ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Performance Analysis and Air Flow Optimization of Radiator Using Simulation

K.Ganesan*, P.Ravikumar
*Assistant Professor, Department of Mechanical Engineering, Nandha Engineering College, Erode, Tamil Nadu, India
PG Scholar, Engineering Design, Department of Mechanical Engineering, Nandha Engineering College, Erode, Tamil Nadu, India

Abstracts

Automotive engine cooling system takes care of excess heat produced during engine operation. It Regulates Engine surface temperature for engine optimum efficiency. Recent advancement in engine for power forced engine cooling system to develop new strategies to improve its radiator performance efficiency. Also to reduce fuel consumption along with controlling engine emission to indicated environmental pollution norms. This paper throws light on parameters optimization flow changes analysis which influences radiator performance along with reviews some of the systematically with new modern approaches to enhance radiator performance analysis with design and numerical analysis of water heating conductivity to transient analysis single sample tube in different copper graded material analysis in flow passing through the water comparing to the better cost effective and material data its analyzed using in ansys 14.5 version.

Keywords- Radiator, flow geometry, cross section, air flow, Ansys..

Introduction

Automotive cooling systems

Modern automotive internal combustion engines generate a huge 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 inside the engine, levering the connecting rods, and turning the crankshaft, creating power. Metal temperatures around the combustion chamber can exceed1000° F. In order to prevent the overheating of the engine oil, cylinder walls, pistons, valves, and other components by these extreme temperatures, it is necessary to effectively dispose of the heat. It has been stated that a typical average-sized vehicle can generate enough heat to keep a 5-room house comfortably warm during zero degree weather. Approximately 1/3 of the heat in 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.

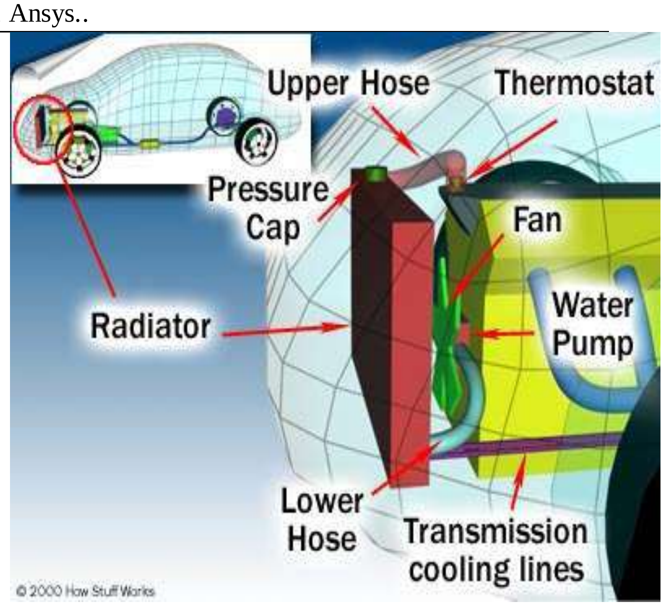

Fig.1Cooling system components

Radiator

A radiator is a type of heat exchanger. It is designed to transfer heat from the hot flows through it to the air blown through it by the fan. Most modern cars use aluminium radiators. These radiators are made by brazing thin aluminium fins to flattened aluminium tubes. The coolant flows from the inlet to the outlet

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

through many conducts the heat from the tubes and transfers it to the air flowing through the radiator.

