fluids with higher volume flow rates.

METHODOLOGY

Expected Procedure to be followed during the complete study:

1. Modeling of concentric tube heat exchanger according to the selected base paper.
2. Further converting the file for compatibility in simulation process.

3. Importing the file in Ansys Fluent for performing the simulation in Ansys.
4. Assigning the name selection to the different parts of concentric tube heat exchanger.
5. Perform the meshing of concentric tube heat exchanger compatible for CFD analysis.
6. Providing the suitable boundary conditions according to the selected base paper.
7. Assigning the material properties.
8. Setting the proper setup for CFD analysis procedure.
9. Evaluating the results after the finish of simulation work.

Mathematical model

For the non-insulated triple concentric pipe heat exchanger, the heat exchange takes place among four fluids with three thermal communication surfaces as shown in Figure6. The fourth fluid is the surrounding ambient.

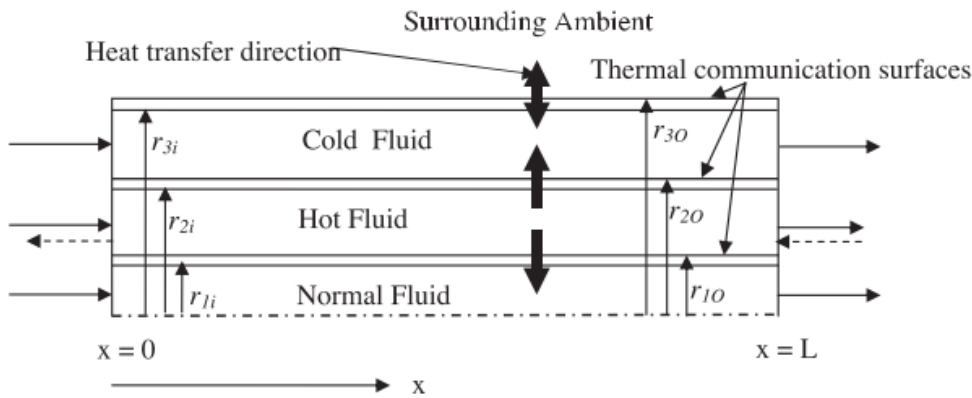


Figure 6:Four fluids exchange heat through three thermal communication surfaces

By applying the energy balance between different fluids, the following governing equations for each of the stream are written:

$$(\dot{M}C_p)_h \frac{dT_h}{dx} = \mp U_2 P_2 (T_h - T_c) \mp U_1 P_1 (T_h - T_n) \tag{1}$$

$$(\dot{M}C_p)_n \frac{dT_h}{dx} = +U_1 P_1 (T_h - T_n) \tag{2}$$

$$(\dot{M}C_p)_n \frac{dT_c}{dx} = +U_2 P_2 (T_h - T_n) + U_0 P_0 (T_\infty - T_c) \tag{3}$$

Where, the negative sign in Eq. (1) is for co-current parallel flow and the positive sign is for the counter-current parallel flow. Using the following dimensionless variables:

$$\theta = \frac{T - T_{c.in}}{T_{h.in} - T_{c.in}}, \text{ and } X = \frac{x}{L_e}$$

The above equations are presented in dimensionless form as:

$$\frac{d\theta_h}{dX} = \pm \frac{U_2 P_2 L_e}{(\dot{m}c_p)_n} (\theta_h - \theta_c) \pm \frac{U_1 P_1 L_e}{(\dot{m}c_p)_n} (\theta_h - \theta_n) \tag{4}$$

$$\frac{d\theta_h}{dX} = + \frac{U_1 P_1 L_e}{(\dot{m}c_p)_n} (\theta_h - \theta_n) \tag{5}$$

$$\frac{d\theta_h}{dX} = + \frac{U_2 P_2 L_e}{(\dot{m}c_p)_n} (\theta_h - \theta_c) \pm \frac{U_0 P_0 L_e}{(\dot{m}c_p)_c} (\theta_\infty - \theta_c) \tag{6}$$

Where, all other symbols are defined in the nomenclature section.

$$NTu_e = \frac{U_2 P_2 L_e}{(\dot{m}c_p)_h}; NTu_{e\infty} = \frac{U_0 P_0 L_e}{(\dot{m}c_p)_h}; R_1 = \frac{(\dot{m}c_p)_h}{(\dot{m}c_p)_c}$$

$$R_2 = \frac{(\dot{m}c_p)_h}{(\dot{m}c_p)_n}; H = \frac{U_1 P_1}{U_1 P_1}; \text{ And } \theta = \frac{T_{n.in} - T_{c.in}}{T_{h.in} - T_{c.in}}$$

Where, positive sign in \pm corresponds to the co-current parallel flow.

Governing equation

The numerical analysis of the fluid flow of the air is focused. The main objective is to analyze the thermal behavior and velocity distribution

generated by the temperature gradient, and to compare the thermal behavior of the reference solar chimney with the solar chimney without draft and drft here. The following

considerations were done for the mathematical model: 1. Steady state flow 2. Boussinesq model 3. k-epsilon turbulence model . The density is calculated using the Boussinesq approximation in the momentum equation in y

direction. The conservation equations are adapted to the model, considering constant properties and including the Boussinesq's equation in the y-component of the Navier-Stokes equation.

Continuity equation

$$\rho \left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} \right) = 0$$

Energy Equation

$$\frac{\partial(\rho H)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \times u_j \times c_p \times T) = \frac{\partial}{\partial x_j} \left(\lambda \cdot \frac{\partial T}{\partial x_j} \right) + S_E$$

Where

ρ is the fluid mass density,

S_E is the source term.

Momentum equations

Component- x

$$u_x \frac{\partial u_x}{\partial x} + u_y \frac{\partial u_x}{\partial y} + u_z \frac{\partial u_x}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{\mu}{\rho} \left(\frac{\partial^2 u_x}{\partial x^2} + \frac{\partial^2 u_x}{\partial y^2} + \frac{\partial^2 u_x}{\partial z^2} \right)$$

Component- y

$$u_x \frac{\partial u_y}{\partial x} + u_y \frac{\partial u_y}{\partial y} + u_z \frac{\partial u_y}{\partial z} = g_y \beta (T - T_\infty) + \frac{\mu}{\rho} \left(\frac{\partial^2 u_y}{\partial x^2} + \frac{\partial^2 u_y}{\partial y^2} + \frac{\partial^2 u_y}{\partial z^2} \right)$$

Component- z

$$u_x \frac{\partial u_z}{\partial x} + u_y \frac{\partial u_z}{\partial y} + u_z \frac{\partial u_z}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{\mu}{\rho} \left(\frac{\partial^2 u_z}{\partial x^2} + \frac{\partial^2 u_z}{\partial y^2} + \frac{\partial^2 u_z}{\partial z^2} \right)$$

The Standard k-ε Model

In these equations, G_k is represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated as described in Modeling Turbulent Production in the k-ε Models.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k$$

and

$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon$$

Design dimension

The three concentric pipe heat exchanger, the exit temperatures of all the fluids flowing through the heat exchanger are to be known. For this, the finite element method is used. In the triple pipe heat exchanger three types of fluids i.e. hot water, cold water and normal tap water flowing in three pipes of the heat exchanger.

