© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

CFD ANALYSIS OF RADIATOR

Bharat Pandagre[1], Rajneesh Kumar Gedam [2]
P.G. Student, Department of Mechanical Engineering, R K.D.F. College of Technology, Bhopal, M.P., India[1]
Associate Professor, Dept. of Mechanical Engineering, , R K.D.F. College of Technology, Bhopal, M.P., India[2]

ABSTRACT: Radiators are miniature version of heat exchangers used for cooling internal combustion engines and other engines like stationary generating plant, railway locomotives, piston engine aircrafts, motorcycles etc. It transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine. Many researches are carried out in order to improve its performance and for having high degree of surface compactness. These heat exchangers have fins, and tubes called compact heat exchangers. In this project we have designed a radiator model with fins. We modified it by changing the tube in tubes. For this purpose a 3D model of radiator was designed using CATIA. After modeling of the geometry the computational analysis tool ANSYS was used to perform a CFD analysis on radiator. The results were verified for Heat transfer rate and Pressure drop, Outlet air temperature, Heat carried by air, Velocity. After the analysis we found that circular tube radiator was the best design on the basis of its heat transfer rate

Keywords: Radiator, Temperature, CFD

1. INTRODUCTION

In today's world, leading automotive companies are manufacturing more powerful and efficient engines. As the engines become more powerful, the energy created by the engine is increases and so does the heat load of the engine. Together with the increase in the heat load, the required cooling capacity of a radiator also increases. Engine manufacturers specify the required cooling capacity according to their engine design parameters. For this reason, in general cooling capacity is a known input quantity. In addition to this, automotive companies also specify the fundamental size limitations of the required cooling system. An appropriate cooling system which fulfills the engine cooling capacity needs to be designed according to the specified input parameters. When the energy generation takes place from fuel energy to mechanical energy in engines, then some amount of energy losses occur due to exhaust heat and cooling system heat load

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

2. LITERATURE REVIEW

Salviochacko, Dr. BiswadeepShome, Vinod Kumar, A.K. Agarwal,D.R. Katkar, [1]Author had designed a radiator cover to increase the radiator efficiency by air flow optimization.

Omprakash Pal, Rajesh Joshi, and Sanjay T. Purkar[2] Author here 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.

Chavan D.K. and Tasgaonkar G.S[3] Author 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.

Bengt Sunden[4]Author in this paper 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. 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. By optimized design with the help of CFD we can reduce this non uniformity in temperature distribution.

J.P Yadav, Bharat Raj Singh[5] Author states that most modern cars use aluminum radiators and they are made by brazing thin aluminum fins to flattened aluminum tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement.

J.R Patel, A.M .Mavani[6]In this author used computational fluid dynamics (CFD) to model the flow of fluid and heat transfer performance characteristics and one design is suggested as a possible replacement to 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. By varying mass flow rate of air, pitch of tube and coolants are analyzed successfully using numerical simulation software.

C. Oliet, A. Oliva, J. Castro, C.D. Pe´rez- Segarra[7]Author studied different factors which influence radiator performance. It includes air and coolant flow, fin density and air inlet temperature. It is observed that heat transfer and performance of radiator is strongly affected by air and coolant mass flow rate. As air and coolant flow increases cooling capacity also increases. When air inlet temperature increases heat transfer and the cooling capacity decreases.

Chacko et al. (2005)[8]Author here used the concept that the efficiency of the vehicle cooling system strongly rely on the air flow towards the radiator core. A clear understanding of the flow pattern inside the radiator cover is required for optimizing the radiator cover shape to increase the flow toward the radiator core, thereby improving the thermal efficiency of the radiator. CFD analysis of the baseline design that was validated against test data showed that indispensable area of re-circulating flow to be inside the radiator cover. This recirculation reduced the flow towards the radiator core, leading to a reputation of hot air pockets close to the radiator surface and subsequent disgrace of radiator thermal efficiency. The CFD make able optimization led to radiator cover configuration that eliminated this recirculation area and increased the flow towards the radiator core by 34%. It is anticipated that this increase in radiator core flow would important to increase the radiator thermal efficiency.

