International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 07 | July -2017 www.irjet.net p-ISSN: 2395-0072

Design and Thermal Analysis of Hydraulic Oil Cooler by Using Computational Fluid Dynamics (CFD)

Ankush S. Patil[1], H. S. Farkade[2]

[1] P.G. Student, Dept. of Mechanical Engineering, Govt. College of Engineering, Amravati, Maharashtra, India.
[2] Assistant Professor, Dept. of Mechanical Engineering, Govt. College of Engineering, Amravati, Maharashtra, India.

---***---

Abstract - *The Hydraulic oil cooler come equipped with the cross flow heat exchanger cores that have a shroud with fans and other brackets and braces to secure the components into the reservoir. In application, heat to be rejected from the oil is flowing into the "core" is placed in the overheating reservoir which insures that the returning oil.. The working fluid in the cross flow heat exchanger rejects its heat to the surrounding forced air respond reduction in the temperatures of oil. This may avoid the fall in viscosity of oil and the problem of overheating of oil. This is passive and efficient heat removal. As long as there is a temperature difference between the oil and the ambient air, the cross flow heat exchanger will remove heat from the oil, thus cooling it down. It is the process that provides the mechanism by which heat is "removed" from one medium to another. In the case of Hydraulic oil coolers – heat is removed from oil into the ambient air.*

Key Words: CFD, Parallel flow, Counter flow, LMTD, Heat transfer, Effectiveness.

1. INTRODUCTION

Heat exchangers are those devices which are used to transfer heat from hot fluid to cold fluid which are of either same or different phases. Heat exchangers are used in wide range for different types of industrial and domestic applications. Some of the heat exchangers are mixing type and some are non-mixing type. The difference between the mixing and non-mixing is that in non-mixing type the fluids are separated by metal wall. In non-mixing type as there is used a metal wall to separate the two different fluids, the heat transfer takes place by convection in each fluid and by conduction through the walls, so that in the analysis of heat exchanger, it is necessary to calculate overall heat transfer coefficient U. Heat load depends on heat rejection required to keep engine surface at optimum temperature. Generally LMTD or ε-NTU method is used to do heat transfer calculations of radiator. Both methods have its own advantages and preferred according to data availability. When radiator inlet and outlet temperature are known LMTD gives faster solution. When any of the temperature is unknown LMTD method undergoes iterations to find solution. In this case ε-NTU is better. In this paper ε-NTU method is described to do heat transfer calculations.

Nomenclature

Re	Reynolds number	h	Convective heat transfer coefficient (W/m2K)
Nu	Nusselt number	K	Thermal conductivity of air (W/mK)
Q	Heat transfer rate	To_1	Inlet oil temperature
Qoil	Total oil volumetric flow rate	To_2	Outlet oil temperature
Pr	Prandtl number	Ta_1	Inlet air temperature
f	Friction factor	Ta_2	Outlet air temperature
fo	Friction factor for plain tube	UA	Overall heat transfer coefficient
η	Thermal performance factor	ε	Effectiveness
m	Mass flow rate of fluid (Kg/s)	Cair	Total air heat capacity
D	Hydraulic diameter (m)	Coil	Total oil heat capacity
W	Width of Helical Tape inserts (m)		
A	Surface area of test section (m2)		
ηfin	Fin efficiency		
ηo	Overall surface efficiency	Greek symbols	
q	Heat flux (W/m2)	ρ	density of fluid, kg/m3
Nfin	Number of fins per tube	μ	viscosity of fluid, N s/m2

2. EXPERIMENTAL SETUP

We proposed to carry out on experimental set-up as shown in the figure 3.5. The experiment setup will be a close-loop experimental facility as shown in figure 3.5. The loop consisted of a hydraulic press bell jinning machine system, accumulator, and heat exchanger along with cooling fan. After treatment of SAE60oil in hydraulic press bell jinning machine it will suddenly get heated up to 60°C. Then oil is recirculating throughout the close loop cycle. The recirculating hot oil at 60°C is stored at accumulator then it is flow toward the inlet of heat exchanger (radiator). After treatment of hot oil by the application of cooling fan whose temperature is at 40°C the hot oil temperature reduces up to 45°C. The heat from the hot oil is gained by the cold fluid air thus the temperature of oil gets reduced. After treatment the oil at 45°C is collected at the oil tank having capacity of 1400 litre. The oil at ambient temperature from the oil tank is further ready for the operation in hydraulic press bell jinning machine.:

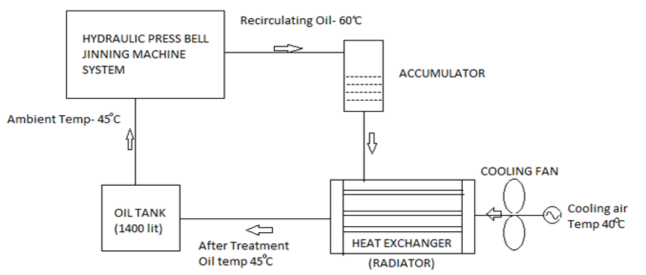

Fig -1: Experimental Setup

2.1 Procedure

The engine ran without the radiator fan until it reached 200 degrees Fahrenheit. Once the engine reached this temperature, the fan on the radiator was turned on to begin cooling the engine. The radiator, along with the fan, then cooled the engine until the temperatures reached steady state, which took approximately six minutes. The inlet and outlet temperatures and the flow rates for air and water were then recorded. This experimental process was repeated four times in order to give a range of results to compare with one another. These experimental results were then compared with the inlet and outlet temperatures found in the theoretical analysis of the problem.

3. SYSTEM DEVELOPMENT

3.1 Designing of Heat Exchanger

Energy equation

We know that total amount of heat is equal to either amount of heat rejected by hot fluid or it is equal to amount of heat absorbed by cold fluid. Mathematically it is written as,

$$q = m_{air} c_{pair} (T_{air,out} - T_{air,in}) = m_{oil} c_{poil} (T_{oil,in} - T_{oil,out})$$

By using above equation, we have

$$q = m_{oil} c_{poil} (T_{oil,in} - T_{oil,out})$$

= 0.13*2047*(65-45)

= 3.410 kw

Now, total amount of heat is equal to heat absorbed by cold medium air. Also consider mass flow rate of hot oil is (m= 0.225 kg/s)

$$q = m_{oil} c_{poil} (T_{oil,in} - T_{oil,out})$$

3410 = 0.33* 1007 * ($T_{air,\ out}$- 40) (Take m= 0.225 kg/s)

$T_{air,\ out}$ = 55 $_0$C

As For Counter Flow heat Exchanger,

The value of LMTD for the counter flow heat exchanger is as follows,

$\theta_1 = (65-55) = 10$

$\theta_2 = (45-40) = 5$

$$LMTD(\theta lm) = \frac{\theta 1-\theta 2}{\ln(\theta 1/\theta 2)} = \frac{(10-5)}{\ln(10/5)} = 7.21°C$$

Now, For Cross flow heat exchanger directly takes the value of correction factor for single cross flow heat exchanger with both fluid are unmixed as below,

Correction Factor, F = 0.85

Overall heat Transfer coefficient,

$$\frac{1}{U} = \frac{1}{hi} + \frac{1}{ho}$$

Internal Flow of Oil

Velocity,

$$v_{oil} = \frac{m_{oil}}{\rho_{oil} A_{tube}} = \frac{0.0833}{864*(\frac{3.14}{4}*0.012)} = 1.22 \text{ m/s}$$

Reynolds No.

$$Re_{oil} = \frac{\rho_{oil} v_{oil} D_{oil}}{\mu_{oil}} = \frac{864*1.22*0.01}{0.0717} = 723$$

Nusselt No.

$$Nu_{oil} = .023 Re^{0.8} Pr^{0.3} = .023*723^{0.8} * 1050^{0.3} = 26.10$$

Heat transfer coefficient,

$$h_{oil} = \frac{Nu_{oil} K_{oil}}{D_{tube}} = \frac{96.4*0.147}{0.01} = 743 \frac{W}{m2k}$$

External flow of Air

Velocity,

$$v_{oil} = \frac{m_{air}}{\rho_{airl} A_{tube}} = \frac{0.2257}{1.16*0.0648} = 3 m/s$$

