OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Performance Analysis of a Copper Radiator Using CFD

R. K. Barathraj[1], Dr. K. S. Amirthagadeswaran[2], M. Safraaj Salaamer[3]

[1]*P.G.Scholar, Department of Mechanical Engineering, Government College of Technology, Coimbatore, India*

[2]*Professor, Department of Mechanical Engineering, Government College of Technology, Coimbatore, India*

[3]*P.G.Scholar, Department of Mechanical Engineering, Government College of Technology, Coimbatore, India*

Abstract: *The thermal performance of an automotive radiator plays an important role in the performance of an automobile's cooling system and all other associated systems. In recent decades, the conventional Copper radiators are being replaced by Aluminium radiators for its lower cost and lesser weight despite significant drop in performance. It has been scientifically proved that copper radiators are better than aluminium. The copper radiator can be made more effective by modifying the geometry suitably. In this study, an attempt has been made to study the performance of a Copper radiator using Computational Fluid Dynamics (CFD). An existing Copper radiator is modelled using Solid Works and its behaviours are simulated using CFD and in future this can be used for performance analysis with some design modifications.*

Keywords: *Radiators, Thermal performance, CFD.*

I. Introduction

A radiator is a type of heat exchanger designed to transfer heat from the hot coolant that flows through it, there by maintaining the engine temperature. This is done by heat transfer from the hot coolant coming from the engine cooling water jacket, flowing into the tubes via the inlet tank. Heat is rejected to the air blown through it by the fan. The radiator plays an important role in the performance of the cooling system of an automobile and all its associated systems. It has the critical function of reducing the temperature of the passing coolant. The cooled coolant continues recirculation throughout the engine and thus removing the waste heat. Radiators are classified by the materials used for the fins and tubes of a core. Conventionally copper radiators were used for most applications. But in later 1970, Aluminium radiators were started replacing copper radiators. There is a much debate over whether a copper or an aluminium radiator is better. Each has its own pros and cons.

It has been scientifically proved that copper transfers heat better than aluminium. It is easier to repair in most cases than aluminium. The main drawback of the copper radiator is the weight difference (Aluminium is much lighter). Also it may be noticed that some white residue growing around the tubes, as a result of chemical reactions from different metals (Brass tubes, Copper header and Lead/Tin solder).

In this study, an attempt has been made to study the performance of a Copper radiator using Computational Fluid Dynamics (CFD).

II. Methodology

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 analyses reduce the development time and increases the reliability of the designs. 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 assembly of the radiator is done by using Solidworks 2013. The specifications of the radiator are given in the Table 1.

Table 1 Specifications of the Radiator

Size	462 x 380 x 55 mm
Fin material	Copper
Tube material	Red Brass
Number of tubes	124
Number of tube rows	3

Number of fins	131
Fins pitch	8 Fins Per Inch
Frontal area	0.175 m^2

The part modelling of the tube and fin is done and then the radiator core is assembled

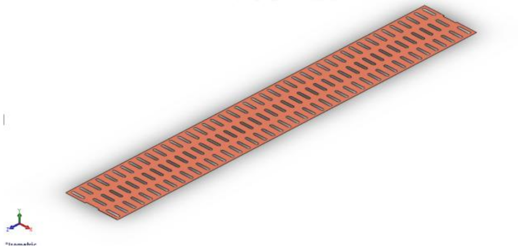

Fig 1 Part modelling of the Fin

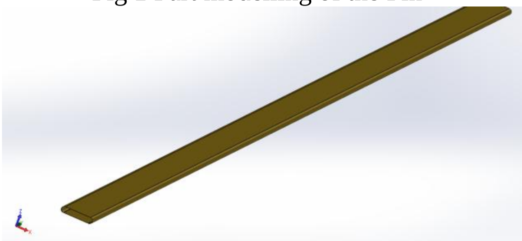

Fig 2 Part modelling of the Tube

The assembling of the fin-tube arrangement is done by using the insert and pattering command by maintaining the fin pitch of 8 per inch.

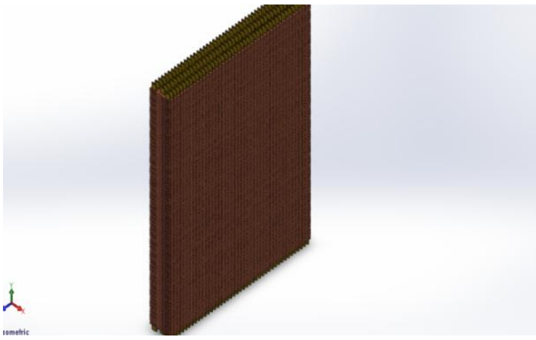

Fig 3 Radiator Core Assembly

The next important step in CFD analysis is meshing. The radiator core assembly is exported as step file and it is imported in Hypermesh 10. The geometry is cleaned up for any free edges and the connectivity is checked. In this work, only the core part is taken for numerical analysis, as the consideration of the side frame assemblies, fan shroud assemblies, header tank and bottom tank would increase the complexity of the problem. This in turn increases the computing processor capacity and computing time. Hence only the core of the radiator is taken for analysis.