Table 1:Parameter of Heat Exchanger Model

S.NO.	Outer diameter	Thickness	Length
1	0.0508m	1.5mm	4m
2	0.0762m	1.5mm	4m
3	0.1016m	1.5mm	4m

Building the Model

The ANSYS Design Modeler provides the following approaches for model generation: Creating a solid model within ANSYS Design Modeler. The diameter of the heat exchanger is 0.0508m, 0.0762m and 0.1016m respectively, and the thickness of the pipe is 1.5mm.

Various steps are used to create heat exchanger model.

Step-1

Draw the sketch of pipe outer diameter and inner diameter as shown in figure 6.

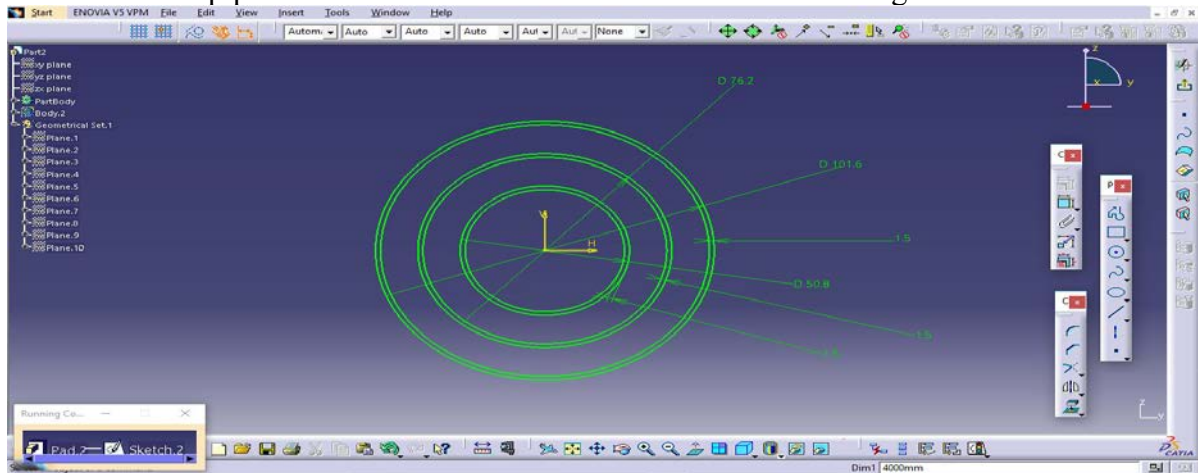


Figure 6: Pipe circle diameter in sketch tool in CATIA

Step-2

Select pad command of sketch of pipe and give 4000mm length as shown in figure 7.

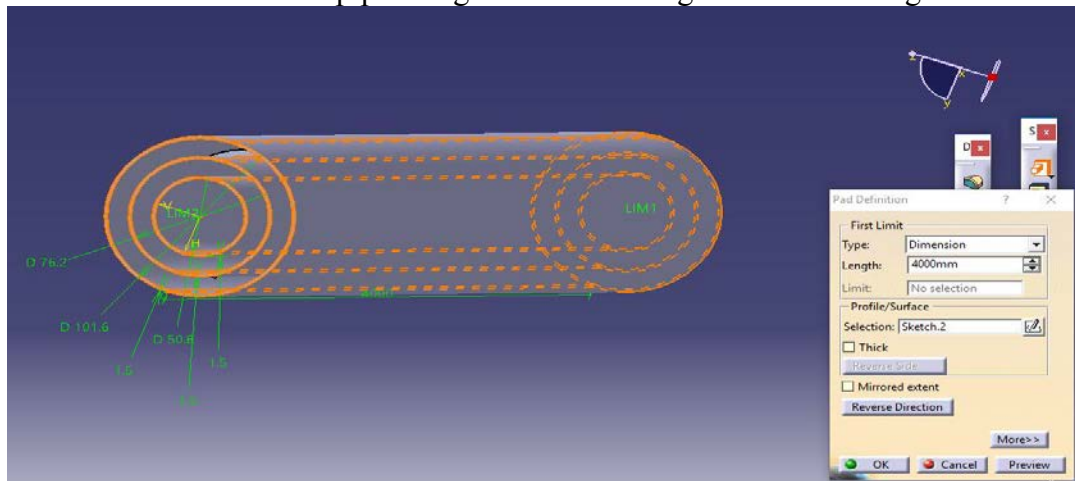


Figure 7: Use pad command in CATIA

Step-3

Fill material inside the pipe and show dimension in pipe (meter) as shown in figure 8.

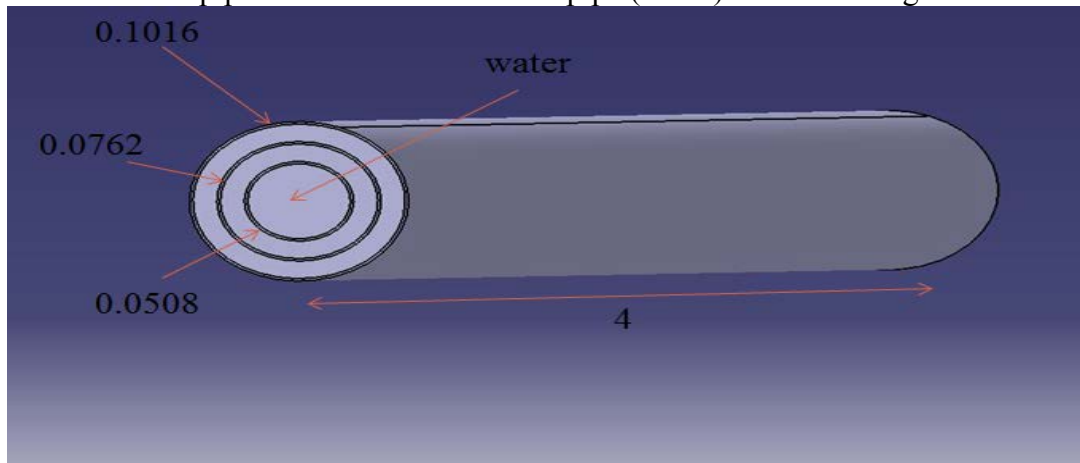


Figure 8: Final model in heat exchanger

Material property

The three pipe are used in fabrication of the heat exchanger are made of stainless steel 40S-

schedule, ANSI / ASME 36.19 as shown in table 2.