Reynolds No.

$$Re_{oil} = \frac{\rho_{airl} V_{air} L_{air}}{\mu_{air}} = \frac{1.16*3*0.}{1.19*10^{-5}} = 87731$$

Nusselt No.

$$Nu_{air} = .664 Re^{\frac{1}{2}} Pr^{\frac{1}{3}} = .664 * 87731^{0.5} * 0.74^{0.33} = 178$$

Heat transfer coefficient,

$$h_{air} = \frac{Nu_{air} K_{air}}{L_{tube}} = \frac{28.54*0.025}{0.01} = 71.35 \frac{W}{m2k}$$

Now, From above Equation the overall heat transfer coefficient is $U = 64.80 \frac{W}{m2k}$

To calculate the total surface area

We have,

$$q = U * As * LMTD * F$$

$$As = \frac{3410}{64.80*7.21*0.85} = 8.678 \text{ m}^2$$

Also,

$$As = \pi * d * L$$

$$L = \frac{8.678}{3.14 * 0.01}$$

$$L = 33.6 \text{ m}$$

No of Tube,

$$N = \frac{\text{Total length of tube}}{\text{Length of single tube}}$$

$$N = \frac{L}{l} = \frac{27.6}{0.6} = 46$$

3.2 System Configuration

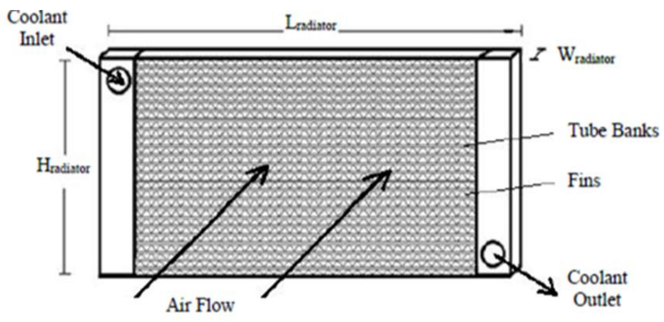

Fig -1: Schematic of radiator

The coolant fluid (water) that passes through the engine block enters on the high side of the radiator as the hot fluid. From here, this hot fluid fills the tube banks of the radiator and makes a single pass across the radiator.

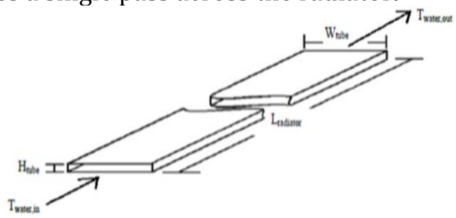

Fig -3: Schematic of single tube

During the entire process two main forms of heat transfer occur: 1) convection due to the internal flow of fluid passing through non-circular tubes and 2) convection due to the external flow of air across the tube bank. Fig. 3 shows the single tube geometry and the internal fluid flow direction through the tubes.

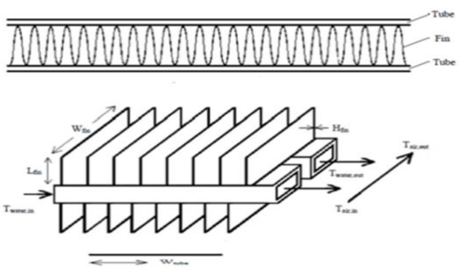

Fig -4: Actual fin geometry (top) and Assume fin geometry (bottom)

Fig. 4 shows a detailed diagram of a segment of the tubes with both heat transfer processes.

3.3 Mathematical Model

The goal of the project is to use the analytical heat transfer process to determine outlet coolant and air temperatures and to compare this data with simulation results. The equations and calculations following are simply used to solve for both outlet temperatures. The initially assumed outlet temperatures and calculated outlet temperatures are then iterated to solve for the actual values for the outlet temperatures for air and oil.

$$q = m_{air} c_{pair} (T_{air,out} - T_{air,in}) = m_{oil} c_{poil} (T_{oil,in} - T_{oil,out})$$

Internal Flow of Oil

The hot water from the engine travels through the tubes of the radiator.

