As per UGC guidelines an electronic bar code is provided to seure your paper

International Journal for Modern Trends in Science and Technology, 7(11): 153-161, 2021

ISSN: 2455-3778 online
DOI: https://doi.org/10.46501/IJMTST0711026
Available online at: http://www.ijmtst.com/vol7issue11.html

The Systematic Comparision on Analysis of Parallel Flow and Counter Flow Heat Exchanger by using CFD and Practicle Methods

Shaik Chand Mabhu Subhani[1] | Pilli Sravani[2]

[1] Department of Mechanical Engineering, Eswar college of Engineering,
[2] Department of Mechanical Engineering, Narasaraopeta engineering college(Autonomous)

To Cite this Article
Shaik Chand Mabhu Subhani and Pilli Sravani. The Systematic Comparision on Analysis of Parallel Flow and Counter Flow Heat Exchanger by using CFD and Practicle Methods. *International Journal for Modern Trends in Science and Technology* 2021, *7*, pp. 153-161. https://doi.org/10.46501/IJMTST0711026.

Article Info
Received: 11 October 2021; Accepted: 19 November 2021; Published: 23 November 2021

ABSTRACT

A Heat Exchanger is a device which is used to transfer heat from one fluid to another, whether the fluids are separated by a solid wall so that they never mix, or the fluids are directly in contact. Every year Heat exchanger technology is growing to develop efficient, compact and economical heat exchangers, all over the world. Updating the community for this development needs an interaction. These days concentric tube heat exchangers are used with forced convection for lowering the working fluid's temperature by raising the cooling medium's temperature. The purpose of this project is to use ANSYS FLUENT software and practical calculations to analyse the temperature drops as a function of both inlet velocity and inlet temperature and how each varies with the other. Each heat exchanger model was designed and simulated for both parallel flow and counter flow heat exchanger models. The results were compared between parallel and counter flow heat exchangers. CFD analysis was utilized to find the outlet temperatures of parallel and counter flow heat exchangers for the inlet velocity and inlet temperature of the fluid medium used. "Computational Fluid Dynamics (CFD) is a science of predicting fluid flow, heat transfer, mass transfer, and related phenomena by solving the mathematical equations which govern these processes using numerical processes". These outlet temperature values obtained were used to determine the overall heat transfer coefficient. Theoretical calculations are done by the values obtained through the experiment conducted on the heat exchanger setup for both parallel and counter flow

KEYWORDS: Heat Exchangers, Parallel flow, Counter flow,temperature , CFD Analysis , ansys.

INTRODUCTION

Today's demand of higher energy consumption and reduced availability of fossil fuel resources increase the impact of thermal performance of heat exchanger day by day. Heat exchangers are very effective for the transfer of heat from one medium to another without even intermixing one fluid with another. One of the most promising devices for heat transfer is the counter flow heat exchanger mostly adapted by the chemical plants, petrochemical plants, oil refineries etc. Reducing the temperature of hot outlet fluid without affecting the cost is a big task for various industries that could be only possible by the proper selection of input. Typically, in a heat exchanger two segregated fluids at different temperature with a solid boundary, exchange thermal energy from one fluid to another via surface without

even intermixing. There are numerous configurations of classifying heat exchanger. In context with the flow configuration, there exists three primary types for heat transfer: parallel flow, counter flow and. cross flow. According to Fourier for the conduction states the more the area of heat exchanger, the more will be the heat transfer rate.

By second law of thermodynamics only transfer of sensible heat occurs in the heat exchanger. One of the greatest advantages of the counter flow heat exchanger is higher uniform temperature difference as well as that the mass flow rate and time for the interaction of one fluid with other increases, the heat transfer also goes up as compared to parallel flow heat exchanger. Maximization of surface area and minimization of flow resistance lead to better effectiveness of heat exchanger, which is the main focus for designing. On the contrary, the increase in area increases the space for the installation and correspondingly manufacturing cost will get increased. On the other hand, reduction in flow resistance can be achieved by improving the surface finishing of the heat exchanger. Many experiments have been carried out on the counter flow heat exchanger citing the flow in either laminar or turbulent manner, for achieving its better configuration. However, very limited CFD simulation has been done on the counter flow heat exchanger at different flow configuration to verify the thermo-hydraulic performance or to check the heat transfer and velocity distribution inside the flow domain.

Direction of Flow: According to the relative direction of two fluid streams the heat exchangers are classified into the following three categories:

1. Parallel flow
2. Counter flow
3. Cross – flow

1. Parallel flow heat exchangers: In parallel flow heat exchangers the fluids both hot and cold travel in same direction. The flow arrangement for hot and cold fluids from inlet to outlet is shown in fig 1.1. In parallel flow heat exchangers the temperature difference from hot to clod fluid decreases. This type of heat exchangers requires large space and hence it is rarely used in practical applications. Eg: Oil coolers, oil heaters, water heaters etc, are examples of parallel flow heat exchanger.