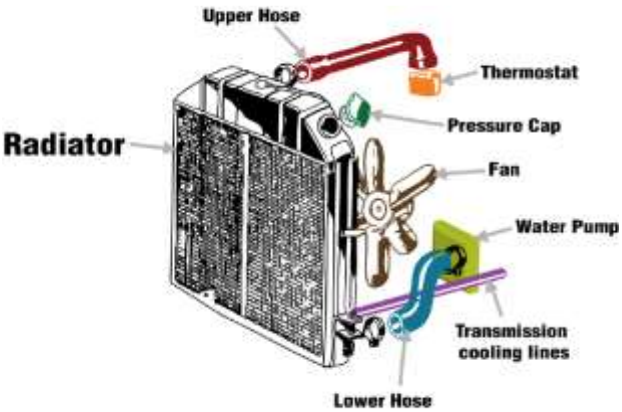

Fig.2 Radiator

Cooling fluids

Cars operate in a wide variety of temperatures, from well below freezing to well over 100˚F (38˚C). So whatever fluid is used to cool the engine has to have a very low freezing point, a high boiling point, and it has to have the capacity to hold a lot of heat. Water is one of the most effective fluids for holding heat, but water freezes at too high a temperature to be used in car engines. The fluid that most cars use is a mixture of water and ethylene glycol ($C_2H_6O_2$), also known as antifreeze. By adding ethylene glycol to water, the boiling and freezing points are improved significantly. The temperature of the coolant can sometimes reach 250˚F to 275˚ F (121˚C to 135˚C). Even with ethylene glycol added, these temperatures would boil the coolant, so something additional must be done to raise its boiling point. The cooling system uses pressure to further raise the boiling point of the coolant. Just as the boiling temperature of water is higher in a pressure cooker, the boiling temperature of coolant is higher if you pressurize the system. Most cars have a pressure limit of 14 to 15 pounds per square inch (psi), which raises the boiling point another 45˚F (25˚C) so the coolant can withstand the high temperatures.

Literature review

C. M. De Silva [1] et al. state that the Formula SAE vehicles, over the program's history have showcased a myriad of aerodynamic packages, each claiming specific quantitative and qualitative features. This paper attempts to critique differing aerodynamic side pod designs and their effect upon radiator heat management. Various features from inlet size, side pod shape and size, presence of an under tray, suspension cover, gills and chimneys are analyzed for their effects. Computational Fluid Dynamics (CFD) analyses are performed in the FLUENT environment, with the aid of GAMBIT meshing software and Solid Works modeling.

Salvio Chacko [2] et al said that the heat exchanger, used in refrigeration unit, air conditioning unit, radiator used with IC engine automobiles is either rectangular or square in shape. But the air blown/sucked by the fan is in circular area developing low velocity zones or high temperature regions are created in the corners. Different heat exchangers/radiators are studied; Radiator is designed, Calculations are done, CAD drawings of radiator and geometrical model are developed.

S.Pawan [3] et al analyzed in automotive engine cooling system takes care of excess heat produced during engine operation. It regulates engine surface temperature for engine optimum efficiency. Recent advancement in engine for power forced engine cooling system to develop new strategies to improve its performance efficiency.

Problem identification

The fluid and flow cooling is generated, produced (or) developed in the system that conducts through the walls or boundaries is to be continuously dissipated to the surroundings or environment to keep the system in relative temperature distribution condition. Large quantities of heat flow thorough a engine have to be dissipated from large area as heat transfer by convection between a air flow and the inside of the cooling fluid internal flow tubes. It can be increased by cooling outer flow on surface of the radiator cooling system. In that case the design model of flow tubes is to be changed in various models in different positions.

Methodology

Fig.3 shows the methodology which is used in this analysis.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

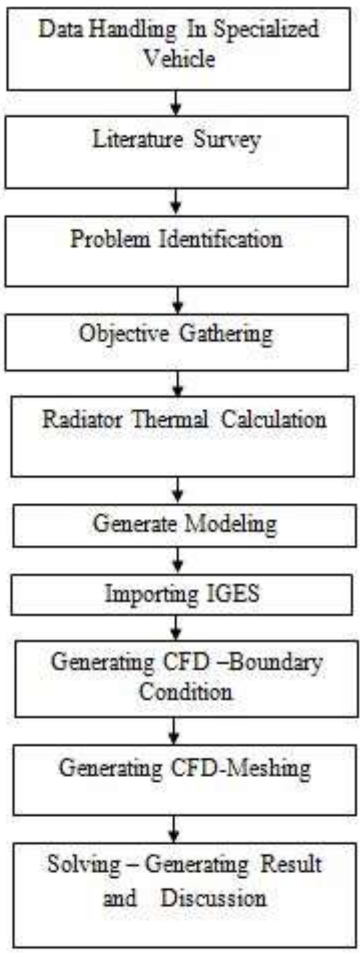

Fig.3 methodology

Data collection

Radiator improves heat conduction in air with internal fluid a cooling system. One way is by creating turbulent flow through Radiator geometry, which reduces the thermal resistance (the inverse of the heat transfer coefficient) through the nearly stagnant flow tubes that forms when a fluid flows parallel to a solid tube surface. A second way is by increasing the radiator flow fluid density, which increases the heat transfer area that comes in contact with the inside fluid tube.

