International Journal of Thermal Technologies
E-ISSN 2277 – 4114
Available at http://inpressco.com/category/ijtt/

Research Article

Numerical analysis of fluid flow and heat transfer in single tube fin arrangement of an automotive radiator

P. Murugesan*, K. S. Ajith, U. Benazir, R. Gopika and S.Arun Aakash

KSR Institute for Engineering and Technology, Tiruchengode, Namakkal, Tamil Nadu, India

Received 20 Jan 2019, Accepted 23 March 2019, Available online 26 March 2019, **Vol.9, No.1 (March 2019)**

Abstract

This paper deals with CFD analysis of the fluid flow and the heat transfer of the automobile radiator. Normally existing radiator has multi tube fin arrangement. In those tubes a spring is inserted to delay the flow of the fluid thus it transfers more amount of heat. In this arrangement the water from the engine flow through these tubes and get cooled. Here the heat transfer takes place only the peripheral surface which is contact with the fins only. To increase the heat transfer rate the multi tube of the radiator arrangement is replaced by a single plate tube with wire mesh arrangement. Wire mesh arrangement is used to delay the fluid flow thereby increasing the heat transfer rate. The heat transfer of the existing radiator is 20.57W and the proposed design has 77.13W where the heat transfer rate is increased by 3.74 times.

Keywords: *Radiator, Multi tube, Single tube, Heat transfer, Convection*

1. Introduction

An automobile engine temperature is to be controlled by a cooling system, as very higher temperature reduces the engine performances. An exhaust temperature from 1500 to 2000 °C can oxidizes the film of the lubricating oil and seizes the engine. Moreover, large temperature differences may setup thermal stresses and reduces the thermal efficiency too. An efficient cooling system in automobile removes not more than 30% of the heat generated in the combustion chamber such that excess cooling reduces the thermal efficiency of the engine. During cold starting, initial heat is required for the different working parts of engine to reach their operating temperature quickly.

In the current study a tube fin arrangement of an existing radiator is analysed for evaluating the fluid flow and heat transfer characteristics. The overall pressure, temperature and mass flow rate distribution of the coolant and air in and around the single tube-fin arrangement with 100 fins are evaluated. The fluid flow simulation is conducted using commercial software FLUENT. The pressure and temperature distribution along the tube length and tube width are presented and analysed. The results obtained serve as good database for the future investigations.

The radiator of a commercially existing vehicle is chosen for the analysis to bring in the practicality to the study. The details of the geometry of the radiator were obtained by the process of reverse engineering. The dimensions of individual components of the radiator were measured using suitable measuring instruments. The measurements obtained were used to generate the CAD model in PTC Creo 4.0.

1.1Heat Transfer

Flow of heat from high temperature region to low temperature region is called heat transfer. Generally heat can transfer was classified into three ways. They are

- Conduction
- Convection
- Radiation

1.1.1 Conduction: It is a mode of heat transfer occurred between same medium. It is determine by "Fourier's law of conduction". This law states that the rate of heat transfer directly proportional to the negative gradient in the temperature and to the area at right angles to that gradient, through which heat flows. Heat transferred in a one dimensional plane is expressed by follow equation.

$$q=-k(dt/dx)$$

The negative sign in the above formula denotes that the heat was transferred from the high heat region to low heat region.

*Corresponding author *P. Murugesan is working as Head of Mechanical Department; K. S. Ajith, U. Benazir, R. Gopika and S.Arun Aakash are Final Year Mechanical Engineering Students
DOI: https://doi.org/10.14741/ijtt/v.9.1.2

1.1.2Convection: It is a mode of heat transfer occurred between two different mediums. For example boiling of water where the heat transfers from the solid vessel to the water. It has two types. They are natural convection and forced convection. In natural convection the heat was transferred by the density difference between the mediums. In forced convection the heat was transferred due to the external force like velocity etc.

1.1.3 Radiation: it is a mode of heat transfer that the heat will transfer between two objects without any medium. For example radiation from the sun.

1.2 Radiator

It is a type of heat exchanger. Heat exchanger has two types. They are evaporator and condenser. Radiator is one of the groups of condenser. Because it reduces the temperature from high to low. Normal radiator has multi tube and fin arrangement. Due to this the heat of the hot water transfers in the form of convection. In this project the multi tube of the radiator was replaced by the single flat tube.

2. Literature survey

Design of the automobile radiator design was concerned on the following parameters like Shape of radiator core, direction flow of working fluid, frontal area of radiator, Space between fins, space between tube, fin & tube size, coolant mass flow rate, material of fins, pitch of tube, velocity of fluid, air inlet temperature (Hardikkumar 2014)

The air that absorbs the heat due to forced convection gains an in increase in temperature. The study forms a foundation for the fluid flow analysis of an automotive radiator. With the computational time and resources available (S.N. Sridhara 2005)

CFD analysis it was found that the temperature at inlet manifold is uniform. Below the inlet flow rate is high so temperature drop is small. Greater temperature change was found at right and left edges where flow rate is small. By optimized design with the help of CFD we can reduce this non uniformity in temperature distribution (Upendra Kulshrestha 2014)

The computational time and resources available the results obtained were found to be satisfactory. However to account for the variation of the inlet conditions with time as in practical cases, transient analysis can be done (P.K. Trivedi 2012).

