International Journal of Engineering Research & Management Technology
ISSN: 2348-4039
April-2021 Volume 8, Issue-2
Email: editor@ijermt.org www.ijermt.org

DESIGN AND THERMAL ANALYSIS OF AUTOMOBILE NORMAL FINS USING CAE TOOLS

Mr. Md. Abdul Mannan
Assistant Professor

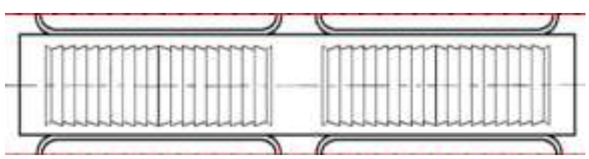
Mr. L. Vemana
Assistant Professor

Dept. of Mechanical Engineering, Kodada Institute of Technology and Science for Women, Kodad, Suryapet (D), Telangana, India

Abstract: Increasing sharp edges work on the speed of force movement or decreasing convection. Equilibrium is only for the clarification of loosening up the surface locale to help with warming transmission like a motor, heat exchanger, focal processor, or to the power-making surface region. The possible increase of doing a warm assessment on a level edge is that it will tell how far heat is scattered. This paper's fundamental occupation is to design and survey the warm properties of the rectangular front with separating estimation and content. In this thought, the moderate power exchangers have rectangular/cross congruity plates and louvered sharp edge plates. The present review uses the assessment gadget to play out a numerical report on an insignificant power exchanger at different mass stream rates. The computational region is seen from making and endorsing out of the present numerical system is spread out first. Later the numerical assessment is loose by changing picked numerical and stream limits like louver pitch, wind stream rate, water stream rate, edge, and louver thickness, by fluctuating every single end, and the results are checked out. Showing is acted in CATIA and the material used for balances of the radiator is Aluminum composite 6061 and evaluation is acted in ANSYS. Ideas should be made on the ideal characteristics and settings will be spread out on the elements tried, for the picked more subtle power exchanger.

I- INTRODUCTION

Radiators are heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. The majority of radiators are constructed to function in automobiles, buildings, and electronics. The geometric construction details of the louvered, finned compact heat exchanger studied presently. The louver with a symmetry pattern domain is chosen for the present numerical study. The fin geometry indicates that it has a periodicity in the tube height-wise (span) direction and in the lateral direction too.



Due to the symmetry of the geometry, the computational domain is confined to one fin pitch in the span-wise direction and one tube pitch in the lateral direction, which is highlighted with a dashed red line. Since the geometry of the heat exchanger is periodic in nature, a smaller portion of the geometry is considered for the computational analysis. It mainly consists of an upper tank and lower tank and between them is a core. The upper tank is connected to the water outlets from the engines jackets by a hose pipe and the lover tank is connect to the jacket inlet through water pump by means of hose pipes. There are 2-types of cores: Tubular and Cellular. When the water is flowing down through the radiator core, it is cooled partially by the fan which blows air and partially by the air flow developed by the forward motion of the vehicle. As shown through water passages and air passages, wafer and air will be flowing for cooling purpose. It is to be noted that radiators are generally made out of copper and brass and their joints are made by soldering.

There are a lot of different factors that contribute to a radiator's ability to cool. Construction materials, methods, and design all play a role. While we're not going to cover all the different aspects of radiator design, we do need to look at how a radiator works. The hot coolant coming from the engine flows into one radiator tank and then passes through the tubes in the radiator core to the other tank. As it flows through the tubes, the heat is transferred to the tube walls and dissipated by the radiator fins. The more tube surface area there is, then the better the radiator can cool.

The process to build either of these radiators is identical. Where the tanks are attached is the only difference. In a down-flow radiator you have a tank attached to the top and bottom of the radiator core. The coolant enters the top tank and flows down to the bottom tank. As you've probably guessed by now, a cross-flow radiator has tanks on the left and right side. Coolant enters one side and flows across to the other.

Assuming you have a cross-flow and down-flow radiator made from the same materials and build methods, then they will provide the same level of cooling. So why the difference? It really comes down to space under the hood. Depending on your vehicle or equipment, you may be able to fit a larger cross-flow radiator instead of a down-flow. This goes back to surface area. If the two radiators are the same size, they will cool the same. In the cases where you can fit a larger radiator of a different flow style, then you can increase the cooling ability. Which radiator to use primarily comes down to the space requirements.

II - LITERATURE SURVEY

The counter current design is the most efficient, in that it can transfer the most heat from the heat (transfer) medium due to the fact that the average temperature difference along any unit length is greater. For efficiency, heat exchangers are designed to maximize the surface area of the wall between the two fluids, while minimizing resistance to fluid flow through the exchanger. The exchanger's performance can also be affected by the addition of fins or corrugations in one or both directions, which increase surface area and may channel fluid flow or induce turbulence.