Radiator geometries and densities that create turbulent flow and improve performance also increase pressure drop, which is a critical requirement in most high performance applications. The optimum flow fluid geometry and flow fluid density combination is then a compromise of performance, pressure drop, weight, and size. A figure-of-merit comparison

Fig.4 existing radiator

based on performance, pressure drop, less weight, and size among common flow tubes in different types is described in " The system in Heat conduction of better Design for advanced air cooling radiator ."Aside from radiator geometry, parameters such as thickness, height, pitch, tubes flow and spacing can also be altered to improve performance. Typically, radiator flow tube flow shapes and pitches vary from 7.60 mm to 0.012 in (0.3 mm), heights vary from 487.50 mm (0.89 mm) to 0.6 in (15.24 mm), and densities vary from 8 to 30 FPI (Tubes Per Inch).

Geometry

Fig.5 shows the typical radiator geometry which is used in this analysis. This geometry is created by using Soild Works software.

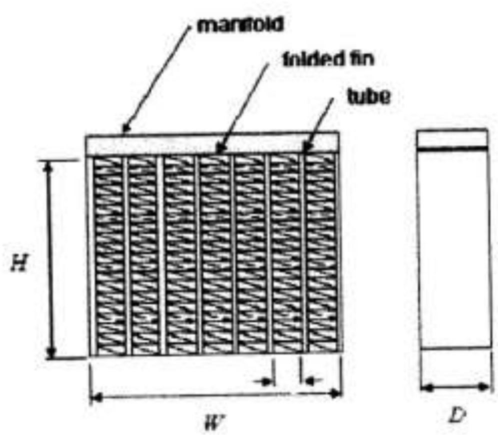

Fig.5 radiator dimensions

Meshing

Fig.6 shows the mesh model of the radiator which is created by using Ansys software. Tetra hectra is used to mesh this radiator model.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

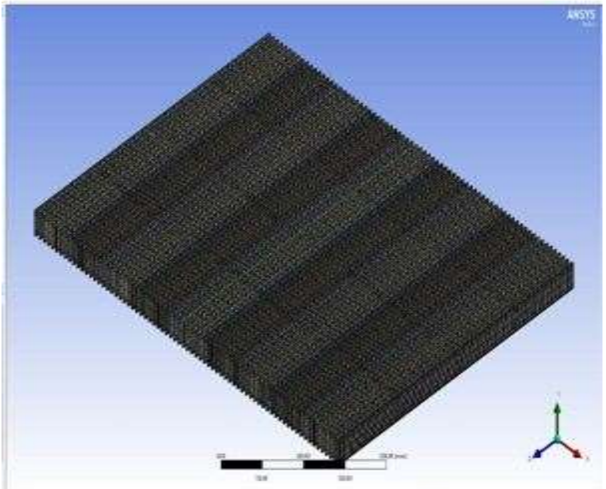

Fig.6 mesh model

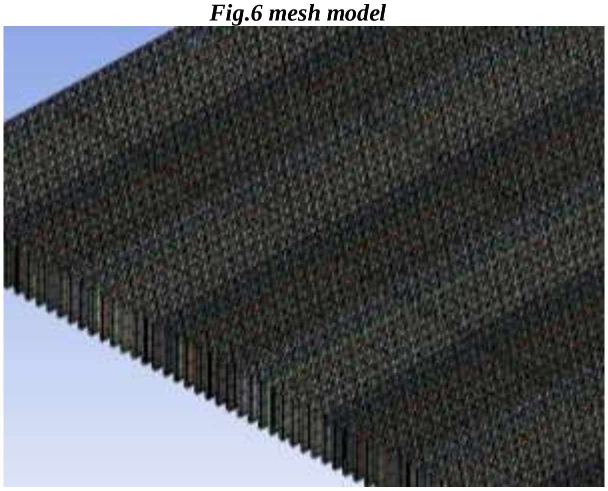

Fig.7 Close up of mesh model

Analysis

The following analysis has been carryout by using Ansys software and the following results were made from post processing results of Ansys.