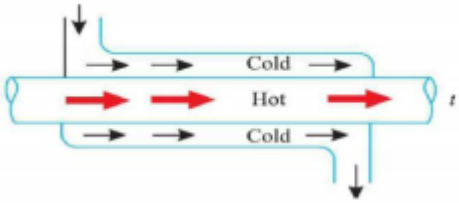


Fig 1.1. Parallel flow heat exchanger

2. Counter flow heat exchangers: In a counter flow heat exchanger, the two hot and cold fluids enter at opposite ends. Theflow arrangement and temperature distribution for such a heat exchanger are shown schematically in fig. 1.2. the temperature difference between the two fluids remains more or less nearly constant. This type of heat exchanger, due to counter flow, gives maximum rate of heat transfer f or a given surface area. Hence such a heat exchangers are most favored for heating and cooling of fluids.

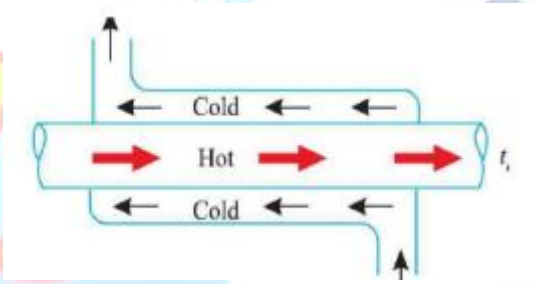


Fig. 1.2. Counter flow heat exchanger

3. Cross - flow heat exchanger: When two fluids crosses one another in space at right angles such type of heat exchanger is known as cross flow heat exchanger. In cross flow heat exchanger there is no mixing of fluid streams and hot fluid flows in spate tubes and cold fluid is mixes perfectly as it flows through the exchanger. The temperature of this mixed fluid will be uniform across any section and will vary only in the direction of flow. The cooling unit of refrigeration system is an example of cross-flow heat exchanger.

OBJECTIVES

The objective of the present study is to provide more complete understanding Flow misdistribution in tubular heat exchanger by studying area weighted and mass weighted temperature profiles for maldistribution without back flow and maldistribution with back flow. And comparison of average temperature profiles of flow maldistribution with the average temperature profiles of uniform mass flow distribution. This numerical investigation was carried out for the

concentric tube arrangement with different diameter of tubes. A finite volume numerical scheme is used to predict the conjugate heat transfer and fluid flow characteristics with the aid of the computational fluid dynamics (CFD) commercial code, FLUENT. An effective model, the standard based k-ε turbulence model was applied in this investigation. the available relevant literature is quite limited With respect to the analytical and it is still difficult to predict the physics of the flow maldistribution within the circular tube banks. Therefore, temperature distributions within the bundle were studied numerically. The objective of this study is to develop a CFD simulation to predict heat Transfer in concentric tube heat exchanger by using different fluids

RELATED WORK

Praveen Kumar Kanti et al, [1] investigated CFD analysis of shell and tube heat exchanger.Analysing shell and tube heat exchanger without baffle plates by changing their outer material.The calculations and simulations are done for counter flow of the heat exchanger.

D.Bhanuchandrarao et al, [2] investigated CFD analysis and performance of parallel and counter flow in concentric tube heat exchangers.The results were compared between eachmodel and between parallel and counter flow with fouled piping. Turbulent flow was alsoanalysed during the development of the heat exchangers to determine its effect on heat transfer. While as expected the fouled heat exchanger had a lower performance and therefore cooled the working fluid less, the performance of the counter heat exchanger unexpectedly of the parallel heat exchanger.

Oon et al. [3] have done a CFD simulation in fluent to avoid the backflow in an annular passage from the sudden expansion of pipe. The Nusselt number and the Reynolds numbers have been considered for the investigation and to show the uniform heat transfer from the sudden expansion. Then the heat flux approach was used in the fluent set up to show the heat transfer in the tubular pipe.

Nagarsheth et al. [4] used a systematic approach on a cross flow water tube in tube heat exchanger with a string of basic instrument like PID controller to regulate the temperature. PID controller is a kind of sensor which works in a closed loop feedback mechanism in industrial control system

Hwang et al [5] measured pressure drop and heat transfer coefficient in fully developed laminar pipe flow using constant heat flux conditions. Based on the experimental results they showed that the experimental friction factor was in good agreement with the theoretical predictions using the Darcy equation.

Bianco et al [6] observed only amaximum of 11% difference between single and two phase results for the laminar regime.

Akbari et al [7] for the first time compared three different two phase models and the single phase model in the laminar regime. Single and two phase models were found to be predicting identical hydrodynamic fields but very different thermal ones.