Area of Tubes,

$$D_{hudraulic} = \frac{4 A_{tube}}{P_{tube}}$$

The hydraulic diameter must be used because it is a non-circular cross section.

Mean Temperature of Oil

The average temperature of water must be calculated to find the fluid's material properties. The properties will be interpolated at this temperature. The properties that are needed are density, Prandtl number, thermal conductivity, dynamic viscosity, and specific heat.

Velocity

$$v_{oil} = \frac{Q_{oil}}{N_{tube} A_{tube}}$$

The velocity of the water through each tube must be found to calculate the Reynolds number. The number of tubes is given by the chosen radiator.

Reynolds Number

$$Re_{oil} = \frac{\rho_{oil} v_{oil} D_{oil}}{\mu_{oil}}$$

Nusselt Number

The Nusselt number was found as a constant for a rectangular cross section from Table 8.1 for fully developed laminar flow. The ratio of width over height of the tube is used in this table to determine the Nusselt number.

Convective Heat Transfer Coefficient for Oil Flow

$$h_{oil} = \frac{Nu_{oil} K_{oil}}{D_{hydraulic}}$$

External Flow of Air

The air flows from the fan across the radiator tubes and through the fins utilizing convective heat transfer. In reality, the flow of air over the tubes will be slightly different due to the fluid flowing around the first tube before reaching the second tube, so calculating the heat transfer coefficient would be very difficult. To simplify the calculations, the flow is assumed to be the same over both tubes. Also, because the height to width ratio of the tubes is so small, the air will be assumed to be flowing on both sides of a flat plate.

Mean Temperature of Air

The average temperature of air must be calculated to find the correct material properties for later use. These properties are specific heat, thermal conductivity, kinematic viscosity, and Prandtl number.

Velocity

$$v_{air} = \frac{Q_{air}}{A_{radiator} - (N_{tube} H_{tube} L_{radiator})}$$

Reynolds Number

$$Re_{air} = \frac{\rho_{air} v_{air} W_{fin}}{\mu_{air}}$$

Nusselt Number

Looking at the geometry of the tubes, it can be assumed that the flow of air is similar to parallel flow over a flat plate. Since the flow never reaches the critical Reynolds number for a flat plate, Re = 5x105, it is said to be laminar for the entire process.

$$Nu_{air} = .664 Re^{\frac{1}{2}} Pr^{\frac{1}{3}}$$

Convective Heat Transfer Coefficient for Air Flow

$$h_{air} = \frac{Nu_{air} K_{air}}{W_{tube}}$$

Fin Dimensions and Efficiency

The geometry of the fins on the radiator is sinusoidal. The troughs of the fins touch the lower adjacent tube and the peaks of the fins touch the upper adjacent tube. The fin efficiency equation takes into account the geometry of the fin and its dimensions to find the efficiency the fin will have.

$$n_{fin} = \frac{\tanh(mL_c)}{mL_c}$$

Overall Surface Efficiency

The overall surface efficiency is needed for the external flow of air because the imperfections of the flow around the fins must be considered

$$n_o = 1 - \frac{N_{fin} A_f}{A_{fin}} (1 - n_{fin})$$

Effectiveness-NTU Method

The Effectiveness-NTU method is used to find the effectiveness of the system. The overall heat transfer coefficient is needed. The surface efficiency is needed for the external flow of air because the imperfections of the flow around the fins must be considered. Using the convective heat transfer coefficients of both the internal and external flows, the UA is calculated. This value is then used to calculate the NTU.

Overall Heat Transfer Coefficient

$$UA = \frac{1}{\left(\frac{1}{n_o h_{air} A_{external}} + \frac{1}{h_{oil} A_{internal}}\right)}$$

Number of Transfer Units

$$NTU = \frac{UA}{C_{min}}$$

Effectiveness

The radiator utilizes a cross-flow single pass design where both fluids remain unmixed. This correlates to a specific equation to calculate effectiveness. However, this equation requires the heat capacity ratio, C_r, to be equal to 1. The calculated heat capacity ratio is 0.455; therefore, the effectiveness is only a close approximation instead of the true value.

$$\varepsilon = 1 - \exp^{[(\frac{1}{C_r} NTU^{0.22})(\exp^{(-C_r NTU^{0.78})} - 1)]}$$

Heat Transfer Rate

The maximum heat transfer rate must be found to find the predicted heat transfer rate. Once this is known, the final output temperature of both the hot and cold fluid is calculated using a modified version of the initial thermal energy equation. These outlet temperatures must be iterated with the initially guessed outlet temperatures until the numbers are equivalent. The iterated outlet temperatures for both air and water are the theoretical values, which are used to compare with the experimental results.

