

# Analysis of Automobile Radiator Using Computational Fluid Dynamics

Rinu Sathyan

*Assistant Professor in Mechanical Engineering, SCT College of Engineering, Trivandrum, Kerala, India*

**Abstract-** Radiators are a type of heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. Upwards of 33% of energy generated by the engine through combustion is lost in heat. Insufficient heat dissipation can result in the overheating of engine, which leads to the breakdown of the lubricating oil, metal weakening of engine parts, and significant wear between engine parts. 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 level of heat transfer components. This lead to the increased demand on the power packed radiators, which can dissipate maximum amount of heat for any given space. This project aims to do a comparison between ordinary straight tube radiator and a new helical tube radiator. The modelling is done using SolidWorks. The fluid flow analysis is done with Ansys Fluent

## I. INTRODUCTION

Most modern cars use aluminium radiators. These radiators are made by brazing thin aluminium fins to flattened aluminum tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator.

The tubes sometimes have a type of fin inserted into them called a tabulator, which increases the turbulence of the fluid flowing through the tubes. If the fluid flowed very smoothly through the tubes, only the fluid actually touching the tubes would be cooled directly. The amount of heat transferred to the tubes from the fluid running through them depends on the difference in temperature between the tube and the fluid touching it. So if the fluid that is in contact with the tube cools down quickly, less heat will be transferred. By creating turbulence inside the tube, all of the fluid mixes together, keeping the temperature of the fluid touching the tubes up so that more heat can be extracted, and all of the fluid inside the tube is used effectively. Radiators usually have a tank on each side, and inside the tank is a transmission cooler. In the picture above, you can see the inlet and outlet where the oil from the transmission enters the cooler. The transmission cooler is like a radiator within a radiator, except instead of exchanging heat with the air, the oil exchanges heat with the coolant in the radiator.

This projects deals with the comparison of a normal straight tube radiator with modified one. Radiator of TATA INDIGO/INDICA is taken as a standard model and its tubes are replaced with helical tubes. Analysis of the radiator is done through CFD (Computational Fluid Dynamics) using ANSYS Fluent software. The temperature distribution and the fluid flow inside the radiator is analyzed and various plots are plotted.

## II. METHODOLOGY

*Generic Steps To Solving Problem in Fluent:*

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

*Build Geometry*

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinates system within ANSYS.

*Define Material Properties*

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

*Generate Mesh*

At this point ANSYS understands the makeup of the part. Now define how the Modeled system should be broken down into finite pieces.

*Define Boundary Conditions*

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

**Obtain Solution**

This is actually a step, because ANSYS needs to understand within what state (steady state, transient... etc.) the problem must be solved

**Present the Results**

After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

**III. SPECIFICATIONS OF THE RADIATOR:**

TABLE 1  
 SPECIFICATION

Details	Dimensions (mm)
Length	644
Width	30
Height	360
Tube length	584
Tube inner diameter	7
Tube outer diameter	9

Tube Material: Aluminum  
 Coolant Fluid: A Mixture Of Ethylene Glycol (25%) & Water (75%)

**IV. BOUNDARY CONDITIONS**

The main boundary condition include, a mass flow rate inlet boundary condition where used in the inlet nozzles. The cylindrical shaped geometries are the wall. At the outlet, the pressure outlet (atmospheric pressure) boundary condition was used. And all other portions are considered as the wall boundary with convective heat transfer surfaces.

**INLET**

Select inlet and change type to mass flow inlet  
 Enter the mass-flow rate  
 Enter the value of inlet gauge pressure =  $1.5e5$  Pa  
 Provide the inlet temperature = 368K  
 Change the option under Direction Specification to "Normal to boundary"

**OUTLET**

Select the outlet and change type to 'Pressure-outlet'  
 Enter the value for outlet gauge pressure = 0 Pa  
 Change the option under Direction Specification to "Normal to boundary"

**WALL**

Select wall and change type to 'Wall'  
 Select the material for the wall:  
 Aluminium  
 Choose convection as the heat transfer mechanism employed in wall.  
 Enter the value for heat transfer coefficient as  $90$  W/m<sup>2</sup>K.  
 Enter wall thickness as 1mm  
 Enter the free stream temperature on wall

**V. CONTOUR PLOTS**

At first the analysis is done by keeping inlet temperature of coolant and free stream temperature constant and temperatures are obtained for different mass flow rates. Varied input mass flow rates are given in the table.