PROBLEM STATEMENT

The double pipe heat exchanger is used in industry such as condenser for Chemical process and cooling fluid process. This double pipe heat exchanger is designed in a large size for large application in industry. To make this small double pipe heat exchanger type become practicality, the best design for this small double pipe heat exchanger is choose. Heat transfer is considered as transfer of thermal energy from physical body to another. Heat transfer is the most important parameter to be measured as the performance and efficiency of the concentric tube heat exchanger. By using CFD simulation software, it can reduces the time and operation cost compared by Analytical calculations in order to measure the optimum parameter and the behaviour of this type of heat exchanger.

SCOPES OF RESEARCH

The scopes of this research are as follows:

i. Study on heat transfer for heat exchanger specific to double pipe heat exchanger types.

ii. Design the double pipe heat exchanger by using ANSYS WORKBENCH.

iii. Simulation in double pipe heat exchanger by using FLUENT software.

iv. Analysis the heat exchanger specific to flow rate of hot and cold fluid.

v. To simulate heat transfer in concentric tube heat exchanger by using CFD-Fluent software.

vi. To analyze the heat transfer in concentric tube heat exchanger by comparing the simulation result to the Analytical calculations. Validate simulation results to the Analytical calculations within 5% error.

GEOMETRY AND MODELING

The layout diagram for practical analysis on parallel flow and counter flow heat exchanger is as shown below

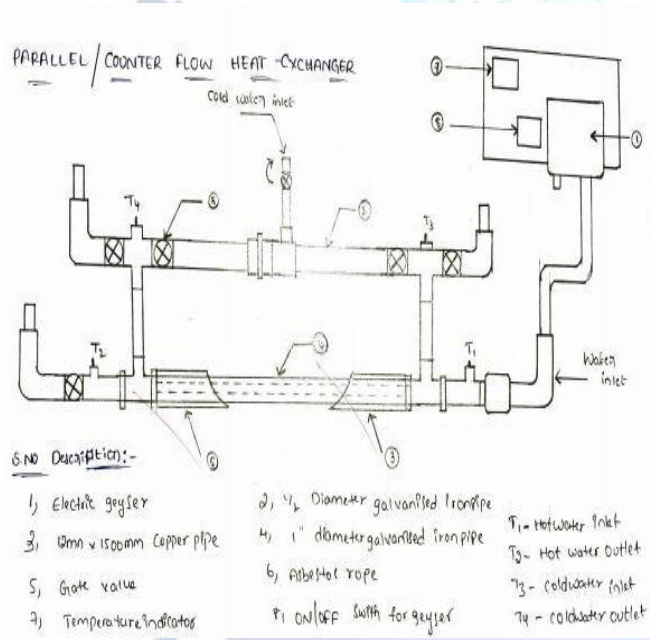


FIG 3.1: LINE DIAGRAM FOR PRACTICAL HEAT EXCHANGER

MODELING OF HEAT EXCHANGER BY USING ANSYS

The geometry made in ANSYS workbench. This geometry imported to ansys fluent and repairs the geometry

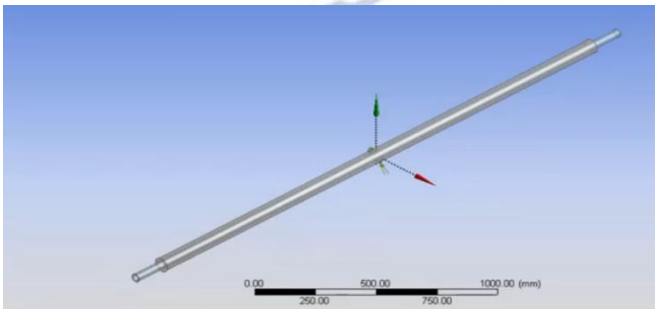


FIG 3.2 : GEOMETRIC MODELING OF HEAT EXCHANGER

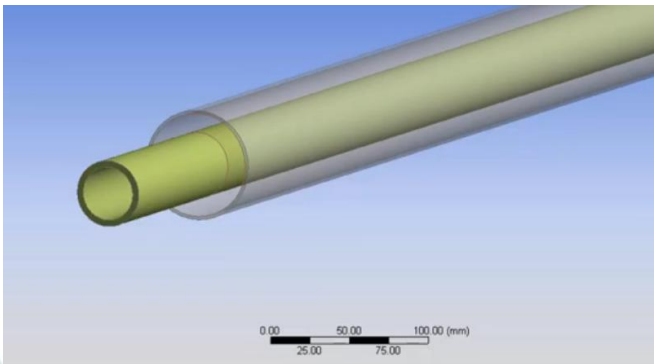


FIG 3.3: GEOMETRIC MODELING OF HEAT EXCHANGER CLOSE VIEW

DEFINING MATERIAL PROPERTIES:

Water was used as the base fluid flowing through tubing or piping. Its material properties were derived from tables based on the temperature which was being calculated in the model. The material was defined in FLUENT using its material browser. For the different flow arrangement problem model certain properties were defined by the user prior to computing the model, these properties were: thermal conductivity, density, heat capacity at constant pressure, ratio of specific heats, and dynamic viscosity. For the modified Graetz problem with pipe wall conduction as well as for the heat exchanger models the material library properties in FLUENT were used.

