International Journal of Engineering Research
Volume No.5, Issue No.9, pp : 778-782

ISSN:2319-6890)(online),2347-5013(print)
1 September 2016

Experimental Study and CFD Analysis of Copper Radiator for Passenger Cars

R.K. Jaya kumar[1],Dr.K.S.Amirthagadeswaran[2]

[2] Department of Mechanical Engineering,Government College of Technology, Coimbatore, Tamilnadu, INDIA
[1] Department of Aeronautical Engineering,Sri Ramakrishna Engineering College, Coimbatore, Tamilnadu,INDIA
jayakumar.rk@srec.ac.in

Abstract: In the present scenario, the usage and importance of radiator is increasing gradually. Engine cooling system takes care of excess heat produced during engine operation and it regulates the engine surface temperature at optimum value. Development in engine cooling system is often required to improve its performance and heat transfer rate. Though aluminium radiators have become common, still many established industries have facility to manufacture copper radiators. Cuprobraze radiators are noted to perform nearer to aluminium radiators. In this work, an attempt has been made to investigate the performance of copper radiator used in passenger cars. Behaviour of a copper radiator at different air velocities was studied by experimental method. The existing copper radiator is modelled using solidworks software and validation of performance done by fluid flow software. The deviation between the experimental and simulation results shows only 8%. Thus it met engineering requirements. So, this method may be taken as an effective approach for the prediction of heat transfer performance in a radiator. This will form as the base to analyse the performance of cuprobraze by modifying the material properties and its parameter.

Keywords: **Radiators, CFD, cooling performance**, **Heat Transfer.**

1.Introduction

A radiator is a cross flow heat exchanger, which transfers heat from hot coolant to air byfins placed on the tube throughout its length via conduction and convection. The coolant circulates over the engine block and absorbs heat from the engine during combustion process. Hot coolant coming from engine is passed to radiator for cooling the coolant. It regulates the engine temperature at optimum value. Radiator is the primary component of the cooling system in automobiles. Failure of engine takes place mainly due to excessive heat produced in the engine components. This can be avoided by employing the proper cooling system. Radiators has been classified depending on flow and type of materials used. Radiators are preferred, based on heat dissipation rate of the engine. Heat transfer rate of copper is higher than aluminium, but the drawbacks of copper are its weight and cost compared to aluminium. And also there may be chance of formation of white residues around the tube due to chemical reactions of different metals.(brass tube, copper fin, lead/ tin solder.).

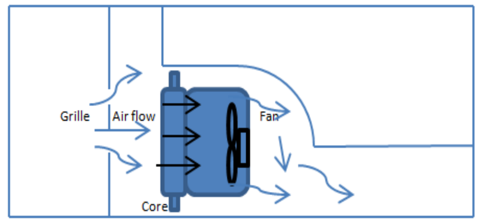

Fig1. Automobile Radiator

J.P Yadav, Bharat Raj Singh[1] states that most modern cars use aluminium radiators and they are made by brazing thin aluminium fins to flattened aluminium tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. J.R Patel, A.M .Mavani[2] used computational fluid dynamics (CFD) to model the flow of fluid and heat transfer performance characteristics and one design is suggested as a possible replacement to the conventional automobile radiators. Fins are used to increase heat transfer area on the air side, since the air has the largest influence on the overall heat transfer rate.By varying massflow rate of air, pitch of tube and coolants are analyzed successfully using numerical simulation software. C. Oliet, A. Oliva, J. Castro, C.D. Pe´rez-Segarra[3] studied different factors which influence radiator performance. It includes air and coolant flow, fin density and air inlet temperature. It is observed that heat transfer and performance of radiator is strongly affected by air and coolant mass flow rate. As air and coolant flow increases cooling capacity also increases. When air inlet temperature increases, heat transfer and thus cooling capacity decreases.