As the geometry of the fin and tubes are not very much complex it was decided to use the structured hexahedral mesh for obtaining a uniform mesh throughout. Also, the element count is reduced comparatively than tetrahedral and hence the computing time. The order of accuracy of the results is more for structured hexahedral element. The volume mesh is created for the tube and air. The fin is 2D Meshed and the thickness was given.

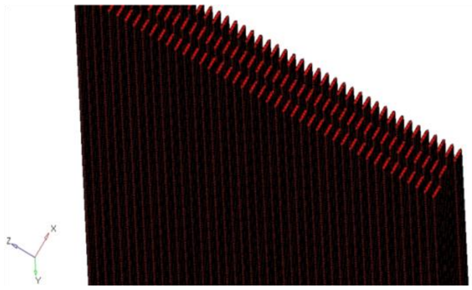

Fig 4 Water Volume Mesh (Tube)

The complete meshed model for the radiator core is shown in fig. 5

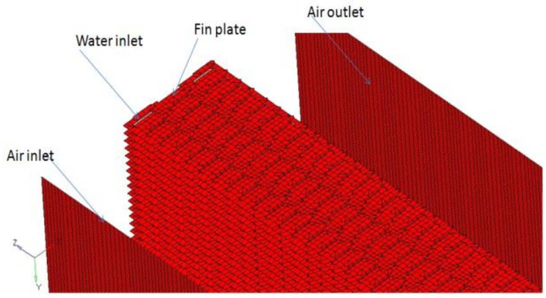

Fig 5 Meshed model of Radiator Core

The mesh count for water volume (tubes) is 308898 hexahedral elements. The volume mesh for air is created to facilitate the air flow through the fins. The mesh count for air volume is 2323809 hexahedral elements. The complete meshed model for the radiator core is shown in fig. 5. Finally the mesh file is exported as a case file.

The final step in the analysis is the solving. ANSYS Fluent 13 was used for the solving and post processing. The following assumptions were made for the numerical analysis.

1. The three dimensional pressure based solver was chosen with steady turbulent flow
2. The working medium is dry air entering the core at 30°C
3. The standard physical properties of air corresponding to 30°C is taken
4. The coolant medium is water entering the radiator at 85°C
5. The standard physical properties of water corresponding to 85°C is taken
6. The standard k-ξ viscous model with standard wall function is chosen
7. The SIMPLE scheme was adopted for Pressure – velocity coupling
8. First order upwind scheme was followed for all variables
9. Air inlet is velocity inlet, air outlet is pressure outlet
10. Water inlet is mass flow inlet and water outlet is pressure outlet

The case file exported in HyperMesh is imported in ANSYS Fluent. The case is checked for the mesh quality. The zone conditions and the boundary conditions are defined. The material properties for the tube and fin are specified. The fin material is Copper and the tube material is Red Brass.

Table 2 Material Properties of Fin – Copper Alloy

Composition	
• **Copper**	**: 99.5 % Min.**
• **Phosphorous**	**: 0.026 % Max.**
Thermal Conductivity	**394 W/m K**
Density	**8.94 g/cm^3**
Specific heat capacity	**394 J/kg K**

Table 3 Material Properties of Tube - Red Brass

Composition	
• Copper	: 85.1 % to 86 %
• Zinc	: 15.15 %
• Lead	: 0.029 %
• Iron	: 0.001 %
Thermal Conductivity	159 W/m K
Density	8.75 g/cm^3
Specific heat capacity	380 J/kg K

The boundary conditions that are given in this problem are velocity inlet for air and mass flow rate for water.

Velocity inlet for air : 6.45 m/s
Mass flow rate of water : 0.8333 kg/s (50 lpm)
Pressure condition : 1 kg/cm^2

The number of iterations to be carried out is specified and solution is initialized and then the case is solved. After the convergence of the solution, the case and data file is saved and the results can be viewed as contour plots.

III. Results And Discussion

The contour plots of temperature for both the air and water are shown in fig 6 and fig 7 respectively.

