International Journal of Engineering and Techniques - Volume 3 Issue 5, Sep - Oct 2017

RESEARCH ARTICLE OPEN ACCESS

Comparative Study and Performance Analysis of Copper, Aluminium and Cuprobraze Radiators Using Flow Simulation Software

R.K Jaya kumar[1], T.Ashokkumar[2]
[1 & 2]Assistant Professors, Department of Aeronautical Engineering, Sri Ramakrishna Engineering College, Coimbatore 22, India
[1]jayakumar.rk@srec.ac.in

Abstract

Cooling system in an automotive vehicle plays an important role in the engine performance. It dissipates excessive heat and maintains the performance of an engine at an optimum level. Size of the radiator mainly depends on heat load and availability of space. In this paper, an attempt has been made to examine the performance of radiators namely copper, aluminium and Cuprobraze employed in various commercial vehicles. Specifications of copper radiator and validation of numerical analysis were identified. After modifying the material properties of a radiator, performance analysis have been carried out using fluid flow software, which is fully integrated in solidworks for computing fluid flows. The result obtained from the analysis is commendable and this method may be taken as an effective approach for the prediction of heat transfer in radiators. Though aluminium has noted to perform better than copper and Cuprobraze, but it has limitations such as irreparability, non recyclability. This paper proposes an alternative solution for existing aluminium radiator with suitable cuprobraze radiator to provide required cooling effect to the engine, thereby overcoming the drawbacks of the radiator.
.

Keywords — **Heat Transfer, Radiators,CFD, Flow simulation**

1. Introduction

A radiator is a cross flow heat exchanger, which transfers heat from hot coolant to air by fins 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 have 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.[1]

Yadav J.P, Bharat Raj Singh [2] 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. Patel J.R, Mavani A.M [3] 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 mass flow rate of air, pitch of tube and coolants are analyzed successfully using numerical simulation software. Oliet C, et al [4] 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. If inlet temperature of air increases, heat transfer rate and cooling capacity decreases. Smaller fin spacing and higher louver fin angle have higher heat transfer. Fin density can be increased till it blocks the air flow and heat transfer rate decreases. Zunjin Fan, ZhengqiGu,et al,[5] studied experimentally and numerically the heat transfer coefficient and surface pressure drop of the dump-truck radiator fin increases with the increasing incoming velocity. Heat transfer coefficient of low temperature radiator is higher than the high temperature radiator, because it has an extra row of fins, which increases the pressure loss of air. Numerical results coincide with the experimental data perfectly. The accuracy well meets the engineering requirement. Hence this approach can be applied in the prediction of heat dissipation effect for radiator. Trivedi P.K.,[6] discussed heat transfer increases as the surface area of the radiator assembly is increased. This leads to a change in the geometry by modifying the arrangement of tubes in automobile radiator to increase the surface area for better heat transfer. The modification in arrangement of tubes in radiator is carried out by studying the effect of pitch of tube by CFD analysis using CFX. Results show that as the pitch of tube is either decreased or increased than the optimum pitch of tubes, the heat transfer rate decreases.

1.1 Cuprobraze Materials

CuproBraze fin material is made of copper that is alloyed with chromium. The chromium precipitates grow in the copper metal matrix of the fins during the brazing process. Although these chromium–copper precipitates are approximately 3 nm or 0.003 mm in diameter and they play a vital role in strengthening the fin alloy through a well-known mechanism known as precipitation hardening. Ironically, the precipitates strengthen the fins at temperatures that in the past would have seriously weakened fins made from conventional alloys. Interestingly, the precipitation of the chromium in the copper fins restores the electrical conductivity. Since thermal conductivity on this type of materials tracks electrical conductivity, the thermal conductivity of the fins is also restored. The electrical conductivity is only 60% of the conductivity of pure copper (IACS) before brazing but after brazing the conductivity is restored to a minimum of 90%,because the chromium atoms (which interfere with the conduction of electrons) are precipitated out of most of the copper–metal matrix. The thermal conductivity of the CuproBraze copper-fin alloy after brazing is 377 W/mK, which compares to 222 W/mK for aluminium. Because, soldering temperatures are not high enough to raise the thermal conductivity, this new copper–alloy fin material must not be used to make conventional soldered radiators. It should only be used for CuproBraze heat exchangers. The CuproBraze brazing operation is needed to restore the thermal conductivity. Similarly, CuproBraze tubes are made from a brass (i.e. copper that is alloyed with zinc) material that is approximately 85% copper with approximately 1% iron to prevent softening of the material during the brazing operation.[7]