Max Heat Transfer Rate

$$q_{max} = C_{min}(T_{oil,out} - T_{air,in})$$

Predicted Heat Transfer

$$q_{predicted} = \varepsilon q_{max}$$

Temperature Out

$$T_{oil,out} = 1 - \frac{q_{predicted}}{C_{oil}} \qquad T_{air,out} = 1 - \frac{q_{predicted}}{C_{air}}$$

3.4 Theoretical Calculation

Table -1: : Basic Radiator Dimensions

Lrad	Hrad	Wrad	Wtub	Htub	Lfin	Wfin	Hfin	Ntube	Nfin
inch	inch	inch	inch	inch	inch	inch	inch		
26.125	18	2.375	1	0.083	0.156	2.3	0.003	36	780

Table -2: : Properties of Oil at Average Temperature

ρoil	cp,oil	Proil	koil	μoil
kg/m3	J/kgK	unitless	W/mK	kg/sm
864	2047	1050	0.147	0.0717

Table -3: : Properties of Air at Average Temperature

ρair	cp,air	Prair	kair	vair
kg/m3	J/kgK	unitless	W/mK	m2/s
1.16	1007	0.74	2.50E-2	1.19E-5

Table -4: : Final Results of Radiator

hoil	hair	U	NTU	ε	qpredictd
W/mK	W/mK	W/mK	unitless	%	W
748	72.2	70.6	0.59	45	3786

Table -5: : Theoretical Temp of Fluids through Radiator

Toil,in	Toil,out	Tair,in	Tair,out
K	K	K	K
338	321.25	313	328.02

4. CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

4.1 Pre –Processor

Geometry

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinates system within ANSYS. The design modeller opens as a new window as the geometry is double clicked

Table -6: : Specification of the Radiator

Details	Dimensions (mm)
Length	644
Height	457
No of Tube	36
Tube length	584
Tube inner diameter	7
Tube outer diameter	9

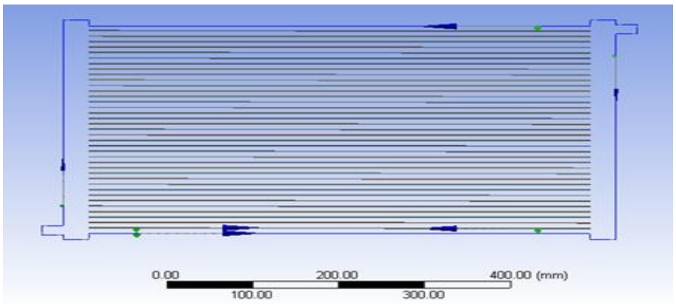

Fig -5: 2D Geometry of Radiator

2. Mesh:

Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. At this point ANSYS understands the makeup of the part. Now define how the Modeled system should be broken down into finite pieces. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region.

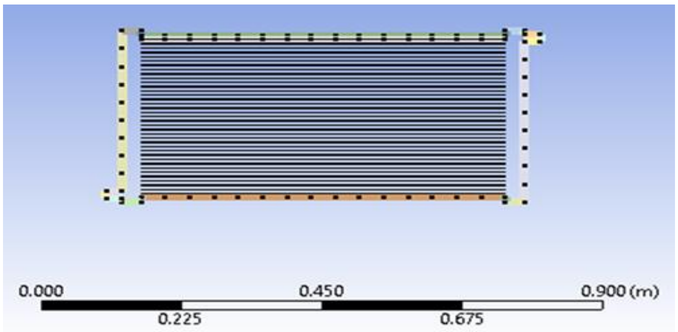

Fig -6: Meshing of Radiator

4.2 Solver

1. Problem Setup

The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state.

