INTERNATIONAL RESEARCH JOURNAL OF AUTOMOTIVE TECHNOLOGY (IRJAT)

http://www.mapletreejournals.com/index.php/IRJAT
Received 21 June 2018 ISSN 2581-5865
Accepted 12 July 2018 2018; 1(4);1-5
Published online 30 July 2018

Design and analysis of air cooled radiator

H. Bahuruteen Ali Ahamadu[1]*, M.D Raj kamal[1], S. Kaliappan[2]

[1]Asst Professor, Dept of Mech Engg, Velammal Institute of Technology Chennai,Tamilnadu, India.

[2]Asso Professor, Dept of Mech Engg, Velammal Institute of Technology Chennai,Tamilnadu, India.

**Corresponding author E-Mail ID: kamalaerodynamics@gmail.com, Mobile: 9092672367*

ABSTRACT

In this work discussed about the problems associated with treatment and disposal of water have become more costly with government regulations and environmental concerns. The objective of the work is to increase the performance of thermic fluid heat exchanger by utilizing the single thermic fluid instead of double fluid presently used in the process industries. Thermic fluid offers outstanding high temperature performance including thermal stability and low vapor pressure. The project design comprises the idea of implementing two 3 way control valves to by – pass the inlet of thermic fluid to the air cooled radiator when the temperature of thermic fluid has increased after cycling through the process. The fluid flow simulation is conducted using commercial software, fluent. The pressure and temperature distribution along the tube length and tube width are presented and analyzed.

Keywords: Heat Exchanger, Fluent, Thermic fluid

1. INTRODUCTION

The Air-cooled heat exchanger is a device for rejecting heat from a fluid or gas directly to ambient air. When cooling both fluids and gases, there are two sources readily available, with a relatively low cost, to transfer heat to air and water. The obvious advantage of an air cooler is that it does not require water, which means that equipment requiring cooling need not be near a supply of cooling water. In addition, the problems associated with treatment and disposal of water have become more costly with government regulations and environmental concerns. The air-cooled heat exchanger provides a means of transferring the heat from the fluid or gas into ambient air, without environmental concerns, or without great ongoing cost. An air-cooled heat exchanger can be as small as your car radiator or large enough to cover several acres of land, as is the case on air coolers for large power plants where water is not available.

2. MODELLING AND ANALYSIS

A CFD analysis of fluid flow and heat transfer is carried out in Carbon Steel radiator, in which FLUENT's pre processing (ICMCFD), Post Processing (CFD) with incompressible heat transfer, is used as the tool. In this study, shell side airflow pattern and tube side Therminol oil 66 flow pattern are studied and presented the variation of overall heat transfer coefficients across the radiator of 523.014 W/m2-K. This study established the capability of FLUENT code to handle such problems. I made an experimental investigation of the heat transfer characteristics of a tube-and-fin radiator for Process and developed the regression equations of heat dissipation rate, therminol oil 66 pressure drop and temperature. The influences of the air velocity, inlet therminol oil 66 temperatures and volume flow rate of therminol oil 66 on heat dissipation rate, therminol oil 66 pressure drops have been discussed in detail by means of the numerical analyses.

2.1 Modelling

The Numerical experimentation of the work carried out involved reverse engineering of a Process Industry radiator for the required fluid domain, discretising the fluid domain, simulation of the fluid flow and heat transfer at steady state and post processing the results and drawing suitable conclusions. The details of the geometry of the radiator were obtained by the process of reverse engineering. The dimensions of individual components of the radiator were measured using suitable measuring instruments and given in Table 1. The measurements obtained were used to generate the CAD model in CATIA V5 R9 and shown in Figure 1.

Table 1: Specifications of the radiator

Radiator type	Cross Flow
Tube size	19.45 mm
Core rows	6
Core columns	20
Core dimensions (L x W x H)	1.5 m x 1.2 m x 0.4 m
Tube and fin material	Carbon steel
No. of tubes	120