TABLE 3.1: FLUID PROPERTIES

Different fluids properties	Density (ρ) kg/m3	Thermal conductivity(K) W/mk	Specific heat CP j/kgK	Dynamic viscosity (μ) kg/m
Water	998.2	0.6	4182	0.001003
Transformer oil	826	0.134	2328	0.009

TABLE 3.2: MATERIAL PROPERTIES

Different material properties	Density (ρ) kg/m3	Thermal conductivity(K) W/mk	Specific heat CP j/kgK
Copper	8978	387.6	381
Galvanized iron	7850	520.1	470

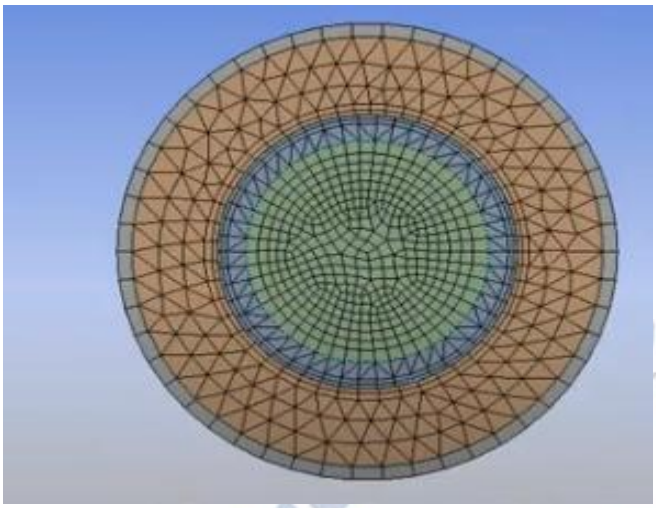


FIG 3.4: MESHED MODEL OF HEAT EXCHANGER

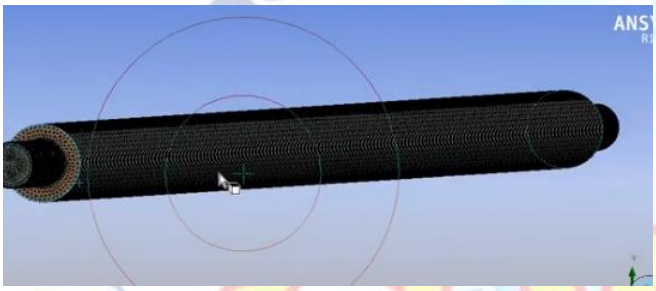


FIG 3.5: .MESH STRUCTURE OF COMPUTATIONAL DOMAIN

FINITE VOLUME METHODS

The mass, momentum, and scalar transport equations are integrated over all the fluid elements in a computational domain using CFD. The finite volume method is a particular finite differencing numerical technique, and is the most common method for calculating flow in CFD codes. This section describes the basic procedures involved in finite volume calculations.

The finite volume method involves first creating a system of algebraic equations through the process of discretising the governing equations for mass, momentum, and scalar transport. To account for flow fluctuations due to turbulence in this project, the RANS equations are discretised instead when the cases are run using the k-epsilon turbulence model. When the equations have been discretised using the appropriate differencing scheme for expressing the differential expressions in the integral equation (i.e. central, upwind, hybrid, or power-law, or other higherorder differencing schemes), the resulting algebraic equations are solved at each node of each cell.

NUMERICAL PROCEDURE AND COMPUTATIONAL METHODOLOGY:

The governing differential transport equations were converted to algebraic equations before being solved numerically. After the specification of the boundary condition, the solution control and the initialization of the solution have to be given before the iteration starts. The solution controls like the pressure velocity coupling and the discrimination of the different variables and the relaxation factors have to be specified.

