ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Flow Analysis on Automobile Radiator

Ranam Prathyusha
M.Tech -Thermal Engineering,
Department of Mechanical Engineering,
Ellenki College of Engineering & Technology,
Hyderabad, Telangana, India.

K.Aparna
Associate Professor,
Department of Mechanical Engineering,
Ellenki College of Engineering & Technology,
Hyderabad, Telangana, India.

G.Vinod Reddy
Associate Professor & HOD,
Department of Mechanical Engineering,
Ellenki College of Engineering & Technology,
Hyderabad, Telangana, India.

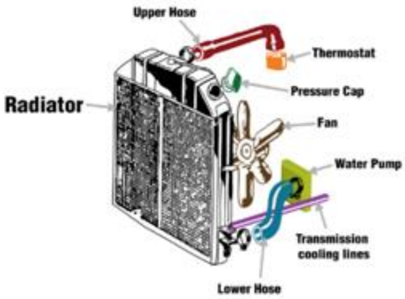
Abstract:
The main aim of the paper is to enhance the performance of fan assisted radiator by maximizing the utilization of the fan behind it. It is proposed to eliminate low velocity zones in the corners of conventional rectangular/square shape radiator and develop circular radiators. The objective of work is to have a circular radiator which is compact made, minimum material, less costly and more efficient, that will work with minimum power consumption of fan and maximum utilization of air flow. Another objective the present work is to develop tangential tube layout. Minimum Pressure drop and flow uniformity has been achieved using tangential tubes over circular tubes.

Numerical simulations have been carried out using commercially available analysis package of CFX of Workbench 14.5. Geometry and meshing has been done by using ICEM CFD. From the flow analyses the results of velocity, pressure, and temperature contours at different locations of the radiator are compared. The circular cross sectional radiator showed better performance than conventional rectangular shape radiator and the pressure drop has been minimized by 33.33% by using tangential tubes over circular tubes. The results are then validated with the experimental results.

INTRODUCTION:

Automobile radiator is used to cool down automotive engine. If it is 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. An improved heat exchanger (Radiator) such as Fan assisted Air cooled Radiator for automobile (I.C. Engine) is proposed. Different types of heat exchangers are now in use, in which air is used as heat transfer medium because air is freely and abundantly available.



In available heat exchangers, flow of air is naturally induced or by the use of fan/s. Fan assisted air cooled heat exchangers are more popular in present scenario. In available air-cooled heat exchangers, fan either forces or draws the air through the radiator. As per present system, the fan is placed behind the heat exchanger (radiator) to force / draw the atmospheric air. These Exchangers use a shroud which directs the air over the entire area of the heat exchanger. Generally, all conventional heat exchangers (Radiators) are either square or rectangular in shape and the fans with circular blades are used to create the flow of air through them.

Several drawbacks or disadvantages of present Radiators are as follows

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

1. Air delivered by fan with circular blade is in circular area even when the
Radiators are square or rectangular in shape.
2. Velocity of air flow generated by the fan is not constant or uniform along its entire axial direction and is almost zero at the centre and gradually increases at the rate of square of the radius.
3. Use of square or rectangular shrouds, to convert the circular flow of air by fan into required shape which increases the cost.
Further the known available equipments consume more power, more material and are therefore not much cost effective. Therefore, it has been proposed to develop a new heat exchanger which would avoid the disadvantages of the available equipments known.

OBJECTIVES OF PRESENT WORK

• Design improvement leads to optimization of fan assisted Radiator (H.E)
• More efficient and compact H.E. is to be designed
• To design a heat exchanger that will work with minimum power consumption for the fan and maximum utilization of air flow.
• Min. Material => Less cost of Radiator H.E
• Excluding the central hub area, the material saving is about 25%, saving in the cost of production on mass scale basis once the dies are manufactured will be about 22 %.
• Considering the number of vehicles used at national and International levels, slight improvement in efficiency and reduction in cost will add to the economy by a great extent

Figure 1: Radiator core model (front view)

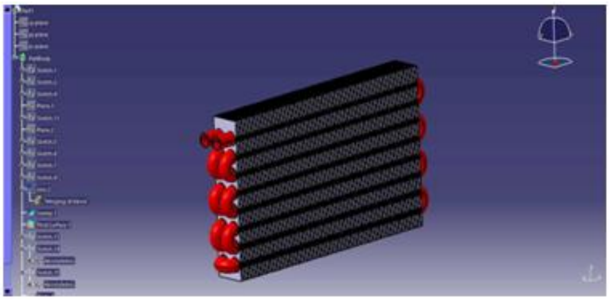


Figure 2: 3D model of radiator

Radiator CAD Drawing: To perform CFD analysis of automotive radiator fan and establish correlation with the experimental data

