Design And Heat Flow Analysis Of Radiator Cooling System

Basawaraj S Hasu[1], G Srikanth Reddy[2], A Vishnu Prasad[3]
[1, 2, 3] Department of Mechanical Engineering
[1, 2, 3] AVN Institute of Engineering & Technology, Hyderabad, India

Abstract- *A radiator is a type of heat exchanger. It is designed to transfer heat from the hot coolant that flows through it to the air blown through it by the fan. Most modern cars use aluminum radiators. Brazing thin aluminum fins forge, these kinds of radiators to flattened and pulverized aluminum tubes. The flow of coolant from the inlet to the outlet through tubes mounted in a parallel arrangement. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator. In the designing stage, dimensions and features for the radiator will be totally defined and measured. Choosing the right radiator's material required and deciding if the material that is chosen is good enough to keep the radiator running without any overheating during the real conditions, is not an easy task. Automotive engine cooling system takes care of excess heat produced during engine operation. It regulates engine surface temperature for optimum engine efficiency. Recent advancement in an engine for power forced engine cooling system to develop new strategies to improve its performance efficiency.*

In the radiators cooling system, the main concept is tubes and fins, in the radiator that tubes carry the coolant. Our first idea in this category was to make the tubes smaller and increase the width of the tubes. By increasing the width, we can also increase the surface area of the tube. For this case, we can optimize the radiator model by changing the number of fins and increasing the width of the fins. This increased surface area allows for more heat transfer by convection. In this project, the 3D model of radiator cooling system is developed by using NX-CAD software, and it is imported into ANSYS CFD software to perform heat flow analysis and determine the heat reduction through the tubes and fins for different cases by changing the tubes and fins design.

I. INTRODUCTION

1. COOLING SYSTEMS

Modern automotive internal combustion engines generate an enormous amount of heat. This heat is created when the gasoline and air mixture is ignited in the combustion chamber. This explosion causes the piston to be forced down into the engine, forcing the connecting rods to reciprocate, and turning the crankshaft (rotate), creating power. Temperatures of the metal around the combustion chamber may exceed 1000° F. to prevent the overheating of the engine oil, cylinder walls, pistons, valves, and other components by these extreme temperatures, it is necessary to dispose of the heat efficiently. It states that a typical average-sized vehicle can generate enough heat to keep a 5-room house comfortably warm during zero degree weather (and I'm not talking about using the exhaust pipe). Approximately 1/3 of the heat of combustion is converted into power to drive the vehicle and its accessories. Another 1/3 of the heat is carried off into the atmosphere through the exhaust system. The remaining 1/3 must remove from the engine by the cooling system.

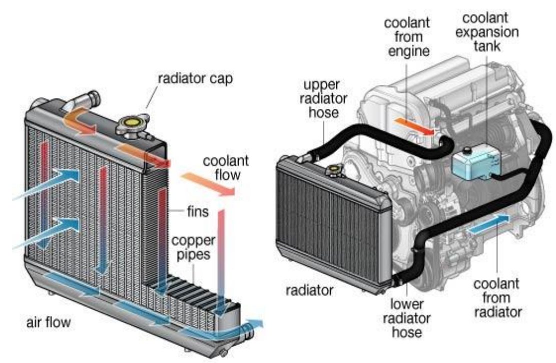


Figure 1. automotive internal combustion engines cooling system

3D MODEL OF RADIATOR COOLING SYSTEM:

The radiator consists of forty-eight copper tubes, thin fins between the tubes and two plastic covers. The radiator consists of forty-five copper tubes, fins between tubes, upper & lower cover had been made from plastic, see figure (13). The overall dimensions of radiator assembly were obtained. Next, radiator disassembly is carried to measure and obtain the actual dimensions to be used in solid model development.