Jain et al. (2012)[9] Author presented a computational fluid dynamics (CFD) modeling of air flow to divide among several from a radiator axial flow fan used in an acid pump truck Tier4 Repower. CFD analysis was executed for an area weighted average static pressure is variance at the inlet and outlet of the fan. Pressure contours, path line and velocity vectors were plotted for detailing the flow characteristics for dissimilar orientations of the fan blade. This study showed how the flow of air was intermittent by the hub obstruction, thereby resulting in unwanted reverse flow regions. The different orientation of blades was also considered while operating CFD analysis. The study revealed that a left oriented blade fan with counterclockwise rotation 5 performed the same as a right oriented blade fan with rotating the clockwise direction. The CFD results were in accord with the experimental data measured during physical testing.

Singh et al. (2011)[10] Author studied about the issues of geometric parameters of a centrifugal fan with backward- and forward-curved blades has been inspected. Centrifugal fans are used for improving the heat dissipation from the internal combustion engine surfaces. The parameters studied in this study are number of blades, outlet angle and diameter ratio. In the range of parameters considered, forward curved blades have 4.5% lower efficiency with 21% higher mass flow rates and 42% higher power consumption compared to backward curved fan. Experimental investigations suggest that engine temperature drop is significant with forward curved blade fan with insignificant effect on mileage. Hence, use of forward fan is recommended on the vehicles where cooling requirements are high. The results suggest that fan with different blades would show same an action below high pressure coefficient. Increase in the number of blades increases the flow coefficient follow by increase in power coefficient due to better flow guidance and reduced losses.

Kumawat et al. (2014)[11]Author illustrated about the axial flow fans, while incapable of increasing high pressures, they are well relevant for handling large volumes of air at comparatively low pressures. In general, they are low in cost, possess good efficiency and can have blades of airfoil shape. Axial flow fans show good efficiencies, and can to work at high static pressures if such operation is necessary. The presentation of an axial fan was simulated using CFD results were presented in the form of velocity vector and streamlines, which provided actual flow characteristics of air around the fan for different number of fan blades. The different parameters similar temperature, pressure, fan noise, turbulence and were also considered while performing CFD analysis. The study exposed that a fan with an

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

optimum number of fan blades performed well as compared to the fan with less number of fan blades. In general, as a compared between the efficiency and cost, five to 12 blades are good practical solutions.

Barve et al. (2014)[12] Author here illustrated about design the fan and analyze it for its strength in structure using the Finite Element Method (FEM) and the flow of air all side it using Computational Fluid Dynamics (CFD) approach. The design of the fan was conducted in phases, starting with calculating to need all dimensions followed by analytical models to prove the concept. The results obtained from the analytical studies determined a potential for a successful design that met greatest of the above outlined parameters. The calculations of the Flow Rate, Static Pressure, Velocity Vectors, and Safety in Structural were made. The structural analysis of the fan represents its strength structurally. The shear stress, Von-Misses stresses approve the safety of the design in structural

3. NUMERICAL CALCULATION FOR RADIATOR

In order to perform CFD analysis certain variables needed to be calculated. Those are mentioned below

Flow rate calculation: The flow rate depends on the area of the circular pipe and velocity of water vapour is per given data velocity of the water be 0.5m/s.

From the above formula we calculate the flow rate = **1.57mm^2.**

Mass flow rate of water:

Velocity of water vapour is 0.5m/s

As per we can calculate the mass flow rate of water vapour.

$m = \rho * V * A$

From the above formula we calculate mass flow rate =**15700kg/sec.**

Reynolds number for water=((ρwa*Vwa.*Di)/μ)

Reynolds number = 8.03x10^6.

