Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 4, Issue 8, August -2017

CFD ANALYSIS OF AN AUTOMOBILE RADIATOR WITH AND WITHOUT LOUVERED FINS USING NANO FLUIDS

Yeduru Jyothi [1], M. Rajesh [2], M.Ch Naidu

[1,2 3] *Dept. Of Mechanical Engineering, Visakha Technical Campus, Visakhapatnam*

Abstract — *Radiators are heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. The majority of radiators are constructed to function in automobiles, buildings, and electronics. The radiator is always a source of heat to its environment, although this may be for either the purpose of heating this environment, or for cooling the fluid or coolant supplied to it, as for engine cooling. Despite the name, most radiators transfer the bulk of their heat via convection instead of thermal radiation. Spacecraft radiators necessarily must use radiation only to reject heat.*

A fin is a thin component or appendage attached to a larger body or structure. Fins typically function as foils that produce lift or thrust, or provide the ability to steer or stabilize motion while traveling in water, air, or other fluid media. Fins are also used to increase surface areas for heat transfer purposes, or simply as ornamentation.

In this project, we intend to develop a model of fin with louvered section. The two variations of fins are taken such that initial is a rectangular fin and the latter is a trapezoidal form. The tubular of 5 sections are developed for the fins. The models are modeled in Catia V5 R20 and analyzed in fluid dynamics computational analysis is done in Ansys 15.0 with fluent interface.

Keywords*- Fins, Heat Exchangers, Ornamentation, Radiators, tubular.*

I. INTRODUCTION

The growth of technology found in high tech industries, such as automobiles, microelectronics, transportation, and manufacturing has created a specific way for scientific advances that would have wide ranging effects on many obstacles facing today's scientific world such as system efficiency, reliability and pollution. However, many factors are underlined in the development of the automotive industries among them as one point is the ability to rapid cooling of the products. Cooling is one of the important process for maintaining and enhancing the operational performance of the system as a result caused by the increase in power and reduces in sizes and weight in future products. Thus researchers are starting to invest more specific way to efficient and effective heat transfer processes.

Continuous technological development in automotive industries has demand for high efficiency engines. A high efficiency engine is not only based on the performance of radiator but also depends on better fuel economy and less emission rate. Reducing the vehicle weight by optimizing design and size of a radiator is a capital feature. Addition of fins is one of the approaches to increase the heat transfer rate of radiator, provides greater heat transfer area and enhances the air convective heat transfer coefficient.

Radiators are heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. The majority of radiators are constructed to function in automobiles, buildings, and electronics. The radiator is always a source of heat to its environment, although this may be for either the purpose of heating this environment, or for cooling the fluid or coolant supplied to it, as for engine cooling.

II. REVIEW OF LITERATURE

J.M. Gorman et. Al. [1] in their paper discussed that Louvered fins constitute a major methodology for heat transfer enhancement. Of critical significance in evaluating the worthiness of such fins is the comparison between the heat transfer and pressure drop for a thus-finned heat exchanger with the base line case of a counterpart plain- finned heat exchanger. Up to the present, it appears that such comparisons are confined to heat exchangers in which one of the participating fluids passes through circular tubes. In another basic geometry in which louvered fins have been employed, therefore mentioned participating fluid passes through flattened tubes which are virtually rectangular in cross-section.

The focus of the present paper is to obtain results for the latter basic geometry for both louver-fin-based heat exchangers and counterpart plain-fin-based heat exchangers.

Rajashekhar Pendyala [2] in his study described that the Nano fluids are the new developed thermal fluids with enhanced thermo physical properties which can improve heat transfer performance of various applications. By introducing nanoparticles with high thermal conductivity in the car radiator coolant can enhance the effective thermal conductivity of coolant which improves the performance of cooling system. Alumina, silica and copper oxide nanoparticles with ethylene glycol-water mixture (60:40) have been used in 3-dimentional car radiator simulations to study fluid flow patterns and heat transfer performance. Heat transfer performance for ethylene glycol-water mixture based Nano fluids at different nanoparticle concentrations has been studied.

