

Optimization of Circular Shaped Automobile Radiator

Ebin Jose ¹, A.V Ramesh ² and Nidheesh P ³

P G Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Arakkunnam, Ernakulam, Kerala, India ¹

Associate Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Arakkunnam, Ernakulam, Kerala, India ²

Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Arakkunnam, Ernakulam, Kerala, India ³

ABSTRACT: Radiator is the major part of the cooling system in an automobile. This paper mainly deals with the effect of shape of the radiator in heat transfer. A circular shaped radiator is proposed instead of the conventional rectangular one. The objective is to optimize the circular shaped radiator for maximum heat transfer by modifying the parameters like tube geometry, tank shape etc. using CFD analysis. About 21% material saving is possible with the new shape and the space requirement is less for circular radiator. Also the heat transfer rate obtained is comparable or slightly higher than that of rectangular radiator.

KEYWORDS: Circular radiator, Automobile, Simulation, Heat transfer rate.

I. INTRODUCTION

In an automobile, fuel and air produce power within the engine through combustion. Only a portion of the total generated power is used to move the automobile, the rest is wasted in the form of exhaust and heat. If this excess heat is not removed, the engine temperature becomes too high which results in overheating of the engine. Thus a cooling system is essential to remove this excessive heat. About 33% of the energy generated by the engine through combustion is lost in heat. Insufficient heat dissipation can result in the overheating of the engine, which leads to metal weakening of engine parts, significant wear between engine parts and finally breakdown of engine. To minimize the stress on the engine as a result of heat generation, automotive radiators must be redesigned to be more compact while still maintaining high levels of heat transfer performance. Now a day the demand is for small cars with high power engine. So size of radiator is an important factor.

The figure1 shows the working of an automobile cooling system. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. As the coolant flows through the tubes of the radiator, heat is transferred through the fins and tube walls to the air by conduction and convection. Different studies have been already done for increasing the heat transfer performance of radiator by taking parameters like coolant flow rate, air velocity, fin configuration etc.

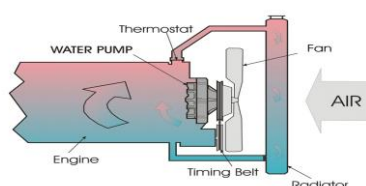


Fig.1: Working of cooling system

Mao-Yu Wen et al. [2] studied three fin configurations to improve the heat transfer of a fin and tube heat exchanger. Out of flat, wavy and compounded fin they have got higher heat transfer for compounded fin. K.Y Leong et al. [3] conducted the performance study of car radiator working with nano fluid based coolants, and have got improved heat transfer. A. Vaisi et al. [4] studied the heat transfer performance of heat exchanger with symmetrical and unsymmetrical fin configurations. They observed higher heat transfer rate for symmetrical arrangement. Chavan D.K et al. [8] studied the heat transfer performance of different radiators and the power consumed by the fan of radiator and they suggested that for optimum efficiency eliminate the corners of the radiator. In this paper optimization is done by changing the shape of radiator. Analysis was done by using CFD software package ANSYS FLUENT 14.

II. COMPUTATIONAL DOMAIN

The figure shows computational domain selected. The computational domain consists of the air volume and water volume.