Nusselt number for water=0.023*((REwa).^0.8)*((PRwa)^0.3)

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

= 8.6295

Convective Heat Transfer calculation:

q = hcAdT

heat transfer= 973.4(w)

Fin parameter

Length 120 mm

Height 1mm

Width 20 mm

And number of fins 16mm, space between fins 4.7mm

Thermal conductivity of fin,k =70 W/m².k

Corrected fin length Lc=Lf+(Hf/2) mm =160

The flow is viscous flow and in cfd calculation using k-epsilon (2 eqn).and standard model. For cfd analysis.

METHODOLOGY

1. <u>Designing of the model:</u> CATIA software is powerful modeling software for making 3D model proposed by Dassault System, I used CATIA V5 20.0.

Three different geometries were created by changing the design of Radiator to cheek the best one for efficiency performance. After creating the models all the files were saved in .Stp format for further thermal Analysis. The designed models of radiator are as follows

1. Existing model of radiator
2. Radiator with circular fins

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

3. Radiator with elongated circular design

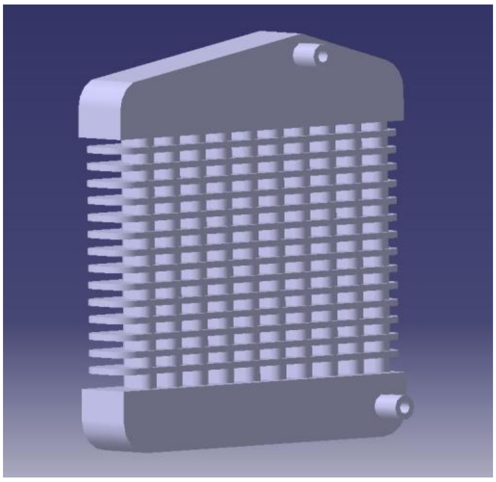


Fig:1 Catia model of existing radiator

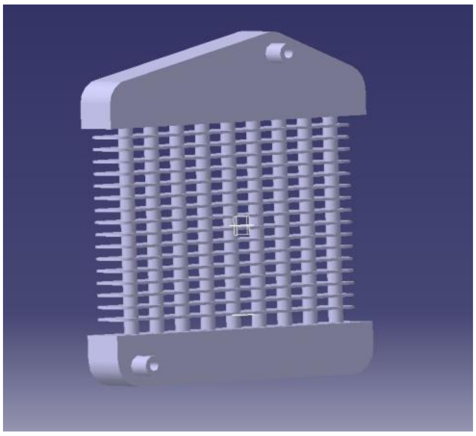


Fig: 2 Catia model of radiator with circular fins.

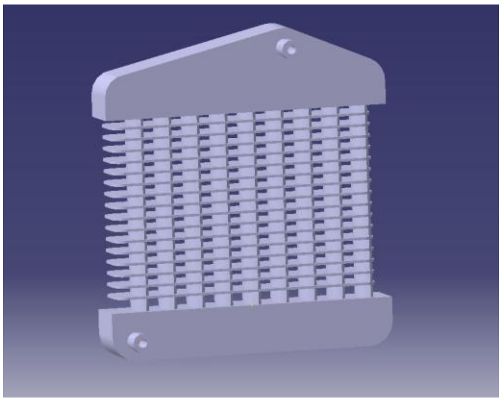


Fig: 3 1.Catia model of Radiator with elongated circular design

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

2. Computational Fluid Dynamics: the 3D models were then imported in ANSYS for thermal analysis. Boundary conditions were applied on the geometry for simulation of actual working condition. After that meshing of the models were done.

