RESEARCH ARTICLE OPEN ACCESS

CFD Analysis of Automobile Radiator – A Review

[1]Upendra Kulshrestha*, [2]Gaurav Kumar [3]Manu Augustine and [4]Sanjay Mittal
[1] Ph.D. research scholar, [2] M.Tech. scholar, [3]Assistant Professor,[4] Lecturer [1,2,3]Department of Automobile Engineering,
School of Engineering & tech., Manipal University, Jaipur, Rajasthan, India.
[4]BSF Polytechnic College Tekanpur Gwalior M.P. India
*Corresponding Author: upendra_78@yahoo.co.in

ABSTRACT
This review focuses on the various research papers regarding CFD analysis to improve automobile radiator efficiency. Different research papers have applied different methodology and different tools for modeling, meshing and numerical solution. Various results suggest that CFD have been proved very effective in reducing concept-to-production time and cost. CFD results have high correlation level with the actual experimental results.

Index Terms: CFD computational fluid dynamics, Automobile radiator, turbulence models

I. Introduction

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.

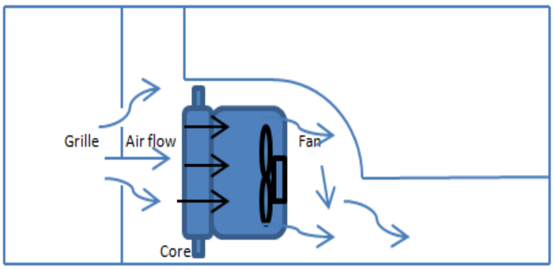

Fig. 1 Automobile radiator

Design of new radiator includes: air flow optimization as it's a very important criteria in convective heat transfer by designing various panels (radiator cover, fins, core, grills etc.) which come in between the path of air flow when air flow from atmosphere through the radiator assembly hence effect the amount of air which can be made to flow through it, design fan blades to suck more air through it by creating more suction, air flow Optimization by reducing air recirculation and air leakage through radiator core, design of radiator core in circular shape, overall size of radiator core, direction of flow of working fluid in vertical, horizontal or radially outward etc. wedge shaped frontal area of radiator, space between fins and space between tube, fin and tubes size, number of tubes ,shape of fins and tubes, coolant mass flow rate, material of fins, tube and panel on which their physical and thermal properties are depended, air inlet temperature which can be reduced by installing intercooler in front of radiator core, These are some possible design parameter which can be kept in mind to design a better automobile radiator.

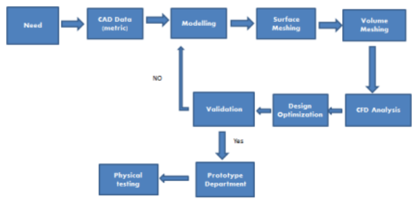

Fig. 2 Process Sequence of CFD Analysis

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

[1]**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 34^0C, 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.

[2]**Omprakash Pal, Rajesh Joshi, and Sanjay T. Purkar** also designed radiator cover or under-hood by different methodology and different tools to reduce hot air recirculation in radiator cover by its better design options and IRFM sealing around radiator in new vehicle for air flow optimization to increase radiator efficiency. Computational domain of under-hood portion for vehicle was done using HYPERMESH 11. Analysis was done for minimum speed at maximum power and torque conditions by CFD solver ACUSOLVE 1.8a which was FEM based. Forced convection was considered for cold simulation for baseline model to analyses flow and velocity distribution and results obtained were modified to increase flow characteristic and heat dissipations. Boundary conditions to virtual wind tunnel domain or computational domain: front face- constant velocity which is equal to velocity of vehicle of interest, bottom face- no slip boundary condition which represent road surface, side and top faces- zero shear boundary condition to prevent boundary layer growth, rear face- zero gradient along flow direction, under-hood vehicle idle condition- equal atmospheric pressure at both front and rear face and for other faces it remained unchanged. Non conformal meshing technique was used for radiator modeling and mesh generation. Radiator model- ungrouped macro based model and fix inlet temperature. input condition- 1-D KULI software for computing heat rejection, outlet temperature of coolant and charged air. Under simulation was done for maximum power and torque situation when cooling load is maximum. Steady flow is assumed and RAN's one equation turbulence (Spalart-Allmaras) is used to model turbulence.

IRFM sealing was applied to intercooler and radiator which resulted in improved flow, reduced recirculation and elimination of leakage around them. Average velocity of air and outlet temperature of coolant both increased as compared to baseline model and above 80% correlation was obtained against test data by CFD analysis.

