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 9, September -2017

MODELING & ANALYSIS OF LOUVERED FIN RADIATOR WITH NANOFLUIDS USING CFD

Sai Kiran.S[1], Dr.M.V.A.Raju Bahubalendruni[2]

[1]PG Scholar (Thermal Engg.), Dept. Of Mechanical Engineering, GMRIT, Rajam,
[2]Associate Professor, Dept. Of Mechanical Engineering, GMRIT, Rajam,

Abstract —*There is always need for improve automobile thermal management system to attain maximum efficiency driven by the trend that the automobile powertrain system operates towards higher power output with smaller system size. The intensified heat dissipation due to the powertrain system needs more efficient cooling solutions. In automobiles for cooling internal combustion engines, the radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine. Traditionally forced convection heat transfer is performed to cool circulating fluid which consists of water or a mixture of water and anti-freezing materials like Ethylene Glycol (EG). These heat transfer fluids offer low thermal conductivity, with the advancement of nanotechnology, the new generation of heat transfer fluids called, "NANOFLUIDS" have been developed and researchers found that these fluids offer higher thermal conductivity compared to that of conventional coolants, which further improve cooling efficiency.*

In this research work, the computational analysis tool ANSYS is used to perform a CFD analysis on a radiator at different mass flow rates, normally radiator has no louvered fins; in this research work, the radiator has louvered fins. The CFD analysis will be performed for radiator with and without louvered fins. To examine the heat transfer rate using nanofluids Al_2O_3*. The different concentrations of nanofluids in the range of 0.1%, 0.3%, 0.5%, 0.7% & 1% of volume percent of base fluid (Ethylene Glycol (EG), Water, Ethylene Glycol (EG) –Water (EG:W=60:40)) will be modeled by the addition of Al2O3 nanoparticles into the coolant flows through the radiator. Relevant input data, nanofluids properties and empirical correlations were obtained from literatures to investigate the heat transfer enhancement of an automotive car radiator operated with nanofluid-based coolants. The material used for fins of radiator is Aluminium Alloy 6061. Modeling and Analysis is performed in CFD.*

Keywords*-Automobile Radiator, Louver fins, Nanofluids (*Al_2O_3*), CFD.*

I. INTRODUCTION

Heat dissipation is probably one of the most important considerations in engine design. All in all, radiator is a simple and lasting technology that will likely be around as long as we use internal combustion engines. There is always need for improve automobile thermal management system to attain maximum efficiency. The radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine. Automotive cooling devices, such as radiators, condensers and evaporators, have a significant effect on a vehicle's efficiency. Especially, a radiator exhausting heat from the engine is closely related to the engine performance and mechanical failure. To enhance efficiency of a radiator, various studies have been done on the additional devices such as a rectangular fin, a plate, a circular tube, a flat tube, an elliptic tube and a louver fin. Among various types of radiators, louver fin radiators are often used in commercial vehicles. A louver fin radiator provides a high heat transfer rate but results in a significant friction loss due to the complex coolant passage. So far, significant efforts have been made to predict the characteristics of heat transfer and flow friction in a louver fin radiator. The louver fin radiator is characterized by two geometric features: complex flow passages to enhance heat transfer and a very large difference in geometric scales between the radiator and a fin element. Because these geometric features from louvers, fins and tubes pose a great difficulty in a numerical analysis, many previous studies were done with experiments.

The numerical studies mentioned so far focused on either very small or large geometric scales in a louver fin radiator. The studies on the microscopic geometries investigated very small louver fin elements and can provide the data for a group of combined louver fins. However, they have a limitation in extending their methods to an entire louver fin radiator. In contrast, the previous studies on large geometric scales considered the entire cooling system in an automobile. However, they require either experimental or empirical correlations for the heat transfer rate and flow friction to model the louver fin region with very small geometric features. We present a numerical method with a reduced cost and sufficient accuracy to analyze a full-scale louver fin radiator for under hood thermal management in an automobile.