Figure 2. Current radiator

II. LITERATURE REVIEW

Water-cooled petrol engines: a review of considerations in cooling systems calculations with variable coolant density and specific heat by Tony. K. Jack, Mohammed, A quick evaluation approach to internal combustion (IC) engine's radiator cooling system analysis is presented. A computer program in Microsoft Excel is developed to assist in the calculations and analysis of engine cooling parameters such as fluid flow rate, effective cooling surface area, coolant passage tubes, and rate of heat dissipation when the density and specific heat at constant pressure varies due to the change in temperature. Derived curve-fitted correlations allow for proper considerations of fluid physical properties. Selection and application of a conservative heat transfer coefficient relationship based on the Nussle relation allow for determining an effective heat transfer area, taking into consideration the inter-relationships of all applicable parameters in the heat flow area. A method for estimating the number of tubes in the radiator for proper coolant circulation is shown. The positive side of using Water/ethylene-glycol mixture versus pure water used as a coolant is discussed through a numerical example. The automotive cooling system in the industry in this project, an existing "Daihatsu" Sirion's cooling system will be studied and evaluated. In the designing stage, dimensions and features for the radiator will be fully defined and measured. Choosing the right radiator's material and deciding if the material has been chosen is good enough to keep the radiator running without any overheating during the real conditions, is not an easy task. 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. It is reported that Copper radiator is more efficient when compared with the Aluminum radiator due to the higher temperature drop. However, The Aluminums radiator is much cheaper. It's recommended to use Copper radiator rather than Aluminum radiator due to the higher temperature drop. The aluminum radiator is recommended due to lower cost. Also, the modified design is preferred due to the low temperature where as the old model is preferred due to low cost and low weight.

III. 3DMODEL OF RADIATOR COOLING SYSTEM DEVELOPMENT

Pipelines are a safe and economical means of transporting gas, water, sewage and other fluids. They are usually buried in the ground to provide protection and support. Due to the soil, external pressure is applied to the pipe and an internal pressure is applied due to fluid flowing in the pipe. Due to these internal and external pressure loads, pipe undergoes some deformation. To check the structure behaviour analysis is carried out on the buried pipe for external and internal pressure loads for two composite materials.

The methodology in this project as follows:

3d model of the buried pipe is generated by using the NX-CAD software.
3d model is converted to Parasolid file.
The Parasolid file is imported to ANSYS software to perform analysis on buried pipe.
Static analysis is done on the buried pipe for pressure loads considering soil as rectangular part.
Static analysis is done on buried pipe for pressure loads considering soil as circular part.
Compare the results of both and best one is selected.
An analysis is done for best-buried pipe for pressure loads by changing the layer orientation for E-glass/Epoxy material.
An analysis is also done for best-buried pipe for pressure loads by changing the layer orientation for Carbon/Epoxy material.
Results of both materials are compared, and best material is selected.

IV. 3DMODEL OF RADIATOR COOLING SYSTEM DEVELOPMENT

The 3D model of the radiator is created using NX-CAD software from the 2D drawings. NX-CAD is the world's leading 3D product development solution. This software enables designers and engineers to bring better products to the market faster. It takes care of the entire product definition to serviceability.

The radiator consists of forty copper tubes, thin fins between the tubes and two plastic covers. The radiator consists of forty copper tubes, fins between tubes, upper & lower cover had been made from plastic. The overall dimensions of radiator assembly were obtained. Next, radiator disassembly is carried to measure and obtain the actual dimensions to be used in solid model development.

Figure 3. Current radiator

Isometric view of radiator tube and fins:

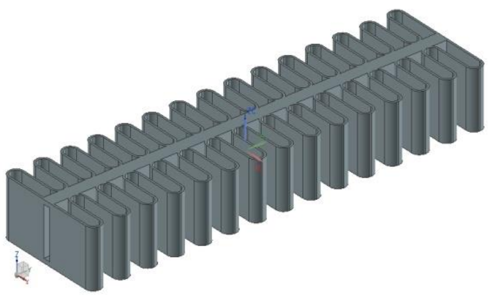


Figure 4. Show the 3D model of radiator tube and fins.

V. FLOW ANALYSIS OF RADIATOR COOLING SYSTEM

Introduction to Computational Fluid Dynamics (CFD):

CFD is a persuasive tool that is used to model the real life behavior of fluids at present situations. The optimization of design parameters is done without the need of the costly testing of multiple prototypes. What is more, it is also a powerful graphical tool for visualizing flow patterns that give insight into flow physics that otherwise would be very challenging and costly to discover experimentally, if possible. Governing equations exists to model fluid behaviour, but it is not every time possible to apply them to many of the complex flow patterns we can see in the real world directly as there would be too many unknown variables.