Table 2: Material property

Material	Density kg/m ³	Specific heat J/kg-k	Thermal conductivity W/m-k
Steel	8030	502.48	16.27
Water	998.2	4182	0.6

Boundary condition

The three fluids having the same volume flow rates as, $V_h = V_c = V_n = 35$ l/min under co-current parallel flow arrangement of the insulated heat exchanger. The inlet temperatures of the hot water, cold water, and normal water are 51.22 °C, 10.93 °C, and 27.47 °C

respectively. The developed computer program is used to predict the temperature distributions of the three fluids along the length of the heat exchanger. All simulations were performed using ANSYS FLUENT that can simulate a two type of model of a Heat Exchanger. These are given in table 3.

Table 3 Inlet boundary condition

S.NO	Condition	Inlet temperature (°C)	Inlet velocity (l/min)
1	Cold	10.23	35
2	Hot	52.11	35
3	Normal	28.23	35

After applying the proper boundary condition the simulation is run. And the results were recorded.

by using contour definition at different section of cold, hot and normal pipe.

RESULTS

After performing CFD analysis the result are obtained as a term of temperature difference outlet to inlet section. The temperature is major

Result for simple pipe

The figure shows temperature contour obtained for the case of simple pipe without any porosity. The temperature is shown at the outlet of the pipe in figure 9.

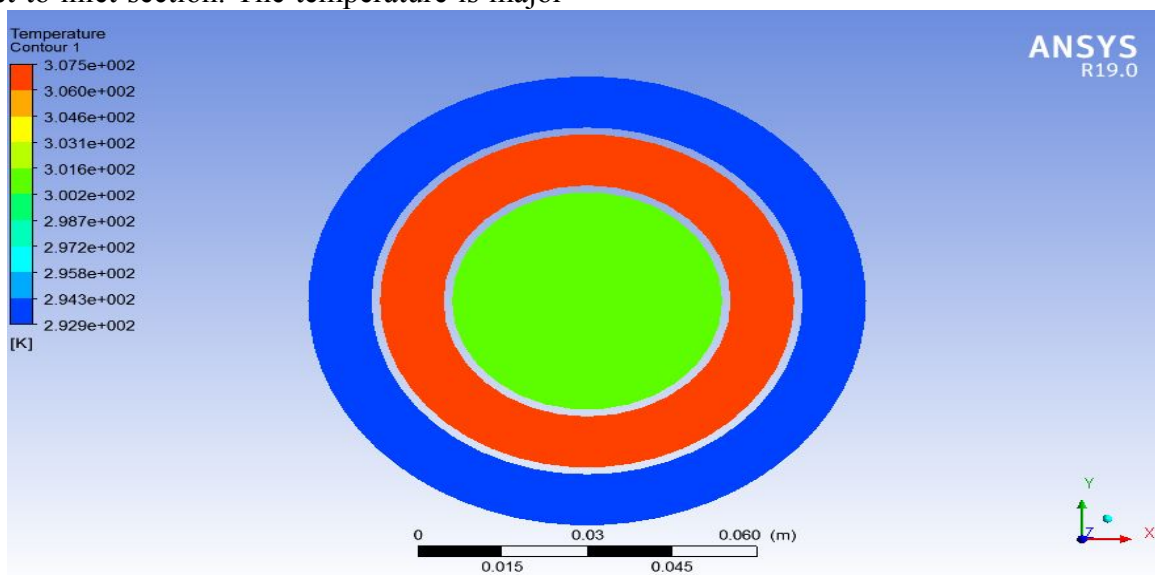


Figure 9: Temperature of simple pipe at outlet

The temperature volume rendering of complete geometry of the pipe is shown in figure 10.

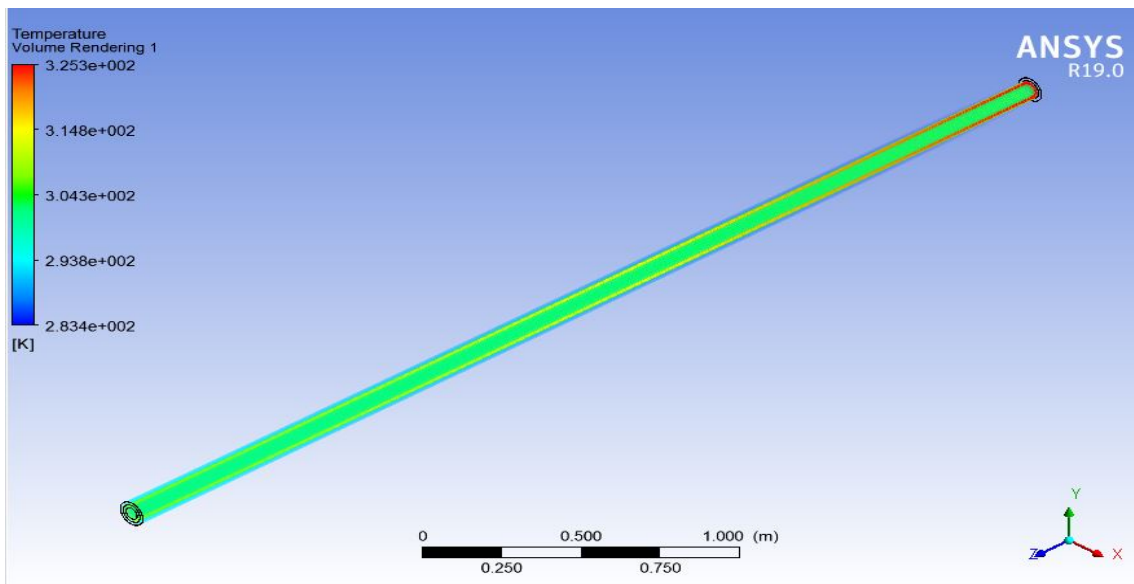


Figure 10:Temperature volume rendering in simple pipe

Figure 11 shows velocity streamline in the complete geometry of the simple pipe.