II. REVIEW OF LITERATURE

VahidDelavari [1] in his paper described that The present numerical study simulated turbulent and laminar flow heat transfer in nanofluids Al_2O_3particles in water and ethylene glycol-based fluid) passing through a flat tube in 3D using computational fluid dynamics (CFD) for single and two-phase approaches. The advantages over pure base fluids were evaluated. Empirical correlations were used to calculate nanofluid viscosity and thermal conductivity as a function of the volumetric concentration of the nanoparticles. First, the Nusselt numbers of the pure water and pure ethylene glycol in flat tubes were compared with the experimental data. Next, the Nusselt numbers for both approaches were compared with those for experimental data at the same Reynolds number for different concentrations of nanoparticles. A small difference in the friction factors of the tube was observed between the two approaches and the Nusselt number for the two-phase model was markedly different from that for the single-phase model; however, the volumetric flow for the same heat transfer rate decreased and less pumping power was required for the nanofluids.

Sang Hyuk Leeet. al [2] in their paper discussed that A numerical method to efficiently predict heat transfer phenomena of a louver fin radiator was presented – multi-scale semi microscopic heat exchange (SHE) method. This method consists of microscopic analysis and semi microscopic analysis. To predict heat transfer characteristics of a louver fin element, the microscopic analysis employs modeling of the detailed geometry of a fin element. Numerical models for the heat transfer rate and flow friction derived from the microscopic analysis are then used for simulations of the full radiator model in semi microscopic analysis. In the semi microscopic analysis, conjugate heat transfer is analysed for the domain with the radiator whose louver fin area is replaced by a porous media. The results with the proposed method show a good agreement with the experimental data. The proposed method can be used to predict flow and heat transfer characteristics of a realistic louver fin radiator with a reduced cost and sufficient accuracy.

III. MODELING OF RADIATOR FIN GEOMETRIES

The modeling of a Radiator fins is done in CATIA V5 R20 modeling software. The model of a Radiator fins is as shown in the Fig. Dimensions for modeling are shown below Table 1.