3. Materials

The create/edit option is clicked to add water-liquid and copper and stainless steel and galvanized iron to the list of fluid and solid respectively from the fluent database.

5. Boundary Conditions

The main boundary condition include, a mass flow rate inlet boundary condition where used in the inlet nozzles. The cylindrical shaped geometries are the wall. At the outlet, the pressure outlet (atmospheric pressure) boundary condition was used. And all other portions are considered as the wall boundary with convective heat transfer surfaces. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition.

Inlet

- Select inlet and change type to mass flow inlet
- Enter the mass-flow rate
- Enter the value of inlet gauge pressure = 1.5e5 Pa
- Provide the inlet temperature = 338K
- Change the option under Direction Specification to "Normal to boundary"

Outlet

- Select the outlet and change type to "Pressure-outlet"
- Enter the value for outlet gauge pressure = 0 Pa
- Change the option under Direction Specification to "Normal to boundary"

Wall

- Select wall and change type to "Wall"
- Select the material for the wall: Aluminum
- Choose convection as the heat transfer mechanism employed in wall.
- Enter the value for heat transfer coefficient as 92 W/m^2K.
- Enter wall thickness as 1mm Enter the free stream temperature on wall

4.3 Post –Processor

At first the analysis is done by keeping inlet temperature of coolant and free stream temperature constant and temperatures are obtained for different mass flow rates. Varied input mass flow rates are given in the table.