The solutions sequential algorithm (called the segregated solver) used in the numerical computation requires less memory that the coupled solver. Since we are using the segregated solver for our problem, the default under relaxation factors are used and the SIMPLE scheme for the pressure velocity coupling is used and the second discrimination is used for the momentum and the standard scheme is used for the pressure

RESULTS AND DISCUSSIONS

4.1 EXPERIMENTALVALUES OBTAINED FOR WATER FROM PRACTICAL HEAT EXCHANGER:

For parallel flow:

Hot water flow rate :50CC

Cold water flow rate: 40cc

Inlet temperature Thi = 490 C

Outlet temperature Tho = 420C

Inlet temperature Tci = 250 C

Outlet temperature Tco = 300C

For counter flow:

Hot water flow rate :50CC

Cold water flow rate: 40cc

Inlet temperature Thi = 490C

Outlet temperature Tho = 410 C

Inlet temperature Tci = 250 C

Outlet temperature Tco = 30.50 C

4.2 EXPERIMENTALVALUES OBTAINED FOR TRANSFORMER OIL FROM PRACTICAL HEAT EXCHANGER :

For parallel flow:

Hot Transformer oil flow rate :50CC

Cold water flow rate: 40cc

Inlet temperature Thi = 490 C

Outlet temperature Tho = 450C
Inlet temperature Tci = 250 C
Outlet temperature Tco = 280C
For counter flow:
Hot transformer oil flow rate :50CC
Cold water flow rate: 40cc
Inlet temperature Thi = 490C
Outlet temperature Tho = 430 C
Inlet temperature Tci = 250 C
Outlet temperature Tco = 28.50C.