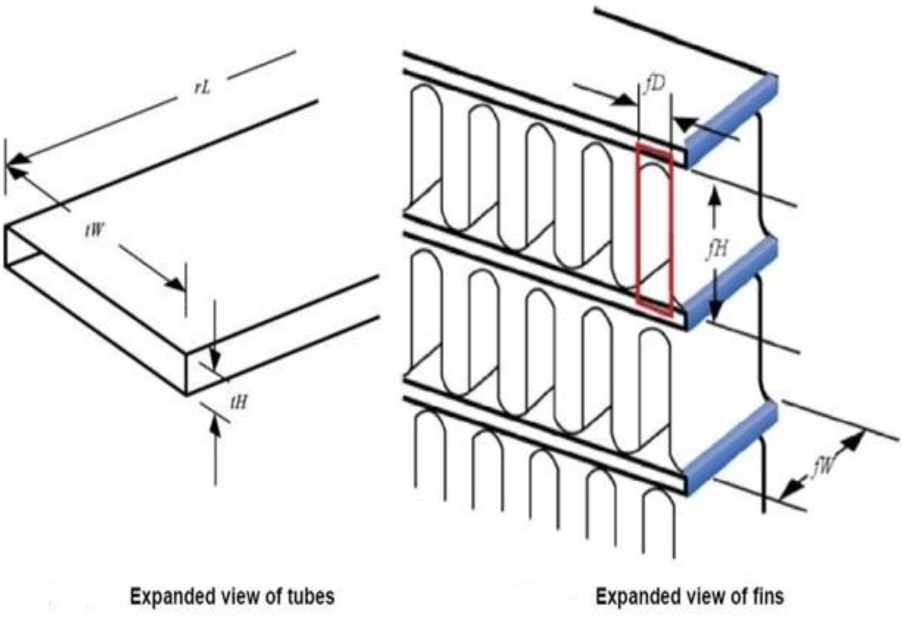


Fig.01 Layout of Radiator without Louvered Fins

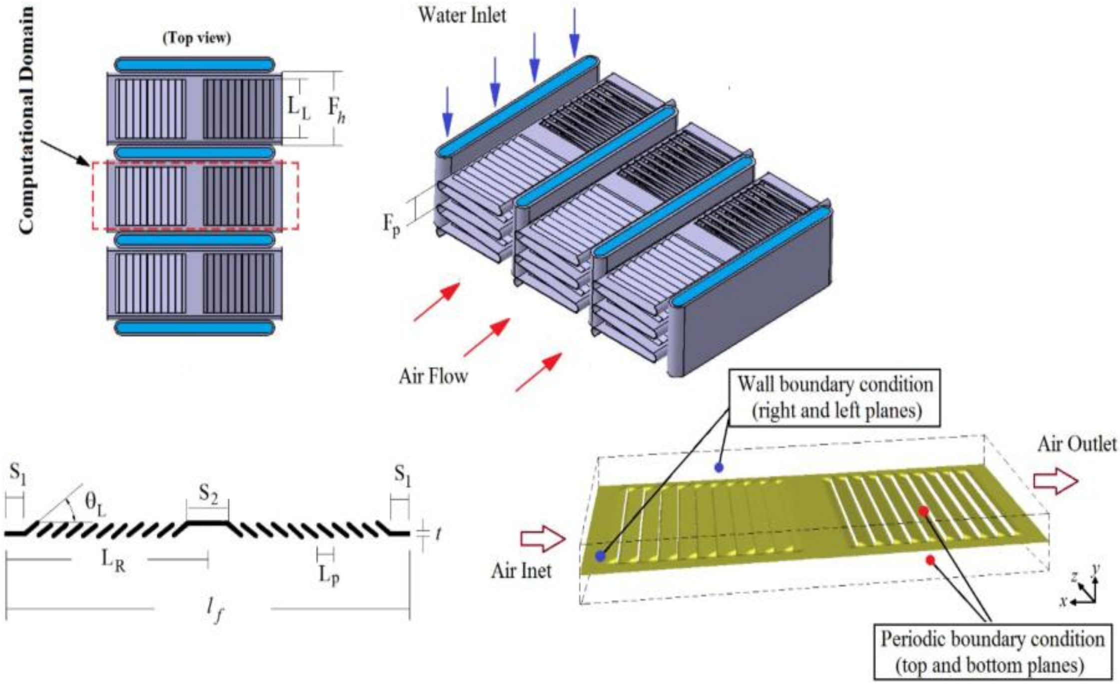


Fig.02 Layout of Radiator with Louvered Fins

Louver Angle Θ	28 degrees	Louver Pitch L_p	1.42mm
Louver Length L_i	17.18mm	Fin Thickness	0.16mm
Fin Pitch F_p	1.8mm	Fin Length F_l	19mm
Fin Depth F_d	22mm	Hydraulic Dia D_h	3.041mm
Tube Pitch T_p	24mm	Tube Depth T_d	22mm

