HEAT TRANSER AND EXERGY ANALYSIS OF A DOUBLE PIPE HEAT EXCHANGER

Ch. Praveen Kumar[1], L.Narsinga Rao[2], M.Raghu[3], Y.Pavan Kumar[4]

[1,2,3,4] M.Tech Dept. of Mechanical Engineering, V.R.Siddhartha Engineering College-520007, A.P., India.

Abstract-*Heat exchangers are the important engineering systems with wide variety of applications including power plants, nuclear reactors, refrigeration and air-conditioning systems, heat recovery systems, chemical processing and food industries. In the present study was to analyze the fluid flow in double pipe heat exchangers and the subsequent performance of these heat exchangers. To facilitate this analysis, the FLUENT 14.5 was used to perform the modeling and calculations. In order to verify the development of each model, the models were built in stages and each stage analyzed and verified. The first stage was a theoretical validation of problem for given specifications and determine the outlet temperatures of hot and cold fluid in parallel and counter flow using NTU method. In the second stage heat exchanger modeling and analysis was carried out. The results show that, good agreement with theoretical values by varying mesh quality and size. After preparing the method in Ansys fluent, the outlet temperatures for different materials and different fluids were determined. Using this outlet temperatures compactness of heat exchanger, entropy generation, exergy loss in a heat exchanger was investigated.*

Key Words: Heat exchanger, Compactness, Irreversibility, Exergy, CFD, Fluent etc.

1. INTRODUCTION

This paper discussed about heat transfer in engineering, definition of heat exchanger, classification of heat exchangers, type of heat exchanger, construction of double pipe heat exchanger, overall heat transfer coefficient of double pipe heat exchanger, log mean temperature difference (LMTD) method for double pipe heat exchanger, effectiveness-NTU method for double pipe heat exchanger, applications of heat exchanger.

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer (not for analysis of chemical reactions). Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.

2. LITERATURE SURVEY

Naphon (2006) studied ,the theoretical and experimental results of the second law analysis on the heat transfer and flow of a horizantal concentric tube heat exchanger are presented. Hot water and cold water are used as working fluids. The test runs are done at the hot and cold water mass flow rates ranging between 0.02 and 0.20 kg/s and between 0.02 and 0.20 kg/s , respectively.The predicted results obtained from the model are validated by comparing with the present measured data.

Gupta and Atrey (2000) studied , counter flow heat exchangers are commonly used in cryogenic systems because of their high effectiveness.In addition to operating and design parameters, the thermal performance of these heat exchangers is strongly governed by various losses such as longitudinal conduction through wall , heat in leak from surrounding, flow maldistribution,etc. In this the numeriocal model is developed earlier is extended to take in to consideration the effect of heat in leak and the predictions are compared with the experimental results.

2.1 Objectives

- Study on heat transfer for heat exchanger specific to double pipe heat exchanger types.
- Design the double pipe heat exchanger by using ANSYS workbench.
- Simulation in double pipe heat exchanger by using FLUENT software.
- Analysis the heat exchanger specific to flow rate of hot and cold fluid.
- To simulate heat transfer in concentric tube heat exchanger by using CFD-Fluent software.
- To analyse 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.

Table1: Material Properties

Material	Density(kg/m3)	Thermal conductivity(w/mk)	Specific heat(J/kgK)
Copper	8978	387.6	381
steel	8030	16.27	502.48

3. Methodology

3.1 Finite volume analysis modelling

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.

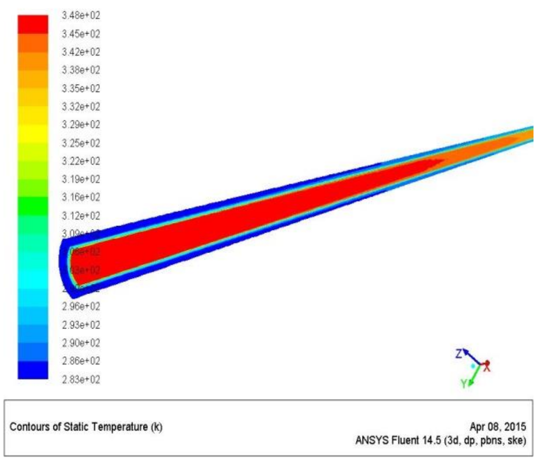
The finite volume method involves first creating a system of algebraic equations through the process of discretizing the governing equations for mass, momentum, and scalar transport. To account for flow fluctuations due to turbulence in this project, the RANS equations are discretised instead when the 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 higher-order differencing schemes), the resulting algebraic equations are solved at each node of each cell.