1. CONSTRUCTING THE DOMAIN AND MESHING

CFD allows virtual experimentation with and consequently optimization of the design parameters such as furnace tube diameter and a wide range of operating conditions. It is very attractive to industry as it saves both time and effort during the design process when compared alongside traditional experimental methods. However, the degree of confidence in the results is dependent on many factors and as a result.For the flow analysis of the radiator cooling system is generated in NX-cad and mesh is created in the ICEM CFD software.

The domain was meshed using tetrahedral meshing technique. The below figure shows the geometry and meshed body of the radiator cooling system.

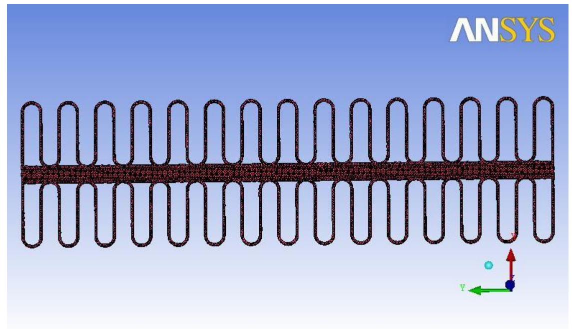
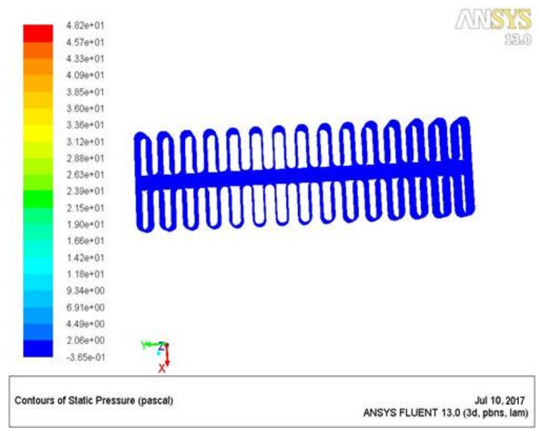


Figure 5. shows the radiator cooling system with volume Mesh and section view in CEM CFD

2. RESULTS

The case-1heat flow analysis results are shown as follows:

CONTOUR PRESSURE:



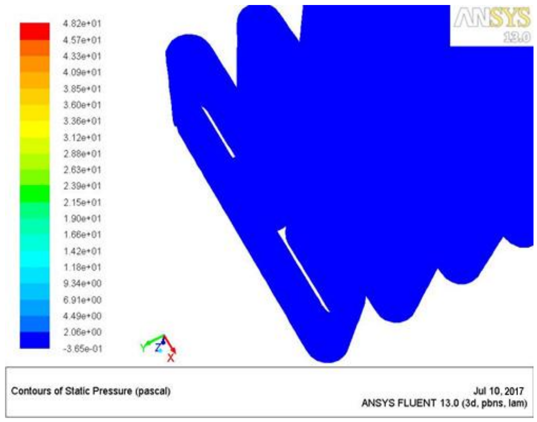


Figure 6. Shows the Contour Pressure occurred on the radiator cooling system.

VELOCITY COUNTOUR:

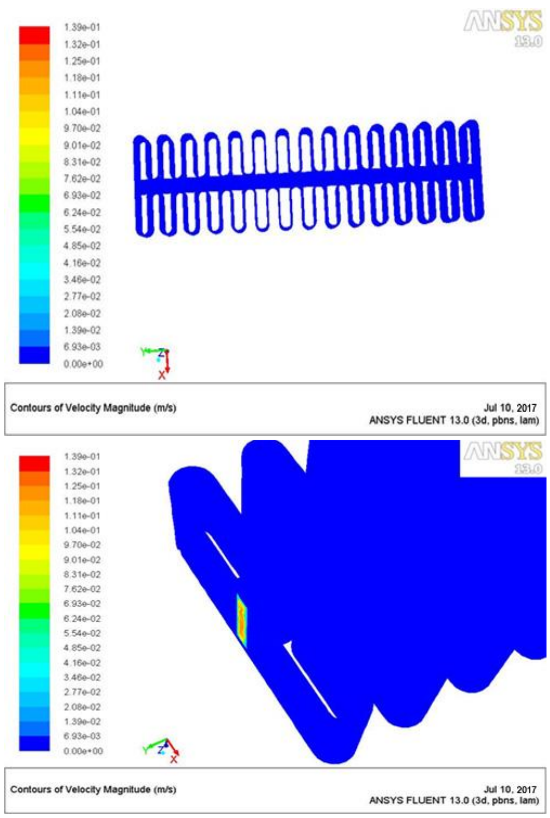


Figure 7. Shows the Contour velocity occurred on the radiator cooling system

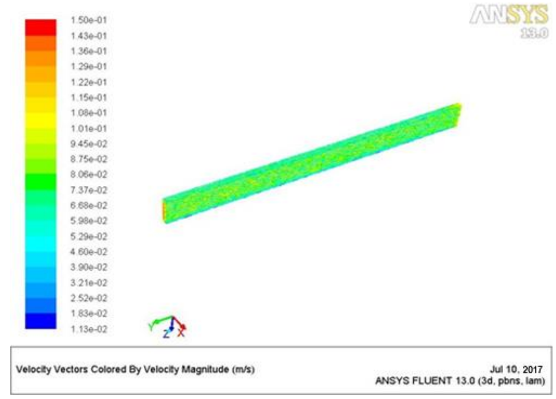


Figure 8. Shows the Vector velocity occurred on the radiator cooling system

From the above figures, we observed that the maximum velocity (0.15m/s) occurred on the outlet.

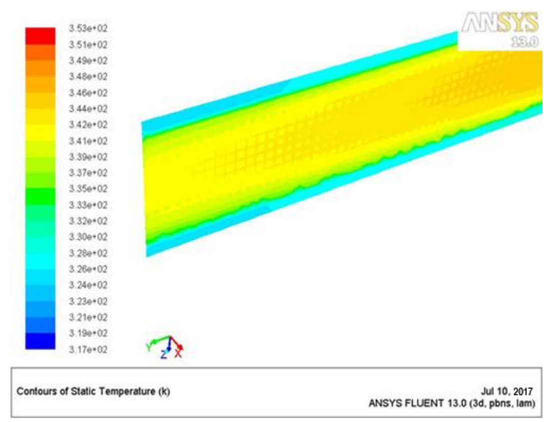


Figure 9. shows the cross section view of Contour temperature occurred on the radiator cooling system.

CASE-2

In this case, the 3D model of modified radiator cooling system is developed by using NX-CAD software, the tube width is increased to 1.5mm to 2mm for better flow transfer through the fins, and the remaining parameters and boundary conditions are same as above case-1 model. The heat flow analysis is carried out on a modified model to evaluate a design. Below figures show the heat flow analysis results.

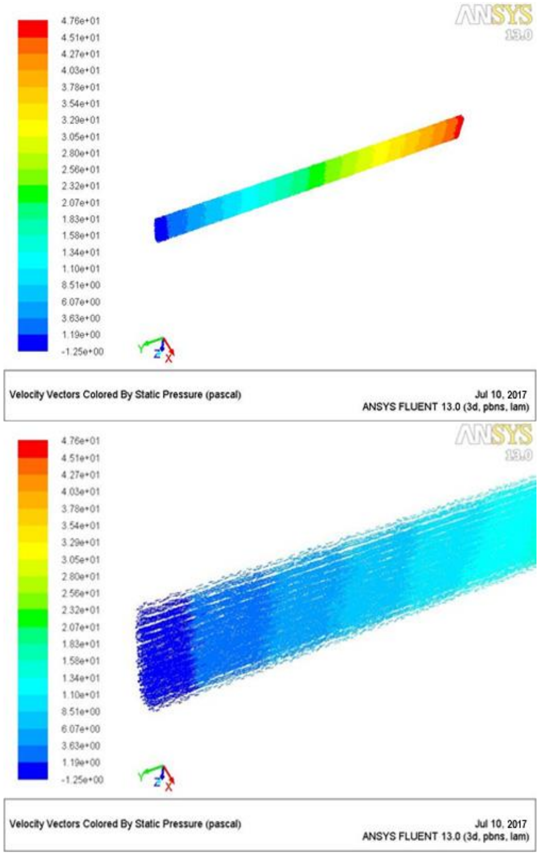
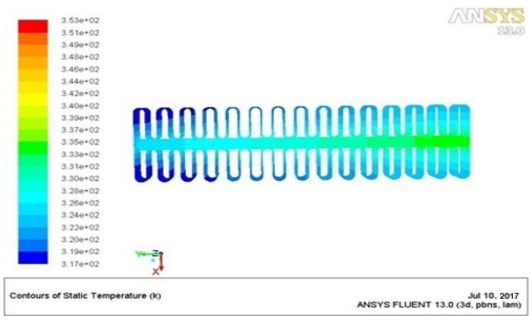


Figure 10. Shows the Vector Pressure occurred on the radiator cooling system

From the above figures, we observed that the maximum pressure (47Pa) occurred on the radiator cooling system inlet areas.