Boundary condition applied are tabulated below

Parameter	Range
Air velocity	0.5m/s
Air temperature	0 to120º

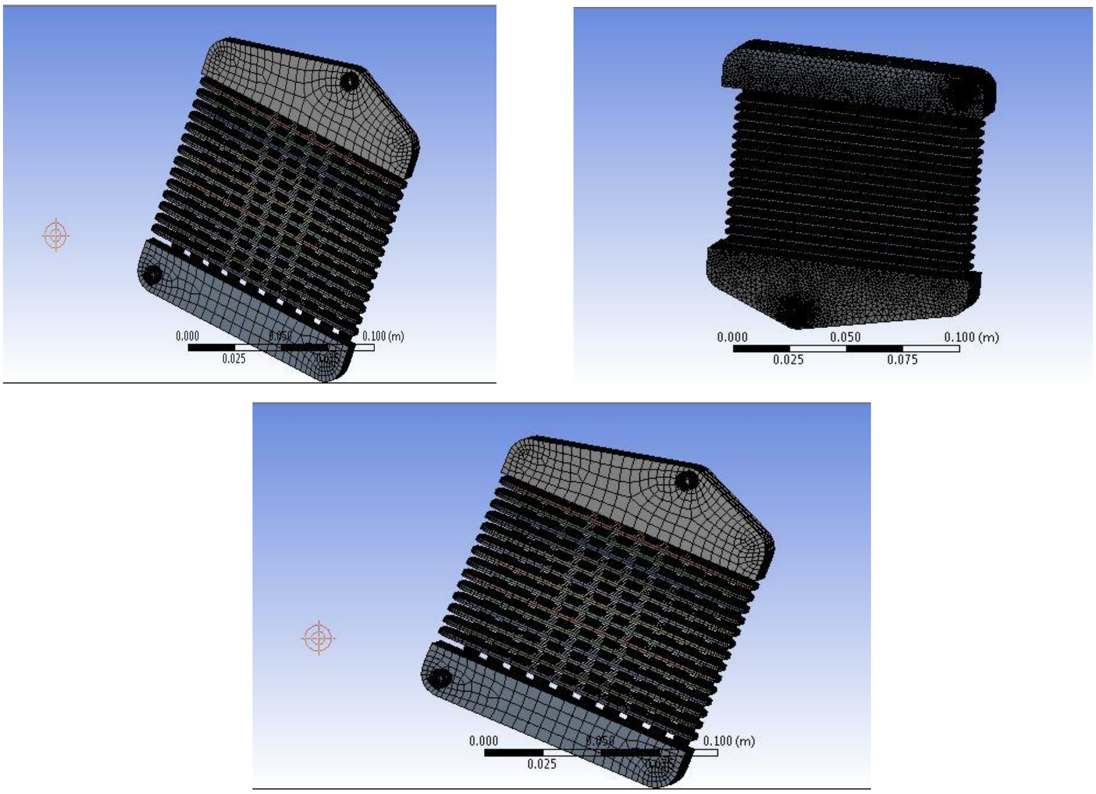
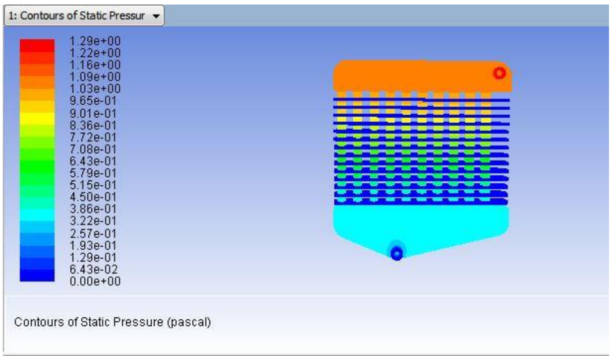