[3]**Chavan D.K. and Tasgaonkar G.S.** proposed to have circular radiator core where all conventional are in either square or rectangle shaped which have following disadvantages; fan deliver air flow in circular shape so its air distribution is not uniform over the entire core less at corners and almost zero at center along axial direction. For CFD analysis, model of radiator and the fan was made in CATIA V5 and then exported to CFD analysis software. CFD model had following new specifications: no material at center area which is equivalent to fan's hub area, design of tubes and the fins are so arranged that the outlet air had nearly constant velocity, fins of varying depth, maximum at the outer periphery which reduces along the inner periphery. Various design data were obtained by CFD analysis. With the help of these data by LMTD and NTU method effectiveness of radiator was calculated assuming cross flow with both fluids unmixed. Following conclusions were made: compact design, less material hence less cost, more efficient, less power consumed by fan.

However this model was having some limitations like bending of tubes in circular shape which increase loss of head in flow, inserting of fins in circular shaped tubes is itself a problem and dies needed to be manufactured for circular shaped radiator which increase its production cost.

[4]**Bengt Sunden** determined the performance of compact heat exchanger in which he also took example of automobile radiator. He used engineering methodology based on thermal balances and correlations and CFD methods based on the finite control volume approach. In engineering approaches he mentioned about LMTD method, NTU method, use correlations for finding heat transfer coefficients, calculation of pressure drops by: frictional losses, from area change, due to acceleration of fluid, inlet, outlet losses due to turning of fluid at curved which result in centrifugal forces on fluid particles. In CFD method he explained in brief about various turbulence models: zero equation, one equation, two equation, Reynolds stress, algebraic stress and large eddy simulation models. Then he explained general

concepts about FVM method, boundary conditions and about discretization. Automotive radiator have flat tubes with louvers on inner surface to introduced low Reynolds number turbulence and a swirling motion which result in increase in heat transfer with increase in pressure drop also as compared to smooth duct. For simulation low Reynolds number k-e model was used. The copper fins are brazed to the brass tubes due to which alloyed zone may be produced near the joint. This joint has different mechanical and thermal properties than the base material i.e. copper and brass. Air gaps may also appeared at joint. With the help of CFD temperature distribution was analyzed for the case of air at joint and it was concluded that although temperature difference between root and tip is not but local variation is high. The effect on fin efficiency was not large but increases as the convective heat transfer coefficient increases. By CFD analysis it was found that the temperature at inlet manifold is uniform. Below the inlet flow rate is high so temperature drop is small. Greater temperature change was found at right and left edges where flow rate is small. By optimized design with the help of CFD we can reduce this non uniformity in temperature distribution.

III. Conclusion And Future Scope

CFD analysis has reduced the cost, time in design and development of radiator as compared to conventional methods. It also reduces the need of prototype during design process while we do iterations to get optimized design. Now we need to make prototype of the optimized design only for physical testing. In past years we have seen very increasing trend in the use of CFD in many fields over worldwide and in India also. Its effect is reaching in Indian universities at very hopeful rate and scholars are showing very interest in it. Today in India over 500 scholars are doing work regarding CFD and numbers are increasing. However the need of high processing capabilities computer and generation of codes is still a big problem in India. Students may have good grip on the required mathematics and fluid mechanics concepts but they lack knowledge regarding computer languages to generate codes. So we feel before increasing scope in CFD at graduation course, first we should focus on computer programming course in engineering colleges.

IV. 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] Omprakash Pal, Rajesh Joshi, and Sanjay T. Purkar, VSRD International Journal of Mechanical, Civil, Automobile and Production Engineering, Vol.III Issue X October 2013 e-ISSN:2249-8303, P-ISSN: 2319-2208

[3] Chavan D.K., and Tasgaonkar G.S., International Journal of Mechanical and Production Engineering Research and Development, ISSN 2249-6890, Vol. 3, Issue 2,June 2013, 137-146,

[4] BengtSunden,DepartmentofEnergySciences,Heat Transfer,LundUNniversity,Lund,Sweden,InternationalJournalofNumericalMethodsforHeatandFluidFlow,Vol20No.5,2010,pp551-569

[5] Chen Sun, *(*2004),*"*Fast Beamforming of Electronically Steerable Parasitic Array Radiator Antennas: Theory and Experiment" IEEE transactions on antennas and propagation, vol. 52.

[6] JP Yadavand ,Bharat Raj Singh,(2011) "Study on Performance Evaluation of Automotive Radiator", S-JPSET :ISSN : 2229-7111, Vol. 2, Issue 2.

[7] E. H. Twizell and N. J. Bright, "Numerical Modelling of Fan Performance," in Applied Mathematical Modelling, Vol. 5, 1981.

[8] J. R. Bredell, D. G. Kroger, and G. D. Thiart, "Numerical Investigation of Fan Performance in a Forced Draft Air-Cooled Steam Condenser," inApplied Thermal Engineering, Vol. 26, pp. 846–852, 2005.

[9] J.P.Holman, 2002, Heat transfer, Tata-McGraw-Hill Publications.

[10] Incropera, F.P.; and DeWitt, D.P. (2002). Fundamentals Of heat and mass Transfer. (5th Ed.), Wiley, New York.