A heat exchanger is a piece of equipment built for efficient heat transfer from one medium to another. The media may be separated by a solid wall, so that they never mix, or they may be in direct contact. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air.

Detail History

Oliet et al. (2007) studied different factors which influence radiator performance. It includes air, fin density, coolant flow and air inlet temperature. It is catch that heat transfer and performance of radiator strongly affected by air & coolant mass flow rate. As air and coolant flow increases cooling capacity also increases. When the air inlet temperature increases, the heat transfer and thus cooling quantity decreases. Smaller fin spacing and greater louver fin angle have higher heat transfer. Fin density may be increased till it blocks the air flow and heat transfer rate reduced. Sulaiman et al. (2009) use the computational Fluid Dynamics (CFD) modeling simulation of air flow distribution from the automotive radiator fan to the radiator. The task undertook the model the geometries of the fan and its surroundings is the first step. The results show that the outlet air velocity is 10 m/s. The error of average outlet air velocity is 12.5 % due to difference in the tip shape of the blades. This study has shown that the CFD simulation is a useful tool in enhancing the design of the fan blade. In this paper this study has shown a simple solution to design a slightly aerodynamic shape of the fan hub.

Chacko et al. (2005) used the concept that the efficiency of the vehicle cooling system strongly rely on the air flow towards the radiator core. A clear understanding of the flow pattern inside the radiator cover is required for optimizing the radiator cover shape to increase the flow toward the radiator core, thereby improving the thermal efficiency of the radiator. CFD analysis of the baseline design that was validated against test data showed that indispensable area of re-circulating flow to be inside the radiator cover. This recirculation reduced the flow towards the radiator core, leading to a reputation of hot air pockets close to the radiator surface and subsequent disgrace of radiator thermal efficiency. The CFD make able optimization led to radiator cover configuration that eliminated these recirculation area and increased the flow towards the radiator core by 34%. It is anticipated that this increase in radiator core flow would important to increase the radiator thermal efficiency. Jain et al. (2012) presented a computational fluid dynamics (CFD) modeling of air flow to divide among several from a radiator axial flow fan used in an acid pump truck Tier4 Repower. CFD analysis was executed for an area weighted average static pressure is variance at the inlet and outlet of the fan. Pressure contours, path line and velocity vectors were plotted for detailing the flow characteristics for dissimilar orientations of the fan blade. This study showed how the flow of air was intermittent by the hub obstruction, thereby resulting in unwanted reverse flow regions. The different orientation of blades was also considered while operating CFD analysis. The study revealed that a left oriented blade fan with counterclockwise rotation 5 performed the same as a right oriented blade fan with rotating the clockwise direction. The CFD results were in accord with the experimental data measured during physical testing.

Singh et al. (2011) studied about the issues of geometric parameters of a centrifugal fan with backward- and forward-curved blades has been inspected. Centrifugal fans are used for improving the heat dissipation from the internal combustion engine surfaces. The parameters studied in this study are number of blades, outlet angle and diameter ratio. In the range of parameters considered, forward curved blades have 4.5% lower efficiency with 21% higher mass flow rates and 42% higher power consumption compared to backward curved fan. Experimental investigations suggest that engine temperature drop is significant with forward curved blade fan with insignificant effect on mileage. Hence, use of forward fan is recommended on the vehicles where cooling requirements are high. The results suggest that fan with different blades would show same an action below high-pressure coefficient. Increase in the number of blades increases the flow coefficient follow by increase in power coefficient due to better flow guidance and reduced losses.

III - PROBLEM STATEMENT AND OBJECTIVES

The objective of this project work is to successfully develop a design of a heat exchangers have fins, louvers and tubes. The mechanism is to be reliable, simple, cost-effective and feasible. The aim of this paper is to provide and to perform a numerical study on a compact heat exchanger at different mass flow rates. So as to enable added by modifying chosen geometrical and flow parameters. In this project, there is the comparison between Louvered Fin and Cross Fin of the heat flow of the Heat Exchanger. This system is also supposed to enhance engines efficiency as the side force felt by a car engine temperature is comparatively less.

The methodology adopted to use standard and presently used components in design rather than to design all components from ground up. The advantage of this method is that, you do not have to spend ridiculous amount and time in testing the integrity of each part as they have already proved their worth in real world applications. Initially the frame design was adopted from already existing fins of radiator and minor changes were made to suite our purpose, the radiator mechanism first devised was based on using power screw driven by condenser and lowering each area of fin of the radiator. This mechanism was later dropped in testing phase due to following disadvantages.