Vishwa Deepak Dwivedi [3] in his paper described that Researches in heat transfer have been carried out over the previous several decades, leading to the development of the currently used heat transfer enhancement techniques. The water and ethylene glycol as conventional coolant have been widely used in an automotive radiator in many years. With the advancement of nanotechnology, a coolant are invented which is known as "Nano fluids".

P Vijaya sagar [4] in his paper discussed that Radiators are used to transfer thermal energy from one medium to another for the purpose of cooling. Radiators are used for cooling internal combustion engines, mainly in automobiles but also in piston-engine aircraft, railway locomotives, motorcycles, stationary generating plant. The radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine.

III. MODELING OF RADIATOR FIN GEOMETRIES

The modeling of a Radiator fins is done in Creo Parametric 3.0 modeling software. The model of a Radiator fins is as shown in the Fig. 1.

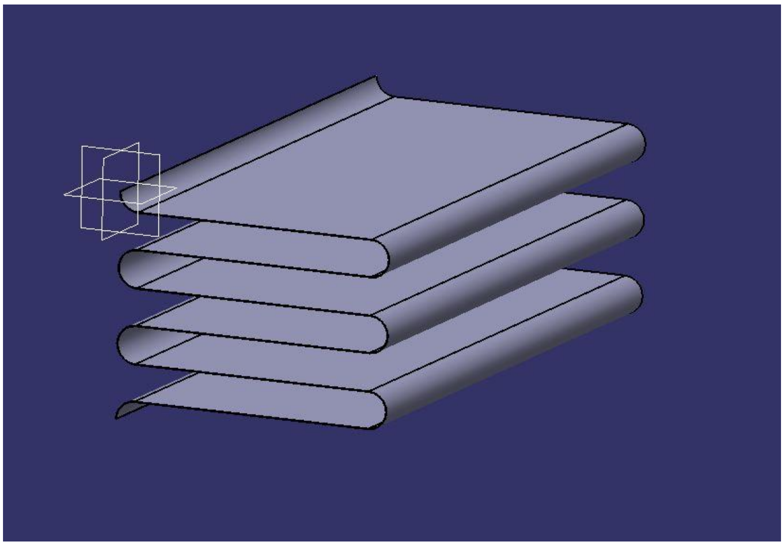


Fig. 1 Model of a Radiator fins

The drawing specifications of straight fins are as shown in the Fig. 2.