COUNTOUR TEMPERATURE:



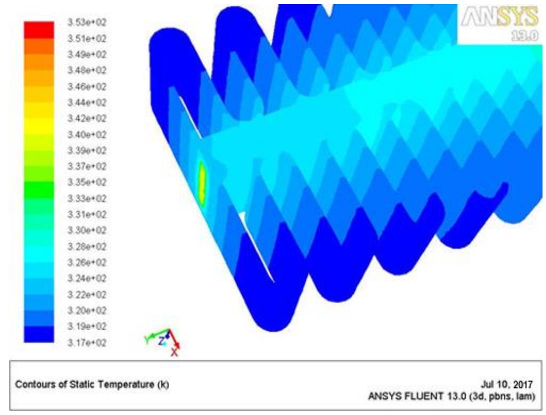


Figure 11. Shows the Contour temperature occurred on the radiator cooling system.

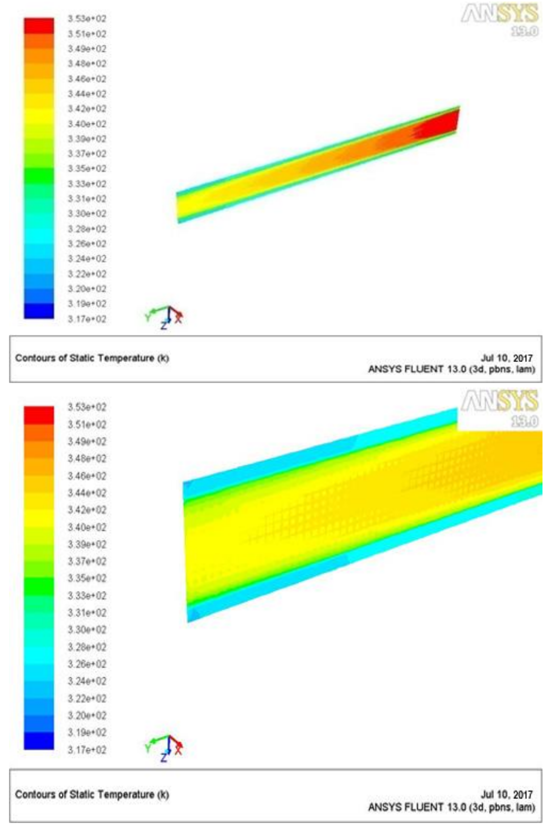


Figure 12. Shows the cross section view of Contour temperature occurred on the radiator cooling system.

CASE-3

In this case, the 3D model of modified radiator cooling system is developed by using NX-CAD software, the number of fins is increased to 15to 20 for better flow transfer through the fins, and the remaining parameters and boundary conditions are same as above case-2 model. The heat flow analysis is carried out on a modified model to evaluate a design. Below figures show the heat flow analysis results.

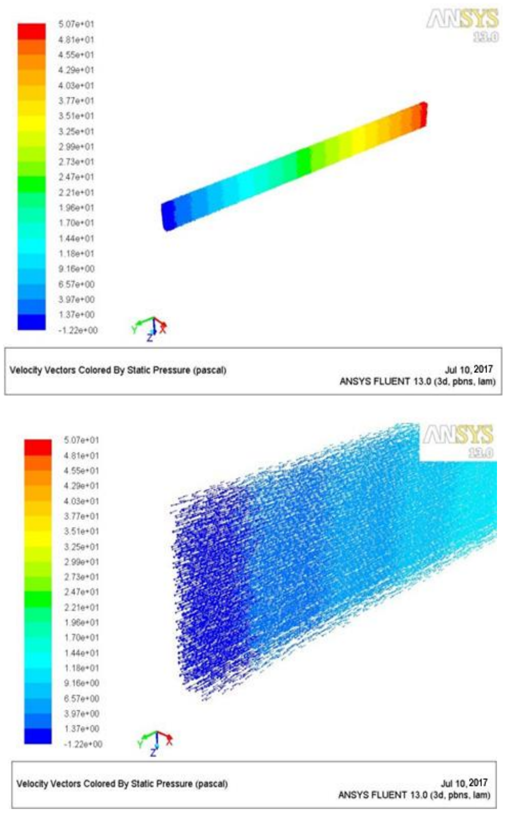


Figure 13. Shows the Vector Pressure occurred on the radiator cooling system

From the above figures, we observed that the maximum pressure (50Pa) occurred on the radiator cooling system inlet areas.