Temperature Counter Plot

1. Mass flow rate 0.083kg/s

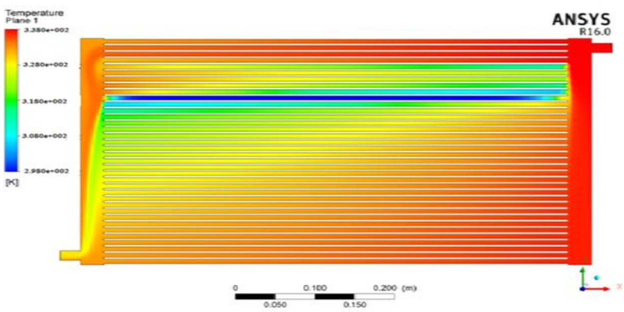

Fig -7: Temperature contour for mass flow rate 0.083kg/s

2. Mass flow rate 0.093kg/s

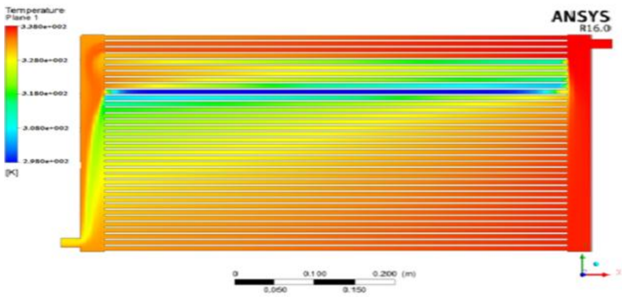

Fig -8: Temperature contour for mass flow rate 0.093kg/s

3. Mass flow rate 0.103kg/s

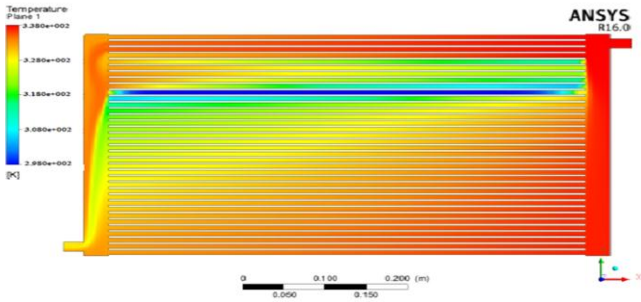

Fig -9: Temperature contour for mass flow rate 0.103kg/s

4. Mass flow rate 0.113kg/s

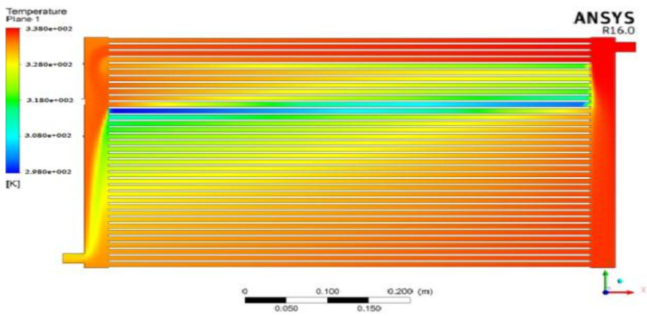

Fig -10: Temperature contour for mass flow rate 0.113kg/s

5. Mass flow rate 0.123kg/s

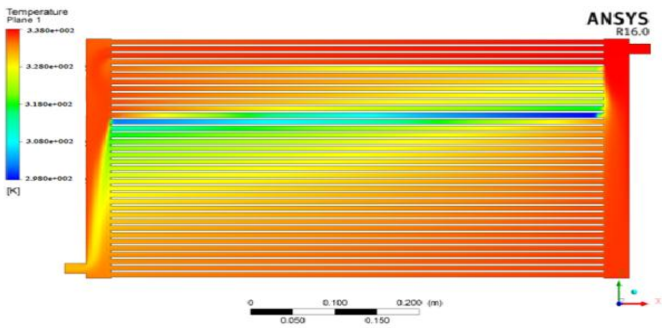

Fig -11: Temperature contour for mass flow rate 0.123kg/s

Result of Temperature Plot

Table -7: : Reading of temp for various Mass Flow Rates

Mass flow rate (Kg/s)	Mass flow rate (Kg/s)	Oil inlet temp (K)	Oil outlet temp (K)	Air inlet temp (K)	Air outlet temp (K)
0.083	0.225	338	322.62	313	326
0.093	0.225	338	324.84	313	325
0.103	0.225	338	328.24	313	321
0.113	0.225	338	329.38	313	320
0.123	0.225	338	331.92	313	318

5. DATA VALIDATION

Comparison of analytical and simulation results

Table -8: : Analytical results

Parameter		Unit	Value
Total heat transfer	Q	W	3786
Oil outlet temperature	Toc	K	321.3
Air outlet temperature	Toa	K	328.82

Table -9: : Simulation results

Parameter		Unit	Value
Total heat transfer	Q	W	3528
Oil outlet temperature	Toc	K	322.46
Air outlet temperature	Toa	K	326.25

Comparison shows that both results closely matched with each other. Thus theoretical thermal analysis of radiator using ε-NTU method is validated using simulation approach.

5.1 Discussion of Results

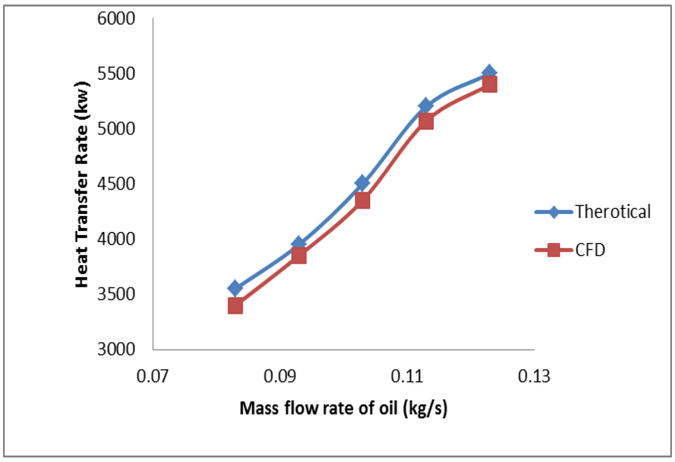

Chart -1: Mass flow rate v/s. heat transfer rate

The relationship of above graph shows that increase in mass flow rate of oil results in increase in heat transfer rate. The nature of graph is linear for flow rate range from 0.083 Kg/s to 0.103 Kg/s. After that point because of sudden increase in mass flow rate the nature of graph diverts.