4. Results and Discussion

Table 4.1: Fluid properties

Different fluids properties	Density (ρ) kg/m^3	P_r	Thermal conductivity(K) W/mk	Specific heat C_P j/kgK	Dynamic viscosity (μ) kg/m-s
Benzene	875	6.51	0.159	1759	0.00058
Glycerin	1259.9	6780.3	0.286	2427	0.799
Water	998.2	6.99	0.6	4182	0.001003

4.1 Flow in a parallel flow heat exchanger- copper-water



$m_h < m_c$

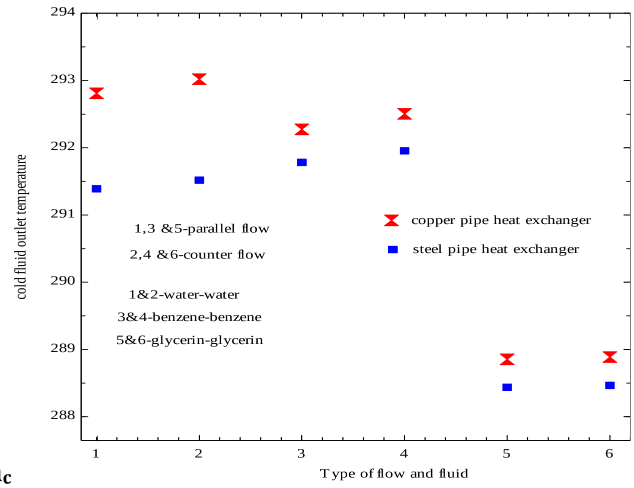


Fig4.1: Turbulent temperature profile

Fig4.3: Cold fluid outlet temperature for PF and CF

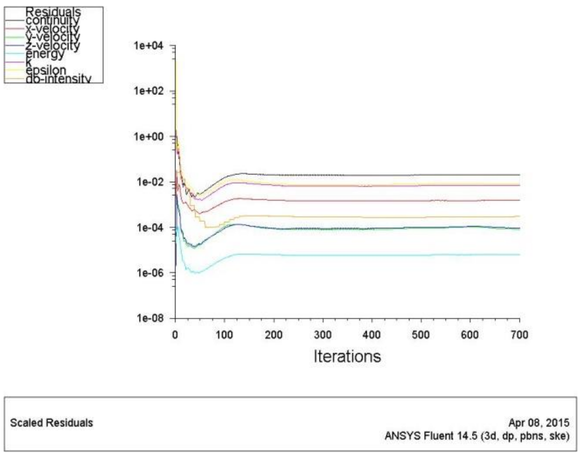


Fig4.2: Scaled Residual

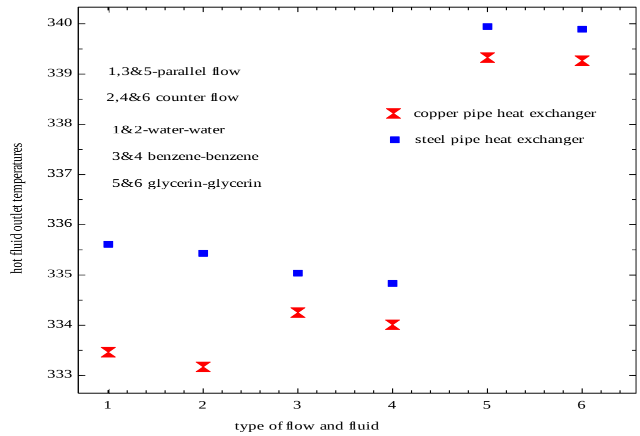


Fig4.4: Cold fluid outlet temperature for PF and CF

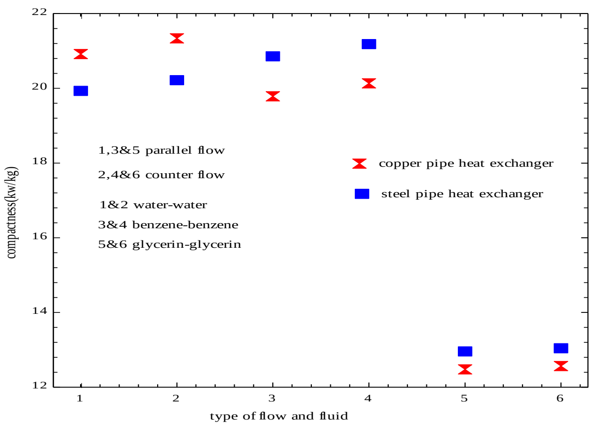


Fig4.5: Compactness for PF and CF

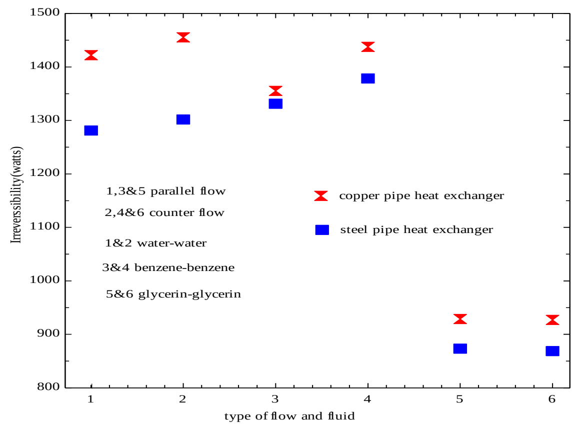


Fig4.6: Irreversibility in PF and CF

m_h>m_c

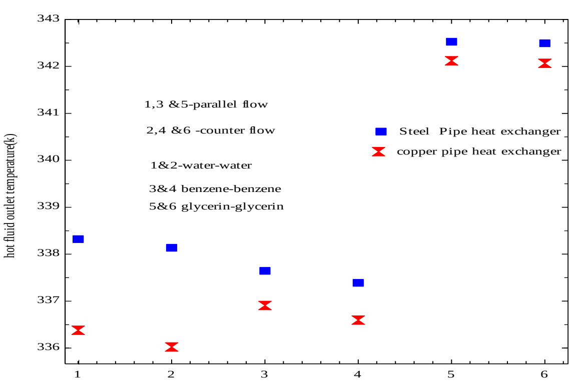


Fig4.7: Hot fluid outlet temperature for PF&CF

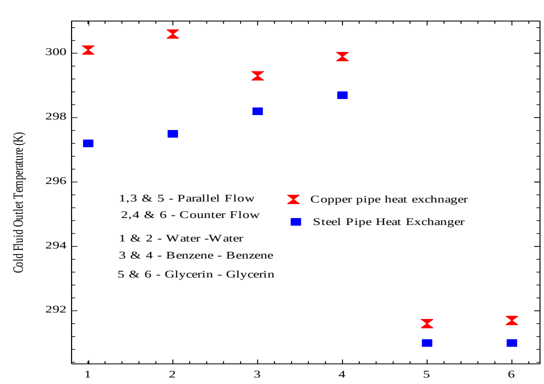


Fig4.8: Cold fluid outlet temperature for PF&CF

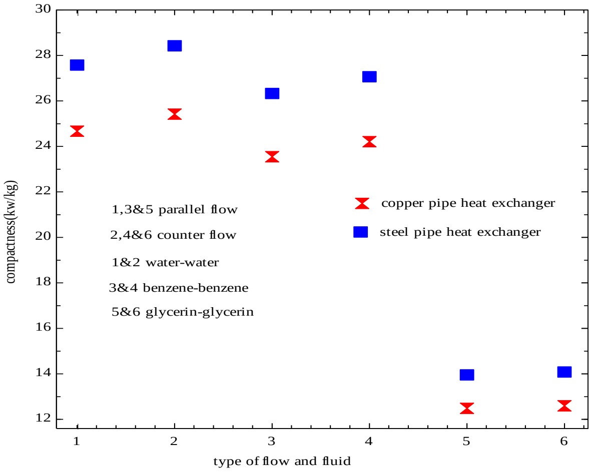


Fig4.9: Compactness for PF&CF

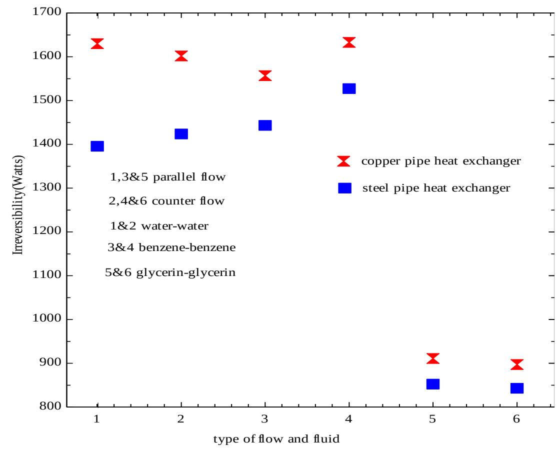


Fig4.10: Irreversibility of PF&CF

mh=mc

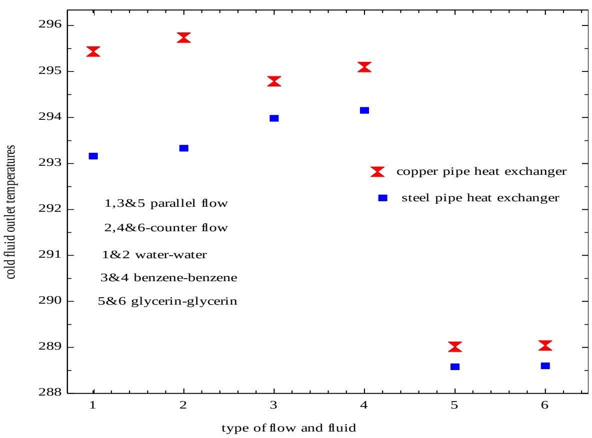


Fig4.11: Cold fluid outlet temps for PF & CF

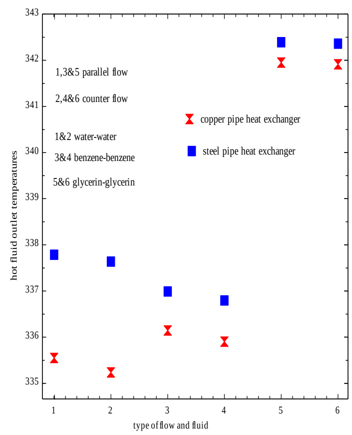


Fig4.12: Hot fluid outlet temps for PF & CF