**Temperature Contour**

*1. Mass flow rate 500kg/h*

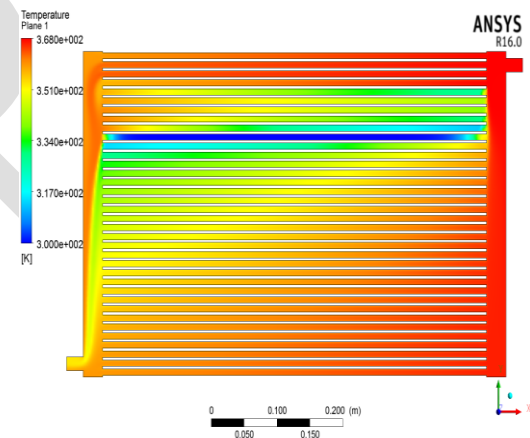


Fig. 1. Temperature contour for mass flow rate 500kg/h

*2. Mass flow rate 1000kg/h*

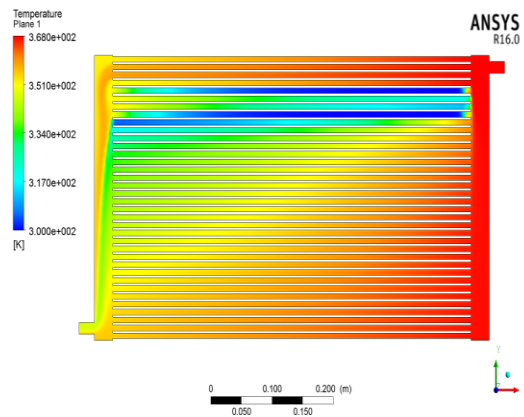


Fig. 2. Temperature contour for mass flow rate 1000kg/h

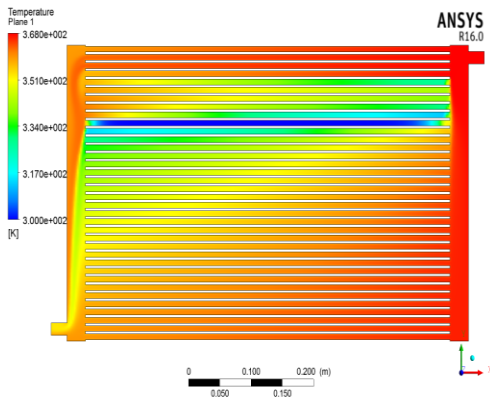


Fig. 3. Temperature contour for mass flow rate 1500kg/h

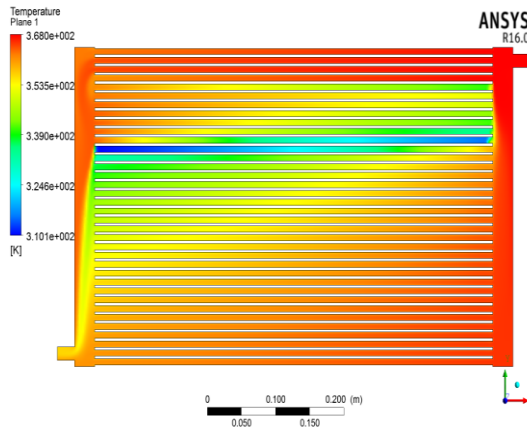


Fig.4. Temperature contour for mass flow rate 2000kg/h

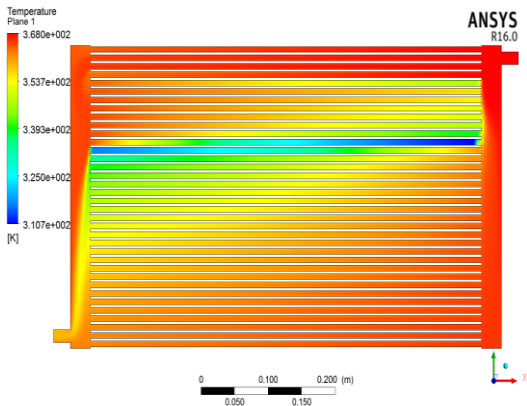


Fig.5.. Temperature contour for mass flow rate 2500kg/h

The temperature of coolant at inlet and outlet corresponding to various mass flow rates obtained during analysis are tabulated below