Results

The following results have been obtained by using CFD analysis.

Flow velocity

Fig.8 shows the flow velocity of fluid which is flow inside the radiator tube with curved type geometry. It shows clear flow velocity is maximum in this type of geometry with a higher magnitude of 118m/s compared with other two geometry models.

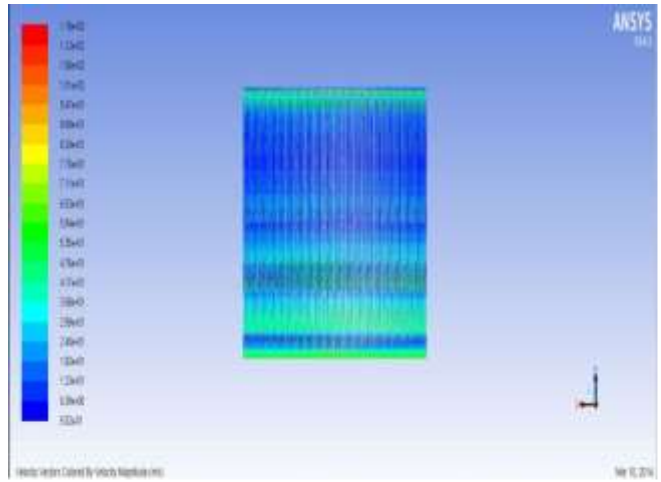

Fig.8 Flow velocity with curve type profile

Fig.9 shows the flow velocity of fluid which is flow inside the radiator tube with curved type geometry. It shows the velocity is maximum in curve type profile with a higher magnitude of 118m/s compared with other two geometry models.

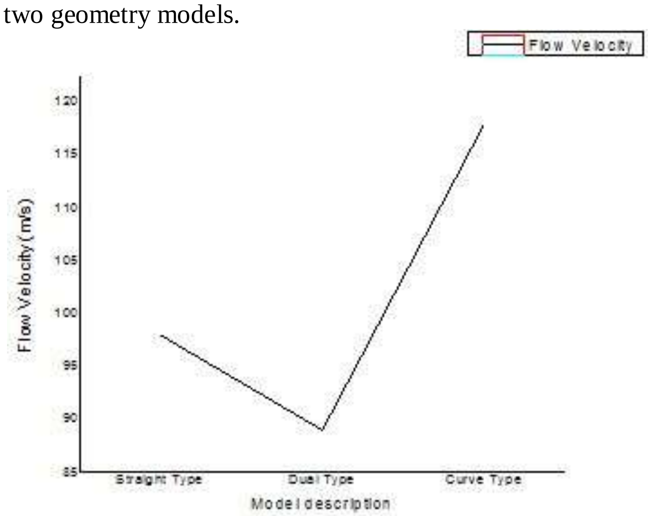

Fig.9 Flow velocity for different flow tube geometry

Table.1 shows the different flow velocity of fluid flow inside the radiator tube. It shows that the radiator tube with curve type profile gives the maximum flow velocity.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

Table.1 flow velocity for different flow tube geometry

Sl.No.	Model description	Flow Velocity (m/s)
1	Straight Type	98
2	Dual Type	89
3	Curve Type	118

Wall temperature

Fig.10 shows the temperature of fluid which is flow inside the radiator tube with curved type geometry. It shows clear that the fluid temperature is minimum in this type of geometry with a lower magnitude of 385 K compared with other two geometry models.

