Optimization and Performance Analysis of An Automobile Radiator Using CFD - A Review

Hardikkumar B. Patel
P G Student
Department of Mechanical Engineering (I.C & AUTO.)
Sardar Patel Institute of Technology, Piludara

Mr. Deepu Dinesan
Assistant professor
Department of Mechanical Engineering
Sardar Patel Institute of Technology, Piludara

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. For a number of years, this component has suffered from little attention with very little changing in its manufacturing cost, operation and geometry. As opposed to the old tubular heat exchangers currently form the backbone of today's process industry with their advanced performance reading levels tubular heat exchangers can only dream of. This review focuses on the various research papers regarding CFD analysis to improve automobile radiator efficiency. The aim of this paper is study about the effect Various parameters like Shape of radiator core, direction flow of working fluid, frontal area of radiator, Space between fins, space between tube, fin & tube size, coolant mass flow rate, material of fins, pitch of tube, velocity of fluid, air inlet temperature are kept in mind to improve automobile radiator efficiency and find optimized geometry from above parametric analysis.Computational fluid dynamics (CFD) programme were directed comparing the heat transfer & pressure drop of heat exchanger with different parameters for optimum performance. CFD results have high correlation level with the actual experimental results. Various results suggest that CFD have been proved very effective in reducing concept-to-production time and cost.

Keywords: Automobile Radiator, Performance parameters, CFD analysis, optimization.

I. INTRODUCTION

In an automobile, fuel and air produce power within the engine through combustion. Only a portion of the total generated power actually supplies the automobile with power -- the rest is wasted in the form of exhaust and heat. If this excess heat is not removed, the engine temperature becomes too high which results in overheating and viscosity breakdown of the lubricating oil, metal weakening of the overheated engine parts, and stress between engine parts resulting in quicker wear, among other things. A cooling system is used to remove this excess heat. Most automotive cooling systems consist of the following components: radiator, water pump, electric cooling fan, radiator pressure cap, and thermostat. Of these components, the radiator is the most prominent part of the system because it transfers heat.

As coolant travels through the engine's cylinder block, it accumulates heat. Once the coolant temperature increases above a certain threshold value, the vehicle's thermostat triggers a valve which forces the coolant to flow through the radiator. As the coolant flows through the tubes of the radiator, heat is transferred through the fins and tube walls to the air by conduction and convection.

Automobile radiator is used to cool down automotive engine. If it's not done various problems like knocking, piston deformation, cylinder deformation etc. can happen. If radiator works properly cooling system will work properly in turn engine performance will increase.

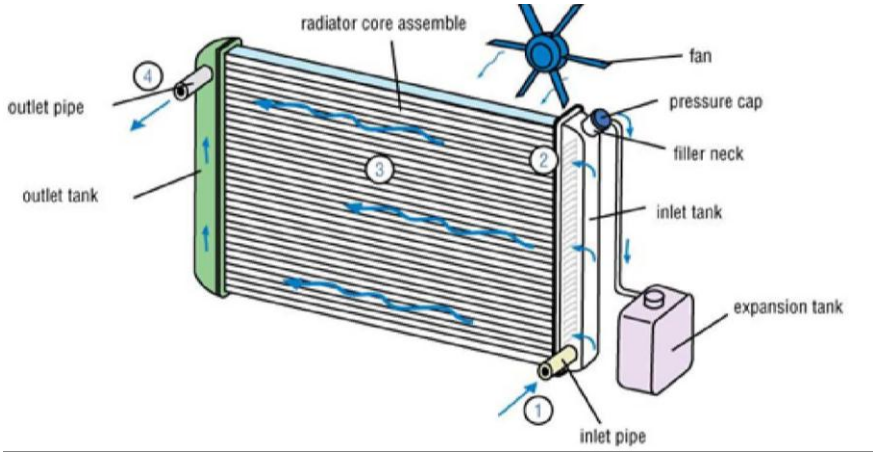

Fig. 1: Components within an Automotive Cooling System

Design of new radiator includes: radiator cover, fins, core, grills etc. which come in between the path of air flow when air flow from atmosphere through the radiator assembly.

Such parameters like; Shape of radiator core, direction flow of working fluid, frontal area of radiator, Space between fins, space between tube, fin & tube size, coolant mass flow rate, material of fins, pitch of tube, velocity of fluid, air inlet temperature are kept in mind to design a better automobile radiator.