Overall heat transfer coefficient is the function of heat transfer coefficient of the air as well as the coolant used (P.S. Kishore 2016)

The efficiency of the internal combustion engine cooling system depends on mainly on the performance of its units. The main unit in his system is radiator (Rinu Sathyan 2016)

Due to convection temperature of fluid flow inside the radiator will decrease, values temperature, velocity and pressure of fluid after analysis are monitored (B. Kishore Kumar 2017)

The varying inlet mass flow rate of coolant the temperature distribution was analyzed and it has been founded that, when the coolant mass flow rate increases the outlet temperature of coolant increase(C. Uma Maheshwari 2016)

3. Design &simulation analysis

The design was created in PTC Creo 4.0 software. First the upper and lower tank of the radiator drawn. Then middle part of radiator core was replaced by a single flat tube. For reducing the fluid flow a wire mesh is kept inside the tube. Then the fins were drawn separately and assembled to the body of the radiator. The assembled radiator was exported as a solid model. The wire mesh option was given by the "Porous zone" option in the meshing section in ANSYS Fluent.

Table 1 Dimension of the radiator

S. No	Parts	Dimensions(all in mm)
1.	Inlet diameter	10
2.	Outlet diameter	10
3.	Tube Dimension (L×B×W)	400×280×20
4.	No. of fins	400
5.	Vertical fins dimensions(L×B×W)	400×24×5
6.	Horizontal fins dimensions (L×B×W)	280×24×5

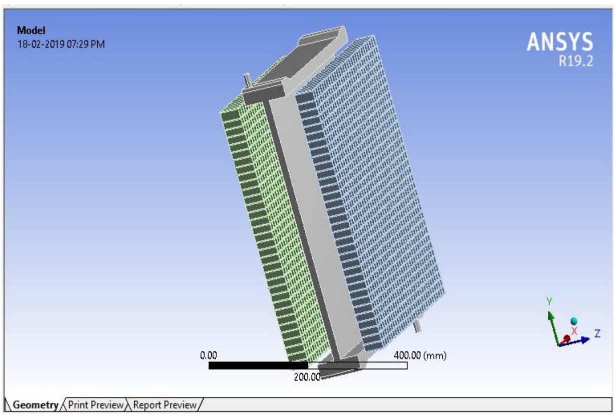

Fig. 1 Exploded view of the modified radiator

Figure no-1 shows the proposed design of our project. The drawn file was saved as an "IGES" format and exported as solid model. Then the drawing was imported in the ANSYS Fluent. The material of the body and the fins were selected. Then mesh was created in the meshing section. After that name for each section in the radiator was given. Then the boundary conditions were selected. Then the number of iterations was given. Then the result was taken from the result section.

3. Boundary Conditions

Boundary conditions are the conditions which describes the working condition of the system. The below table shows the boundary conditions of the radiator.

Table 2 Boundary conditions

S.No	Parameters	Data
1.	Inlet temperature	90°C
2.	No. of elements	15124964
3.	No. of nodes	3496470
4.	Shape of element	Triangle
5.	Velocity of fluid	0.5 m/s
6.	Tube material	Aluminum

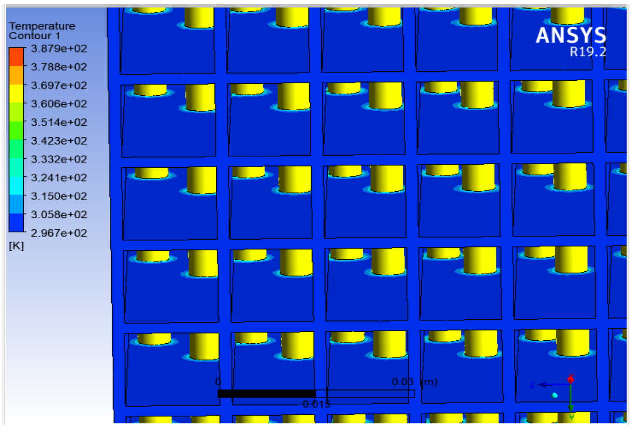

Fig. 2

The figure no-2 shows the heat distribution of normal existing radiator. The above figure shows that the heat will distribute around peripheral of the tubes.

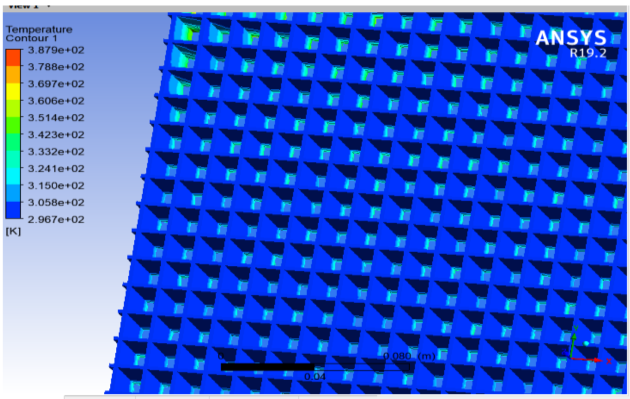

Fig. 3

The figure no-3 shows the heat distribution of the proposed design. In this the temperature was distributed all over the fins. Because the flat plate has more contact than the tubes of normal radiator

Fluid flow is very important for the heat transfer. The velocity of the fluid flow in the existing radiator was slow down by a spring arrangement. For reducing the fluid flow in the proposed model we used a wire mesh arrangement. This arrangement makes the fluid to flow in slow.