1.2 Filler materials

The brazing filler metal that was developed for joining CuproBraze fins and tubes belongs to the CuSnNiP family. This filler metal, called OKC600, is composed of 4.2-wt.% nickel (Ni), 15.6-wt.% tin (Sn) and 5.3-wt.% phosphorus (P) with the balance copper (f75-wt.% Cu). This alloy has been patented (US Patent Number 5,378,294) but can be freely used for automotive and heavy-duty industrial heat exchanger applications. The melting temperature is 600 ^{o}C with a melting range of 10 ^{o}C. [7]. In this study, an attempt has been made to study the performance of various radiators using Computational Fluid Dynamics (CFD).

2. Methodology

Software methodology was adopted for the progress of the work. Computational Fluid Dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to analyze and solve the problems that involve fluid flows. CFD has become an integral part of the engineering design cycle. CFD analysis reduces the development time and increases the reliability of the designs. It makes it possible to evaluate and predict the working parameters such as velocity, pressure and temperature throughout the solving process. The fundamental basis of any CFD problem is the Navier Stokes equations, which define any single phase fluid flow. It works by solving the equations of the fluid flow over a region of interest, with the specified known conditions on the boundary of that region. Conservation of matter, momentum and energy must be satisfied throughout the region of interest. The modelling and numerical analysis of the radiator is done by using Solidworks.[1]

2.1 Flow simulation

Flow Simulation is capable of calculating flow of fluids of different types in the same analysis, but fluids of different types must be separated by walls. A mixing of fluids may be considered, only if the fluids are of the same type. Flow Simulation has an integrated database containing properties of several liquids, gases and solids. Solids are used in conjugate heat conduction analysis. For each analysis maximum of ten liquids or gases can be chosen. It can analyse various types of flows namely turbulent flows, Laminar flows and Laminar or Turbulent flows. The turbulent equations can be disregarded, only if the flow is entirely laminar. It will analyse and simulate the effects of fluid flow, heat transfer, and related forces on immersed or surrounding components.

Table.1. Specifications of the Radiator

Size	350×350×55mm
Type of fin	serpentine
Number of tubes	66
Number of tube rows	2
Fin pitch	2.8mm
Frontal Area	.12m^2

Table.1. indicates the specifications of the copper radiator and the type of fin arrangement

The solid models of fin and tube have chosen to be modelled in solidworks software. Instead of modelling a complete geometry of the radiator core, three set of tubes have been drawn to reduce complexity. Modelling of fin over the tube has been done using pattern command is shown in fig1. Hence this model is subjected to flow environment.

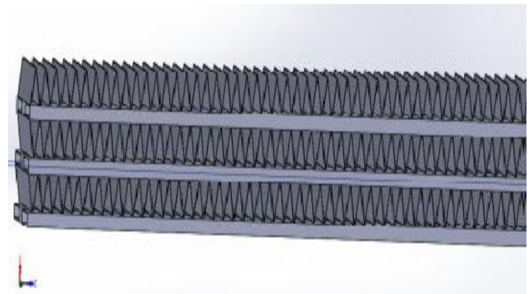

Fig.1. Three set of tubes and fin arrangement

Fig.2. depicts the symmetrical section arrangement of fins over a length of the tube. The inlet and outlet of the tube is sealed with a lid, where the internal fluid flow occurs. The entire arrangement is enclosed in a fluid domain, where meshing and solving takes place.

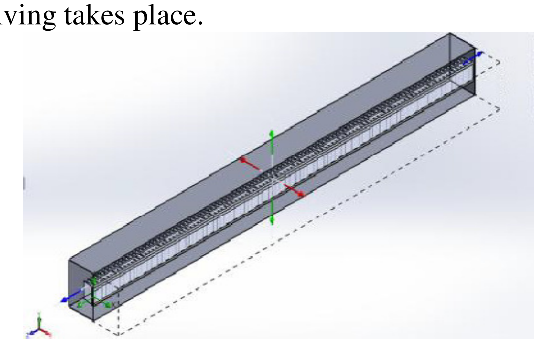

Fig.2. Symmetrical section of tube and fin

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, increase in accuracy, fins over the single tube is considered for analysis. Further simplification can be done by considering the symmetry of the tube over its length. Hence the fluid volume is meshed into a grid for the simulation.[1]