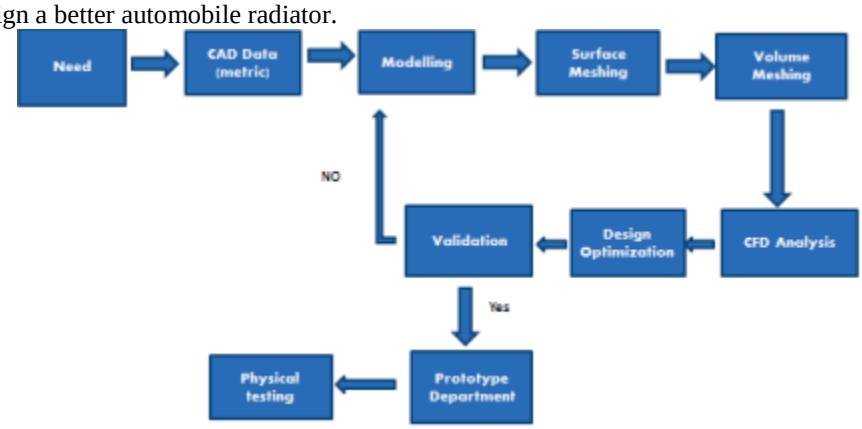

Fig. 2: Process Sequence of CFD Analysis

Role of CFD is very vital nowadays as a design tool. For CFD simulation various commercial software are available in market. Modeling is done by CAD then whole discretization model is resolved into small cells by discretization. Apply governing equation to discrete element and solve them by CFD solver. Numerical solutions are obtained regarding pressure distribution, temperature distribution, air flow distribution etc. then result is optimized and that result is validated against base data. If this model is as per our requirement its prototype will be cast and test then produce in real world application.

II. LITERATURE REVIEW ON PLATEN OF INJECTION MOLDING MACHINE

A. "Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization"
Engineering Research Center, Tata Motors Limited, Pune, India
Engineering Automation Group, Tata Technologies Limited, Pune, India

Salvio chacko, Dr. Biswadeep Shome, Vinod Kumar, A.K. Agarwal, D.R. Katkar had designed a radiator cover to increase the radiator efficiency by air flow optimization. They started with CFD model of baseline model and it was validated against test data. It suggested some good design of radiator cover which was described in four case models. Use CFD made process easy and also completed in only four iterations. CAD data were imported from AVE (Advanced Vehicle Engineering) and clean up using ANSA and surface mesh generation using ANSA then volume mesh generation using TGRID. CFD analysis was done by FLUENT and design optimization by ANSA level. Final optimized design CAD data were sent to AVE for validation. This process went up to four iterations. In fourth iteration they get optimized data and its prototype was developed then its physical testing was done. In fourth iteration the hot air recirculation was reduced to maximum extent which result into increase of average velocity through radiator core from 4.2m/s to 5.6m/s that is 34% against baseline case. But CFD model was done on many assumptions which restrict its applicability to more generalized cases. Assumptions like steady turbulent flow, incompressible fluid, dry air as working medium and physical properties at 340C, interior details of radiator core are neglected and assumed porous medium, radiator walls are thermally insulated and adiabatic. So further work can done by removing some assumptions and make it applicable to more general case.

B. "CFD analysis of fluid & heat transfer in patterned roll-bonded aluminium plate radiators", 3rd International conference on CFD, CSIRO, Melbourne, Australia, pp. 12-19.

A.Witry, M.H. Al-Hajeri and Ali A. Bondac state that for the internal flow, heat transfer augmentation caused by the repetitive impingement against the dimple obstructions renders such geometries equal to those of aerospace industry pin-fin whilst lowering pressure drops due to the wider cross-sectional areas. For the external flows, the wider and wavy nature of the surface area increases heat transfer leaving the addition of extra surface roughness add-ons as an option.

C. "Experimental study of operating performance of a tube-and-fin radiator for vehicles" , Proceedings of Institution of Mechanical Engineers, Republic of China, 215: pp. 2-8.

J A Chen, D F Wang and L Z Zheng described that the radiator's working performance including heat dissipation rate, coolant pressure drop and air pressure drop mainly depend on it's operating parameters such as the inlet coolant temperature, coolant flow volume rate and air velocity under given structural parameters and geometry. In order to investigate the effects of inlet coolant temperature, coolant volume flow rate and cooling air velocity on heat performances of the radiator, the experimental scheme was designed with the ternary quadratic form polynomial regression combinatorial design method. On the basis of the experimental results, it was concluded that the heat performances of the radiator clearly depended on its operating parameters. In the range of given experimental parameters the following conclusions could be obtained:

(1) The heat dissipation rate of the radiator grew as the inlet coolant temperature, coolant volume flow rate and cooling air velocity increased.
(2) The results also revealed that the air pressure drop and the coolant pressure drop were directly proportional to the air velocity and the coolant volume flow rate respectively. Also, the air pressure drop of the radiator clearly the radiator had little to do with the inlet coolant temperature and volume flow rate. Meanwhile, the inlet coolant temperature and the air velocity hardly affected the coolant pressure drop of the radiator.
(3) Using the optimization technique of the experiment in the wind tunnel test of the radiator to investigate the effects of the operating parameters on the working performances of radiator was feasible. It was also found that this method had some advantages in decreasing the experimental workload and saving experimental costs.