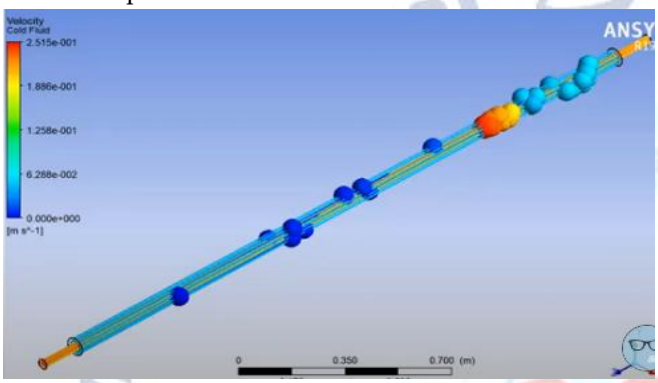


FIG 4.1: VELOCITY DISTRIBUTION OVER PIPES FOR WATER –WATER IN TUBES

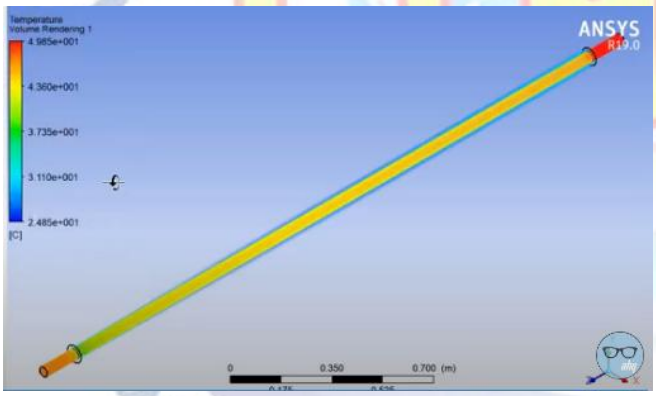


FIG 4.2: TEMPERATURE DISTRIBUTION FOR PARALLEL FLOW HEAT EXCHANGER

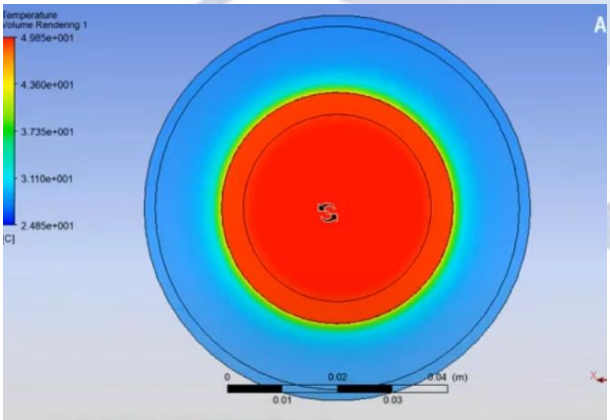


FIG 4.3: TEMPERATURE DISTRIBUTION FOR PARALLEL FLOW HEAT EXCHANGER

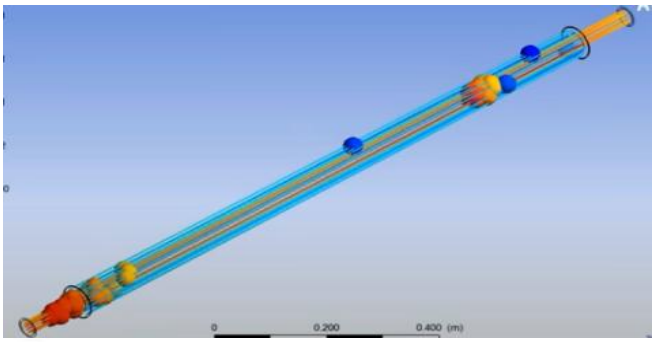


FIG 4.4: VELOCITY DISTRIBUTION OVER PIPES FOR OIL AND WATER IN TUBES

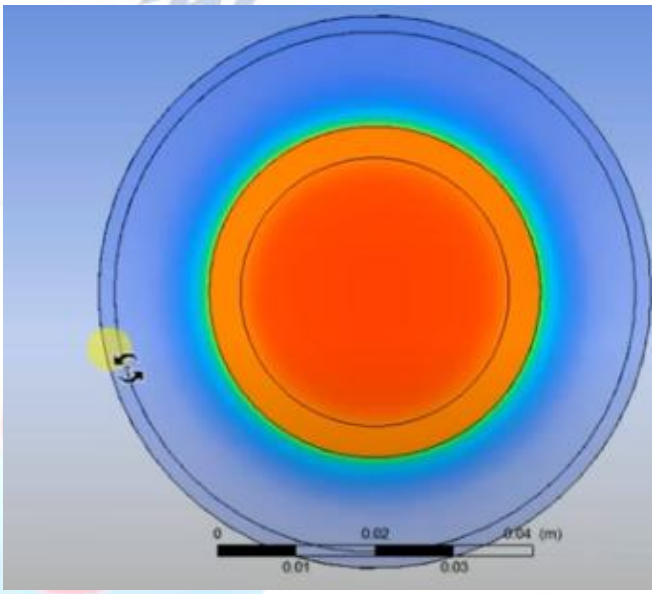
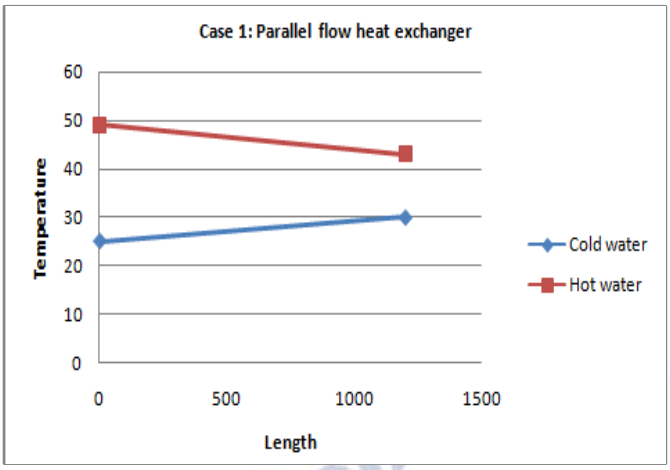


FIG 4.5: TEMPERATURE DISTRIBUTION FOR PARALLEL FLOW HEAT EXCHANGER

4.3 RESULTS FOR PRACTICAL ANALYSIS ON HEAT EXCHANGER

TABLE 4.1 : CASE 1- PARALLEL FLOW HEAT EXCHANGER WATER - WATER

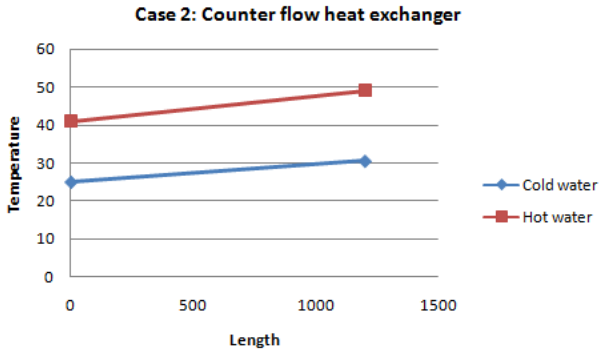
s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	30	5
Hot water	50	49	43	6



GRAPH 4.1: CASE 1- PARALLEL FLOW HEAT EXCHANGER WATER – WATER

TABLE 4.2: CASE 2 - COUNTER FLOW HEAT EXCHANGER WATER - WATER

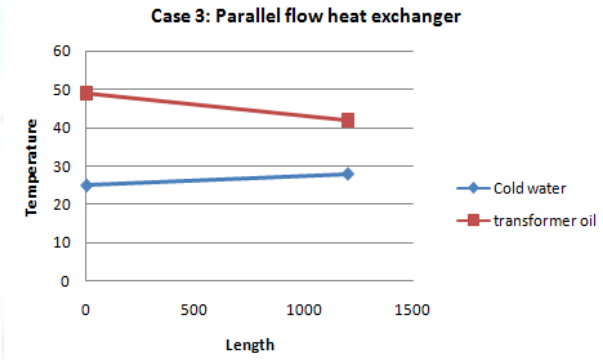
s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	30.5	5.5
Hot water	50	49	41	7