TABLE 2  
 VARIOUS MASS FLOW RATES

Mass flow rates in Kg/hr	Mass flow rates in Kg/s	Radiator inlet temperature (K)	Radiator outlet temperature (K)
500	0.138	368	340.45
1000	0.277	368	354.45
1500	0.416	368	358.39
2000	0.555	368	359.9
2500	0.694	368	361.07

*Analysis by changing free stream air temperature*

The second analysis was done by changing free stream temperature (that is temperature of cooling air) for about five values which is listed below.

TABLE 3

VALUES OF INPUT FREE STREAM TEMPERATURE

Air inlet temperature °C	Air inlet temperature K
0	273
10	283
20	293
30	303
40	313

The temperatures of coolant at inlet and outlet corresponding to various free stream air temperatures obtained during analysis are given below

TABLE 4

TEMPERATURE OF COOLANT AT INLET AND OUTLET CORRESPONDING TO VARIOUS MASS FLOW RATES

Air inlet temperature K	Coolant inlet temperature K	Coolant outlet temperature K
273	368	329.57
283	368	333.62
293	368	337.68
303	368	341.73
313	368	345.8

*New design with standard dimensions*

The radiator model was redesigned. The straight tubes were replaced with helical tubes. The length, width and height were kept constant. The boundary conditions were kept the same. As the tubes were replaced with helical its overall length increases.

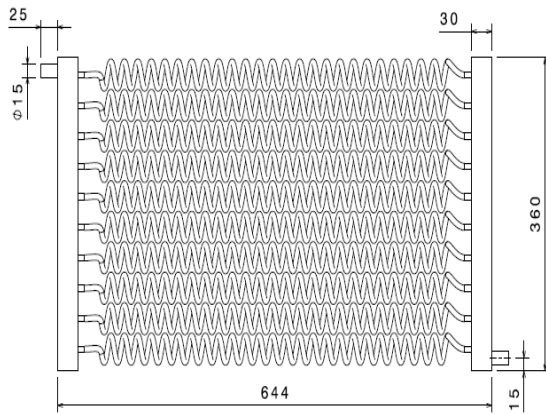


Fig.6. Helical tube Radiator

TABLE 5  
 DIMENSIONS OF NEWLY DESIGNED RADIATOR

Details	Dimensions (mm)
Length	644
Width	30
Height	360
Tube length	2813.16
Tube inner diameter	7
Tube outer diameter	9
No. of tubes	10
Diameter of helix	30
Pitch of helix	20
Number of turns of helix	26

While analyzing the results we obtained, the following things were observed

- The temperature at the outlet is found to be 356°C.
- The temperature drop of fluid in the tubes is due to the increase in surface area. The increase in surface area results in increase in heat transfer thus resulting in better cooling effect.

TABLE 6  
 COMPARISON OF STRAIGHT TUBE AND HELICAL TUBE RADIATOR

Radiator models	Mass flow rate (Kg/s)	Coolant inlet temperature (K)	Coolant outlet temperature (K)	Overall heat transfer coefficient (W/m <sup>2</sup> K)	Effectiveness %
For straight tube radiator	0.555	368	359.9	882.20	33
For helical tube radiator	0.555	368	356	820.05	46

## VI. RESULT AND DISCUSSION

- From the two readings it is found that temperature drop is 4° higher for helical tube model than a straight tube model
- Hence heat transfer rate is higher for a helical tube model
- Its effectiveness gets increased to 13% more than straight tube model

Our objective is to design a helical radiator which is suited for TATA INDICA with same efficiency of its original radiator but with reduced size. For this reverse engineering has been done. It is found that overall heat transfer coefficient ranges in same value. We assumed the overall heat transfer coefficient of helical tube radiator as 820 W/m<sup>2</sup>K. The inlet and outlet conditions are given same as that of straight tube radiator. That is

- Air inlet temperature= 308K
- Air outlet temperature= 333K
- Inlet temperature of coolant= 368K

From the listed details total surface area of helical tubes are found. Then overall length of tubes is found out. For obtaining new design first we assumed that there are 9 rows of tubes. Then the overall length is divided into nine parts which gives length of a single tube. This dimension is divided with length of a single helix to obtain number of turns. And thus we obtained a new radiator with reduced dimensions.