Table 01. Geometric Parameters of Louver Fin Radiator

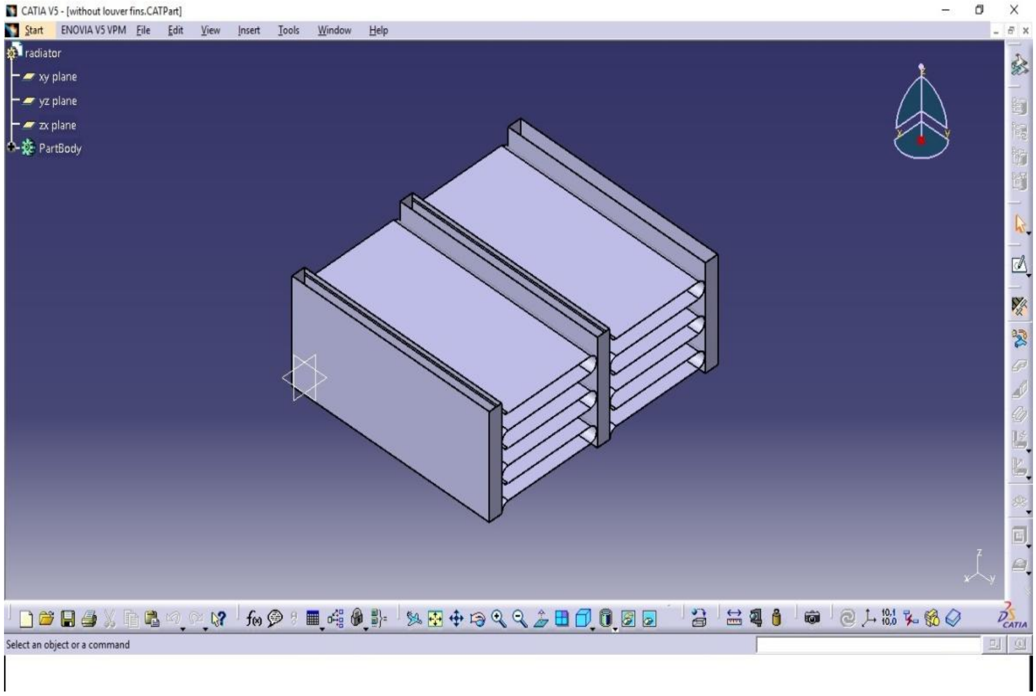


Fig. 03Model of a Radiator fins

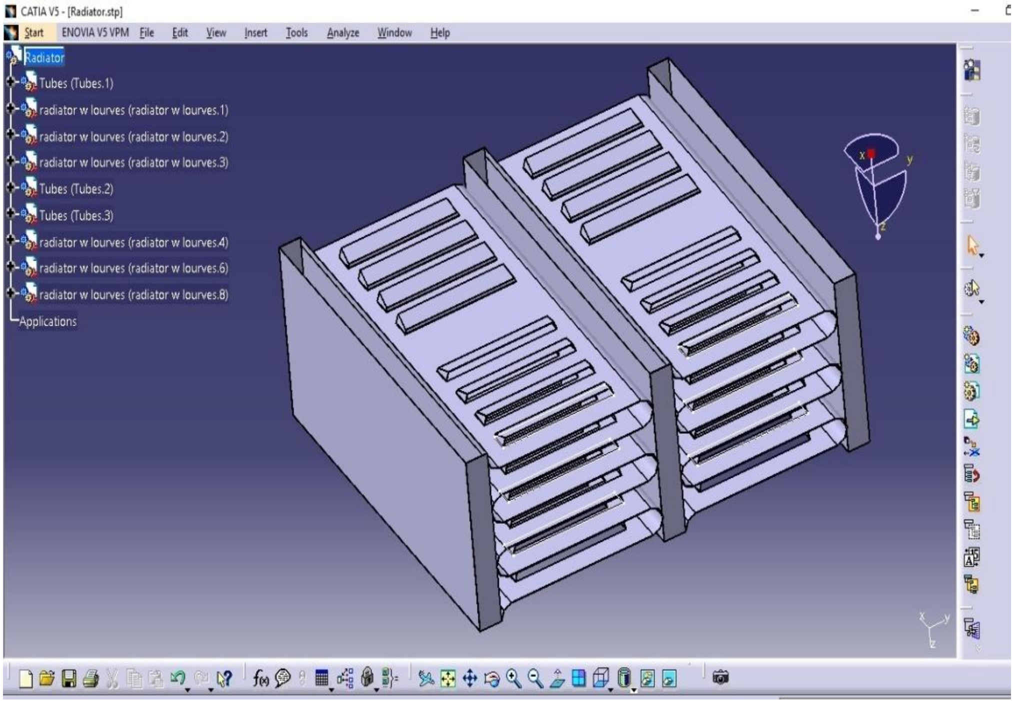


Fig. 04Model of LouveredRadiator fins

Computational analysis tool ANSYS (Fluent) 15.0 is used to perform a CFD analysis on a radiator at different mass flow rates. Normally radiator has no louvered fins; in this research work, the radiator has louvered fins. The CFD analysis will be performed for radiator with and without louvered fins. To examine the heat transfer rate using Nano fluids Al_2O_3. Different concentrations of Nano fluids in the range of 0%, 0.1%, 0.3%, 0.5%, 0.7% & 1% of vol. % of base fluid (Water, Ethylene Glycol & Ethylene Glycol (EG) - Water) will be modeled by the addition of Al_2O_3 Nano particle into the coolant flows through the radiator.

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 CATIA V5 R20 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. 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-є 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.

Domain is 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 grid of the radiator fin body is as shown in the Fig. 3.

a. Models → Viscous→ Laminar.
b. Materials → Fluent Database→ Air.
c. Operating Conditions→ Reference Pressure Location→ 5m.
d. Boundary Conditions.