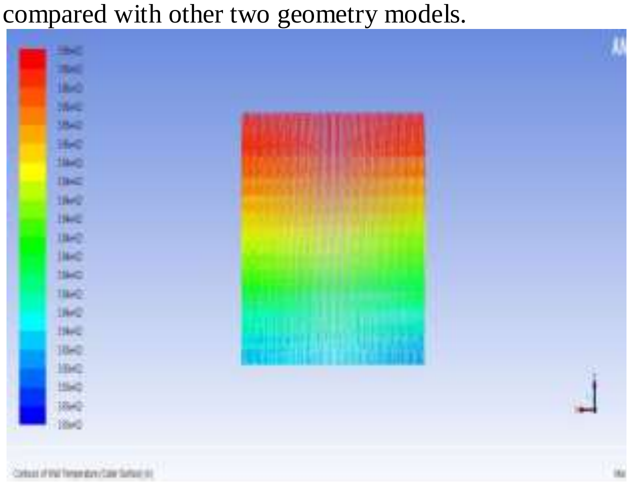

Fig.10 wall temperature for curve type profile

Fig.10 shows the temperature of fluid which is flow inside the radiator tube with curved type geometry. It shows clear that the fluid temperature is minimum in this type of geometry with a lower magnitude of 385 K compared with other two geometry models.

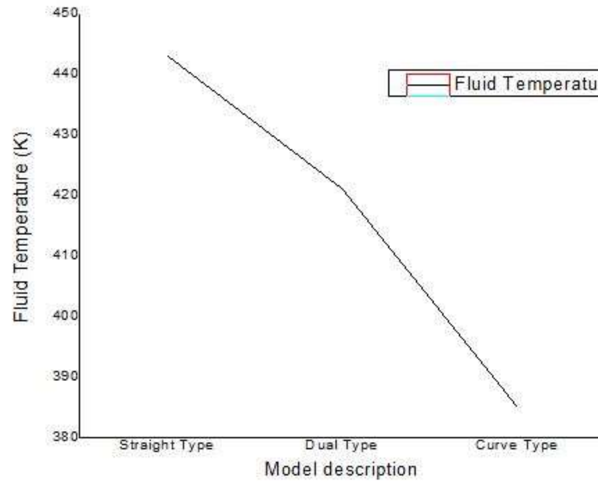

Fig.11 wall temperature for different flow tube geometry

Fig.11 shows the temperature of fluid which is flow inside the radiator tube with curved type geometry. It shows clear that the fluid temperature is minimum with the radiator tube posses the curve type geometry with a lower magnitude of 385 K compared with other two geometry models.

Table.2 shows the different wall temperature of fluid flow inside the radiator tube. It shows that the radiator tube with curve type profile gives the minimum wall temperature.

Table.2 wall temperature for different flow tube geometry

Sl.No.	Model description	Wall Temperature (K)
1	Straight Type	443
2	Dual Type	421
3	Curve Type	385

Conclusion

Thus the analysis has been done on the rate of heat convection of radiator flow tubes and curved tubes using ansys simulation software.

We found that the rate of heat dissipated is high in present type flow tube when compared to single flow tube with curved flow tube. So usage of air type cooling flow dual type flow tube in compared in to best performance to satisfaction in curved type flow tube in our project. Finally radiator tube with curve type gives the maximum velocity distribution and minimum wall

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

temperature compared with other two different geometry.

Future scope for the project

The numerical research studies are going over the radiator full assembly model. Most of the works are concentrated on the design and operating parameters influencing on the performance of the radiator flow tubes.

The numerical model analysis work like this could be proto type of model having to taken the validation result in velocity of air flow and fluid properties with conventional air engine heat conduction values are simulated. The analysis could also reveal the influence of radiator flow tubes section like straight type half round with round shape and different types of flow tubes like present type flow tubes and curved flow tube the numerical model analysis could be done. At next level that multi fluid properties are concerned to flowing at different numerical analyzed to simulate experimentally with practically.

References

1. C.M.Desilva, M.Norazmi, T.Christie, E. Abou-Saba, A. "Computational Flow Modelling of Formula-Sae Side Pods for Optimum Radiator Heat Management" Journal of Engineering Science and Technology, Vol.6, No.1 (2011), 94-108.
2. Salvio Chacko, Dr.Biswadip Shome, Vinod A. Kumar, K.Agarwal, D.R.Katkar "Numerical simulation for improving radiator efficiency by airflow optimization" Engineering Automation Group, Tata Technologies Limited, Pune, India Engineering Research Center, Tata Motors Limited, Pune, India.
3. Pawan.S.Amrutkar, Sangram.R.Patil "Automotive radiator performance–Review" International Journal of Engineering and Advanced Technology (IJEAT)ISSN:2249–8958,Volume-2,Issue- 3,February2013.