CASE-4

In this case, the 3D model of modified radiator cooling system is developed by using NX-CAD software, the fins width is increased to 1mm to 1.5mm for better flow transfer through the fins, and the remaining parameters and boundary conditions are same as above case-3 model. The heat flow analysis is carried out on a modified model to evaluate a design. Below figures show the heat flow analysis results.

VECTOR PRESSURE:

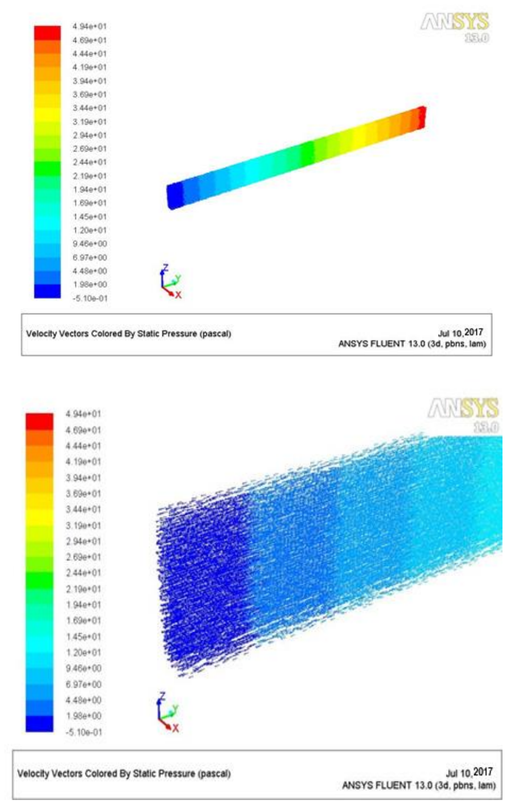


Figure 14. Shows the Vector Pressure occurred on the radiator cooling system

From the above figures, we observed that the maximum pressure (49Pa) occurred on the radiator cooling system inlet areas.

VI. RESULTS AND DISCUSSIONS

In this project, the 3D model of radiator cooling system is developed by using NX-CAD software, and it is imported into ANSYS CFD software to perform heat flow analysis. In the radiators cooling system, the main concept is tubes and fins, in the radiator that tubes carry the hot water flow and it was analyzed for heat flow boundary conditions as velocity and temperature which are developed from an engine. In this project 3D model of radiator cooling system taken as tube and fins assembly for an easy computational domain and it was analyzed for four different cases by changing the width of the tubes and number of fins. Heat flow analysis results for all the cases are shown below.

Case-1:

In this case, the radiator tubes and fins parameters are taken from the reference journals and in this preferred model we are considering 15 fins on each tube, a width of the fin is 1mm and width of the tube are 1.5mm.

From the case-1 flow analysis results, we can observe that the average temperatures are 353K, 330K, 324K and 338K respectively occurred on the inlet, interface, fins and outlet areas of radiator cooling system. In this case, we observed that the average temperature is on interface and fins are high. The total mass of radiator tube and fins are 0.41kg only. So we can change the radiator cooling system by increasing the tube width, because of the change in the tube we can increase the temperature distribution by conduction heat transfer.

Case-2:

In this case, the tube width is increased to 1.5mm to 2mm for better flow transfer through the fins, and the remaining parameters and boundary conditions are same as above case-1 model.

From the above flow analysis results, we can observe that the average temperatures are 353K, 329K, 323K and 337K respectively occurred on the inlet, interface, fins and outlet areas of radiator cooling system. In this case, we observed that the fewer temperature values on interface, fins and outlet by increasing tube width. The total mass of radiator tube and fins are 0.48kg only. But the average outlet temperature is also having very high. So we can change the radiator cooling system by increasing the number of fins and perform heat flow analysis on modified-3 model as case-3

Case-3:

In this case, the number of fins is increased to 15to 20 for better flow transfer through the fins, and the remaining parameters and boundary conditions are same as above case-2 model.