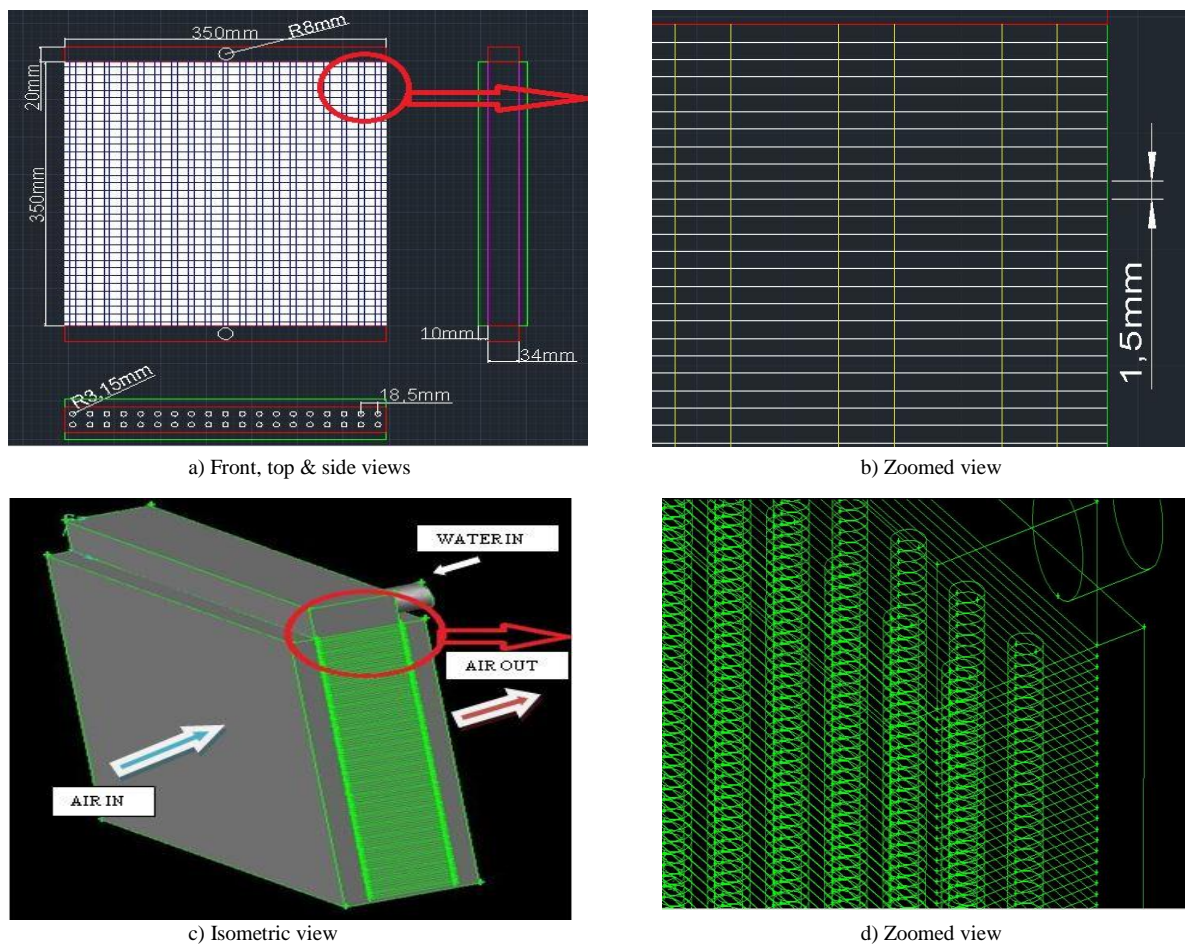


Fig. 2: Computational domain

In fig.(a) the front view, top view and side view of the fin and tube arrangement of the radiator is shown. Fig.(b) represents the zoomed view of fin arrangement. The fig (c) shows the flow direction of water and air and fig (d) shows the zoomed view of the fin and tube arrangement. The above computational domain is taken for rectangular radiator. The diameter of circular shaped radiator is equal to the length of rectangular one and the computational domain is taken accordingly. The arrangement of fin is same for both. The optimization parameters for

circular radiator are coolant inlet tank shape, tube arrangement and number of tubes. Considering a square shaped radiator, if 'D' is the length of one side then area will be D^2 . If we consider a circular shape for the radiator having diameter D, area will be $\pi/4(D^2)$. So while selecting the circular shape for the radiator, about 21.46% reduction in area is possible.

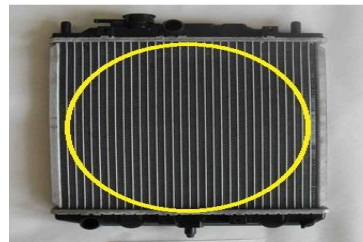


Fig.3: Area comparison

III. EQUATIONS

a) CONTINUITY EQUATION

In fluid dynamics, continuity equation represents the conservation of mass. For a steady incompressible 3 dimensional flow, the continuity equation in Cartesian co-ordinate is given by:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

Where, u, v and w are the velocity components in x, y and z directions respectively.

b) MOMENTUM EQUATION (NAVIER-STOKES EQUATION)

The momentum/Navier-Stokes equations are the basic governing equations for a fluid flow. The following are the scalar form of Navier Stokes equation in x, y and z directions.

i) Along x-direction

$$\left[\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right] \times \rho = - \frac{\partial p}{\partial x} + \rho g_x + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right]$$

ii) Along y-direction

$$\left[\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right] \times \rho = - \frac{\partial p}{\partial y} + \rho g_y + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \quad (3)$$

iii) Along z-direction

$$\left[\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right] \times \rho = - \frac{\partial p}{\partial z} + \rho g_z + \mu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right] \quad (4)$$

Where u, v, w - Velocity components in x, y, z directions respectively. 'p' represents pressure, ρ is the density, μ is the dynamic viscosity and g is the acceleration due to gravity.

c) ENERGY EQUATION

(5)

Where 's' is the dissipation rate and 'k' represents the thermal conductivity

$$\rho C_p \left[\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right] = k \left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right] + S$$

IV. BOUNDARY CONDITIONS AND NUMERICAL METHODOLOGY

The water and air inlet are taken as velocity inlet condition and outlet for both are taken as pressure outlet condition. The radiator top, bottom and sides are taken as wall. In this study, due to lack of computer configuration, quarter portion of radiator is taken for analysis and radiator right side and bottom is taken as symmetry boundary condition. The inlet water velocity is taken as 0.2072m/s and inlet water temperature is given as 328 K. Air velocity is taken from 0 m/s to 2.4 m/s. The inlet air temperature is 304 K. At walls no slip condition (u=v=w=0) is taken. The algorithm used is SIMPLE (Semi Implicit Method for Pressure Linked Equation) and turbulent model used is K- ϵ SST.