2. Methodology

For the progress of this work, two methodologies were adopted. They are

1. Experimental Methodology
2. Software Analysis Methodology

Experimental Methodology

Experimental evaluation was carried out in centrifugal blower. Road testing offers a direct assessment of the true behaviour of engine cooling performance as well as provides airflow over the vehicle under “real” conditions. On the other hand, centrifugal blower provides an approximation of on-road flow conditions that are replicated in controlled environment (involving a stationary vehicle with respect to movement of air). The main advantage of centrifugal blower testing over wind tunnel testing is that there is a lesser pressure drop across the radiator in the case of centrifugal blower. It leads to improved

International Journal of Engineering Research
Volume No.5, Issue No.9, pp : 778-782

ISSN:2319-6890)(online),2347-5013(print)
1 September 2016

cooling of the radiator. This method of testing is more convenient, time-effective and accurate.

Experimental setup

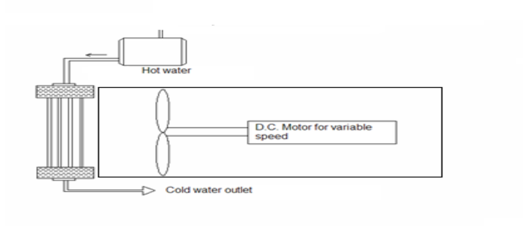

Fig 2. Schematic layout of centrifugal blower

Table1. Specifications of the centrifugal blower

Area of the Duct	$0.1m^2$
Torque arm distance	0.2m
Type of the drive	D.C motor
Power	3.7kW
Speed	3000rpm

Fig3. Digital Anemometer

Table2. Specifications of the Digital Anemometer

Parameters	Range	Resolution	Accuracy
Air velocity	0.40 – 45 m/s	0.1 m/s	±(2%+0.1m/s)
Air Temperature	0 - 60°C	0.1°C	0.5°C

Procedure for Experimental setup

- The initial setup and arrangements for the experiment is done.
- Copper radiator is fixed in front of the discharge area of the blower.
- Connection from the boiler to the radiator upper tank is done.
- By using electrical heater, water is heated to a temperature of 80°C is allowed to flow through the tubes.
- Thermometer is used, to measure the temperature of water.
- The centrifugal blower is started and operated at an rpm of 1400 – 1500 to obtain the required velocity of air.
- Velocity and temperature of air passed through the radiator is measured using digital anemometer.
- Velocity of air is varied by changing the rpm of the blower, which is noted from the LED display of the centrifugal blower.
- Outlet temperature of water is noted.
- Repeat the above steps to know the behaviour of a radiatorat different air velocities.

Table 3. Tabulation

S.no	**Air speed (km/hr)**	**Inlet temp (°C)**		**Outlet temp (°C)**	
		Water	**Air**	**Water**	**Air**
1	40	80	27	60	38
2	50	80	27	59	41
3	60	80	27	57	44

The above table 3.shows the readings were taken during experimentation.

Radiator performance

Two common methods exist for expressing the heat transfer characteristics of a given heat exchanger surface geometry. These are known as LMTD approach and ε-*NTU* approach. The performance of a heat exchanger can be determined by examining the heat loss and heat gain that takes place between the working fluids.

Heat lost by the coolant can be expressed as:

$$Q_h = m_h C_{ph}(T_{hi} - T_{ho})$$

Heat gained by the air can be expressed as:

$$Q_c = m_c C_{pc}(T_{co} - T_{ci})$$

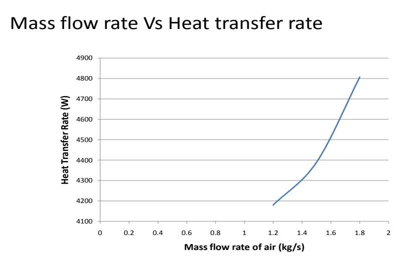

Fig 4. Effect of mass flow rate Vs Heat transfer rate.

The above graph clearly shows that, heat transfer rate increases with increase in air velocity.