From the above flow analysis results, we can observe that the average temperatures are 353K, 326K, 321K and 335K respectively occurred on the inlet, interface, fins and outlet areas of radiator cooling system. In this case, we observed that the fewer temperature values on interface, fins and outlet by increasing number of fins. The total mass of radiator tube and fins are 0.59kg only. But the average temperature is still having high. So we can change the radiator cooling system by increasing the width of fins and perform heat flow analysis on modified-4 model as case-4.

Case-4:

In this case, the fins width is increased to 1mm to 1.5mm for better flow transfer through the fins, and the remaining parameters and boundary conditions are same as above case-3 model.

From the above flow analysis results, we can observe that the average temperatures are 353K, 324K, 321K and 335K respectively occurred on the inlet, interface, fins and outlet areas of radiator cooling system. In this case, we observed that the fewer temperature values on interface and fins by increasing width of fins. The total mass of radiator tube and fins are 0.79kg.

Table 1.

CASE	AVERAGE TEMPERATURE DISTRIBUTION				
	INLET (K)	INTERFACE (K)	FINS (K)	OUTLET (K)	WEIGHT (Kg)
1	353	330.4	324.5	338.02	0.41
2	353	329	323	337	0.48
3	353	326.06	321.2	335.19	0.59
4	353	324	321.01	335.02	0.79

Table12 shows the comparison of heat transfer in all cases

From the above comparison of results, we can observe that the case-3 and case-4 have the fewer temperature values on interface, fins and outlet areas than the other cases. But comparing within these case-3 and case-4, case-4 (0.79kg) having heavy weight than the case-3 (0.59kg). Hence we can conclude that the case-3 modified model of radiator cooling system is better than the other cases.

In this case, we had taken off a length of the tube and fins model for better computational analysis, and we are getting 353k to 335k temperatures from the inlet to outlet area. Hence we can estimate the final temperature value of outlet is 315k by the adding tube length and increasing fins as per the final model.

VII. CONCLUSION

In this project, the 3D model of radiator cooling system is developed by using NX-CAD software, and it is imported into ANSYS CFD software to perform heat flow analysis. In the radiators cooling system, the main concept is tubes and fins, in the radiator that tubes carry the hot water flow and it was analyzed for heat flow boundary conditions as velocity and temperature which are developed from an engine. In this project 3D model of radiator cooling system taken as tube and fins assembly for an easy computational domain and it was analyzed for four different cases by changing the width of the tubes and number of fins.

From the above comparison of results, we can observe that the case-3 and case-4 have the fewer temperature values on interface, fins and outlet areas than the other cases.

But comparing within these case-3 and case-4, case-4 (0.79kg) having heavy weight than the case-3 (0.59kg). Hence we can conclude that the case-3 modified model of radiator cooling system is better than the other cases.

REFERENCES

[1] Water-cooled petrol engines: 'a review of considerations in cooling systems calculations with variable coolant density and specific heat' by Tony. K. Jack.

[2] 'Study of the Effect of Mass flow Rate of Air on Heat Transfer Rate in automobile radiator by CFD simulation using CFX' by P. K. Trivedi.

[3] Performance Improvement of A Louver-Finned Automobile Radiator Using Conjugate Thermal CFD Analysis by Jun Janna G.C.

[4] Automotive Radiator Sizing and Rating – Simulation Approach by P. S. Amrutkar, S. R. Patil.

[5] Hybrid Radiator-Cooling System by David M. France, High-Efficiency Radiator Design for Advanced Coolant by Scott Janowiak, Yunus A. Cengel and Robert H. Turner, 2005. Fundamentals of Thermal-Fluid Sciences.

[6] Brace C. J, Burnham-Slipper H, Wijetunge R. S, Vaughan N. D., Wright K., Blight D., Integrated Cooling Systems for Passenger Vehicles, SAE O1HX-8, 2001.

[7] Robinson, K., Hawley, J G., Hammond, G P., Owen, N J., Convective coolant heat transfer in internal combustion engines, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering.

[8] William H. C. and Donald L. A,1981. Automotive Fuel, Lubricating, and Cooling Systems.

[9] Randy Rundle, 1999. Automotive Cooling System Basics.

[10] Ray T. Bohacz, 2007. Engine Cooling Systems.