V. CODE VALIDATION

Code validation is the process of conformation of CFD algorithm used. Actually code validation means that the CFD algorithm used for solving a fluid problem is correct. In CFD, a given problem can be solved using a wide range of algorithms. Some examples of such algorithms are SIMPLE (Semi Implicit Method for Pressure linked equations), SIMPLEC (Semi Implicit Method for Pressure Linked Equations-Corrected), PISO, COUPLED etc.

The total heat transfer rate and outlet temperature of water and air were the parameters taken for code validation. In this process the experimental results were compared with the results obtained from CFD using a particular algorithm. After validating and finding that the algorithm chosen is suitable for the given case, the same algorithm is used for the optimization purpose also. The suitable algorithm obtained from code validation was SIMPLE (Semi-Implicit Method for Pressure Linked Equations).

VI. RESULTS AND DISCUSSION

Radiators	Heat transfer rate (Watts)	Pressure drop (Pascal)
Conventional rectangular radiator Number of tubes - 38 tubes Tube arrangement - Parallel	893.47 W	29.25 Pa
Normal circular shape Number of tubes - 38 tubes Tube arrangement - Parallel	753.82 W	30.58 Pa
Modified circular radiator Number of tubes - 40 tubes Tube arrangement - Parallel	839.20 W	32.86 Pa
Modified circular radiator Number of tubes - 37 tubes Tube arrangement - Alternative	925.65 W	32.10 Pa
Modified circular radiator Number of tubes - 38 tubes Tube arrangement - Parallel Tank shape changed	963.17 W	30.58 Pa (< 5%)

Table 1. Comparison of CFD results

The conventional rectangular/square radiator has a heat transfer rate of 893.47 watts. When the shape is changed to circular one heat transfer rate is reduced to 753.82 watts due to reduction in total heat transfer area. When number of tubes increased to 40 heat transfer rate also increased and is equal to 839.20 watts. But the pressure drop increased more than 10%. Alternative tube arrangement has got more heat transfer rate compared to 40 tubes geometry. Also it has got higher heat transfer than the conventional rectangular radiator. The pressure drop value is higher for this tube arrangement. The highest heat transfer rate is obtained for the modified tank shape geometry and the pressure drop is also comparable with the conventional rectangular one. The following table shows a comparison of the CFD results (heat transfer rate and air pressure drop) obtained from conventional rectangular radiator and modified circular radiators.

The fig 4 shows the temperature and pressure contours of rectangular and optimized circular radiator. Fig (a) and (b) shows the pressure contour and temperature contour of rectangular radiator. Fig (c) and (d) shows the same for the optimized circular radiator.