FINITE- ELEMENT ANALYSIS:

The proposed design of radiator is done as per the standard designing procedure for our project work. It includes the design of radiator model on 3D modeling mechanical software (CATIA), its manual calculations, CFD analysis on ANSYS software and its results. CAD Model: The designed model of radiator is made with the help of CATIA software as per dimensions and calculations carried out for our project work. Figure shows the 3D model of radiator. CFD modeling proposed in this thesis is composed of three phases. The preprocessing phase involves fin-side (air-side) porous medium modeling, water-side modeling, meshing and setting up the necessary parameters. In the solution phase, the solution method is selected, relaxation factors are tuned up and solution is performed. Finally, in the post-processing phase, the results are processed. For CFD analysis, commercial software ANSYS 14.5 workbench is used with FLUENT 14.5. The water domain of the radiator was modeled as a regular fluid domain while the air domain was modeled as a porous medium due to the complex and repeating geometry of fins. Implementation of fins into air domain is maintained by using porous media on the air side. In order to obtain the necessary input parameters and coefficients for porous medium, separate simulations were performed on a unit cell with straight fin and wavy fin structures.

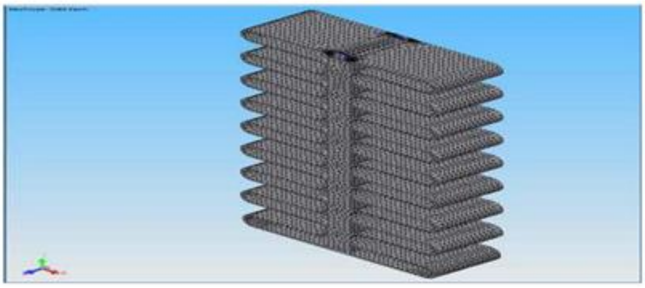
ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Procured porous medium parameters are implemented into full-sized radiator model and fluid flow and thermal analysis is utilized. Outcome of the study presents the prediction of a thermal performance of an industrial vehicle radiator. Modeling of unit cell straight fin and wavy fin is covered. Extraction of porous medium flow and heat transfer parameters are discussed in this chapter. the analysis of full-sized 2x10-tube reduced radiator model and 4x39-tube real life application model are presented under both hydro-dynamical and thermal point of views. In Chapter 4, the conclusion of the study is presented.

CFD ANALYSIS OF RADIATOR:

Modeling& Analysis: The preliminary design of the fluid and solid volumes were done in CATIA for a single tube fin arrangement. The model was then imported into GAMBIT and the complete design of the single tube and fin arrangement was done. The model was then meshed and the meshed model was expanded for full length of 360mm comprising of 120 fins. The symmetric arrangement of the tube and fin can be considered and one half the tube with fins on top of one surface can be modeled to represent one single tube. The heat transfer across the entire radiator will be same as that of one single tube and thus the entire heat transfer can be calculated by modeling a single tube itself. The meshed model is then imported into FLUENT. The fluid properties and boundary conditions are specified and solved. The solver chosen for the analysis is segregated, implicit, 3D, steady state solver in FLUENT 6.3. The segregated solver has been used for incompressible and mildly compressible flows by many investigators and has been shown to depict the results with better accuracy. It solves the energy and flow equations sequentially. The continuity, momentum and energy equations of fluid flow are solved in the process of obtaining temperature profiles. The segregated approach solves for a single variable field (e.g., pressure, p) by considering all cells at the same time. It then solves for the next variable field by again considering all cells at the same time, and so on.

Standard k-ε model is chosen to account for turbulent flow. FLUENT also takes care of the temperature dependent variation of fluid properties.