ZONE	TYPE
Inlet	Velocity(m/s)
Inlet	Pressure(Pa)
Inlet	Temperature(Air & Coolant)(Kelvin)
Inlet	Fluid Mass flow Rate(Air & Coolant)(Kg\Sec)
Boundary	Wall
Outlet	Pressure Outlet
Solid	Fluid

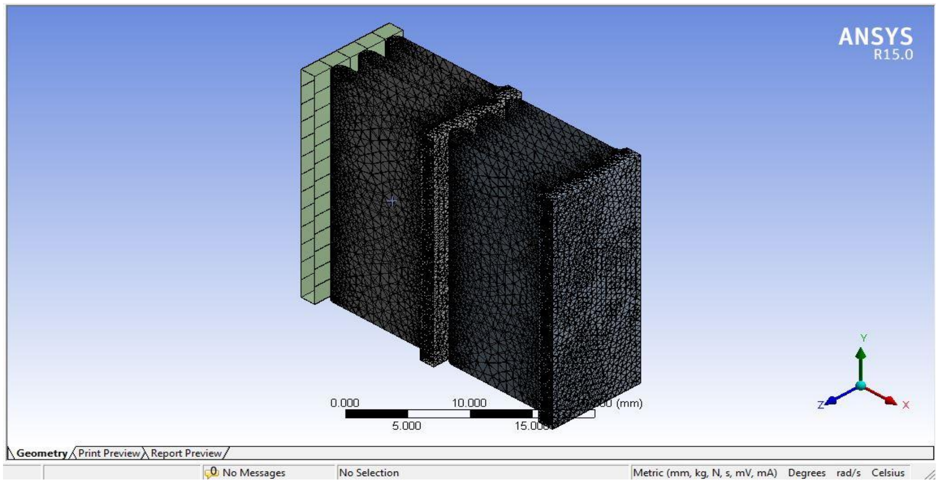


Fig. 05 Grid of the radiator without Louver fin body