Software Analysis Methodology

Flow Simulation is a software fully integrated in Solidworks for computing fluid flows inside and outside Solidworks models, as well as heat transfer to (from, between, in) these models due to convection, radiation, and conduction with a proven computational fluid dynamics (CFD) technology. Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. CFD analyses reduce

ISSN:2319-6890)(online),2347-5013(print)
1 September 2016

development time and increases the reliability of design. Navier stoke equation is the basis of all CFD problems. CFD provides the ability to simulate any physical condition. Simulation of a model can be executed in a short period of time with help of high speed computers.[14]The modelling of radiator is done using Solidworks flow simulation software.

Fig 5.copper radiator

Table 4. Specifications of the radiator

Size	350×350×55mm
Fin material	Copper
Tube material	Brass
Type of fin	Serpentine
Number of tubes	66
Number of tube rows	2
Fin pitch	2.8mm
Frontal Area	.12m^2

In table 4, radiator specifications are mentioned. Brass is an alloy of (70% cu, 30% zn).
The part modelling of the tube is shown in fig6

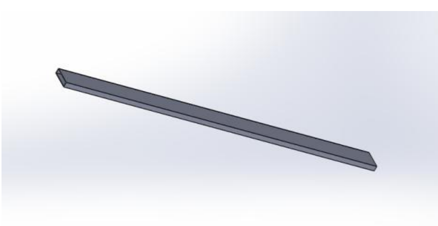

Fig6.Part modelling of the tube

Instead of modelling a complete geometry of the radiator core, three set of tubes have been drawn to reduce complexity.

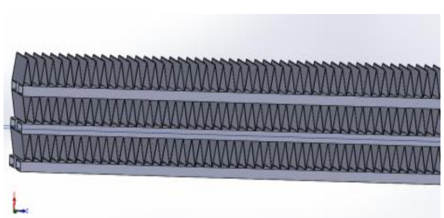

Fig7. Three set of tubes for radiator core

Modelling of the fin over the length of tube has been done using pattern command as shown in fig7.

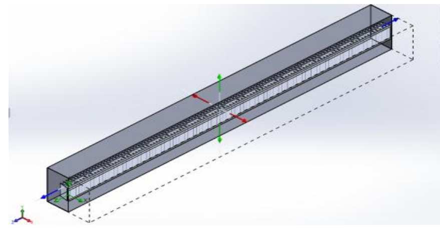

Fig 8. Symmetrical section of tube

In this type of radiator, serpentine type of fins is used. Due to unavailability of high speed computer and in order to reduce the computational time and to increase accuracy, fins over the single tube are considered for analysis. Further simplification has been done by considering the symmetry of the tube over its length. Result obtained in this section is replicated the whole model of the radiator core.

Next step in the analysis is meshing. During meshing, the software model is divided into a number of small components, in order to increase the accuracy of results. Every small component was analysed to obtain the desired result. By using Solid works fluid flow software, the analysis was carried out.

The complete meshed model is shown in fig 9.

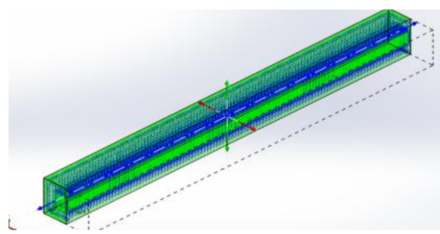

Fig9. Meshed model of tube and fin

Generation of automesh over the surface of fin and tube can be done easily. This is the main advantage of using fluid flow software over others. Coarse mesh will be generated on the domain surface to reduce the computational time for meshing and fine mesh will be generated on the surface of the tube and fin for accuracy in results.