GRAPH 4.2: CASE 2- COUNTER FLOW HEAT EXCHANGER WATER - WATER

TABLE 4.3 : CASE 3- PARALLEL FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

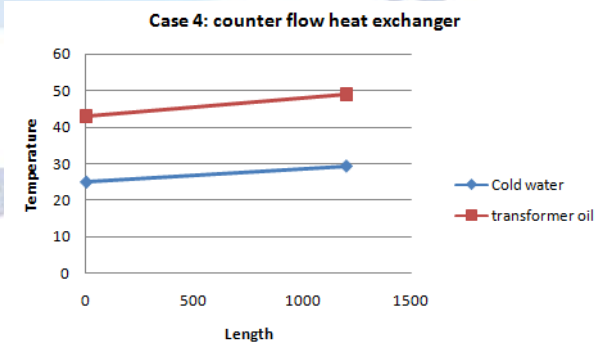
s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	28	3
Transformer oil	50	49	45	4



Graph 4.3 : Case 3- Parallel flow heat exchanger Transformer oil - water

TABLE 4.4: CASE 4 - COUNTER FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	29.5	4.5
Transformer oil	50	49	43	6

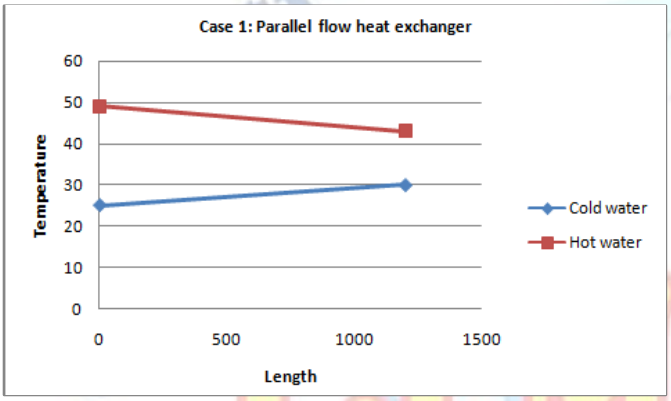


GRAPH 4.4: CASE 4 - COUNTER FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

Results for CFD analysis on heat exchanger

TABLE 4.5 : CASE 5 - PARALLEL FLOW HEAT EXCHANGER WATER - WATER

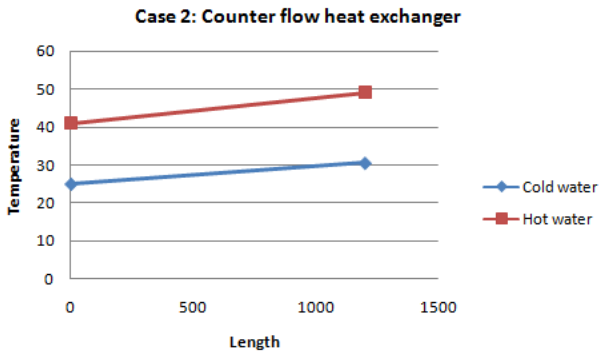
s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	30.5	5.5
Hot water	50	49	43	6



GRAPH 4.5: CASE 1- PARALLEL FLOW HEAT EXCHANGER WATER – WATER

TABLE 5\4.6: CASE 6 - COUNTER FLOW HEAT EXCHANGER WATER - WATER

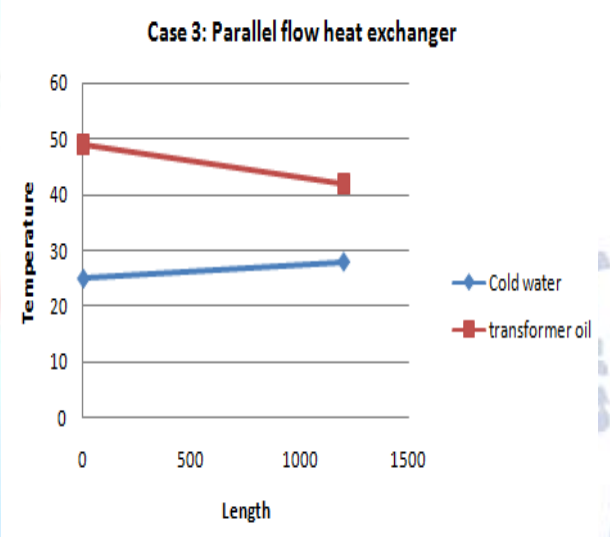
s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	31	6
Hot water	50	49	42	7



GRAPH 4.6: CASE 6- COUNTER FLOW HEAT EXCHANGER WATER - WATER

TABLE 4.7 : CASE 7- PARALLEL FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