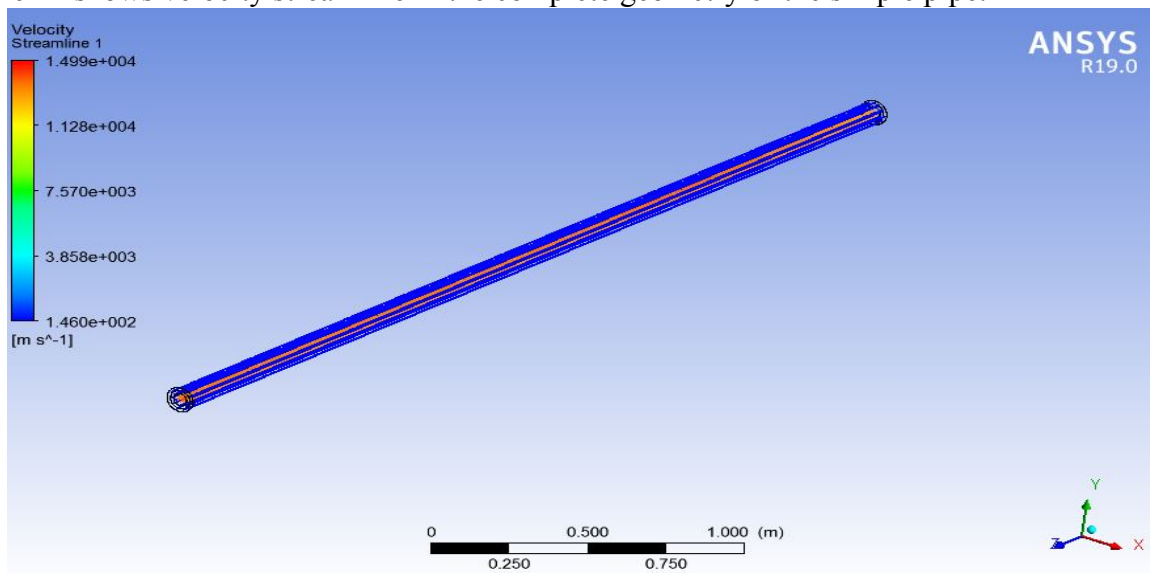


Figure 11:Velocity streamline of simple pipe

Figure 12 shows a close look at velocity vector in the complete geometry of the simple pipe.

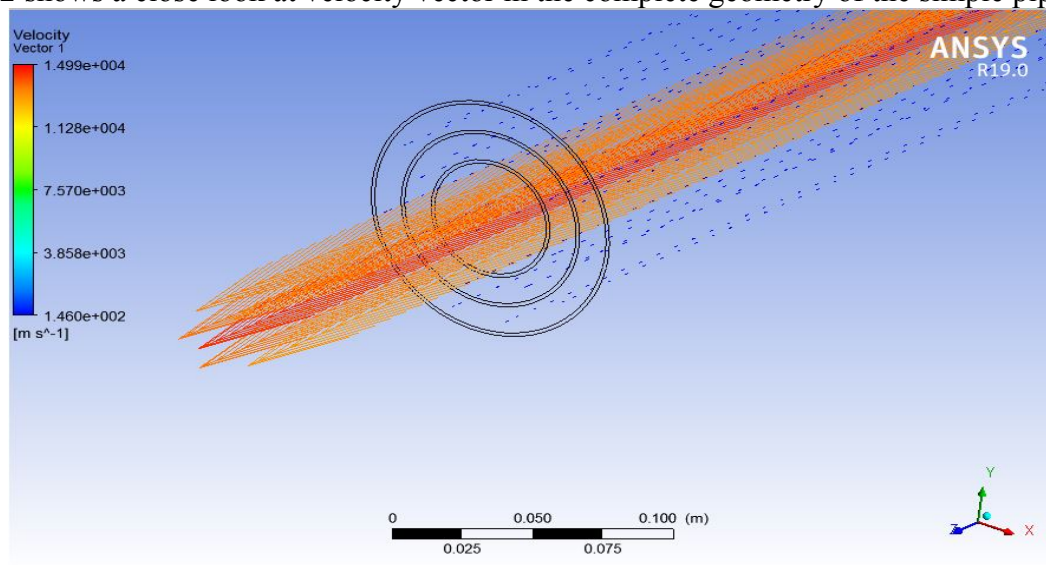


Figure 12:Velocity vector of simple pipe

Table 4 shows tabulated data of cold, hot and normal showing inlet and outlet temperature of simple pipe.

Table 4 Temperature at Inlet and Outlet for simple pipe

	Inlet	Outlet
Cold	283.38	293.419
Hot	325.26	307.366
Normal	301.38	301.543

Figure 13 shows the trend of temperatures of hot, normal and cold liquid. Hot liquid shows a decrease in temperature whereas cold liquid shows increase in temperature. And normal liquid temperature remains same.

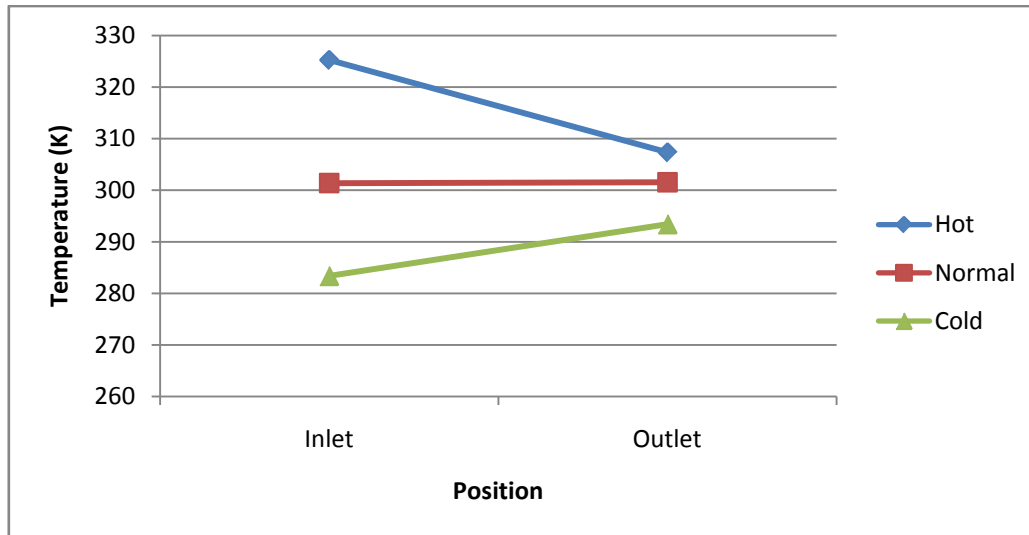


Figure 13: Graph for inlet and outlet temperature

Pipe with 25% porosity

The figure shows temperature contour obtained for the case of pipe with 25% porosity. The temperature is shown at the outlet of the pipe in figure 14.



Figure 14: Temperature at outlet of pipe with 25% porosity

Figure 15 shows the temperature volume rendering of complete geometry of pipe with 25% porosity.