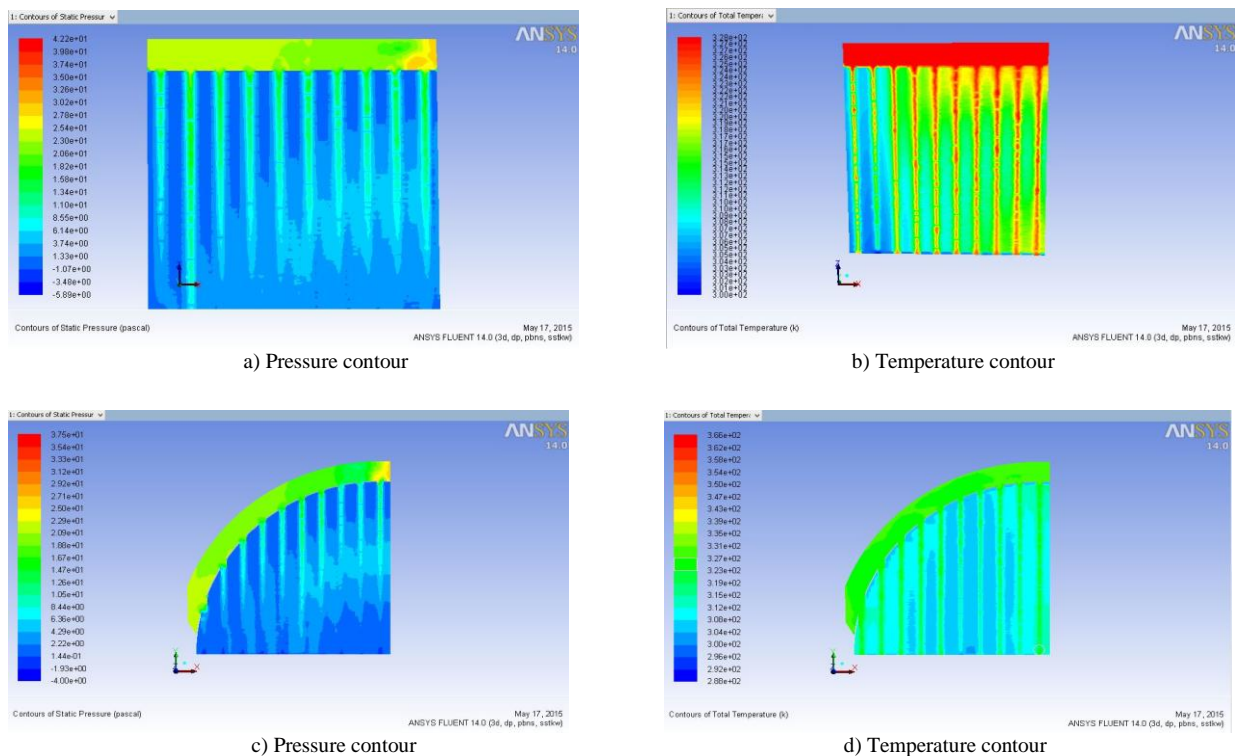


Fig. 4: Pressure and temperature contour for rectangular and optimized circular radiator

VII. CONCLUSION

For CFD analysis the geometry model and mesh were created in Gambit and the code validation is successfully done by comparing experiment and CFD results. The optimization is successfully done by modifying the number of tubes, tube arrangement and coolant tank shape. The optimized circular radiator requires 21.46% less occupancy area compared to rectangular one. While comparing with the rectangular radiator, the optimized model requires less material for fabrication. Hence the cost of production will reduce, once the dies are created. The optimized model has got 7.8% higher heat transfer rate than the conventional model. The pressure drop is also comparable with the conventional one.

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization

Vol.4, Special Issue 12, September 2015

2nd International Conference on Emerging Trends in Mechanical Engineering (ICETME-2015)

On 3rd, 4th & 5th September 2015

Organized by

Department of Mechanical Engineering, Toc H Institute of Science & Technology, Ernakulam-682313, India

REFERENCES

1. S. Vithayasai, T. Kiatsiriroat, A. Nuntaphan; *Effect of electric field on heat transfer performance of automobile radiator at low frontal air velocity*; *Applied thermal engineering*; 2006
2. Mao-Yu Wen, Ching-Yen Ho; *Heat-transfer enhancement in fin-and-tube heat exchanger with improved fin design*; *Applied Thermal Engineering*; 2008, 1050–1057
3. 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*; 2010, 2685-2692
4. A. Vaisi, M. Esmailpour, H. Taherian; *Experimental investigation of geometry effects on the performance of a compact louvered heat exchanger*; *Applied Thermal Engineering*; 2011, 3337-3346
5. Junjanna G.C, Dr. N. Kulasekharan, Dr. H.R. Purushotham; *Performance improvement of a Louver-finned automobile radiator using CFD analysis*; *International journal of engineering research and technology*, Vol. 1, Issue 8, 2012
6. S.M Peyghambarzadeh, S.H Hashemabadi, M. Naraki Y. Vermahmoudi; *Experimental study of overall heat transfer coefficient in the application of dilute nanofluids in the car radiator*; *Applied thermal engineering*; 2012.
7. S.M Peyghambarzadeh, S.H Hashemabadi, S.M. Hosini, M. Seifi Jamnani; *Experimental study of heat transfer enhancement using water/ethylene glycol based nano fluids as a new coolant for car radiators*; *International communications in heat and mass transfer*; 2012.
8. Chavan D.K, Tasgaonkar G.S; *Study, analysis and design of automobile radiator proposed with CAD drawings and geometrical model of fan*; *International journal of mechanical and production engineering research and development*; Vol. 3, Issue 2, 2013, 137-146
9. Pawan S. Amrutkar, Sangram R. Patil; *Automotive Radiator Performance – Review*; *International Journal of Engineering and Advanced Technology (IJEAT)*; 2013
10. Deepak Chintakayala, Rajamanickam C S; *CFD analysis of fluid flow and heat transfer of an automotive radiator with nano fluid*; *IEEE*; 2013.