Flow and temperature maldistribution

While calculating heat exchanger performance, it is usually assumed that flow is uniform in both rate and temperature distribution. The causes for maldistribution are as follows.[13]

- Non-uniform inlet conditions caused by the radiator grille.
- Inherent mismatch between the annular flow of the fan and the rectangular heat exchanger. (The fan shroud will never be deep enough to allow full transition).
- The objects placed downstream of the fan, cause asymmetric fan airflow.
- Low pressure created between the grille and radiator, because of hot air circulation

The final step in this analysis is solving. The following assumptions were made during the analysis. They are [14]

International Journal of Engineering Research
Volume No.5, Issue No.9, pp : 778-782

ISSN:2319-6890)(online),2347-5013(print)
1 September 2016

1. The working medium is dry air entering at 27°C.
2. The standard physical properties of air at 27°C have been taken.
3. Coolant entering the radiator is 80°C.
4. The standard physical properties of air at 80°C have been taken.
5. There is no phase change in the fluid.
6. All dimensions are uniform throughout the radiator.
7. Air inlet is velocity inlet, air outlet is pressure outlet
8. Water inlet is mass flow inlet and water outlet is pressure outlet.

Table 5. Material properties of fin- copper

Thermal conductivity	394 W/mK
Density	8.94 g/m^3
Specific heat	394 J/kgK

Table 6. Material property of tube-brass

Thermal conductivity	110 W/mK
Density	8.75 g/m^3
Specific heat	380 J/kgK

Boundary conditions for the analysis has taken as
Velocity of inlet air : 16.6m/s
Mass flow rate of water : 0.05kg/s
Pressure condition : 1bar
For convergence of the solution, number of iterations takes place during the solving process. Results can be viewed in contour plots.

3. Results and discussion

The temperature of coolant and air is observed from the contour plot. Heat lost by the coolant is not equal to heat gained by the air, because air has very low thermal conductivity.

The contour plot showing the temperature distribution over fin and tube is shown in fig10.

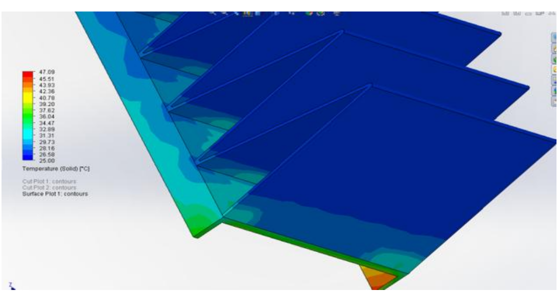

Fig 10. Temperature contour for air side

From the above fig10.temperature contour of the air side (through fins), the outlet temperature of air is observed as 32° C on average.

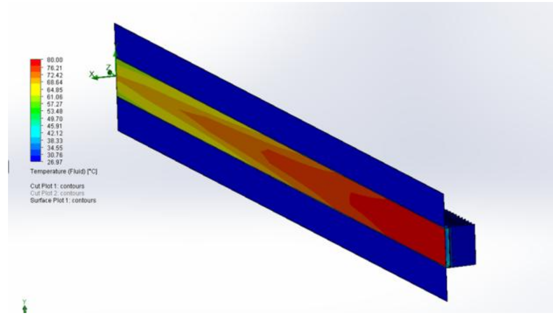

Fig11.Temperature contour for water side

From fig 11.the temperature contour plot, outlet temperature of water is observed as 64°C. Drop in temperature shows that, heat is removed from the water.

4. CFD Validation

By using experimental results, CFD results were validated. With the help of a centrifugal blower, the radiator is tested. The inlet temperature of air and water were kept constant and the readings were observed. There is a slight variation in the results obtained by experimental and software models, approximately 8%.

5. Conclusion

In this work, an attempt has been made to investigate the performance of copper radiator used in passenger cars. Heat transfer rate, Heat transfer coefficient and surface pressure drop increase with increase in air velocity. Deviation in results is mainly due to friction loss, quality of mesh and its accuracy. Further modification and optimization can be done by changing the material properties and the design parameters. This might be a suitable approach to find the heat dissipation from the radiator.

Acknowledgement
The authors are grateful to Government College of Technology, Coimbatore for allowing us to conduct experiments in the Thermal laboratory.