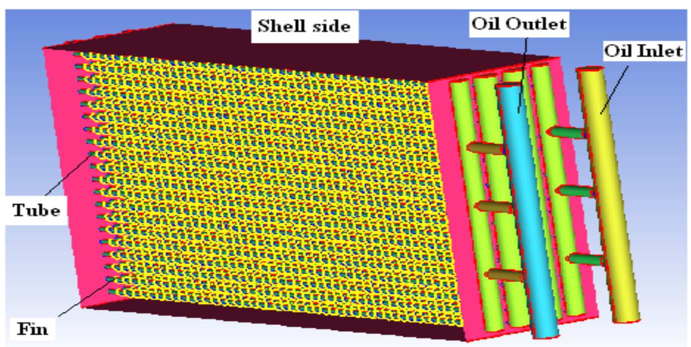


Fig 1. Assembled CATIA model of the radiator

2.2 Analysis

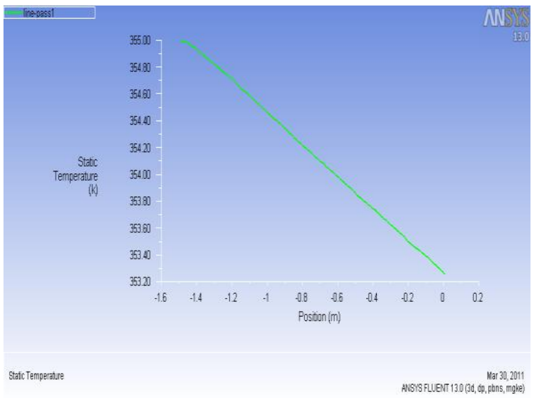
The CFD analysis is carried out using fluent software. The analysis in carried out for every pass separately. The output results obtained from one pass is used as an input for the subsequent pass. The temperatures contours, variation and pressure contours are obtained for every tube pass in the radiator.

3. RESULTS AND DISCUSSION

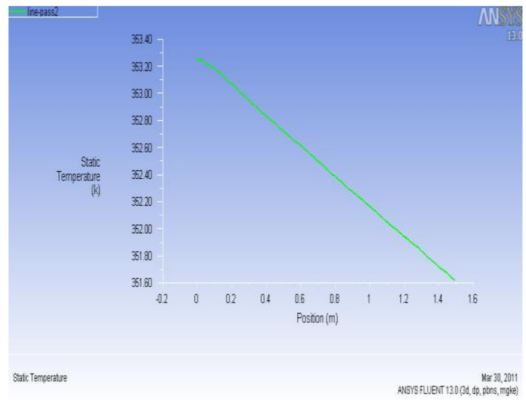
The results obtained from the simulation analysis are presented in Figures 6.1 to 6.3. Figure.6.1 shows the temperature contours of the coolant along the direction of flow for various tube passes. A drop in temperature of the coolant from 82 ^{o} C to 72 ^{o} C is observed in these plots. In every pass, approximately a 2°C drop is observed. The analytical calculation made for the heat exchanger analysis assuming a temperature drop of 10°C shows a surface area requirement of 10.25 m2.

In the CFD analysis considering this surfaces area with the same configuration assumed in the analytical calculation, approximately a similar temperature drop is observed which shows the correctness and capability of the CFD analysis. Hence this model and analysis procedure adopted can be extended for simulation and optimization analysis. The temperature drop obtained in every pass is also shown separately in Fig. 2.

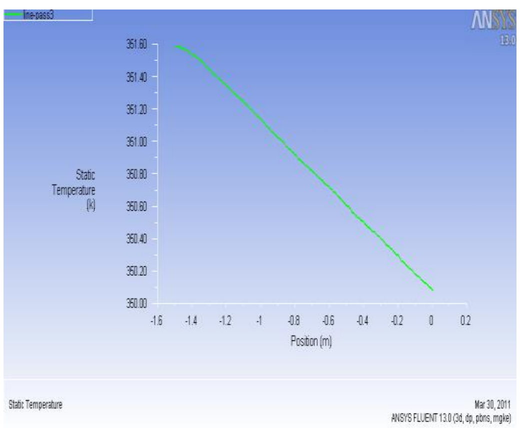
Fig. 3. Shows the pressure contour of the HTF region of analysis. The pressure shows a steady variation across the length of the tube. This trend is expected results in any radiator performance. A total pressure drop of 0.827 kg / cm2 in observed from the analysis.