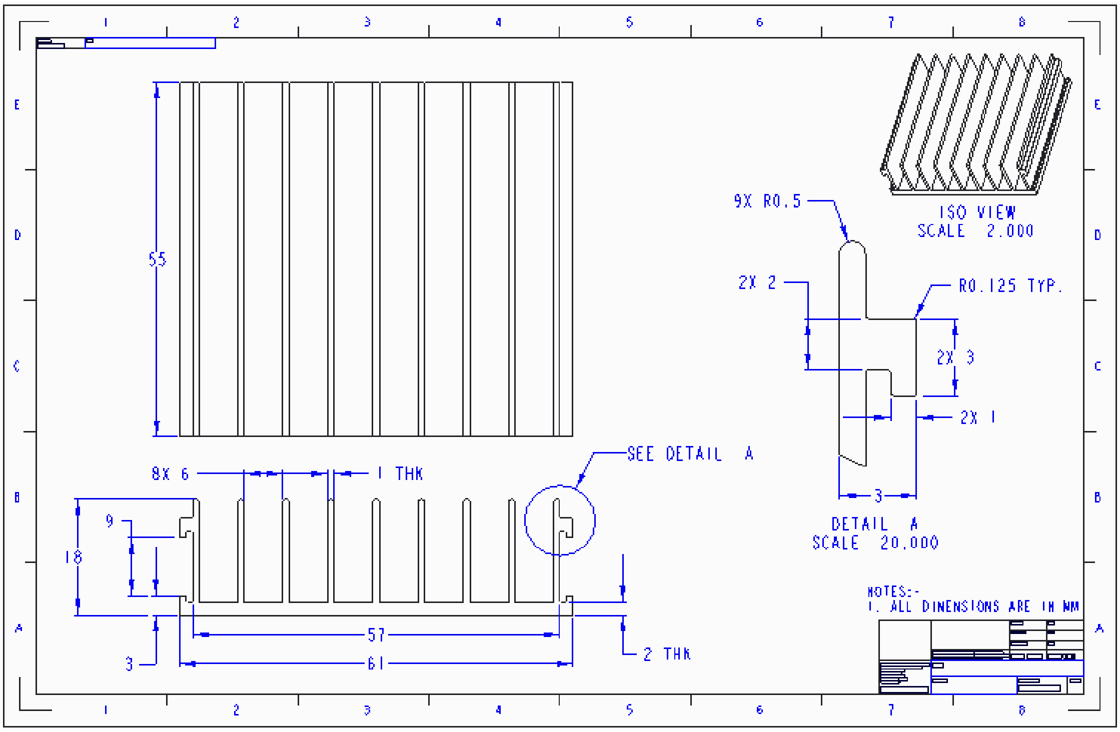


Fig. 2 Drawing Specifications of straight fins

IV. COMPUTATION ANALYSIS

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reaction (e.g., combustion), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

All the CFD codes contain three main elements. They are as follows,

- Preprocessor.
- Solver.
- Post processor

The geometry is created in ANSYS ICEM CFD as per the given data for each of the model and a domain is created to encompass the flow inside the domain to the walls of the body. In order to study domain independence, three cylindrical domains are considered in trial and error method taking the distances from nose and tail ends of the model and taking the radius from the axis of the model. Three dimensional hexahedral grids were generated to discretize the body and the domain.

Three dimensional segregated implicit solvers is used in the present analysis, the k-ω, k-ϵ turbulence models in addition to the continuity and momentum equations were used as governing equations. Boundary conditions used in the present analysis are inlet as velocity inlet, outlet as Pressure Outlet, far field, and body as walls. All the three models are computed in the solver Fluent. The solution was stopped when changes in solution variables from one iteration to the next is negligible. Solution is iterated till the convergence is observed. Then forces and moments results were extracted from it. This data is saved as the data file in the solver itself.

- Solve the momentum equations to find the velocity components (Uo, Vo).
- Solve the pressure-correction equation to find 'p' at each grid point.
- Replace the previous intermediate values of pressure and velocity with the new corrected values and return to the original step. Repeat the step until the step converges.

Geometry and Domain are created in ANSYS 15.0. Blocking and Meshing is done. Checking the mesh quality and saving the file to solver Fluent. Export it into fluent software. Computing and monitoring the solution in Fluent. Examine and save the results. The Temperature for the outlet tube side is as shown in the Fig. 3.