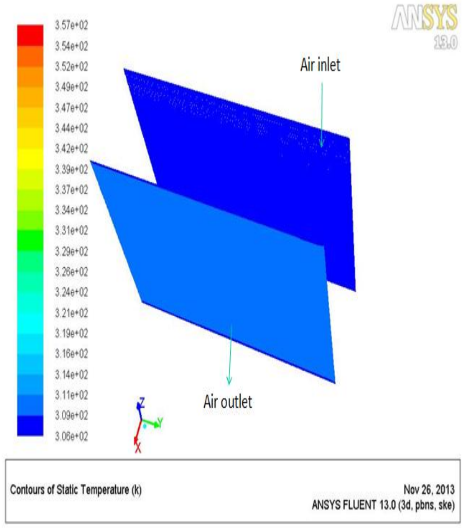

Fig 6 Temperature Contour for Air Side.

From the temperature contour of the air side the outlet temperature of air varies can be read as 42°C on average.

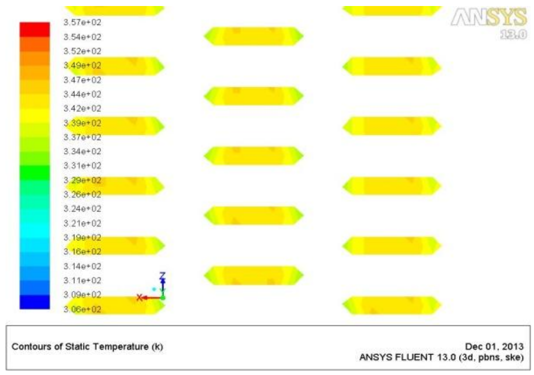

Fig 7 Temperature Contour for Water side.

From the plot, it is found that the outlet temperature of water on average can be read as 72ºC. This drop in temperature signifies the heat removed from the water.

IV. CFD Validation

The CFD results are validated with the help of experimental results. The closed type wind tunnel setup was used to test the radiator. The input conditions of flow rate and temperature remains the same for the air and water and the outlet conditions were observed.

From the experimental results, the average outlet temperature of air is 51°C and the average outlet temperature of water is 78°C.

The percentage error in prediction of outlet temperature of air and water are 7.69% and 19.61% respectively. With these reasonable deviations, the CFD model can be taken as the real replicating model.

V. Conclusion

In this study, an attempt has been made to study the behaviour of a conventional Copper radiator using Computational Fluid Dynamics (CFD). The numerical results were in reasonable agreement with the experimental results. The deviation in the numerical values can be related in the actual system to various factors [1], [2] such as friction losses, ambient surrounding conditions, loss of velocity of air at the corners of the cores, etc. Also, in numerical modelling, the various input parameters of the materials, mesh size and quality, and method of solving affects the accuracy of the solution. The factors affecting the heat transfer rate of the includes fin and tube material, geometry of the fin and tube, mass flow rate of the fluids, fin arrangements, type of fin used such as flat of serpentine or box, etc. Since the numerical model is replicating the real working model, the model can be modified to according to these factors and its performance can be studied and optimized design can be obtained. It is concluded that the numerical model can be used for further analysis with any design modifications for obtaining optimum performance of the radiator and if found better, it can incorporated in the actual system.

REFERENCES

[1] **Chavan, D.K.** and **Tasgaonkar, G.S.** (2013), "Study, Analysis And Design Of Automobile Radiator (Heat Exchanger) Proposed With Cad Drawings And Geometrical Model Of The Fan", International Journal of Mechanical and Production Engineering Research and Development, ISSN 2249-6890, Vol. 3, Issue 2, Jun 2013, pp. 137-146

[2] **Junjanina, G.C., Kulasekharan, N.** and **Purushotham H.R.** (2012), "Performance Improvement of a Louver-Finned Automobile Radiator Using Conjugate Thermal CFD Analysis", International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 8, October – 2012, ISSN: 2278-0181

[3] **Trivedi, P.K** and **Vasava, N.B.** (2012), "Study of the Effect of Mass flow Rate of Air on Heat Transfer Rate in automobile radiator by CFD simulation using CFX" International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 6, August – 2012, ISSN: 2278-0181

[4] **Witry, A., Al-Hajeri, M.H.** and **Ali A. Bondok** (2005), "Thermal Performance of automotive Aluminium plate radiator", Applied Thermal Engineering, pp. 1207-1218

[5] **Shabtay, Y.L., Ainali, M.,** and **Lea, A.** (2004), "New brazing processes using anneal-resistant copper and brass alloys", Materials and Design, Volume 25, pp. 83-89