For the new dimension the percentage reduction in length is about 31.6% & height of about 8%. This result shows that the newly designed radiator is compact and consumes only less space.

*New design with reduced dimensions*

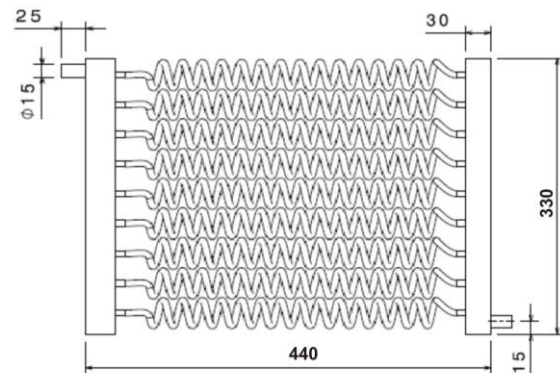


Fig.7. Sketch of newly designed Radiator

TABLE 7  
 DIMENSIONS OF NEWLY DESIGNED RADIATOR

Details	Dimensions (mm)
Length	440
Width	30
Height	330
Tube length	1731.17
Tube inner diameter	7
Tube outer diameter	9
No. of tubes	9
Diameter of helix	30
Pitch of helix	20
Number of turns of helix	16

## VII. CONCLUSION

From the above results, the following conclusions can be made

- 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.
- The length of radiator gets decreased by 204mm from its original dimensions.
- Height also gets decreased to 30mm from its original dimensions.
- It is not suitable to increase or decrease the pitch of helical tubes which will affect efficiency.

- Width of the radiator was not changed and can be modified to lesser dimensions.
- Also the modified design is preferred due to better performance and size whereas the old model is preferred due to low cost.

## REFERENCES

- [1]. C. Oliet, A. Oliva, J. Castro & C.D. Pe´rez-Segarra, Applied Thermal Engineering 27 (2007) 2033–2043 Science Direct, ‘**Parametric studies on automotive radiators**’.
- [2]. Hardikkumar B Patel & Deepu Dinesan, IJRST Volume I Issue 12 May 2015, ‘**Performance Analysis of an Automobile Radiator using CFD**’.
- [3]. Brandon Fell, Scott Janowiak, Alexander Kazanis, Jeffrey Martinez, ‘**High Efficiency Radiator Design for Advanced Coolant**’.
- [4]. Krunal Suryakant Kayastha, Volume 11, Issue 01 (January 2015), PP.24-35 IJERD, ‘**CFD Simulation and Heat Transfer Analysis of Automobile Radiator using Helical Tubes**’.
- [5]. P. K. Trivedi, N. B. Vasava, IJEAT ISSN: 2249 – 8958, Volume-1, Issue-6, August 2012, ‘**Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by CED Analysis**’.
- [6]. Pawan S. Amrutkar, Sangram R. Patil & S. C. Shilwant, ISSN 2277-4785 Vol. 3, Issue 4, Oct 2013, 1-10; ‘**Automotive Radiator - Design And Experimental Validation**’.
- [7]. Understanding Automotive Cooling Systems [www.secondchancegarage.com/public/cooling-system-2.cfm](http://www.secondchancegarage.com/public/cooling-system-2.cfm)
- [8]. How Car Cooling Systems Work by karim nice [www.auto.howstuffworks.com/cooling-system6.htm](http://www.auto.howstuffworks.com/cooling-system6.htm)
- [9]. Introduction To ANSYS Software and why is it used [www.mechtutorials.net/2016/01/introduction-to-ansys-software-and-use-of-ansys.html](http://www.mechtutorials.net/2016/01/introduction-to-ansys-software-and-use-of-ansys.html)
- [10]. Designing a More Effective Car Radiator The challenge: To determine the design parameters of a smaller radiator assembly capable of dissipating the same amount of heat as the original assembly. © Maplesoft, a division of Waterloo Maple Inc., 2008 [www.maplesoft.com/applications/view.aspx?SID=6403&view=html](http://www.maplesoft.com/applications/view.aspx?SID=6403&view=html)