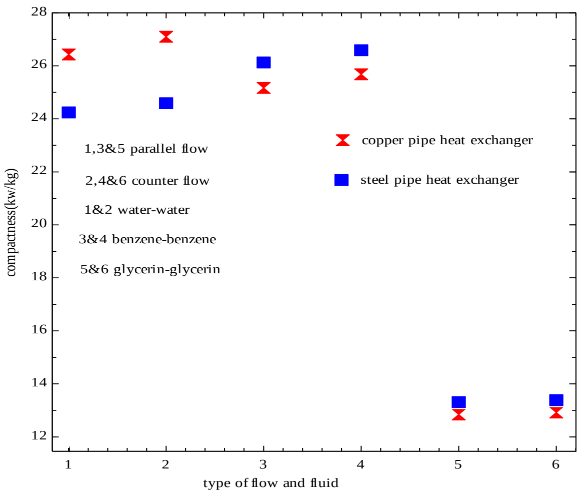


Fig4.13: Compactness for PF & CF

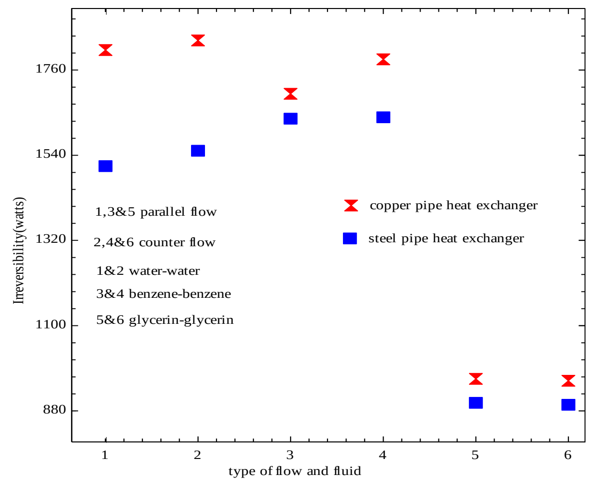


Fig4.14: Irreversibility of PF & CF

5.CONCLUSIONS

From fig 4.3& fig 4.4 conclude that achieving the lowest values of hot fluid temperatures in all three fluids compared to other two cases. From fig 4.5 in terms of compactness graph conclude that less compactness compared to other two cases. From fig 4.6 in terms of irreversibility graph conclude that exergy losses are less in water and benzene compared to all three cases. In case of glycerin exergy losses are more compared to second case and less compared to first case.

From fig 4.7& fig 4.8 conclude that achieving the highest values of cold fluid temperatures in all three fluids compared to other two cases copper, steel as a pipe material. From fig 4.9 in terms of compactness graph conclude Design of heat exchanger based on this flow rate case having less compactness compared to third case and more compact compared to first case From fig 4.10 in terms of irreversibility graph conclude exergy losses are more in water and benzene compared to first case, exergy losses less compared to third case in all three fluid.