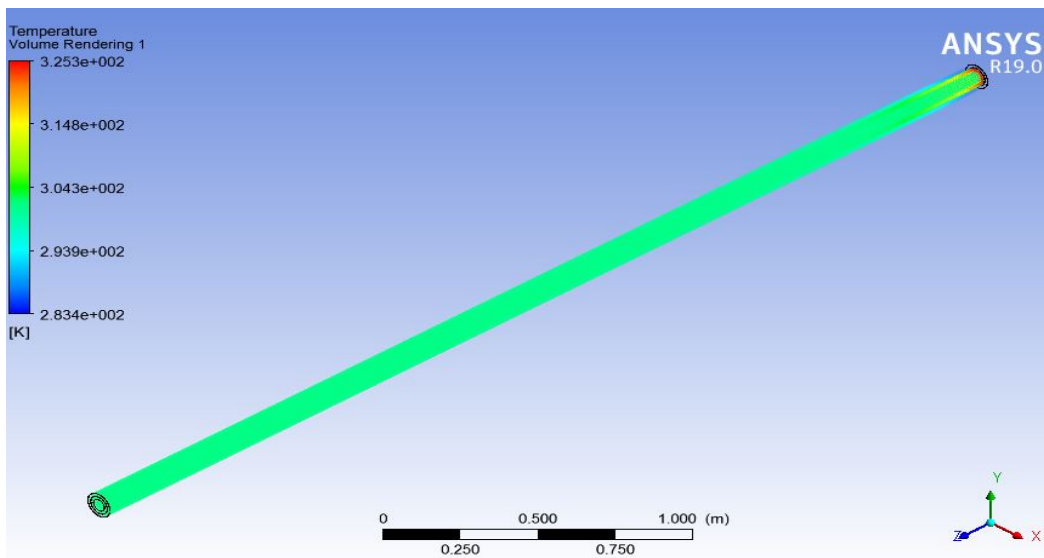


Figure 15:Temperature volume rendering in pipe with 25% porosity

Figure 16 shows the velocity streamline in the complete geometry of pipe with 25% porosity.

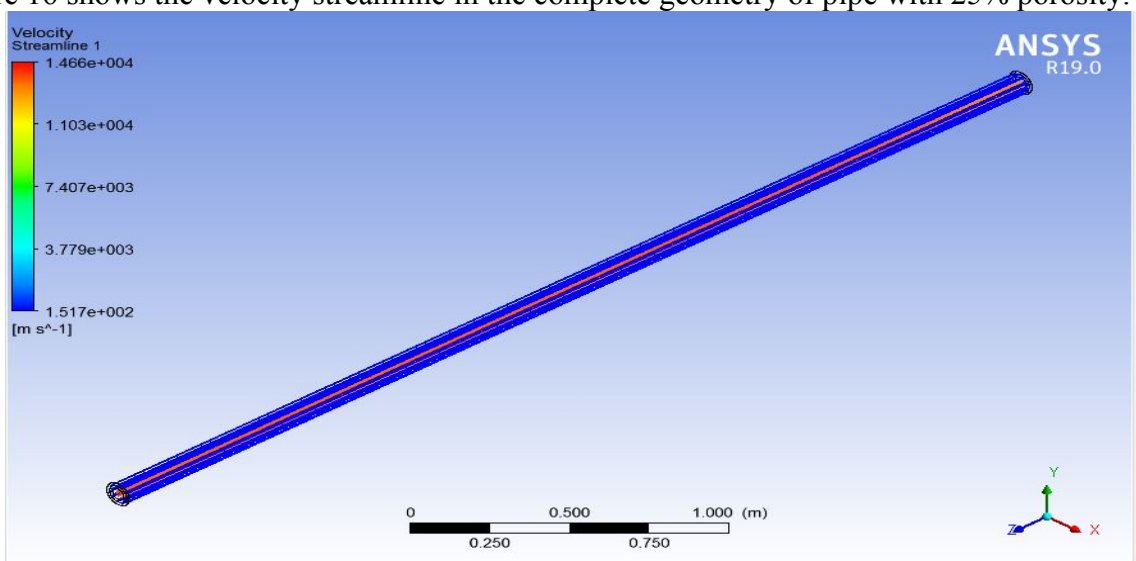


Figure 16:Velocity streamline pipe with 25% porosity

Figure 17 shows the velocity vector in the complete geometry of pipe with 25% porosity.

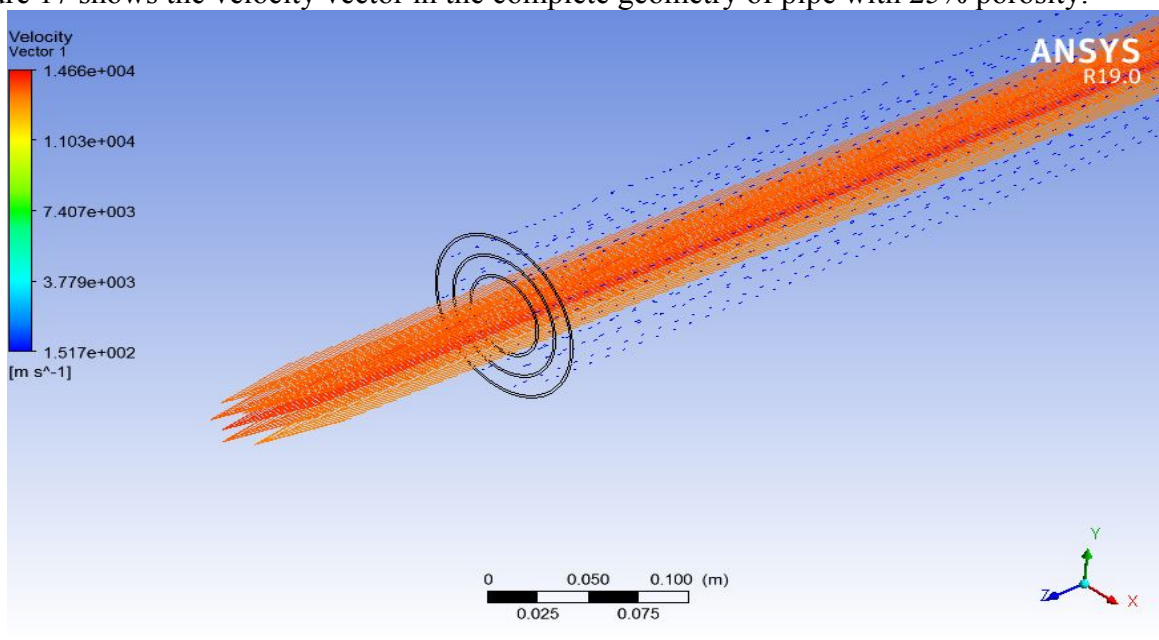


Figure 17:Velocity vector of pipe with 25% porosity

Table 5 shows tabulated data of cold, hot and normal showing inlet and outlet temperature of pipe with 25% porosity.

Table 5: Inlet and outlet temperature of pipe with 25% porosity

	Inlet	Outlet
Cold	283.38	301.414
Hot	325.26	301.446
Normal	301.38	301.499

Figure 18 shows the trend of temperatures of hot, normal and cold liquid. Hot liquid shows a decrease in temperature whereas cold liquid shows increase in temperature. And normal liquid temperature remains same in the pipe with 25% porosity.