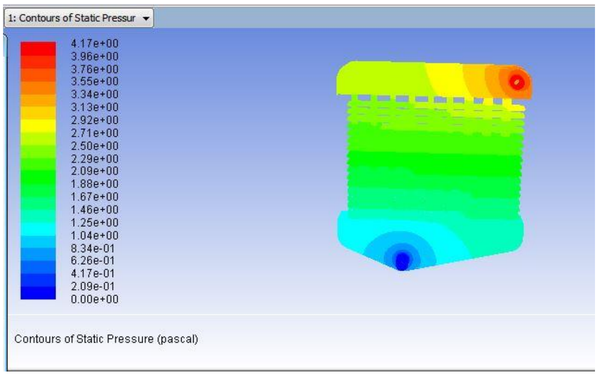
Fig: 4 Meshing of Radiator models

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

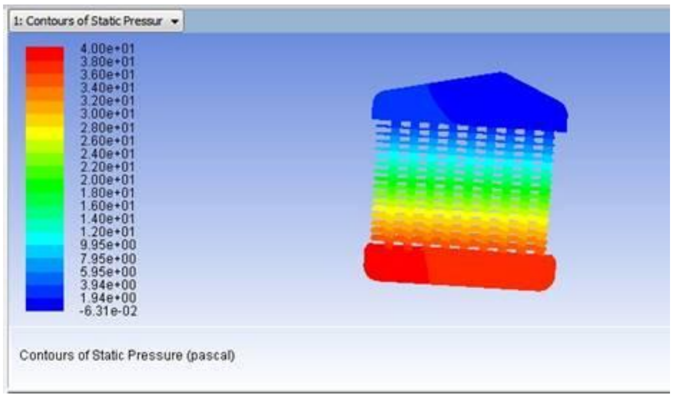
4. RESULTS AND DISCUSSION



a) Existing Radiator

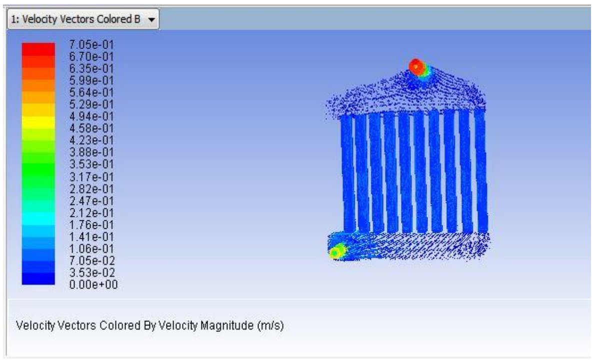


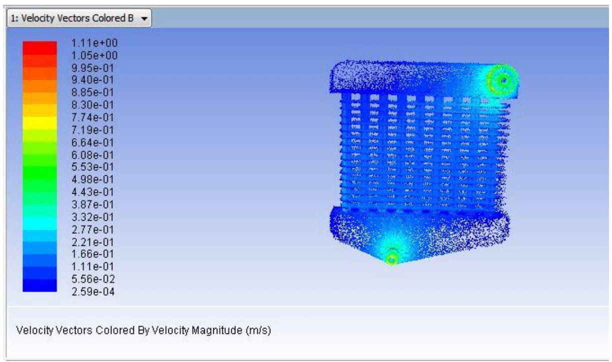
b) Circular Fin Radiator



c) Elongated circular tube radiator

Fig:5 Pressure Contours for Designed models of Radiators

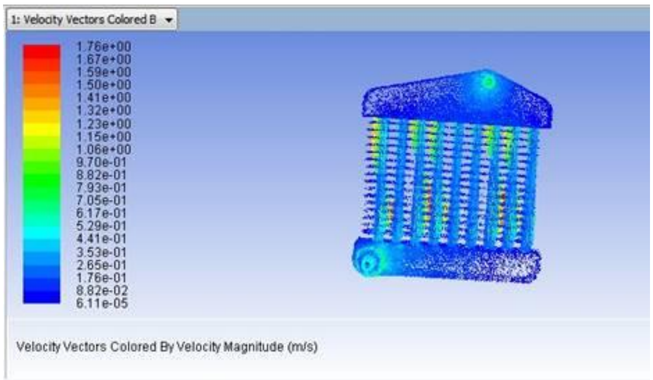




a) Existing Radiator

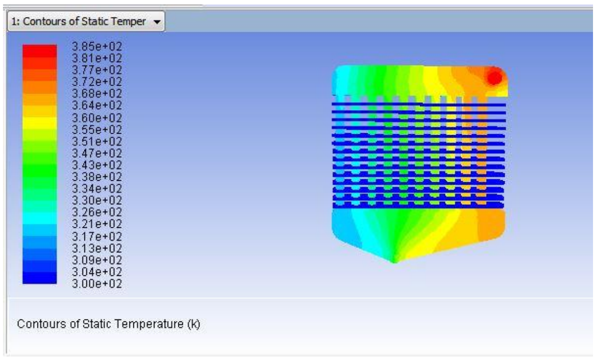
b) Circular Fin Radiator

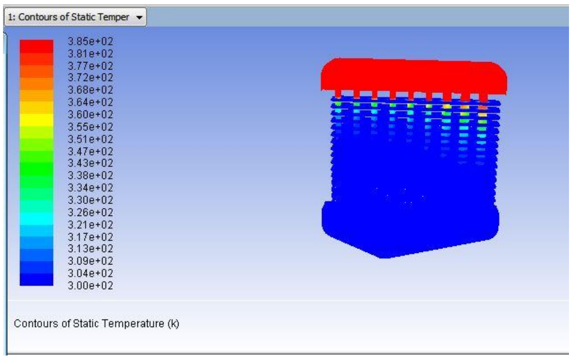
© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in



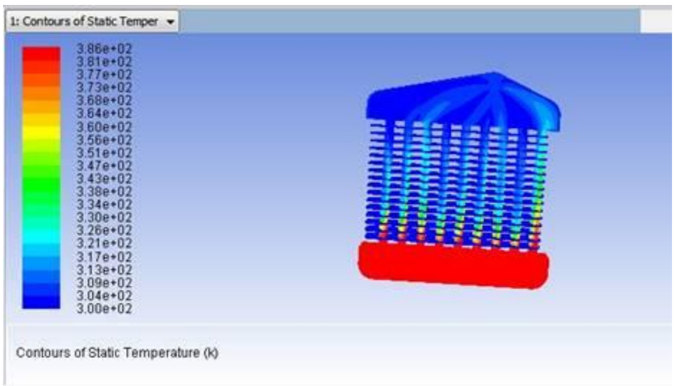
c) Elongated circular tube radiator

Fig:6 Velocity Contour for Designed models of Radiator





a) Existing Radiator b) Circular Fin Radiator



c) Elongated circular tube radiator

Fig:7 Temperature contour of Designed models of Radiator

5. CONCLUSION

On performing the results, found that the result obtained by existing model are velocity (7.05e-01 m/s), Pressure (1.29e+00 Pa), Temperature (3.85e+02 K), Mass flow Rate (9.419 Kg/s) and Total Heat Transfer (0.873 W). In second model after changing in radiator tube design result obtained are velocity (1.11e+00 m/s), Pressure (4.17e+00 Pa), Temperature (3.85e+02 K), Mass flow Rate (3.142 Kg/s) and Total Heat Transfer (4.845 W) and the third one model results obtained as velocity (1.76e+00 m/s), Pressure (4.00e+01 Pa), Temperature (3.86e+02 K), Mass flow Rate (7.984 Kg/s) and Total Heat Transfer (1.848 W). It has been found after performing CFD Analysis of models Total heat mass transfer in circular tubes radiator is larger than other models, the thermal heat transfer from one medium to other medium is larger than other model.

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]Bengt Sunden, Department of Energy Sciences, Heat Transfer, Lund UNniversity,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.

[11]Kumawat H. (2014), 'Modeling and Simulation of Axial Fan Using CFD', Mechatronic and Manufacturing Engineering, vol. 8, no. 11, pp. 1858–1862.

[12] Leong K.Y, Saidur R, Kazi S.N and A.H. (2010), 'Performance investigation of an automotive car radiator operated with nanofluid- based coolants (nanofluid as a coolant in a radiator)', Applied Thermal Engineering , vol. 2, pp. 1-10.