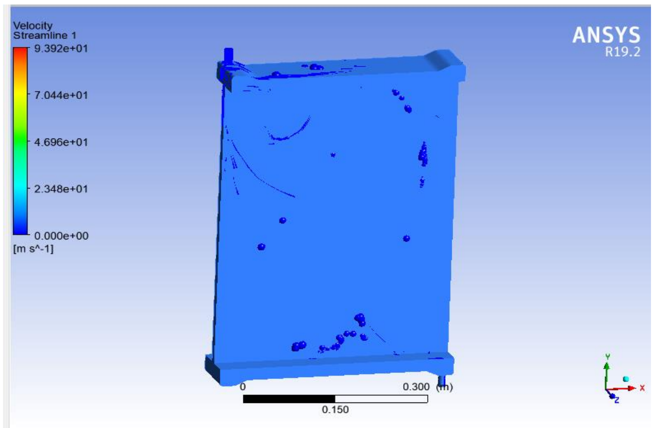

Fig. 4

The figure no-4 shows the fluid flow of the single plate tube radiator. Due to the wire mesh the fluid flow in the radiator was delayed.

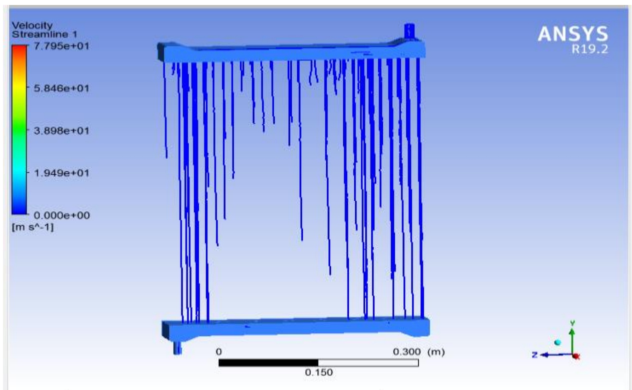

Fig. 5

The figure no-5 shows the fluid flow of the multi tube radiator. Here the water flows from upper tank to the lower tank through the tubes

5. Results

1. The inlet temperature of the normal radiator is 90° and outlet temperature is 86°C.

Heat transfer $Q = m\,C_p\,(T_1\text{-}T_2) = 1.09\times4.718\times(90\text{-}86)$ = 20.57 W

2. The inlet temperature of the proposed radiator 90°C and outlet temperature is 75°C

Heat transfer $Q = m\,C_p\,(T_1\text{-}T_2) = 1.09\times4.718\times(90\text{-}75)$ = 77.13 W

Conclusions

1.) The outlet temperature of the proposed design was 75°C. But in the existing design it reduced only for 86°C. The inlet temperature of the hot fluid is 90°C. Proposed design has the heat transfer rate of 77.13W where the existing radiator has 20.13W which is greater than 3.74 times than the normal radiator.

2.) The flat plate tube has more contact than the radiator core tubes with fin. At the same time the flat plate tube has more area which will be contact with air. So the heat transfer of the proposed design maximum and it can reduce the more temperature of the hot water than the existing radiator.

References

Hardikkumar.B.Patel (2014), Optimization and performance analysis of an automobile radiator using CFD- A review *IJIRST*,Vol.No-1,pp125-126.

S.N. Sridara (2005) CFD analysis of fluid flow and heat transfer in a single tube fin arrangement of an automotive radiator , *ICME*, vol.No-5,pp1-5

Upendra Kulshrestha, (2014) CFD analysis of automobile radiator, *IJERA* ,pp323

C.Uma Maheswari,(2016), CFD analysis of radiator, *IJRTEM,*Vol.No-1,pp61-101

Pawan. S. Amruthkar, (2013)Automotive radiator performance,*IJEAT,* vol.2, pp. 563-565.

Subbareddy. Gangireddy, Kishore Kumar. B (2017), Modeling and CFD analysis of radiator by using nano fluids, *IJPRES,* Vol.No 9, pp.358-368.

Rinu Sathyan,(2016), Analysis of automobile radiator using computational fluid dynamics, *IJLTEMAS*, Vol.No-5, 156-160

P.Mounika, Rajesh K Sharma, P.S. Kishore (2016), Performance analysis of automobile radiator *,IJRMEE* ,Vol.No-3, pp. 35-38.

P.K. Trivedi (2012). Effect of variation in pitch of tube on heat transfer rate in automobile radiator by CFD analysis , *IJEAT,* Vol.No-1, pp. 180-183.