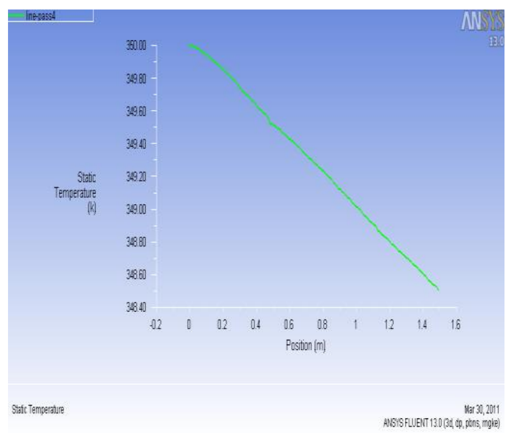
1st Row Tube pass



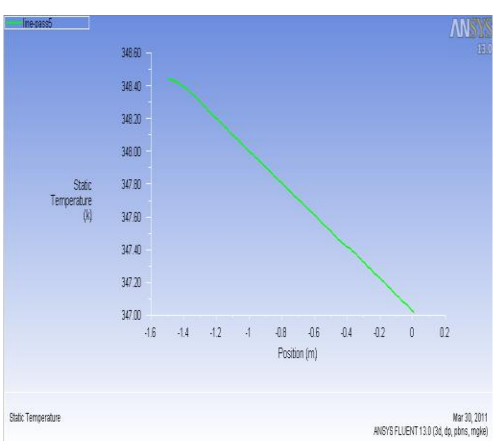
2nd Row Tube pass



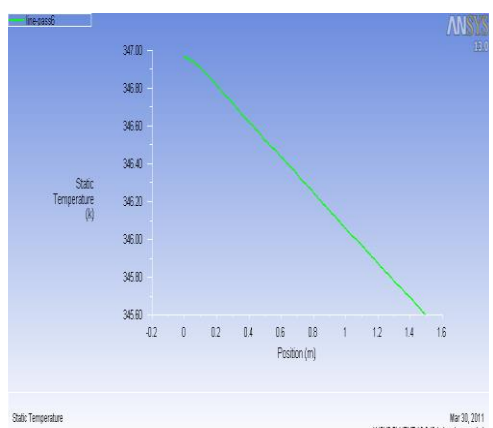
3rd Row Tube pass



4th Row Tube pass



5th Row Tube pass



6th Row Tube pass

Fig 2. Variation of oil temperature (K) across the tube length

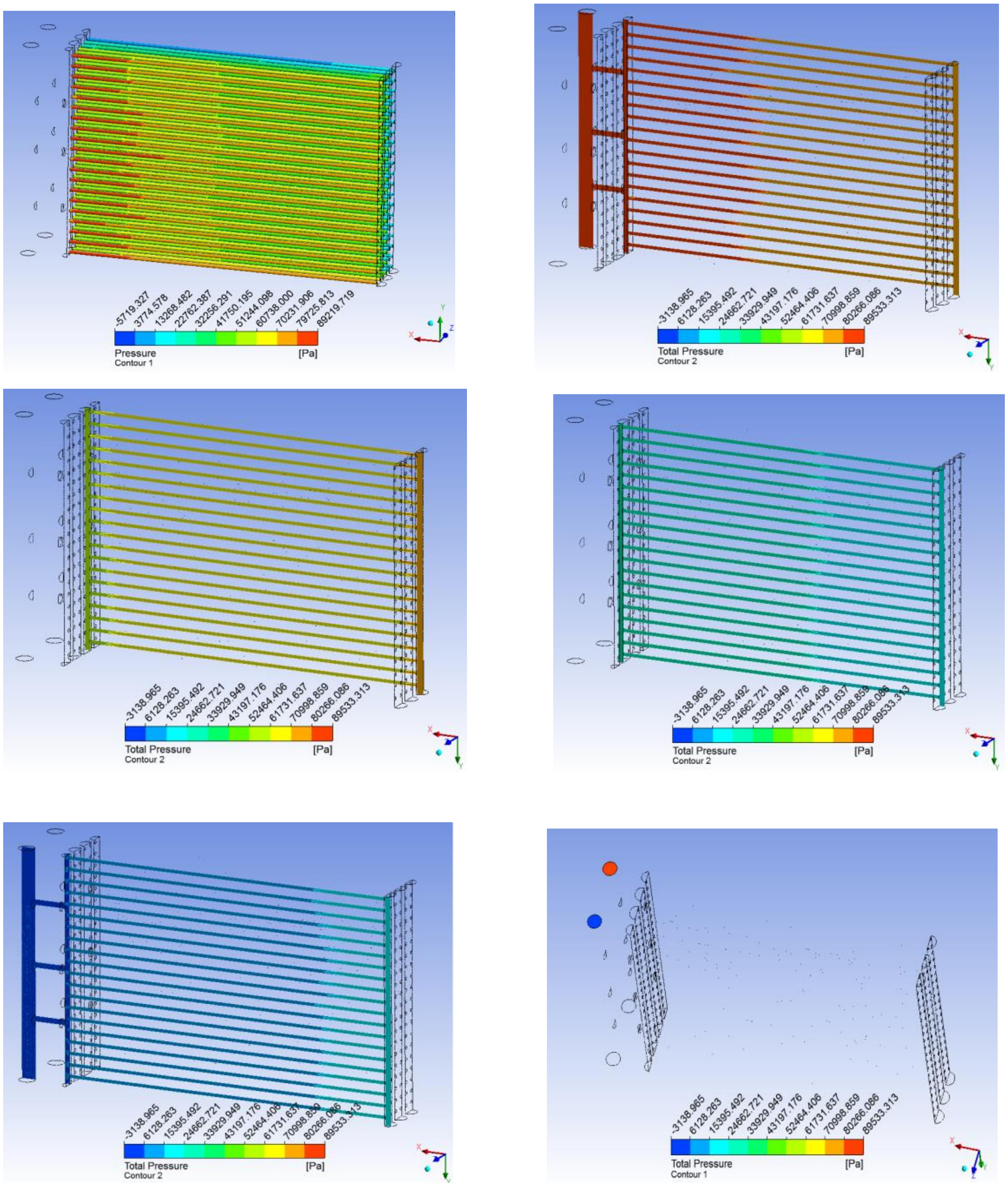


Fig .3 Pressure contours of oil along the tube

4. CONCLUSION

The fluid flow and heat transfer analysis of a multi tube-fin arrangement of the process radiator is successfully carried out using numerical simulation built in commercial software FLUENT. The variations in the pressure, temperature of the heat transfer fluid along the direction of flow and presented and analysed. It is observed that the temperature of Therminol oil drops by 10^0C and the HTF experience a pressure drop of 0.872kg/cm^2 which is in good agreement with the theoretical results. In the present study the fluent software capability is studied for the analysis of a radiator. The software is able to provide accurate results and hence it is construed that the analysis can be extended for optimising the geometrical and flow parameters.

REFERENCES

1. H. Bahuruteen Ali Ahamadu M.E., Daniel S.G, Rajasekaran B, Selvaganesh M, Sujin Prasad J ,Performance Analysis of Single Shell Four Tube Heat Exchanger, International Journal of Pure and Applied Mathematics Volume 119 No. 12 2018, 13729-13734
2. M.D.Rajkamal ,M. Mani Bharathi, Shams Hari Prasad M, Santhosh Sivan.M, Karthikeyan.S r.H.Bahruteen Ali Ahamadu S.KALIAPPAN Dr.T.Mothilal Thermal analysis of shell and tube heat Exchanger International Journal of Pure and Applied Mathematics Volume 119 No. 12 2018, 14299-14306