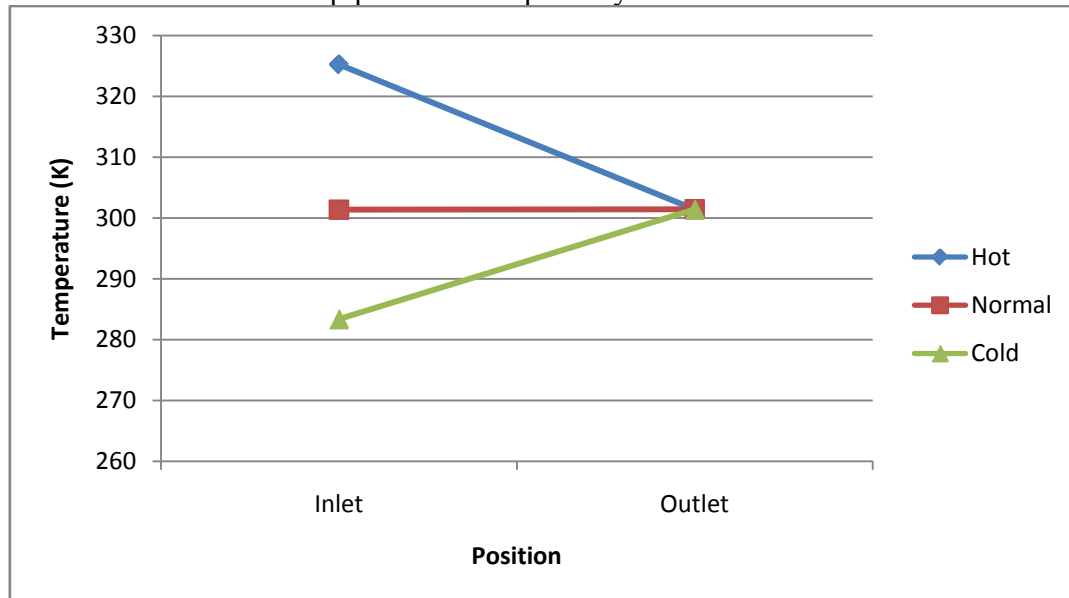


Figure 18: Graph for inlet and outlet temperature

Pipe with 50% porosity

The figure shows temperature contour obtained for the case of pipe with 50% porosity. The temperature is shown at the outlet of the pipe in figure 19.

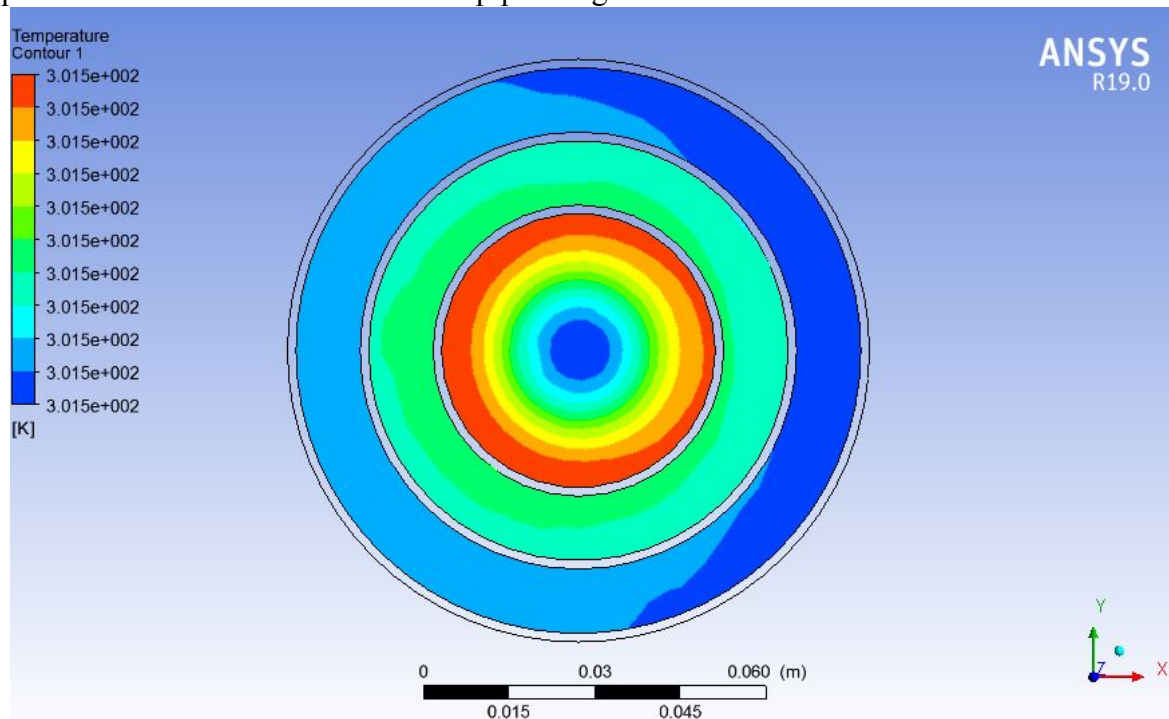


Figure 19: Temperature at outlet of pipe with 25% porosity

Figure 20 shows the temperature volume rendering of complete geometry of pipe with 50% porosity.

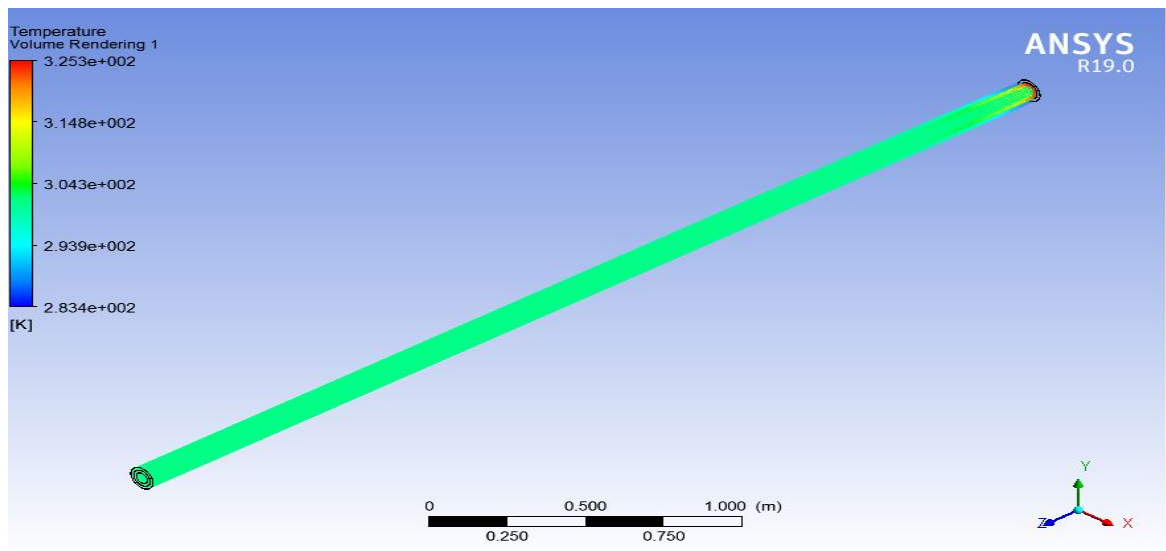


Figure 20:Temperature volume rendering in pipe with 50% porosity

Figure 21 shows the velocity vector in the complete geometry of pipe with 50% porosity.