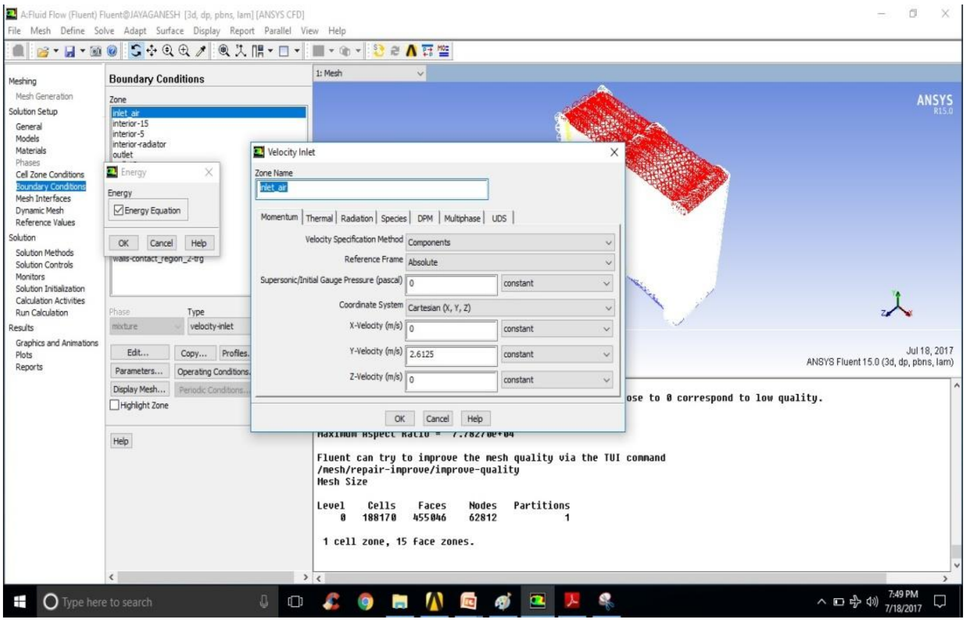


Fig.06 Velocity defined for the boundary conditions

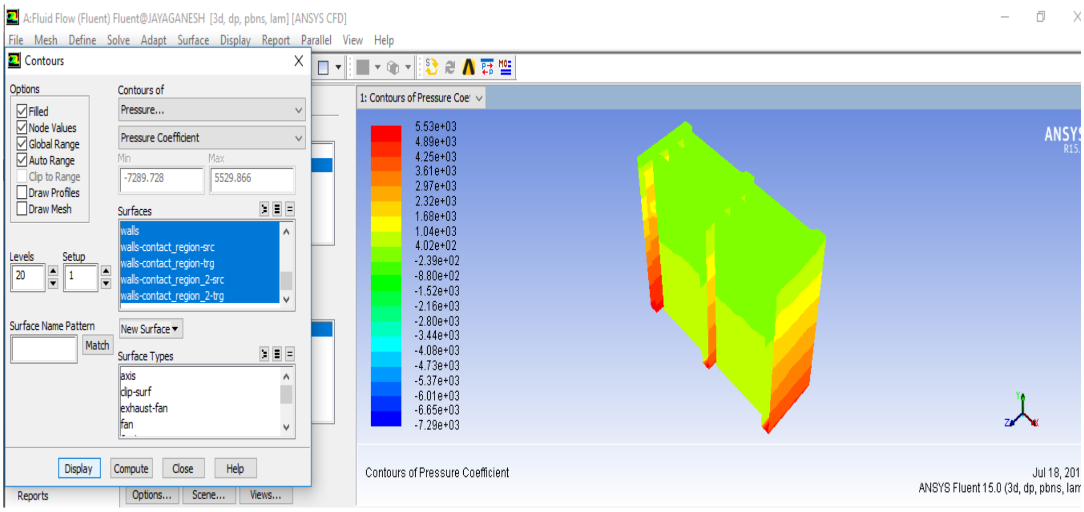


Fig. 07 Pressure Coefficient for the radiator fins

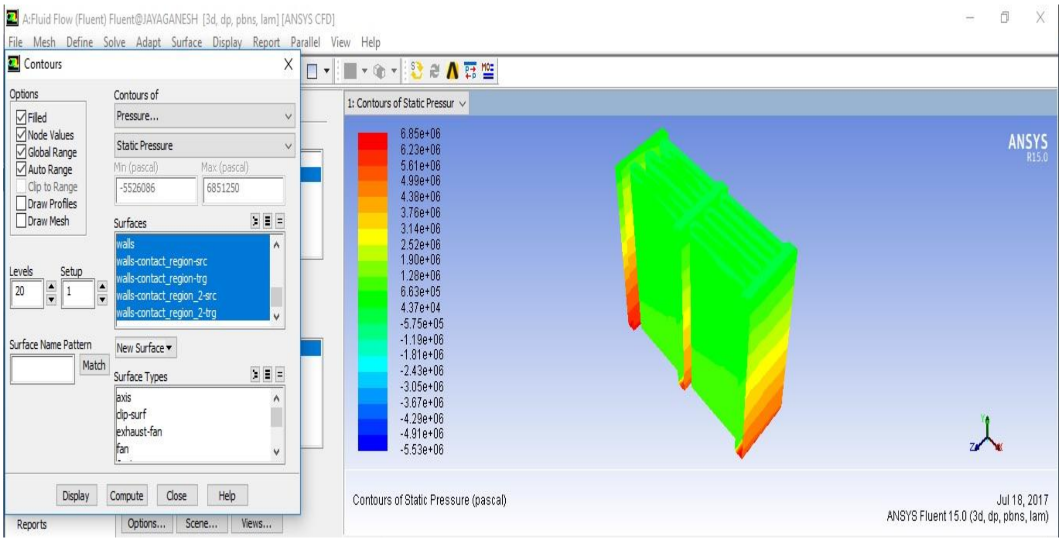
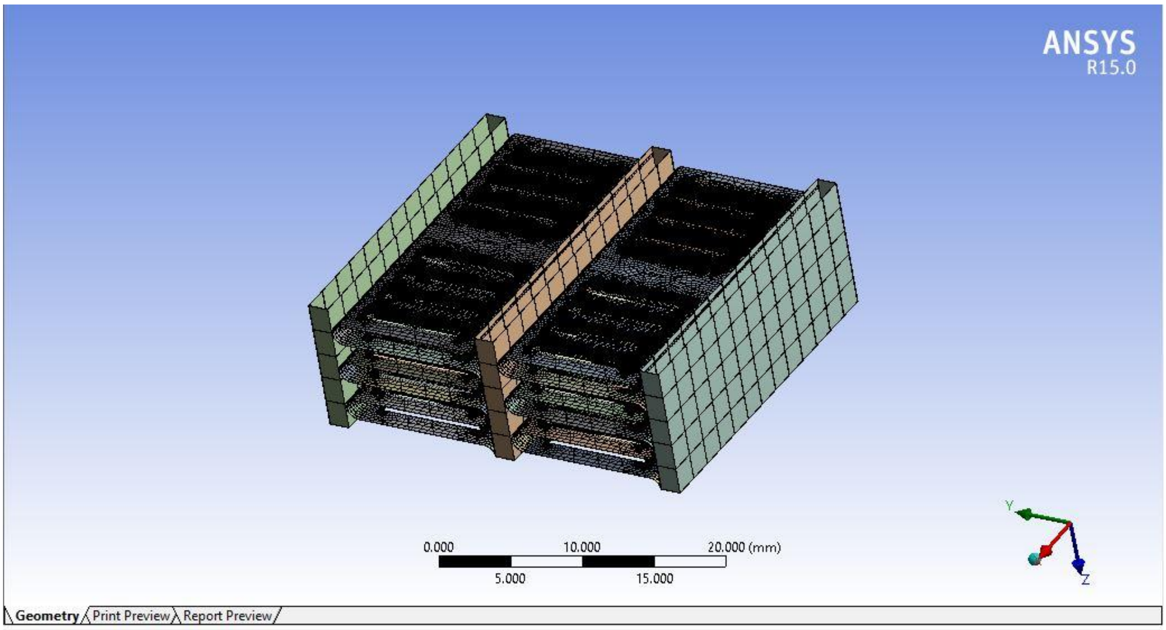