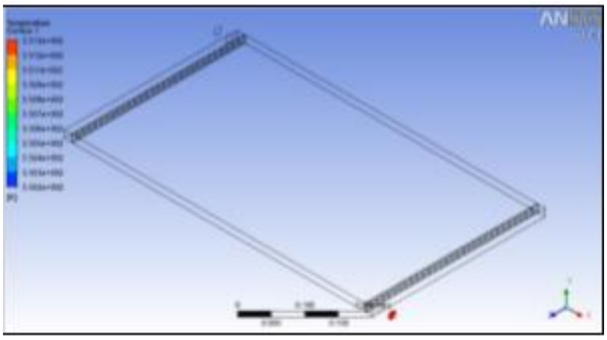
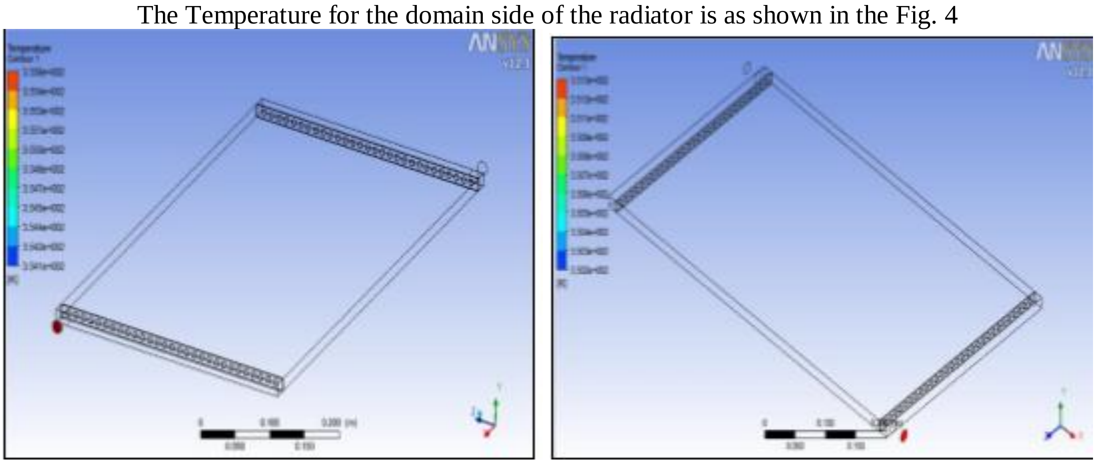


Fig. 3 Geometric model of the circular exhaust diffuser

The Temperature for the domain side of the radiator is as shown in the Fig. 4



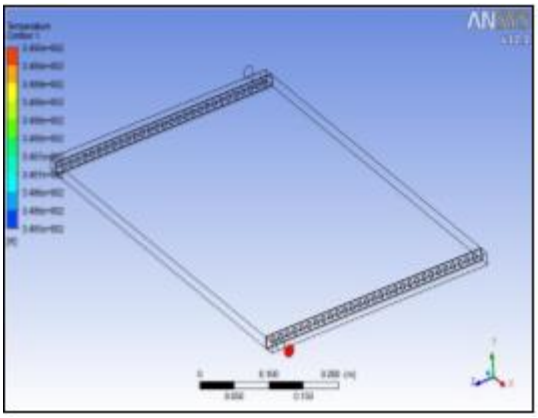


Fig. 4 Temperature for the domain side of the radiator

The Temperature Contour of the interfaced section of the radiator is as shown in the Fig. 5

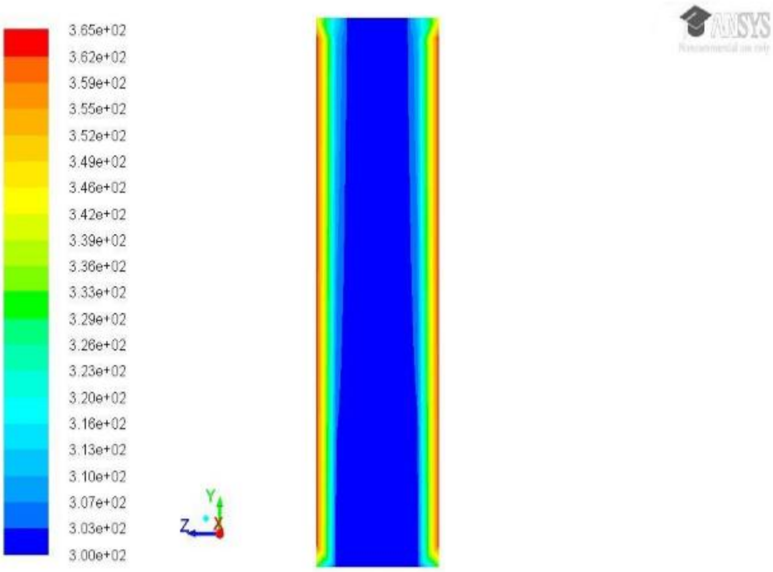


Fig. 5 Temperature Contour of the interfaced section

5.4 Effect of Tube side fluid on radiator Performance:

The Effect of Tube side fluid on Performance of the radiator is as shown in the Fig. 6.