From fig 4.11& fig 4.12 conclude that achieving the highest values of cold fluid temperatures compared to first case and lowest values compared to second case copper, steel as a pipe material. From fig 4.13 in terms of compactness graph conclude design of heat exchanger based on this flow rate case having more compactness compared to all three cases. From fig 4.14 in terms of irreversibility graph conclude that exergy losses are more compared to all the three cases in all fluids and both parallel and counter flow type.

REFERENCES

[1] Naphon P, (2006), Second law analysis on the heat transfer of the horizontal concentric tube heat exchanger, International Communications in heat and mass transfer, Vol.33, pp 1029-1041.
[2] Gupta P, and Atery M.D., (2000), Performance evaluation of counter flow heat exchangers considering the effect of heat in leak and longitudinal conduction for low temperature applications, Cryogenics, Vol.40, pp. 469-474.
[3] Gupta P.K., Kush P.K. and Tiwari A., (2007), Design and optimization of coil finned-tube heat exchangers for cryogenic applications, Cryogenics, Vol.47, pp. 322-332.

[4] Gupta P.K., Kush P.K. and Tiwari A., (2007), Second law analysis of counter flow cryogenic heat exchangers in the presence of ambient heat in leak and longitudinal conduction through wall, International Journal of heat and mass transfer, Vol.50, pp. 4754-4766.
[5] Gheorghian A, and Dobrovicescu A., (2014), Analysis of plate fin surface types performance based on the second law of thermodynamics. A case study, U.P.B.Sci.Bull., Series D, Vol.76 (1).