Fig. 08. Static Pressure for the radiator fins



.

Fig. 09 Grid of the radiator with Louver fin body

International Journal of Advance Engineering and Research Development (IJAERD)
Volume 4, Issue 9, September-2017, e-ISSN: 2348 - 4470, print-ISSN: 2348-6406

V.CONCLUSIONS

The present computational methodology is validated by comparing the results from the present CFD analysis with the experimental results. The results from the computations are found to be in good agreement with that of the experiments. In order to ensure improvement in heat transfer in nanofluid, the Brownian motion of the nanoparticles plays a vital role. The nanoparticles move randomly in the fluid and resulting in the decrease of the boundary layer thickness and thus enhancing the heat transfer from wall to the bulk fluid. The forced convection heat transfer characteristics of ethylene glycol/water based nanofluids are presented. It is observed that nanoparticle concentration plays an important role towards heat transfer enhancement.

	0%	0.10%	0.30%	0.50%	0.70%	1%
Experimental for water base	118	135	140	144	150	161
Computational for water base	126	141	149.6	152.4	163.8	178.1
Experimental for ethylene glycol base	37	39.6	40.8	41.8	43.1	45.6
Computational for ethylene glycol base	34.2	38	42.1	42.6	44.7	47.2

The variation between the experimental results and computational results is varying from 5% to 8% for the Nu.

