Review Paper on CFD Analysis of Automobile Radiator to Improve its Thermal Efficiency

J. R. Patel[1] A. M. Mavani[2]
[1]PG Student(Thermal) [2]Associate Professor
[1, 2] Mechanical Engineering Department
[1, 2]LDRP-ITR, Gandhinagar, Gujarat, India

Abstract—Radiators are heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. Radiators are used for cooling internal combustion engines, mainly in automobiles but also in piston-engine aircraft, railway locomotives, stationary generating plant or any similar use of such an engine. The flow behavior & temperature profile prediction in the radiator tubes are very useful information & is of great importance to the designer. For the preliminary design, the performance of the radiator can be accessed through Computational Fluid Dynamics (CFD). CFD is a science that can be helpful for studying fluid flow, heat transfer, chemical reactions etc by solving mathematical equations with the help of numerical analysis. CFD employs a very simple principle of resolving the entire system in small cells or grids and applying governing equations on these discrete elements to find numerical solutions regarding pressure distribution temperature gradients, flow parameters and the like in a shorter time at a lower cost because of reduced required experimental work. In this paper, research papers has been studied and finalized that the various coolants (Nanofluids) affects efficiency of radiator at various mass flow rate. The CFD gives the exact results to study the effect of mass flow rate, pitch of tubes and coolants (Nanofluids). The coolants properties play significant roles in the improvement of radiator performance.

Keywords: Radiator, Heat transfer, Nanofluids, CFD

I. INTRODUCTION

Automobile Radiator are becoming highly power-packed with increasing power to weight or volume ratio. Increased demand on power packed radiators, which can dissipate maximum amount of heat for any given space. For over 70 year now, small change has occurred to the design and manufacturing of automobile radiator. However, cheaper and better advanced designs of radiators are available to industries. Using computational fluid dynamics (CFD) to model the flow of fluid and heat transfer performance characteristics for one such optimum design as a possible replacement for 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, mass flow rate of air, pitch of tube and coolants are analyze successfully carried out using numerical simulation built in commercial software ansys12.1. In this research paper , coolants used as Nanofluids like cuo/water, water/ethyl one glycol for better radiator performance using Computational Fluid Dynamics (CFD). Nanofluids give higher heat transfer rate than base fluids. This study is CFD simulation of the mass flow rate; fluid flow and heat transfer of shell and tube heat exchanger (radiator) at various coolants (Nanofluids). In the literature research, CFD used to simulate flow and heat transfer in tube and fin heat exchanger. All of them used the fluent CFD programme and were directed at comparing the heat transfer and pressure drop of heat exchanger with different geometrical parameters. CFD analysis gives accurate and exact result.

II. LITERATURE REVIEW

A. Research report belong coolants (Nanofluids)

In this research, the overall heat transfer coefficient of CuO/water nanofluids is investigated experimentally under laminar flow regime (100 > Re >1000) in a car radiator. The overall heat transfer coefficient decreases with increasing the nano fluid inlet temperature from 50º C to 80º C. The addition of nanofluid increases the overall heat transfer coefficient up to 8% at nanofluid concentration of 0.4 vol. % in comparison with the base fluid. Fig. 2.1 shows the overall heat transfer coefficient with nanofluid as a function of nanofluid flow rate at constant air flow rate and the nano particle concentration.

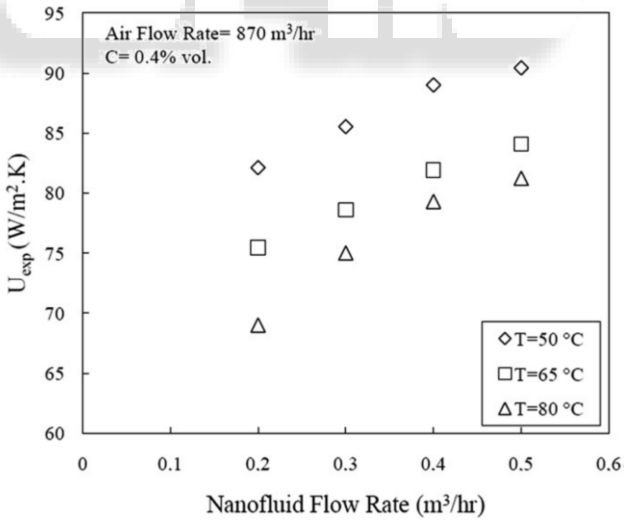

Fig. 1: Effect of nanofluid volumetric flow rate on the overall heat transfer coefficient of CuO/water nanofluid in the radiator.