3. Comparative CFD Analysis of Shell and Serpentine Tube Heat Exchanger by Subin Michael , International Research Journal of Engineering and Technology (IRJET),e-ISSN: 2395 -0056 ,Volume: 04 Issue: 02, Feb -2017.
4. CFD modelling development and experimentalvalidation of a phase change material (PCM) heatexchanger with spiral-wired tubes by W. Youssef, Y.T,Elsevier,2018.
5. Analysis study of shell and tube heat exchanger for clough company with reselect diffrent parameters to improve the design by Ammar Ali Abd,Case Studies in Thermal Engineering, ELSEVIER.
6. CFD of helical coil tube in tube heat exchanger by Vijaya Kumara Reddy, ICMPC 2016.
7. Simulation and CFD Analysis of heat pipe heatexchanger using Fluent to increase of the thermalefficiency by M. H. SABER, H. MAZAHERASHTIANI
8. Thermal analysis of double-pipe heat exchanger inthermodynamic vent system by Zhan Liu , Energy Conservation Management ,ELSEVIER.Forced Convective Heat Transfer of Aqueous Al2O3 Nanofluid
9. Investigation of Jet Noise Reduction Using Fan Flow Deflectors on CFD by Rajkamal M.D, S.Kalippan in International Journal of Applied Engineering, ISSN 0973-4562 Vol. 10 No.33 (2015).
10. Analysis and Optimization of Pressure Reducing and Desuperheating System in Thermal Power Plant in ‘International Journal of Latest Trends in Engineering and Technology by U.Aaditya. Raikamal.M.D
11. Performance Analysis of Gasifier on CFD’, by Raikamal.M.D in International Journal of Applied Engineering Research ,ISSN 0973-4562 Vol. 10 No.33 (2015)
12. Aerodynamic Effects on Formula One Car Using CFD in ‘International Journal of Applied EngineeringResearch. International Journal of Applied Engineering Research ,ISSN 0973-4562 Vol. 10 No.33 (2015)