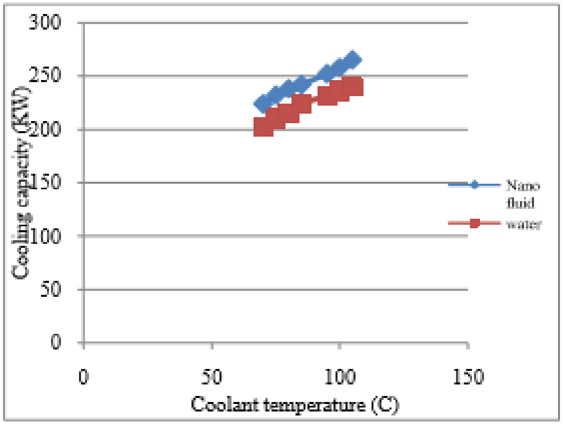


Fig.10. Effect of coolant temperature on cooling capacity

When engines run at high values of rpm to increase the speed of the vehicle, the heat generated in the parts of the engine also increases drastically. Hence, at higher speed the cooling process should also be effective in order to dissipate the heat to the atmosphere. The fluid flow and heat transfer analysis of the automotive radiator (louver fin) with Nano fluid is successfully carried out. The variations in the pressure, temperature and velocity in the direction of coolant flow and air flow is analyzed. With the computational time and available resources, the results obtained are to be satisfied.Based on results and comparisons, following conclusions are obtained; Cooling capacity increases with increase in mass flow rate of air and coolant. Reduction in cooling capacity with the increase in inlet air temperature while cooling capacity increases with the increase in inlet coolant temperature. The pressure drop also increases with the increase in air and coolant mass flow rate through radiator. About 6% increment in cooling capacity with the use of louver fin heat exchanger with nanofluid as compared to conventional coolant with the same model.

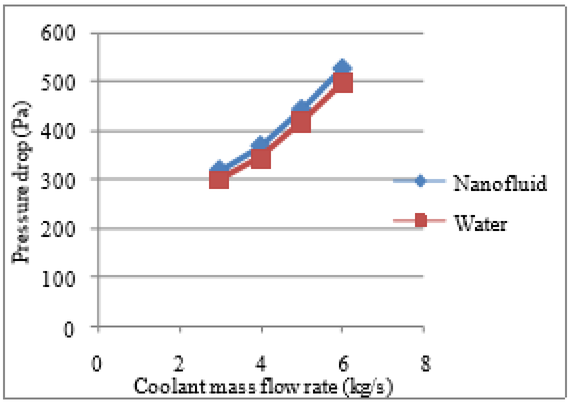


Fig.11. Effect of mass flow rate of coolant on Pressure drop

As a future work, further CFD studies can be performed with different types of radiator configurations and validations can be performed with experimentation Optimization of radiators in terms of size and weight can be performed computationally for a range of cooling capacities. Moreover, fan of the radiator may be implemented into the computational model for more realistic studies especially for other vehicle applications in which the fan is located in front of the radiator. Together with the increasing computational power of the working systems, the current computational model may also be implemented for the under hood domain simulations of the complete cooling system.

REFERENCES

[1] VahidDelavari, Seyed Hassan Hashemabadi, “CFD simulation of heat transfer enhancement of Al_2O_3/water and Al_2O_3/ethylene glycol nanofluids in a car radiator”, Elsevier, 2014.
[2] Sang Hyuk Lee, NahmkeonHur and Seongwon Kang, “An efficient method to predict the heat transfer performance of a louver fin radiator in an automotive power system”, Journal of Mechanical Science and Technology, 2014
[3] 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.
[4] 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.
[5] M. Y. Wen, C. Y. Ho, “Heat transfer enhancement in fin and tube heat exchanger with improved fin design”, Applied Thermal Engineering, vol. 29, pp 1050-1057, 2009.
[6] Y. Varol, H. Öztop, A. Varol, “Effects of Thin Fin on Natural Convection in Porous Triangular Enclosures”, International Journal of Thermal Sciences, pp. 1033–1045, 2007.
[7] J.Yang, M. Zeng, Q. Wang, “Forced Convection Heat Transfer by Porous Pin Fins in Rectangular Channels”, Quiwang Wang, vol. 132, May 2010.
[8] S. Jain, Y. Deshpande, “CFD Modeling of a Radiator Axial Fan for Air Flow Distribution”, World Academy of Science, Engineering and Technology, vol. 71, 2012.
[9] Z. Zhang, Y. Li, “CFD simulation on inlet configuration of plate-fin heat exchangers”, Cryogenics, vol. 43, 673-678, 2003.
[10] K. L. Wasewar, S. Hargunami, P. Atluri, N. Kumar, “CFD Simulation of Flow Distribution in the Header of Plate-Fin Heat Exchangers”, Chem. Eng. Technol., vol. 30, no. 10, 1340-1346, 2007.
[11] B. R. Baliga, R. R. Azrak, “Laminar Fully Developed Flow and Heat Transfer in Triangular Plate-Fin Ducts”, ASME, Journal of Heat Transfer , vol. 108, 1986.
[12] L. Zangh, “Laminar flow and heat transfer in plate-fin triangular ducts in thermally developing entry region”, International Journal of Heat and Mass Transfer, vol. 50, 1637-1640, 2007.