It is shown that the overall heat transfer coefficient significantly increases with increasing flow rate of nanofluid. For example, at the air flow rate of 870m³/h, nanofluid concentration of 0.4 vol.%, and nanofluid inlet temperature of 80°C, for large particles. The maximum value of the overall heat transfer coefficient with nanofluid, effect of each operating parameter on the overall heat transfer coefficient and the optimum values of each parameter are analyzed by CFD simulation.

M. Nasiri et al. [4] experimented heat transfer of nanofluid through annular duct. The nanofluids were Al2O3 and TiO2 with water as the base fluid. The range of the Reynolds number for both the nanofluids were 4000 and 13000. The volume concentration for both fluids was 0.1, 0.5, 1.0, and 1.5% of Al2O3 and TiO2. Both nanofluids shows higher Nusselt number than those of the base fluids and enhancement increases with the particle concentration. At Peclet number about 24400, the enhancement of Nusselt number for Al2O3/H2O nanofluid with concentration of 0.1%, 0.5%, 1.0%, 1.5% are 2.2%, 9%, 17% and 23.8% respectively. For TiO2/H2O nanofluid, at Peclet number 53200 the increment in the Nusselt number with particle concentration of 0.1, 0.5, 1.0, and 1.5% are 1%, 2%, 5.1%, and 10.1%. Relative enhancement in the heat transfer coefficient is increased by increasing in the nano particle concentration for both nanofluids. This may be due to thermal conductivity of the nanofluid, the presence of the Brownian motion, nano particle migration in nanofluid, possible slip at the wall, and thinner boundary layer thickness. Comparison shows similar properties for both nanofluids with the particle concentration are same. This can be related to the higher thermal conductivity and lower particle size of Al2O3 nano particles in Al2O3-water nanofluid.

Javad Bayat and Amir Hossein Niksereht [5] studied numerically the thermal performance and the pressure drop of nanofluids in turbulent forced convection. The study involves the axisymmetric steady, forced turbulent convective flow of nanofluid through the circular tube having diameter D=1 cm and length L=1m, by mathematical modeling. The set of coupled non-linear Navier-Stoke differential equations have been discretized using finite –volume technique. The experimentation was performed on water/ Ethylene Glycol (60:40) by mass with Al2O3 for wide range of Reynolds number of 104<Re<105 and constant wall heat flux condition. The result shows that the volume fraction has great impact on heat transfer, pressure drop, Prandtl number, pumping power. There is large difference in the pressure drop by using the nanofluid and the base fluid for same pumping power. Nanofluids provide higher thermal enhancement at higher Reynolds number but not recommended for the practical application in the turbulent regimes, as pumping power is considerable.

B. Research report belong to CFD Analysis

Salvio Chacko et al [5] used the concept that the efficiency of the vehicle cooling system strongly depends on the air flow through the radiator core. A clear understanding of the flow pattern inside the radiator cover is essential for optimizing the radiator cover shape to increase the flow through the radiator core, thereby increasing the thermal efficiency of the radiator. The CFD analysis was conducted using the commercial software FLUENT™, while the surface and volume mesh were generated using ANSA™ and TGRID™, respectively. Use of ANSA™ for surface meshing was instrumental in reducing the CFD cycle time. CFD analysis of the baseline design that was validated against test data showed that significant regions of re-circulating flow existed inside the radiator cover. This recirculation reduced the flow through the radiator core, leading to a build-up of hot air pockets close to the radiator surface and subsequent degradation of radiator thermal efficiency. The CFD enabled optimization led to radiator cover configuration that eliminated these recirculation regions and increased the flow through the radiator core by 34%. It is anticipated that this increase in radiator core flow would significantly increase the radiator thermal efficiency.

Mohd Yusoff Sulaiman et al [3] presented the computational Fluid Dynamics (CFD) modeling simulation of air flow distribution from the automotive radiator fan. The task undertaken the model the geometries of the fan and its surroundings is the first important step.