[6] Patel H. S., Makadia R. N.,(2013), A Review on Performance Evaluation and CFD Analysis of Double Pipe Heat Exchanger, international journal research Vol.2, (4)
[7] Fundamentals of engineering heat and mass transfer, R.C.Sachdeva, 4th Edition ,New Delhi, New Age International Publications, 2010.
[8] Heat and mass transfer data book, C.P Kothandaraman, S.Subramanyan, 8th Edition, New Delhi, New Age International Publications, 2014.
[9] Transport Phenomena, Bird Byron R, Warren E. Stewart, and Edwin N. Lightfoot. 2nd Edition, New York, John Wiley & Sons, Inc.2007.
[10] Engineering heat transfer, William S. Janna, 2nd Edition, CRC Press, 1999.

BIOGRAPHIES

	Ch. Praveen Kumar, studying M.Tech in V.R.Siddhartha Engineering College, Vijayawada. Research area of interests is Computational fluid dynamics and Thermodynamics.
	L.Narsinga Rao, studying M.Tech in V.R.Siddhartha Engineering College, Vijayawada. Research areas of interests are Computational fluid dynamics, IC Engines and Thermodynamics.
	M.Raghu, studying M.Tech in V.R.Siddhartha Engineering College, Vijayawada. Research areas of interests are Computational fluid dynamics and Thermodynamics.
	Y.Pavan Kumar, studying M.Tech in V.R.Siddhartha Engineering College, Vijayawada. Research areas of interests are Computational fluid dynamics, Heat transfer and Thermodynamics