PRODUCT TYPE	Radiator
Max ENGINE SPEED	6000
CORE SIZE(W X HX T) (MM)	122X360X32
COLLANT TYPE	WATER/ETHYLENE GLYCOL 50-50%
AIR VELOCITY (m^3/s)	0.75
Coolant flow rate (lpm)	60
Max coolant inlet temp	120
Heat rejection (kw)	30

Boundary Conditions

Air flow: Velocity inlet
Flow velocity- 20 m/s
Inlet air temp-303 K
Air density- 0.45 kg/m3
Hydraulic diameter- 2.16mm
Coolant flow: Mass flow inlet
Coolant- water+ ethylene glycol 50%
Coolant temp- 365 K
Flow rate-0.045 kg/s
Hydraulic diameter-4.62 mm

FLUENT – Analysis

The solver used for the analysis is 3D- Pressure based solver with basic K-ε model. The SIMPLE algorithm for pressure-Velocity coupling was used and energy equation was also activated. The second order upwind scheme was used for pressure, momentum, turbulent kinetic energy, turbulent dissipation rate and energy

Fins and coolant tube
No of fins- 120(full length of one tube)
Total length of tube for analysis- 360mm
Depth of fin- 32mm
Material of tube and fin- Aluminium
50% Ethylene Glycol Aqueous Solution Properties

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Density: 1050 kg/m3
Specific heat capacity: 3.55 Kj/Kg-K
Thermal conductivity: 0.395 W/m-K
Viscosity: 0.0157 Kg/m-s

RESULTS

For This Particular Set of Inlet Conditions the following results are obtained

• The temperature of the coolant dropped to 365 K from 359 K across the length of the tube.

• The air temperature rose to a maximum of 325K near the fin walls and to an average of 305K overall.

• The temperature in the front tube is found to drop at a faster rate than the inner tube.

Comparison of CFD Results mathematical model Results and Test results

The test was carried out on a 4-stroke 4-cylinder horizontally opposed light weight engine for aircraft application. The engine is partly air cooled and partly watercooled. The radiator was analyzed using the mathematical model and also in Fluent for CFD analysis. The engine was tested on dynamometer and the values of radiator coolant flow rate, temperatures across the radiator, air flow rate and air temperature were measured at various conditions. The experimental data and results of mathematical model & CFD analysis are tabulated in Table 4.10. The measured data was verified against the simulated results from CFD and from mathematical model as shown in Fig. 5.8. The results show that the mathematical model was able to predict the outlet temperatures better over the entire range of flow rates with lesser variations. The CFD simulation prediction gave more accurate predictions in the higher flow rate region.

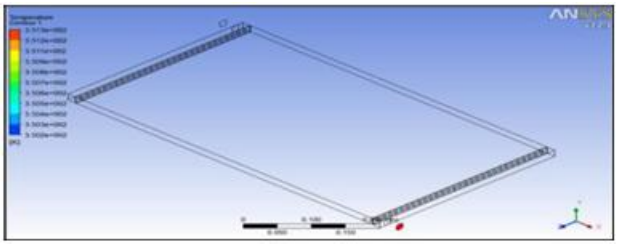


Figure 3: Outlet Temperature Tube side Comparison with Exp. Result

Sr No.	Experimental Results	CFD Analysis Results	% Variation
1	79.12	78.3	1.03640040

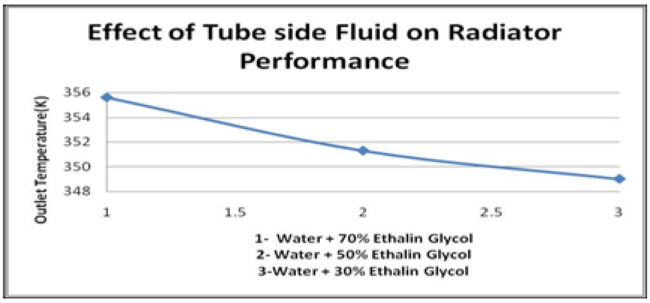
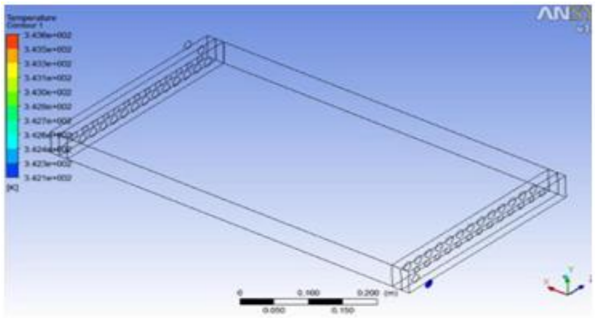


Figure 4: Effect of Tube Side Fluid

Effect of Tube Diameter on Radiator Performance:



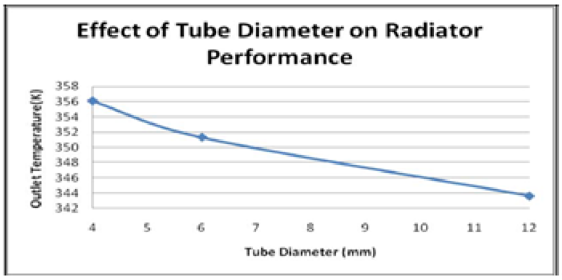
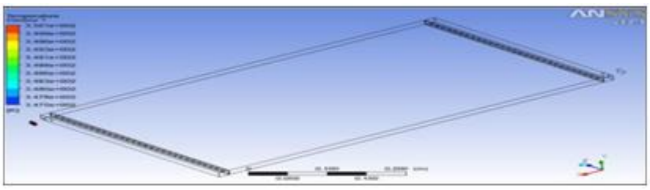


Figure 5: Effect of Tube Diameter

Effect of Coolant (Methanol and Propanol):



Outlet Temp* Water + 50% Methanol

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

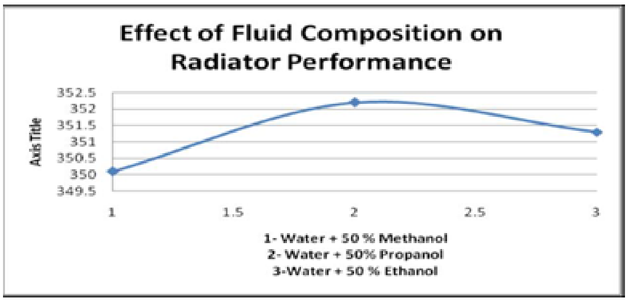
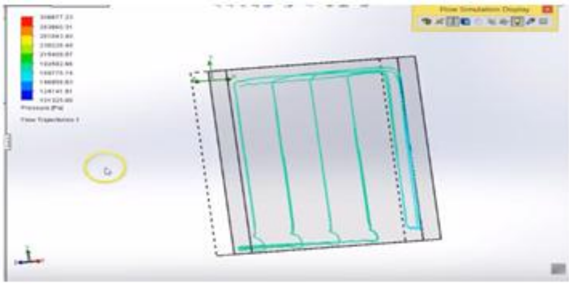
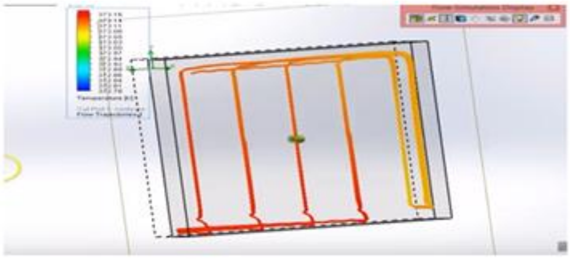


Figure 6: Effect of Fluid Composition

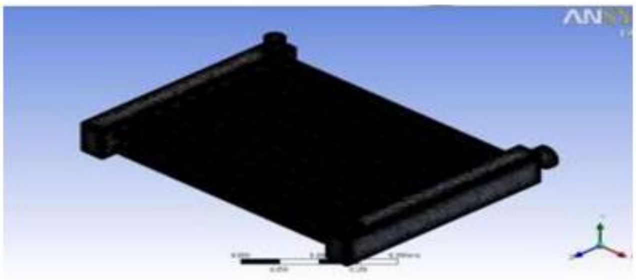
We are increasing the water in the mixture tube side outlet temperature will be decreasing in the radiator 355.6 to 349, So as the higher mixing ration of the water is desirable for achieving better performance of the radiator. If we are decreasing the tube diameter, then the tube side outlet temperature will be increasing in the radiator 343.6 to 356.1. As we are decreasing here the tube diameter, so as the diameter is decreasing the friction force acting at the extreme fiber of the intersection zone of inner fluid and the solid will be increasing and due to that phenomena the temperature rise in the small diameter tube and which is not desirable, it leads to poor performance. Also from the other comparison, we can conclude that by fixing the water proportion and taking the reading with different coolant (IE. Like Methanol, Propanol, Ethanol) in 50 % mixing ration with the water, then the Ethanol gives the highest outlet temperature of 351.3 C among all the mixtures and Methanol gives the Least outlet temperature as 350.1. So from the result, it is desirable to use Methanol with water, which gives better performance but the difference is not much more between all the coolants and that's why by sometimes seeing the toxicity and other property Ethanol is more desirable to use among all the coolants even it gives the high temperature at the outlet.