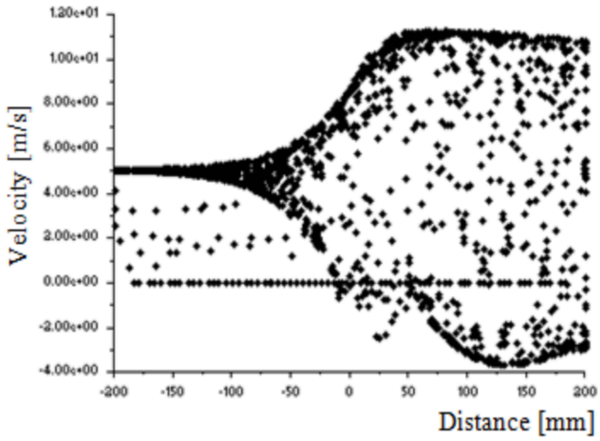

Fig. 2: Diagram of velocities against flow distance.

The results show that the outlet air velocity is 10 m/s. The error of average outlet air velocity is 12.5 % due to dissimilarities in the tip shape of the blades. This study has shown that the CFD simulation is a useful tool in optimizing the design of the fan blade. This study has shown a simple solution to design a slightly aerodynamic shape of the fan hub.

Witry et. al. [3] had carried out CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator, in which FLUENT's segregated implicit 3- D steady solver with incompressible heat transfer, is used as the tool. In this study, shell side airflow pattern and tube side water flow pattern are studied. The authors presented the variation of overall heat transfer coefficients across the radiator ranging from 75 to 560 W/m2-K. This study established the capability of FLUENT code to handle such problems. They used the Aluminium roll bonding technique; it is possible to manufacture a wide range of heat exchanger configuration that helps augment heat transfer whilst reducing pressure drops. Witry et al (1998) showed that such plates can be used for a variety of purposes & can with stand acceptable level of fouling & added advantage with Aluminum's capability to withstand corrosion.

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 fins whilst lowering pressure drops due to the wider cross sectional areas. For external flows, the winder & wavy natural of the surface area increases heat transfer leaving the extra surface roughness add-ons optional. CFD results obtained for a patterned plate heat exchanger using the CFD code fluent show excellence levels of possible performance improvement on both sides the heat exchanger (radiator).

S. N. Sridhara et al [3] stated that the demand of small cars by the customers leads to increased demand on the power packed radiators, which can dissipate maximum amount of heat for any given space. The flow behavior and temperature profile prediction in the radiator tubes are very useful information and is of great importance to the designer. The geometry of the finned-tube heat exchanger is an intricate one and there are no analytical optimization schemes available to optimize their design, while experimental trial and error is far too time consuming. The radiator designs at present depend on the empirical methods, wherein existing experimental data is used as the thumb rules for the design process. However, for any preliminary design the performance of the radiator can be accessed through Computational Fluid Dynamics (CFD) in prior to the fabrication and testing. In the current study a tube fin arrangement of an existing radiator is analyzed for evaluating the fluid flow and heat transfer characteristics. The overall pressure, temperature and mass flow rate distribution of the coolant and air in and around the single tube-fin arrangement with 32 fins are evaluated. The fluid flow simulation is conducted using software FLUENT 6.1

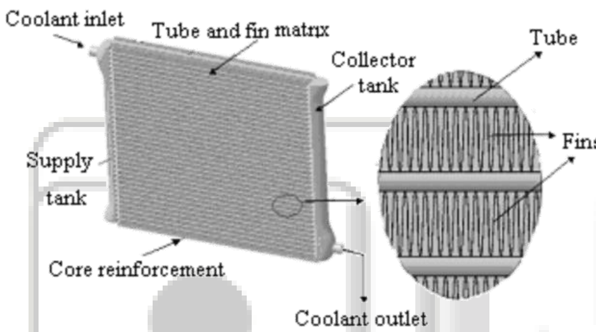

Fig. 3: Assembled CATIA model of the radiator

The pressure and temperature distribution along the tube length and tube width are presented and analyzed. The results obtained serve as good database for the future investigations.