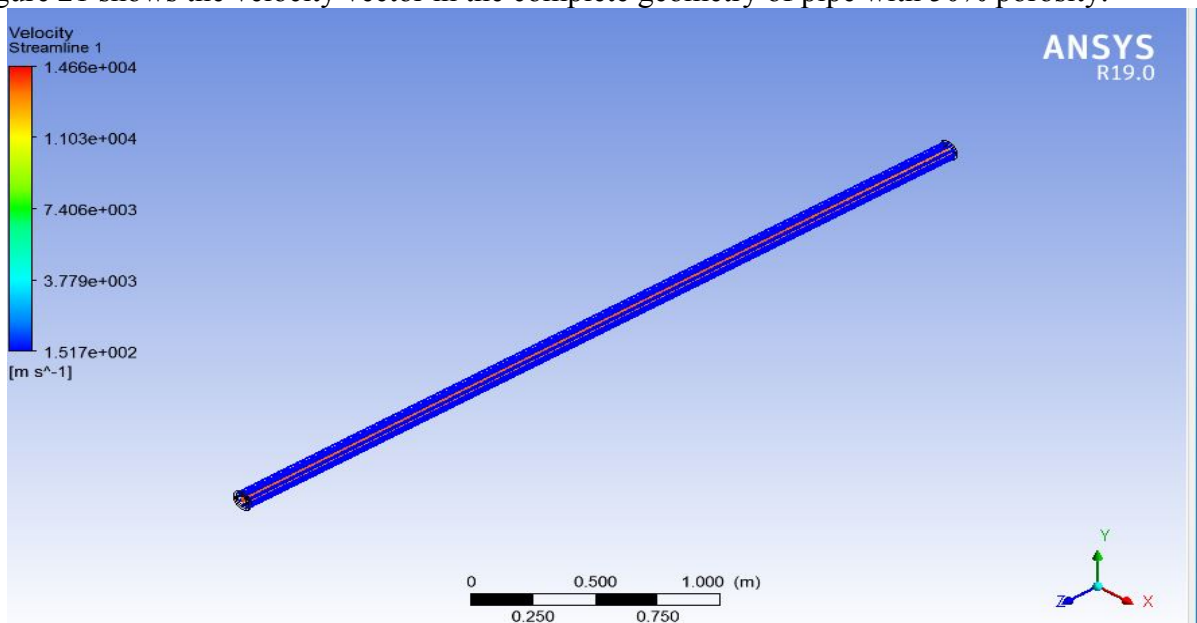


Figure 21:Velocity streamline pipe with 50% porosity

Figure 22 shows the velocity vector in the complete geometry of pipe with 50% porosity.

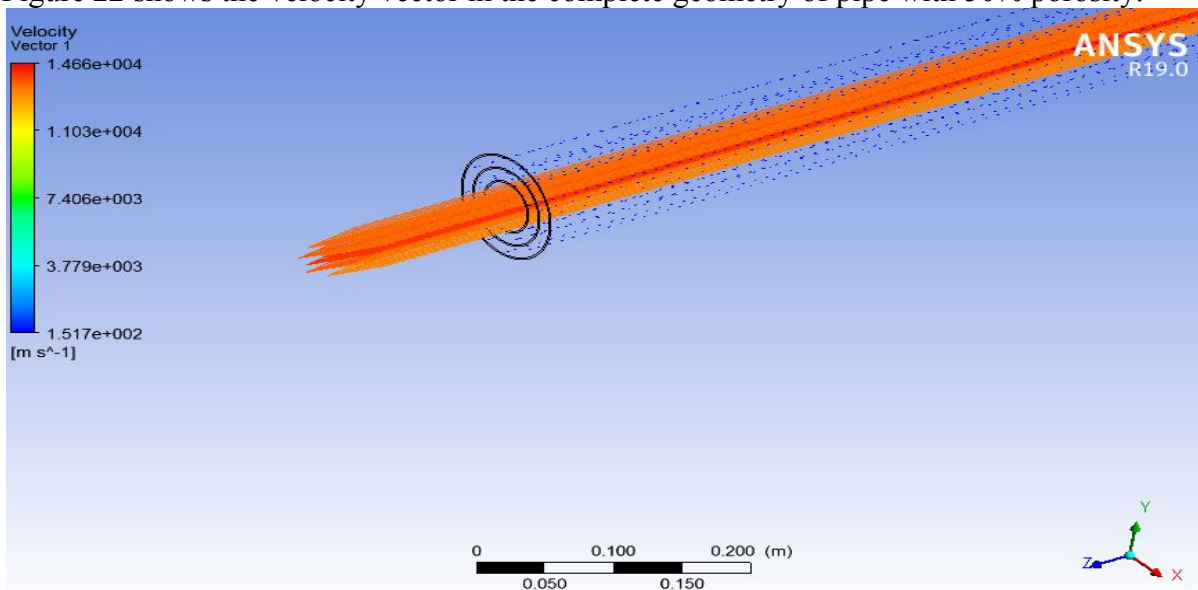
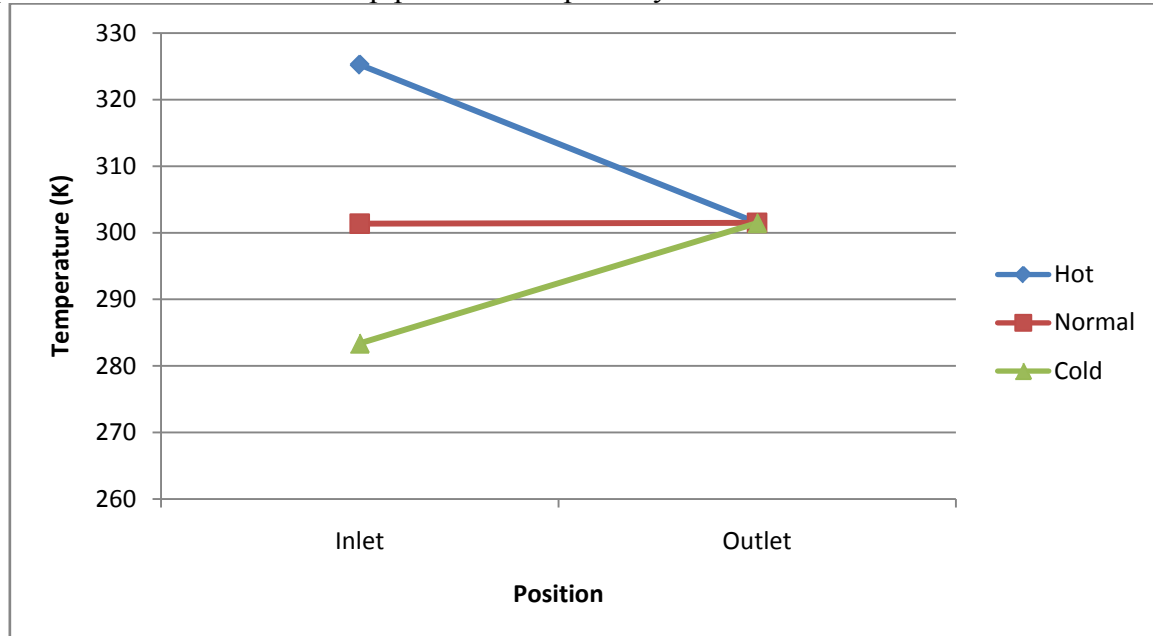