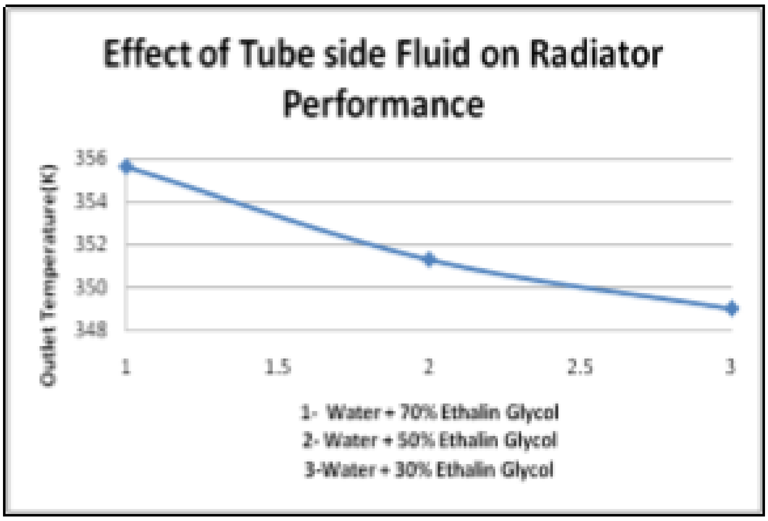


Fig. 5.4 Effect of Tube side fluid on radiator Performance

The Surface Nusselt number variation for tube 1 position of the radiator is as shown in the Fig. 7.

Tube1:

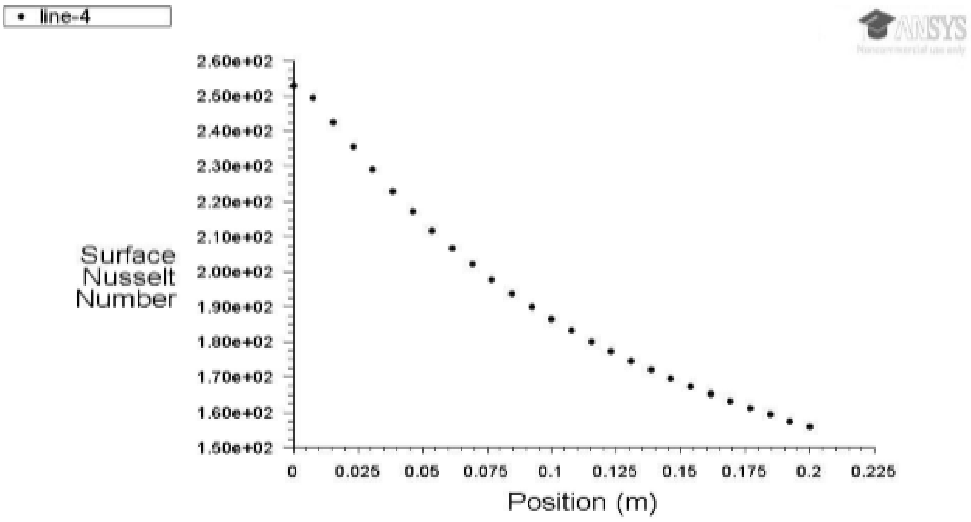


Fig. 7 Surface Nusselt number variation for tube 1 position

The tube1, tube2 and tube3 have been compared on the basis of different graphs governed from the CFD analysis and it has seen that, fin configuration in tube3 is more effective than other two tubes. The geometry of fins used in Tube2 has more restricted path for the air flow which increases the flow resistance and decreases the air flow rate and that downs the heat transfer rate. It can be seen that value of surface Nusselt number has maximum value for tube3 as compared to tube1 and tube2.