D. "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) ISSN: 2249 – 8958, Volume-1, Issue-6, 2012.

P. K. Trivedi, N. B. Vasava illustrated the effect of Tube pitch for best configured radiator for optimum performance. Heat transfer increases as the surface area of the radiator assembly is increased. This leads to change 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 Shows that as the pitch of tube is either decreased or increased than optimum pitch of tubes, the heat transfer rate decreases.

E. "Parametric studies on automotive radiators", Applied Thermal Engineering, 27, 2007

C. Oliet, A. Oliva, J. Castro and C.D. Pe´rez-Segarra 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 strongly affected by air and coolant mass flow rate. As air and coolant flow increases cooling capacity also increases. the effect of various parameters on cooling capacity as in Fig.2

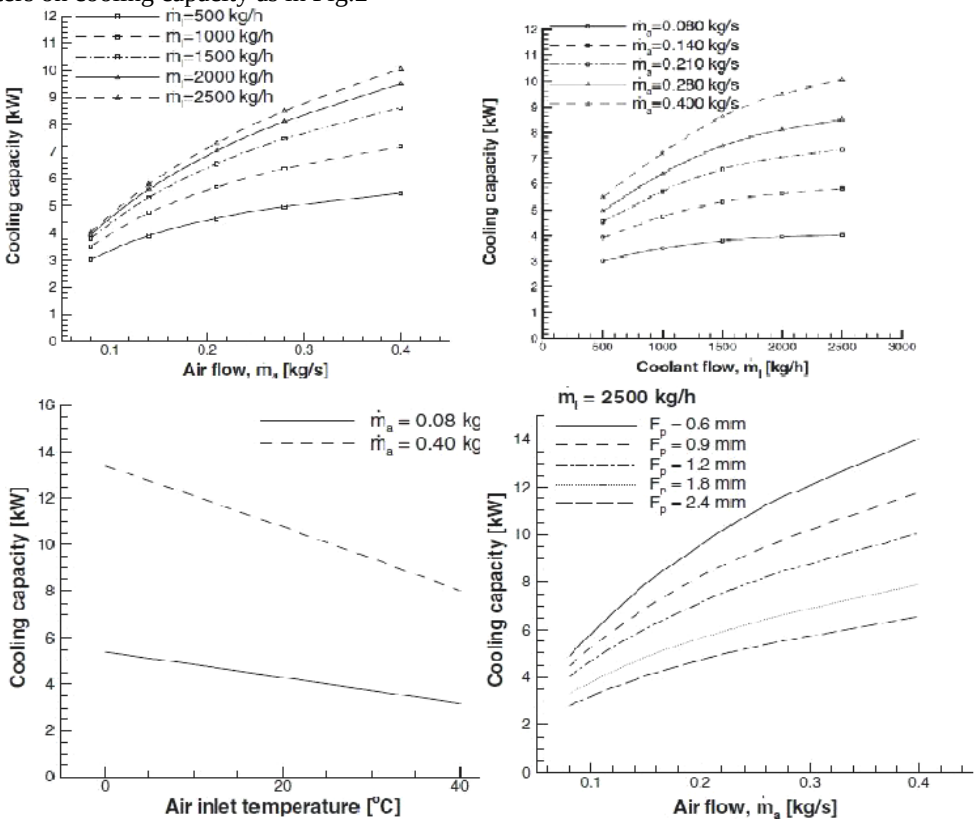

Fig. 3: Effect of Air Flow, Coolant Flow, Air Inlet Temperature and Fin Density (Affecting Air Flow) On Cooling Capacity

III. CONCLUSION

As per review of research papers, we can see that Such parameters like; Shape of radiator core, direction flow of working fluid, frontal area of radiator, Space between fins, space between tube, fin & tube size, coolant mass flow rate, material of fins, pitch of tube, velocity of fluid, air inlet temperature are kept in mind to design a better automobile radiator. Using CFD we were directed comparing the heat transfer & pressure drop of heat exchanger with different parameters for optimum performance And CFD analysis has also reduced the cost & time in design and development of radiator as compared to conventional methods.

REFERENCES

[1] Salvio chacko, Dr. Biswadeep Shome, Vinod Kumar, A.K. Agarwal,D.R. Katkar, "Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization" Engineering Research Center, Tata Motors Limited, Pune, India.

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

[3] J A Chen, D F Wang and L Z Zheng, 2001, "Experimental study of operating performance of a tube-and-fin radiator for vehicles", Proceedings of Institution of Mechanical Engineers, Republic of China, 215: pp. 2-8.

[4] 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) ISSN: 2249 – 8958, Volume-1, Issue-6, 2012

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