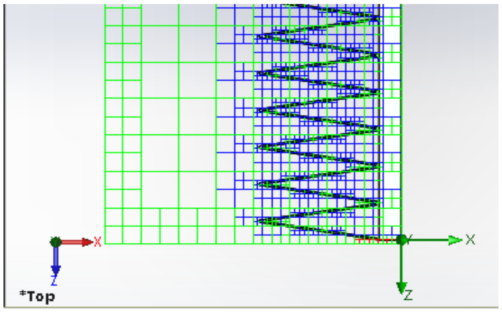

Fig.3. Meshed model of tube and fin

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. Fig3 shows the meshing of the entire model. The final step in this analysis is solving. The radiator is assumed to be in steady state or equilibrium condition for the flow rate of coolant. Fluid flow is considered as an incompressible and the analysis has been carried out to know the performance of the radiator.

The following assumptions are made for the numerical analysis in flow simulation software.

1. The working medium is dry air entering at 27ºC.

2. The standard physical properties of air at 27°C have been taken.

3. Coolant entering the radiator is 80ºC.

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.

2.2 Boundary Conditions

It is significant for the Computational Fluid Dynamics problems to define the boundary conditions correctly, otherwise there may be a chance of error during convergence of result or solving time will tends to increase. Depend upon the applications, boundary conditions will be varied. Some of the commonly used boundary conditions are inlet boundary condition, outlet boundary condition and wall boundary conditions and so on. Flow Simulation software indicates an inappropriate boundary conditions.

Boundary conditions for the analysis has taken as Coolant inlet temperature is 80ºC and the velocity of inlet air is16.6 m/s, Mass flow rate of water entering the tube of the radiator core is 0.05 kg/s and Pressure condition at the exit of the tube is 1bar. The Environment pressure condition is interpreted as a static pressure for outgoing flows.

Table.2. Material properties of fin – copper

Density	8.950 g/m^3
Thermal Conductivity	386 W/mK
Tensile strength	330 MPa
Specific heat	383 J/kg K
Melting point	1083 ˚C

Table.3. Material properties of tube – brass

Density	875 g/m^3
Thermal Conductivity	110 W/mK
Tensile strength	330 MPa
Specific heat	385 J/kg K
Melting point	915 ˚C

Material properties for the copper radiator are tabulated in the above table.2 and table.3. Hence, these properties would be taken during the solving process. Similarly, the properties of aluminium and Cuprobraze have been taken and performance analysis was carried out.

2.3 Flow Trajectories

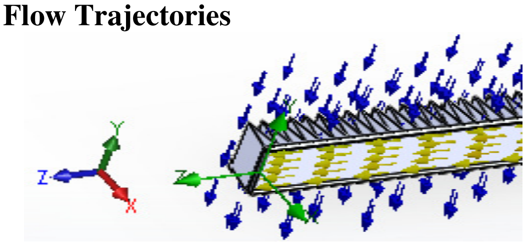

Fig.4. Direction of fluids

The fig.4 depicts the direction of coolant through the tube and direction of air over the fins. Both fluids are flowing perpendicular to each other and the fluids remain unmixed. Transfer of heat from the coolant to air takes place. This type of flow is employed in cross flow heat exchanger. Heat transfer coefficient and pressure drop increases with increasing coolant velocity. While increasing the fluid velocity, temperature boundary layer thickness on the tube will be reduced. Flow trajectories have shown the flow streamlines and it will provide a very good image of the 3D fluid flow.

3. Results and discussion

The radiator was tested at the air velocity of 60km/hr. The deviation between the experimental and simulation results shows only 8%. It is mainly due to frictional loss, quality of mesh and its accuracy. Thus, it met engineering requirements.[1] So, further analysis has been carried out using the same procedure by changing the material properties. Simulation results on geometry for symmetrical models, had given better insight on the performance. The following are the result of analysis of various types of radiators.