References

i. JP Yadav and Bharat Raj Singh, "Study on Performance Evaluation of Automotive Radiator", S-JPSET , Vol. 2 Issue 2, 2011, ISSN : 2229-7111.

ii. J. R. Patel, A. M. Mavani, "Review paper on CFD Analysis of Automobile Radiator to improve its Thermal performance",IJSRD, Vol. 2 Issue 01,2014.

iii. C. Oliet, A. Oliva, J. Castro, C.D. Pe´rez-Segarra, "Parametric studies on automotive radiators", Applied Thermal Engineering, 27, 2007.

iv. Zunjin Fan, ZhengqiGu, etal "Prediction of Heat Dissipation Effect by Simulating Radiator Group of Mining Dump-truck",Applied Mechanics and Materials Vols. 217-219 (2012) pp 2473-2479

v. P. K. Trivedi, N. B. Vasava, "Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by CFD Analysis", International Journal of Engineering and Advanced Technology (IJEAT), Vol.1 Issue-6, 2012, ISSN: 2249 – 8958

ISSN:2319-6890)(online),2347-5013(print)
1 September 2016

vi. *PitambarGadhave, Shambhu Kumar, "Enhancement of forced Convection Heat Transfer over Dimple Surface", International Multidisciplinary Journal, 2012.*

vii. *Prof. D. K. Chavan, Prof. Dr. G. S. Tasgaonkar, "Thermal Optimization of Fan assisted Heat Exchanger (Radiator) by Design Improvements".International Journal of Modern Engineering Research (IJMER), Vol.1 Issue 1, 2011.*

viii. *P. Gunnasegaran, N. H. Shuaib, and M. F. Abdul Jalal."The Effect of Geometrical Parameters on Heat Transfer Characteristics of Compact Heat Exchanger with Louvered Fins", ISRN, Thermodynamics, Volume 2012.*

ix. *SalvioChacko S., et. al., "Numerical Simulation for Improving Radiator Efficiency", Engineering Automation Group, Tata Technologies Limited, Engineering research Centre, Tata Motors Limited, India, 2003.*

x. *Hilde Van Der Vyer, JacoDirker et al,"Validation of a CFD model of a three dimensional tube-in-tube heat exchanger", ThirdInternational Conference on CFD in the Mineralsand Process Industry, CSIRO, Melborne, Australia. pp. 25-32.2003.*

xi. *A.Witry M.H. Al-Hajeri and Ali A. Bondac, "CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator", 3rdInternational conference on CFD, CSIRO, Melborne, Australia, pp. 12-19. 2003.*

xii. *J A Chen, D F Wang and L Z Zheng, "Experimental study of operating performance of a tube-and-fin radiator for vehicles", Proceedings ofInstitution of Mechanical Engineers, Republic ofChina, 215, pp. 2-8, 2011.*

xiii. *P. Kanefsky, Nelson, A. Valerie, M. Ranger, "A Systems Engineering Approach to Engine Cooling Design", SAE, SP-1541, 1999.*

xiv. *Barathraj,R.K, et al, "Performance Analysis of copper radiator using CFD" International Journal on Modern Engineering and Research,2014.*

xv. *Y.L. Lee, Y.T. Yong, "Analysis of Engine Cooling Airflow including Non-uniformity over a Radiator", Int. J. of Vehicle Design, 24, pp. 121-135, 2000.*

xvi. *J.P. Chiou, "The Effect of Flow Non-uniformity on the Sizing of Engine Radiators", SAE 800035, 1980.*

xvii. *Malipeddi, S.R., "Guidelines for Underhood Thermal Management of Off-highway vehicles", IMechE, C599 101 2003, pp.879-888, 2003.*

xviii. *A. R. EsmaeiliSany, 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.*

xix. *J. Stafford, E. Walsh, V. Egan, and R. Grimes, "Flat Plate Heat Transfer With Impinging Axial Fan Flows," in International Journal of Heat and Mass Transfer, Vol. 53, pp. 5629–5638, 2010.*