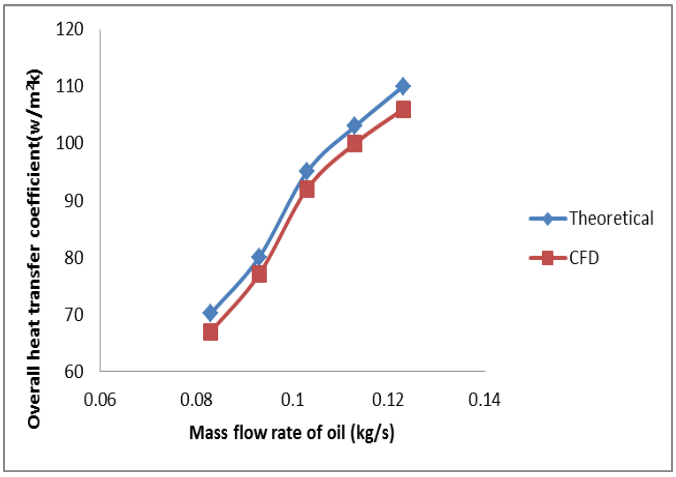

Chart -2: shows that as the mass flow rate increases the overall heat transfer rate (U) increases.

The heat transfer coefficient is maximum i.e. 110 W /m2° K at mass flow rate of 0.123 Kg/s. The nature of graph is linear up to flow rate of 0.093Kg/s. It get diverted at this point and increases sudden up to flow rate of 0.103 Kg/s. From this point graph shows slightly reducing nature up to flow rate of 0.113 Kg/s and again graph shows linear nature.

6. CONCLUSIONS

A CATIAV5r18 and CFD package (ANSYS FLUENT 13.0) was used for modelling and CFD study of heat transfer characteristics of a hydraulic oil cooler by the application of heat exchanger for counter flow. The objectives of the project, to find and compare the inlet and outlet temperatures of the fluids in the radiator were accomplished successfully. Several important conclusions could be drawn from the present simulations and would be presented as follows.

- It is found that overall heat transfer coefficient increases with the increase in mass flow rate. For maximum mass flow rate (0.123 Kg/s) it is found to be increased up to 110 W/m2 ° K which more than theoretical heat load hence system is safe and will not overheat.
- The capacity ratio and effectiveness is increased with the mass flow rates. This shows that the existing cooler is more efficient at higher flow rates. The highest effectiveness is found as 0.725.
- Ansys Fluent is good CFD program to to simulate the heat transfer cases.

REFERENCES

[1] Kansal Santosh ,Fatehmohd.Shabahat,2014 ," Design and Performance Evaluation Of Shell And Tube Heat Exchanger Using CFD Simulation" , International Journal Of Research And Technology ,Volume 3 ,Issue 7 ,PP-1663-1666.

[2] KrunalSuryakantKayastha , Volume 11, Issue 01 (January 2015), PP.24-35 IJERD, "CFD Simulation and Heat Transfer Analysis of Automobile Radiator using Helical Tubes".

[3] M. Carl, D. Guy, B. Leyendecker, A. Miller, and X. Fan, "The Theoretical And Experimental Investigation Of The Heat Transfer Process Of An Automobile Radiator", Proc. ASEE Gulf Southwest Annual Conference, El Pas.o, Texas, 2012, 1-12.

[4] Hardikkumar B Patel &DeepuDinesan, IJIRST Volume I Issue 12 May 2015,"PerformanceAnalysis of an Automobile Radiator using CFD".

[5] A. R. Esmaeili Sany, M. H. Saidi, J. Neyestani,"Experimental Prediction of Nusselt Number and Coolant Heat Transfer Coefficient in Compact Heat Exchanger Performed with ε-NTU Method", The Journal of Engine Research, Vol.18, Spring, 2010, 62-70.

[6] KrunalSuryakantKayastha , Volume 11, Issue 01 (January 2015), PP.24-35 IJERD, "CFD Simulation and Heat Transfer Analysis of Automobile Radiator using Helical Tubes".

[7] Pawan S. Amrutkar, Sangram R. Patil& S. C. Shilwant, ISSN 2277-4785 Vol. 3, Issue 4, Oct 2013, 1-10; "Automotive Radiator - Design And Experimental Validation".

[8] C. Oliet, A. Oliva , J. Castro & C.D. Pe´rez-Segarra, Applied Thermal Engineering 27 (2007) 2033–2043 Science Direct, "Parametric studies on automotive radiators".