Figure 22:Velocity vector of pipe with 25% porosity

Table 6: Inlet and outlet temperature of pipe with 50% porosity

	Inlet	Outlet
Cold	283.38	301.466
Hot	325.26	301.482
Normal	301.38	301.497

Figure 23 shows the trend of temperatures of hot, normal and cold liquid. Hot liquid shows a decrease in temperature whereas cold liquid shows increase in temperature. And normal liquid temperature remains same in the pipe with 50% porosity.

**Figure 23:** Graph for inlet and outlet temperature

Comparisons of result

Table 7 shows the tabulated data of cold, hot and normal outlet temperatures of simple pipe, pipe with 25% and 50%.

Table 7: Comparison of cold, hot and normal outlet temperature of simple, 25% porosity and 50% porosity

	Simple Pipe	25 % Porosity	50% Porosity
Cold outlet	293.419	301.414	301.466
Hot Outlet	307.366	301.446	301.482
Normal Outlet	301.543	301.499	301.497

CONCLUSIONS

CFD simulation is carried out and results are obtained for simple concentric pipe, pipe with 25% porosity and 50% porosity. These results are

- Simulation carried out on simple pipe and cold liquid gives temperature of 293.419 k on outlet, hot liquid shows 307.366 k and normal temperature as 301.543 k.
- When porosity is added to the pipes of 25%, then the outlet temperature of pipe of cold liquid is obtained as 301.414 k, hot liquid is obtained as 301.446 k and for normal liquid; it is obtained as 301.499 k.

- Porosity is further increased to 50%, cold liquid shows temperature at outlet as 301.466, hot liquid as 301.482 and for normal liquid; it is obtained as 301.497 k.

Thus, it is clearly observed that, after adding porosity to the material of pipe, results are greatly improved and it can be concluded that, by increasing porosity, heat exchange process can be accelerated.

REFERENCES

- [1] U. Rehman, "Heat Transfer Optimization of Shell-and-Tube Heat Exchanger through CFD Studies Heat Transfer Optimization of Shell-and-Tube Heat Exchanger," p. 29, 2011.

- [2] P. Dharmik A, V. D. Dhiman, J. J. Patel, and R. Engineer, "CFD analysis of triple concentric tube heat exchanger," *Univ. J. Res.*, vol. 01, no. 11, pp. 0–0, 2015.
- [3] T. kumar Hotta, R. Nanda Sajja, R. Teja Sigireddy, and V. Sai Mugala, "Design of Fins To Maximize the Heat Transfer Rate From the Engine Cylinder," *Int. J. Mech. Eng. Technol.*, vol. 9, no. 94, pp. 213–223, 2018.
- [4] Y. Taamneh, "Thermal analysis of gas turbine disk integrated with rotating heat pipes," *Case Stud. Therm. Eng.*, vol. 10, no. June, pp. 335–342, 2017.
- [5] S. Naveen and S. Bhuvaneshwaran, "CFD Analysis of Concentric Tube Heat Exchanger Using Twisted Tapes," *International J. Adv. Res. Ideas Innov. Technol. ISSN*, vol. 3, no. 1, pp. 870–879, 2017.
- [6] K. SRIDHAR and K. B. 2 1, "Comparative Analysis of Parallel and Counter Flow Heat Exchangers," vol. 06, no. 04, pp. 638–644, 2017.
- [7] D. Dubey, D. Singh, A. Yadav, S. Pal, and H. Thakur, "Thermal Analysis of Engine Cylinder having thick tip fin with varying slot sizes and material," *Mater. Today Proc.*, vol. 4, no. 8, pp. 7636–7642, 2017.
- [8] 2Prasad Ingavale 3Deepak More 1Sagar Jagtap, "Review on triple tube heat exchanger with dimple on internal tube & internal threaded middle tube using CFD and Experimental analysis for heat transfer," pp. 796–798, 2017.
- [9] B. Paul, M. Mathew, and P. G. Shripad, "Modelling and thermal analysis of honey comb heat sink for LEDs in street lighting applications," *Proc. Int. Conf. Inven. Syst. Control. ICISC 2017*, pp. 1–5, 2017.
- [10] D. Saurabh, H. Khond, and M. M. Lele, "CFD Analysis of a Triple Concentric Tube Heat Exchanger having water flowing at three different temperatures," *Int. J. Curr. Eng. INPRESSCO IJCET Spec. Issue*, vol. 4, no. 4, pp. 49–52, 2016.
- [11] M. Z. M. S. Ali, M. Krishna, D. V. V. S. B. Reddy, and R. S. M. Ali, "Thermal Analysis of Double Pipe Heat Exchanger by Changing the Materials Using CFD," vol. 26, no. 2, pp. 95–102, 2015.
- [12] G. A. Quadir, I. A. Badruddin, and N. J. Salman Ahmed, "Numerical investigation of the performance of a triple concentric pipe heat exchanger," *Int. J. Heat Mass Transf.*, vol. 75, pp. 165–172, 2014.
- [13] P. Peigné, C. Inard, and L. Druette, "Experimental study of a triple concentric tube heat exchanger integrated into a wood-based air-heating system for energy-efficient dwellings," *Energies*, vol. 6, no. 1, pp. 184–203, 2013.
- [14] G. A. Quadir, S. S. Jarallah, N. J. S. Ahmed, and I. Anjum, "Experimental investigation of the performance of a triple concentric pipe heat exchanger," *Int. J. Heat Mass Transf.*, vol. 62, pp. 562–566, 2013.