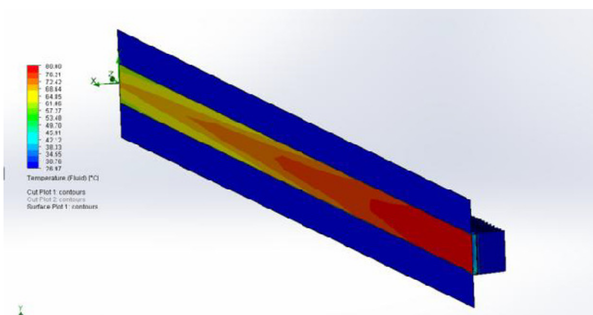

Fig.5. Analysis result of copper radiator

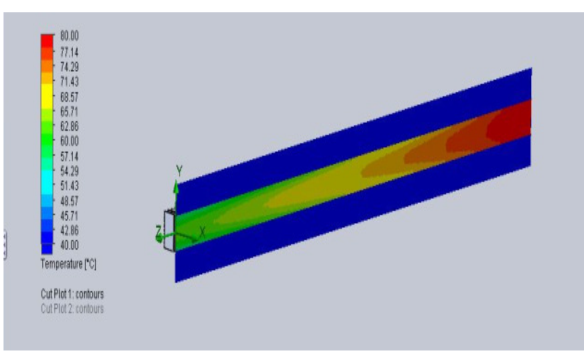

Fig.6. Analysis result of Aluminium radiator

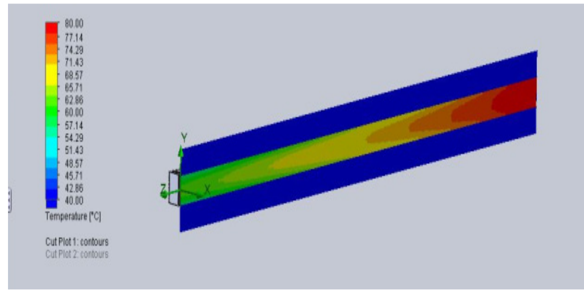

Fig.7. Analysis result of cuprobraze radiator

Temperature of the coolant at the exit of the tube is known from the contours plots of copper, aluminium and Cuprobraze radiators. It was observed that, the performance of aluminium radiator is 5% higher than copper and 3% higher than Cuprobraze radiator. Though the thermal conductivity of copper is higher than aluminium, but the rate of heat dissipation found better in aluminium radiator, because of its density.

4. Conclusion

It is inferred that, cuprobraze has an advantage of 24% lower air side pressure drop, comparing to aluminum and optimum coolant flow rate need to be used for better performance of the engine. From the analysis results, rate of heat dissipation is higher in aluminium radiator when comparing to other two radiators. Though the performance of cuprobraze radiator is marginally lower than aluminium and it can be nullified by reducing the pitch of the fins and changing the shape of fins that induces the heat transfer coefficient. Heat rejection can also be increased by a slight increase in the fin count. It can be feasible for harsh environments, heavy duty heat exchangers and it would be repaired, reused and recycled.

References

[1] Jaya kumar R.K et al, " Experimental study and CFD Analysis of copper radiator for passenger cars" International journal of Engineering Research, Vol.5 Issue 9,2016.

[2] Yadav JP and Bharat Raj Singh, "Study on Performance Evaluation of Automotive Radiator", S-JPSET , Vol. 2 Issue 2, 2011.

[3] Patel J. R, Mavani A. M., "Review paper on CFD Automobile Radiator to improve its Thermal Analysis of Automobile radiator", IJSRD, 2014.

[4] Oliet C et al, "Parametric studies on automotive radiators", Applied Thermal Engineering, 27, 2007.

[5] Zunjin Fan, et al, "Prediction of Heat Dissipation Effect by Simulating Radiator Group of Mining Dump-truck", Applied Mechanics and Materials Vol. 217-219 (2012)

[6] Trivedi P. K et al, "Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by CFD Analysis", International Journal of Engineering and Advanced Technology" Vol.1 Issue-6, 2012.

[7] Shabtay. Y L et al, "New brazing processes using anneal resistant copper and brass alloys" materials and design25 (2004)

[8] Hilde Van Der Vyer, JacoDirker et al, "Validation of a CFD model of a three dimensional tube-in-tube heat exchanger", Third International Conference on CFD in the Mineralsand Process Industry, CSIRO, Melborne, Australia.

[9] Witryet.A et al,, "CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator", 3rdInternational conference on CFD, CSIRO, Melborne, Australia, 2003.

[10] Chen J A, Wang D F and Zheng L Z, "Experimental study of operating performance of a tube-and-fin radiator for vehicles", Proceedings of Institution of Mechanical Engineers, Republic of China, 215, 2011.

[11] Kanefsky P, et al, "A Systems Engineering Approach to Engine Cooling Design", SAE, SP-1541, 1999.

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

[13] Lee Y.L, Yong Y T, "Analysis of Engine Cooling Airflow including Non-uniformity over a Radiator", International. Journal of Vehicle Design, 24, 2000.