For tube3, near the bottom point of the tube, it is more than 300 which are greater than the tube2 which has nearly equal to 250. The surface heat transfer coefficient is compared at different position on the tube, and has more value for tube3, nearly equal to 9W/ -k at the lowest position of the tube, as compared to 5.5W/ -k and 4.5W/ -k for the tube1 and tube2 respectively. Heat transfer rate is 11.05 W, 12.647 W and 16.81 W respectively for the tube1, tube2 and tube3. Tube3 has maximum heat transfer rate. Hence the results showed that, for tubes having different fin configurations, the tube having ten equally spaced internal helical fins is more effective as compared to the tube without fin and tube2 which has one helical fin with large number of turns.

The Surface Heat transfer coefficient variation for tube 1 position of the radiator is as shown in the Fig. 5.10.

Tube1 :

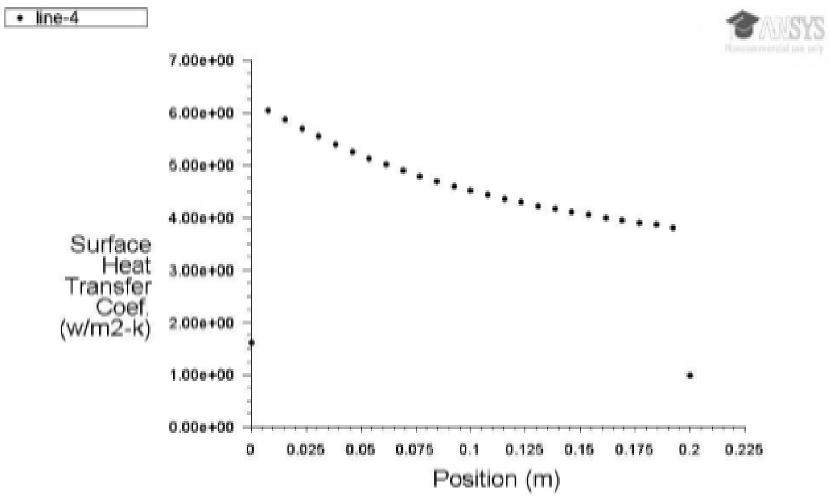


Fig. 5.10 Surface Heat transfer coefficient variation for tube 1 position

Tube3, tube4, tube5 having same fin configuration, which already had been concluded, have been compared for best fin profile. Tube3, tube4 and tube5 have rectangular, trapezoidal and concave parabolic fin profiles respectively. It has seen that at the position of 20mm from the bottom point of the tube the value of surface Nusselt number is 450 for tube3, for tube4 it is more than 600 which is greater than tube5 which has less than 600.

The value of surface heat transfer coefficient has approximately equal values for tube4 and tube5 of approximately equal to 14W/ -k as compared to tube3 of approximately equal to 10W/ -k. Heat transfer rate from tube4 is 18.244 W which is more than 17.061 W and 16.81 W for tube5 and tube3 respectively. Hence the overall performance of the fins and heat transfer rate from different fin profile has maximum value for trapezoidal fins for natural convection through internal fins for the given case.

V. CONCLUSIONS

The Analysis of louver fined automobile radiator lead to the following conclusion. The designing a radiator without louver fins and with louver fins. The original radiator has no louver fins. The computational analysis tool ANSYS is used to perform a CFD analysis on radiator. The initial parameters are Inlet air velocity, Air Inlet temperature. The analysis results, the velocity, pressure and heat transfer rate is more for the radiator with louver fins that of the original model. Heat transfer analysis is done to analyze the heat transfer rate to determine the thermal flux. The material taken is Aluminum alloy 6061 for thermal analysis. The thermal analysis results, thermal flux is more for the radiator with louver fins that of the original model, so heat transfer rate is more. So, the modifying the radiator model with louver fins yields better results. We are increasing the water in the mixture tube side outlet temperature will be decreasing in the radiator 355.6 to 349, So as the higher mixing ration of the water is desirable for achieving better performance of the radiator. If we are decreasing the tube diameter, then the tube side outlet temperature will be increasing in the radiator 343.6 to 356.1.