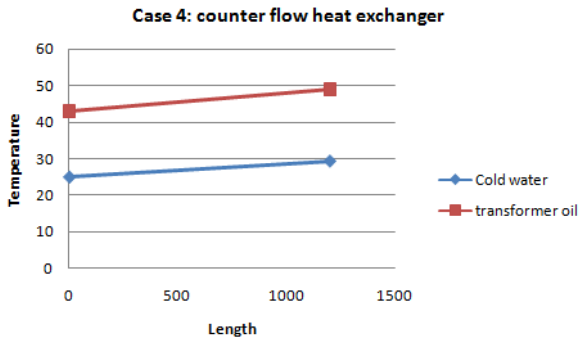
s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	28	3
Transformer oil	50	49	45	4



GRAPH 4.7 : CASE 7 - PARALLEL FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

TABLE 4.8: CASE 8 - COUNTER FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

s.no	Water flow rate in CC	Inlet temperature in 0C	Out let temperature in 0C	Temperature difference in 0C
Cold water	40	25	29	4
Transformer oil	50	49	43	5



GRAPH 4.8: CASE 8 - COUNTER FLOW HEAT EXCHANGER TRANSFORMER OIL - WATER

FUTURE SCOPE AND CONCLUSION

The performance, practical and CFD analysis of different fluids were investigated on parallel and counter flow in concentric tube heat exchanger.

In the present work the counter flow heat exchanger using CFD analysis has been investigated. The heat transfer and flow distribution of 3-D model counter flow heat exchanger is discussed in detail and compared with the previous literature in view of varying the inlet temperature and flow rate. The discretized 3-D model outcome are in agreement with the previous literature and hence are validated for proposed heat exchanger using fluent simulation tool of ANSYS. The present work observes the better heat transfer and comprehensive understanding of a heat exchanger.

The conclusions of the investigating at are as follows.

1. The main objective of this project was to analyses the fluid flow in double pipe heat exchangers and the subsequent performance of these heat exchangers.

2. This review is done on both practical and CFD analysis

3. Modeling the double pipe heat exchanger by using ANSYS WORKBENCH.

4. To simulate heat transfer in concentric tube heat exchanger by using CFD-Fluent software.

5. The ANSYS FLUENT results were found to be• fairly consistent with hard calculations with most of the values within 5% of each other

6. The results in practical and as well as in cfd are nearly equal

7. As the cold fluid temperature goes down, the steady state condition for heat transfer can be achieved at a faster rate

8. The pressure and temperature contours show the higher velocity and pressure magnitudes along the outer pipes as compared to the inner pipe.

Moreover new materials with better thermal properties for the construction of counter flow heat exchanger could lead to bright and improved changes of heat transfer. The present work can further be enhanced by considering Nusselt and Reynolds number stress models to improve the flow characteristics in the system and comparing the results with other computational methods.

REFERENCES

[1] Praveen Kumar Kanti, Karthika.U.P, Sabeer Ali, SanathKumar.N, ShyamChandran C"CFD analysis of shell and tube heatexchanger", ISSN: 2319- 6890)(online),2347-5013, VolumeNo.5Issue:Special 6, pp: 1129 -1254, 20 May 2016

[2] D.Bhanuchandrarao, M.Ashokchakravarthy, Dr. Y. Krishna, Dr. V .V. SubbaRao, T.HariKrishna "CFD Analysis And PerformanceOf Parallel And Counter Flow In Concentric Tube Heat Exchangers", ISSN: 2278-0181,Vol. 2 Issue 11, November – 2013

[3] C. S. Oon, H. Togun, S. N. Kazi, A. Badarudin and E. Sadeghinezhad, "Computational simulation of heat transfer to separation fluid flow in an annular passage", Int. Commun. Heat Mass, Vol. 46, pp. 92-96, (2013).

[4] S. Nagarsheth, U. Pandya and H. Nagarsheth, "Control Analysis Using Tuning Methods for a Designed, Developed and Modeled Cross Flow Water Tube Heat Exchanger", Int. J. Mech., Aero, Ind. Mechatron. Eng., World Academy of Science and Technology, Vol. 8, No. 12, pp. 1889- 1894, (2014).

[5] K.S.Hwang, S.K.Jang, S.U.S.Chio, Flow and convective heat transfer characteristics of water-based Al2O3 nanofluids in fully developed laminar flow regime, International Journal of Heat and Mass Transfer, 52 (2009) 193-199.

[6] V. Bianco, F. Chiacchio, O. Manca, S. Nardini, Numerical investigationof nanofluids forced convection in circular tubes, Applied Thermal Engineering, 29 (2009) 3632 – 3642.

[7] M. Akbari, N. Galanis, A. Behzadmehr, Comparative analysis of singleand two-phase models for CFD studies of nanofluid heat transfer, International Journal of Thermal Sciences, 50 (2011) 1343 – 1354