C. Research report belong to Experimental work

Mr. Krys Bangert et al. looks at how change to a walls surface finish can affect a radiators heat transfer. A series of tests and computer simulations were run comparing the heat loss from a radiator when the wall roughness and emissivity directly behind it was changed. To gather the empirical data needed for the study an existing radiator test rig built to approximate European standard EN 442-2 was used. Thermocouples were mounted on the radiator, wall, inlet/outlet pipes and in the air gap behind the radiator, to gather temperature readings for levels of conduction, radiator and convection in the system. Three tests were carried out under steady state conditions, with a 10ºC drop across the radiator to comply with the British standard. The first test used a plain wall as a control, the second and third tests had sandpaper sheets matching the profile of a radiator attached to the wall behind the radiator. The sheets were sprayed in gloss black and silver paint respectively, to modify the surface emissivity. A computer simulation of the setup and tests was also created for comparison using the CFD (computational fluid dynamics) program fluent. The results from the experiment and model were compared in another spreadsheet. The total power output of the radiator and the thermocouple readings for the top, middle and bottom of the air gap behind the radiator were compared with the CFD analysis. The CFD readings again support the wall emissivity hypothesis but the difference in temperature increase is very small less than 1C between the plain and black walls. With the exception of some simulation results showing that higher emissivity coatings behind a radiator can improve its heat transfer.

Chen et al. made an experimental investigation of the heat transfer characteristics of a tube-and-fin radiator for vehicles using an experimental optimization design technique on a wind tunnel test rig of the radiator. The authors have developed the regression equations of heat dissipation rate, coolant pressure drop and air pressure drop. The influences of the air velocity, inlet coolant temperature and volume flow rate of coolant on heat dissipation rate, coolant pressure drop and air pressure drop have been discussed in detail by means of the numerical analyses. The results provide a basis for the theoretical analysis of heat performances and structural refinement of the tube-and-fin radiator.

III. REVIEW SUMMARY

After a comprehensive study of the existing literature it is observed that the geometric & operating parameters affects the heat transfer rate as well as performance of radiator. The fluid flow & heat transfer analysis of single tube fin Arrangement of automobile radiator is successfully carried out using numerical simulation built in commercial software FLUENT. Significant increases of the total heat transfer rates have been observed with the nano particles and its increases overall heat transfer rate. Optimizing the values of mass flow rates and the power rating of the vehicles by generating CFD codes. So many operational & geometric parameters plays an important role in the performance of radiator but the mass flow rate of air one of the operational parameter is play significant effect as the vehicle speed must be controlled by vehicle speed and it feasible to vary the parameter of mass flow rate of air. Pitch of tube for radiator is feasible to air – volume ratio constant. coolants are easily available and coolants as nano fluids give much higher heat transfer rate than base fluid. It has fluid flow heat transfer characteristics. Most of researchers have investigated on the performance of radiator by different parameter study under CFD analysis for improves its efficiency.

REFERENCES

[1] M. Naraki, S. M. Peyghambarzadeh, S. H. Hashemabadi, Y. Vermahmoudi, "Parametric study of overall heat transfer coefficient of CuO/water nanofluids in a car radiator". International Journal of Thermal Sciences 66 (2013) 82-90
[2] M. Nasiri, S. Gh. Etemad, R. Bagheri, "Experimental heat transfer of nanofluid through an annular duct", International Communication in Heat and Mass Transfer 38 (2011) 958-963.
[3] Javad Bayat, Amir Hossein Nikseresht, "Thermal performance and pressure drop analysis of nanofluids in turbulent forced convective flows", International Journal of Thermal Sciences 60 (2012) 236-243.

[4] Salvio Chacko'S., et. al., "Numerical Simulation for Improving Radiator Efficiency", Engineering Automation Group, Tata Technologies Limited, Engineering research Center, Tata Motors Limited, India, 2003

[5] Mohd Yusoff et. al., "CFD Modelling of Air Flow Distribution from a Fan", University Technical Malaysia Melaka, Malaysia, 2009.

[6] 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.

[7] S. N. Sridhara, S. R. Shankapal and V Umesh Babu, "CFD analysis of Fluid Flow & Heat Transfer in a Single Tube-Fin Arrangement of an Automotive Radiator" International Conference on Mechanical Engineering 2005, Dhaka, Bangladesh, Conference Paper, 2005

[8] Chen et al, "Experimental investigation of the heat transfer characteristics of a tube & fin radiator", Technical paper, China, 2006.

[9] Mr. Krys Bangert et al, "Radiator heat transfer augmentation by changes to wall surface roughness and emissivity, The University of Sheffield, June-August 2010.