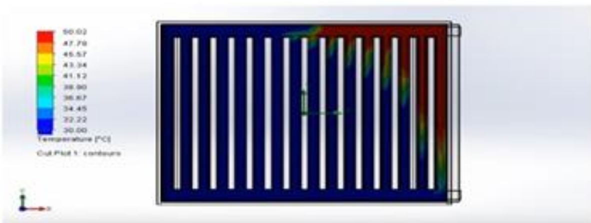


Step 1: Importing Radiator model

The first step involves importing of radiator model in ANSYS software. File format for importing radiator model is iges. After importing the radiator model in to ANSYS surrounding it look like as shown in following figure



Calculation of problem for radiator model

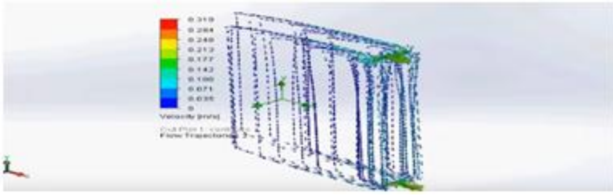


In this step calculation is carried out for problem setup in previous step. It is important step in forming the solution for problem setup

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Solution of Radiator model

The generated solution after calculation of setup problem is shown in followig figure 3.5.



Final solution figure

Comparison

Sr. No.	Parameters	Proposed Model	
		Theoretically	ANSYS
1.	Geometry of Tubes	Elliptical	Elliptical
2.	Number of Tubes	32	32
3.	Heat Transfer Rate	73.072 KW	76 KW
4.	Effectiveness	90.77 %	95 %

SUMMARY

1. The Heat transfer rate through this Radiator is 73.072 KW (Theoretically) & 76 KW through (ANSYS).
2. The Effectiveness of the Radiator is 90.77 % (Theoretically) & 95 % through (ANSYS).

In this way we have studied the design process of radiator and design a model successfully. Also we conclude that the proposed radiator model is more effective

BOUNDARY CONDITIONS

The main boundary condition include, a mass flow rate inlet boundary condition where used in the inlet nozzles. The cylindrical shaped geometries are the wall. At the outlet, the pressure outlet (atmospheric pressure) boundary condition was used. And all other portions are considered as the wall boundary with convective heat transfer surfaces.

INLET

Select inlet and change type to mass flow inlet Enter the mass-flow rate
Enter the value of inlet gauge pressure = 1.5e5 Pa
Provide the inlet temperature = 368K
Change the option under Direction Specification to "Normal to boundary"

OUTLET

Select the outlet and change type to „Pressure-outlet"
Enter the value for outlet gauge pressure = 0 Pa
Change the option under Direction Specification to "Normal to boundary"

WALL

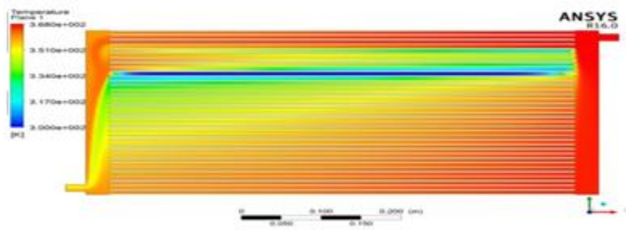
Select wall and change type to „Wall"
Select the material for the wall: Aluminium
Choose convection as the heat transfer mechanism employed in wall.
Enter the value for heat transfer coefficient as 90 W/m²K.
Enter wall thickness as 1mm
Enter the free stream temperature on wall

CONTOUR PLOTS

At first the analysis is done by keeping inlet temperature of coolant and free stream temperature constant and temperatures are obtained for different mass flow rates. Varied input mass flow rates are given in the table.