REFERENCES

[1] J.M. Gorman, M. Carideo, E. M. Sparrow, J. P. Abraham, “Heat transfer and pressure drop comparison of louver-and plain-finned heat exchangers where one fluid passes through flattened tubes”, Case Studies in Thermal Engineering, 122–126, 2015.

[2] Rajashekhar Pendyala, Jia Ling Chong, Suhaib Umer Iyas, “CFD Analysis of Heat Transfer Performance in a Car Radiator with Nano fluids as Coolants”, Chemical Engineering Transactions, 45, 1261-1266, 2015.

[3] Vishwa Deepak Dwivedi, Ranjeet Rai, “Design and Performance Analysis of Louvered Fin Automotive Radiator using CAE Tools”, International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 4, Issue 01, January 2015.

[4] P Vijaya Sagar, K Kiran Chand, “Thermal Analysis of an Automobile Radiator with and Without Louvered Fins”, South Asian Journal of Engineering and Technology (SAJET), 20th – 21st November, 2015.

[5] P. S. Kishore, “Performance Analysis of Automobile Radiator”, International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE), Volume: 3 Issue: 5, May 2016.

[6] K. Priyadharshini, “Finite Element Analysis of Radiator Fins to increase the Convection Efficiency of Radiator by using Al Alloy, Cu and Brass Material”, Journal of Advanced Engineering Research ISSN: 2393-8447 Volume 3, Issue 1, 2016, pp.78-82.

[7] Anant Lal , Devkant Varde, “Design & Performance Analysis Of Automobile Radiator”, Accent Journal Of Economics Ecology & Engineering, Vol.01 , Issue 04, August 2016.

[8] Syed Abuthahir.M, Vivek Sidharth.R, Swaminathan.R, Joseph Manickam. A, “Design Modification and Analysis of Helical Fins Used in Locomotive Engines Radiator”, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5, Issue 5, May 2016

[9] D.Srinivasu, D.Santharao, A.Ramakrishna, “Cfd Analysis to Predict Heat Transfer Performance of Louver Fin Radiator with Water/Eg & Al2O3 Nano Fluid”, International Journal of Science Engineering and Advance Technology, IJSEAT, Vol. 4, Issue 2 ISSN 2321-6905.

[10] S. J. Kim, D. Kim, “Forced Convection in Microstructures for Electronic Equipment Cooling”, ASME, Journal of Heat Transfer, vol. 121, 1999.

[11] D. Kim, S. J. Kim, “Compact Modeling of Fluid flow and Heat Transfer in Straight Fin Heat Sinks”, ASME, Journal of Electronic Packaging, vol. 126, 2004.

[12] T. Jeng and S. Tzeng, “A semi-empirical model for estimating permeability and inertial coefficient of pin-fin heat sinks”, International Journal of Heat and Mass transfer, vol. 48, 3140-3150, 2005.

[13] T. Jeng, S. Tezeng, Y. Hung, “An analytical study of local thermal equilibrium in porous heat sinks using fin theory”, International Journal of Heat and Mass Transfer, vol. 49, 1907-1914, 2006.

[14] F. P. Incropera, D. P. DeWitt, T. L. Bergman, A. S. Lavine, “Fundamentals of Heat and Mass Transfer“, 6th Ed., Wiley, 2007.

[15] N. Kulasekharan, H. R. Purushotham, G. C. Junjanna, “Performance improvement of a louver-finned automobile radiator using conjugate thermal CFD analysis”, International Journal of Engineering Research & Technology, 2278-0181, vol. 1 issue 8, 2012.