Temperature Contour

1. **Mass flow rate 500kg/h**



Temperature contour for mass flow rate 500kg/h

The temperature of coolant at inlet and outlet corresponding to various mass flow rates obtained during analysis are tabulated below

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

VARIOUS MASS FLOW RATES

Mass flow rates in Kg/hr	Mass flow rates in Kg/s	Radiator inlet temperature (K)	Radiator outlet temperature (K)
500	0.138	368	340.45
1000	0.277	368	354.45
1500	0.416	368	358.39
2000	0.555	368	359.9
2500	0.694	368	361.07

Analysis by changing free stream air temperature

The second analysis was done by changing free stream temperature (that is temperature of cooling air) for about five values which is listed below

Air inlet temperature °C	Air inlet temperature K
0	273
10	283
20	293
30	303
40	313

The temperatures of coolant at inlet and outlet corresponding to various free stream air temperatures obtained during analysis are given below

Temperature of Coolant at Inlet and Outlet Corresponding To Various Mass Flow Rates

Air inlet temperature K	Coolant inlet temperature K	Coolant outlet temperature K
273	368	329.57
283	368	333.62
293	368	337.68
303	368	341.73
313	368	345.8

The radiator model was redesigned. The straight tubes were replaced with helical tubes. The length, width and height were kept constant. The boundary conditions were kept the same. As the tubes were replaced with helical its overall length increases.

CONCLUSION:

The efficiency of the internal combustion engine cooling system depends mainly on the performance of its units. The main unit in this system is the radiator. The length of radiator gets decreased by 204mm from its original dimensions. Height also gets decreased to 30mm from its original dimensions.

It is not suitable to increase or decrease the pitch of helical tubes which will affect efficiency. Width of the radiator was not changed and can be modified to lesser dimensions. Also the modified design is preferred due to better performance and size whereas the old model is preferred due to low cost. The reverse engineering has been implemented throughout the present work, in order to achieve design analysis and improvement for the car radiator element. A comparison has been carried out in chapter three between Aluminum and Copper alloy radiator models. It is found that Copper radiator is more efficient when compared with the Aluminum radiator due to higher temperature drop (3.56 %.). However, The Aluminum radiator is much cheaper. A new design of the radiator has been proposed. The radiator dimensions were changed by increasing the width from 20 mm to 40 mm. And this change has reduced in reducing the maximum temperature by 3.9 %.

REFERENCES

[1] C. Oliet, A. Oliva, J. Castro, C.D. Perez Segarra, ―Parametric Studies on Automotive Radiators‖, Applied Thermal Engineering, 27, 2007.

[2] JP Yadav and Bharat Raj Singh, ―Study on Performance Evaluation of Automotive Radiator‖, SJPSET : ISSN : 2229-7111, Vol. 2, Issue 2, 2011.

[3] Mazen Al-Amayreh, ―Experimental Study of Thermal Conductivity of Ethylene Glycol Water Mixtures‖, European Journal of Scientific Research, Vol.44 No.2, 2011.

[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] Prof. D. K. Chavan, Prof. Dr. G. S. Tasgaonkar, ―Thermal Optimization of Fan Assisted Heat Exchanger (Radiator) by Design Improvements‖,

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

International Journal of Modern Engineering Research (IJMER),Vol.1, Issue 1, 2011.

[6] K.Y. Leong, R. Saidur, S.N. Kazi, A.H. Mamun, ―Performance Investigation of an Automotive Car Radiator Operated with Nanofluid-Based Coolants (Nanofluid as a Coolant in a Radiator)‖, Applied Thermal Engineering, 30, 2010.

[7] John Vetrovec, ―Engine Cooling System with a Heat Load Averaging Capability‖, SAE International, 2008.

[8] M.H. Salah, P.M.Frick, J.R.Wagner, D.M.Dawson, ―Hydraulic Actuated Automotive Cooling Systems— Nonlinear Control and Test‖, Control Engineering Practice, 17, 2009.

[9] Y.A.Cengel, Heat Transfer , Tata Mcgraw- Hill publications, New Delhi, 2011.

[10] P.S. Kishore, Experimental and theoretical studies of convective momentum and heat transfer in tubes with twisted tape inserts, Ph.D. Thesis, Andhra University,Visakhapatnam, India, 2001.

[11] P.K.Sarma, C.Kedarnath, V.Dharma Rao, P.S.Kishore, T.Subrahmanyam and A.E.Bergles, ―Evaluation of Momentum and Thermal Eddy Diffusivities for Turbulent Flow in Tubes‖, Published in International Journal of Heat and Mass Transfer, Vol.53, Issues 5-6, pp. 1237-1242, Feb., 2010.

[12] Evaluation of Heat Transfer and Friction Factor on Wavy Fin Automotive Radiator (IJSRD/Vol. 3/Issue 08/2015/037).