1. It has extended by modifying chosen geometrical and flow parameters at the suitable temperature for a car.
2. Wear and tear of material coating and rust formation in the heat exchanger.
3. The system doesn't have compact heat exchanger for high optimal values and settings will be based on the variables.

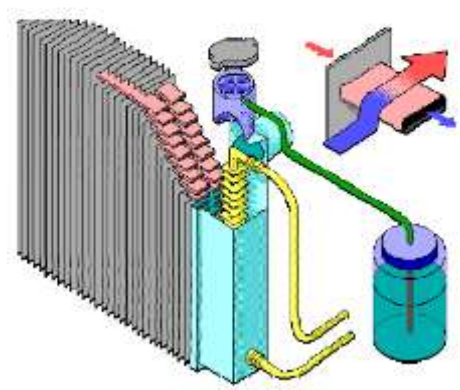
Due to these disadvantages, the power screw design was dropped and a fully new design was defined. The software to be used in design is Catia V5 and testing of design is Ansys.

IV - WORKING MECHANISM

Working of Automobile Radiators

Almost all automobiles in the market today have a type of heat exchanger called a radiator. The radiator is part of the cooling system of the engine as shown in Figure below. As you can see in the figure, the radiator is just one of the many components of the complex cooling system. Coolant path and Components of an Automobile Engine Cooling System Most modern cars use aluminum radiators. These radiators are made by brazing thin aluminum 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.



The louver arrangement in a fin used in an automotive radiator. Although lot of work has been done so far in the computational analysis for the compact heat exchangers, validation of an experimentally tested domain and conducting analysis of modified designs to optimize the design and improve performance on the same domain was not reported so far. This forms the motivation of the present work.

Thus, the objective of the present work is to identify an experimental work from literature, perform computational analysis for the domain studied experimentally to validate the present numerical work. The second objective is to perform geometrical and flow parameter study on the domain identified by varying louver pitch, air flow rate, water flow rate, fin and louver thickness, one parameter at a time. Comparison of these numerical results will help in identifying the optimal combination of geometrical and flow parameters for the domain selected.

V - DESIGN METHODLOGY OF AUTOMOBILE RADIATOR FINS

Introduction to CATIA

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systems. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systems product lifecycle management software suite. CATIA competes in the high-end CAD/CAM/CAE market with Cero Elements/Pro and NX (Unigraphics).

CATIA can be applied to a wide variety of industries, from aerospace and defense, automotive, and industrial equipment, to high tech, shipbuilding, consumer goods, plant design, consumer packaged goods, life sciences, architecture and construction, process power and petroleum, and services. CATIA V4, CATIA V5, Pro/ENGINEER, NX (formerly Unigraphics), and Solid Works are the dominant systems.

Fig: Model design of Automobile Radiator in CATIA-V5

VI - ANALYSIS OF AUTOMOBILE RADIATOR FINS

Procedure for FE Analysis Using ANSYS:

The analysis of the Automobile Radiator is done using ANSYS. For compete assembly is not required, is to carried out by applying moments at the rotation location along which axis we need to mention. Fixing location is bottom legs of rod assembly machine.

Preprocessor

In this stage the following steps were executed:

- **Import file in ANSYS window**
  File Menu > Import> STEP > Click ok for the popped up dialog box > Click Browse" and choose the file saved from CATIAV5R20 > Click ok to import the file

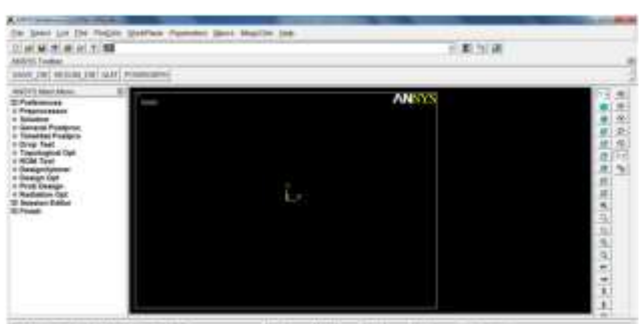


Fig. Import panel in Ansys.

VII - DISCUSSION ON ANALYSYS RESULT

Results of Nodal Temperature analysis:

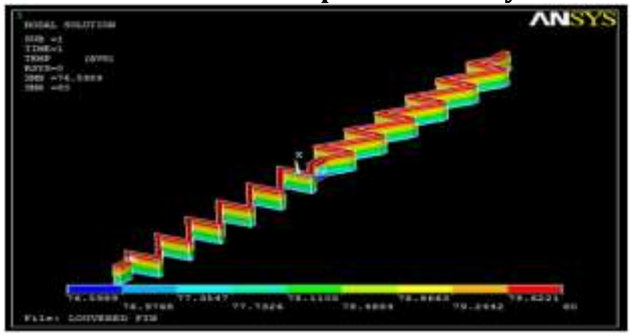


Fig: Nodal Temperature of LOUVERED FIN

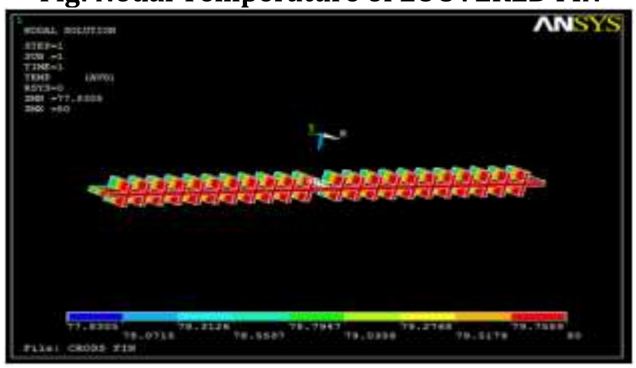


Fig: Nodal Temperature of CROSS FIN

Results of Thermal Gradient analysis:

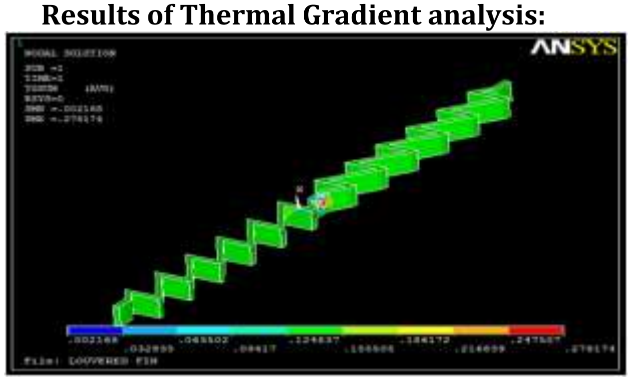


Fig: Thermal Gradient of LOUVERED FIN

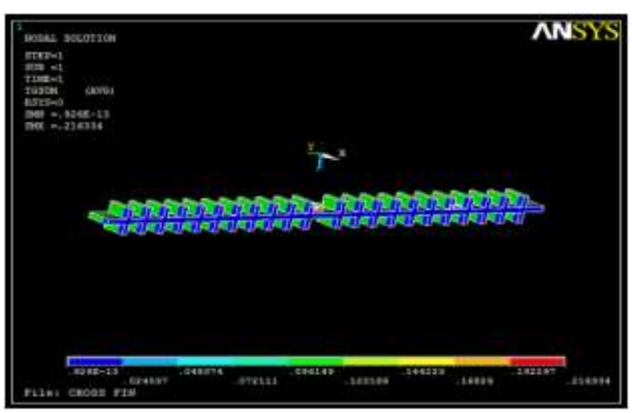


Fig: Thermal Gradient of CROSS FIN

Results of Thermal Flux analysis:

Fig: Thermal Flux of LOUVERED FIN

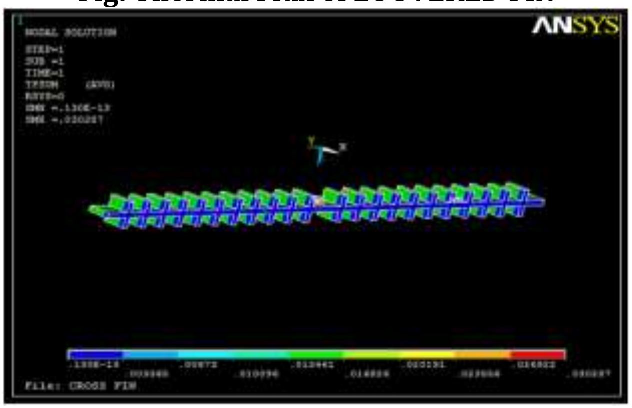


Fig: Thermal Flux of CROSS FIN

Results of Heat Flow analysis:

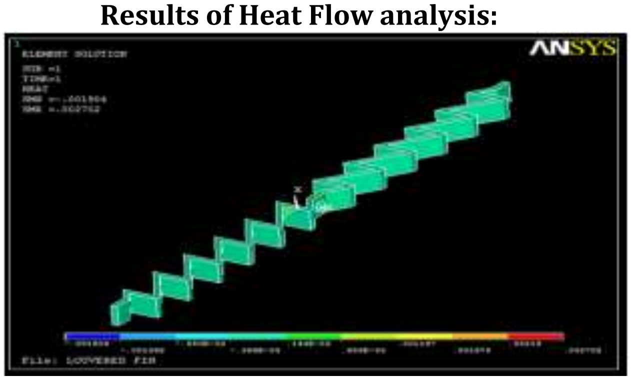


Fig: Heat Flow of LOUVERED FIN

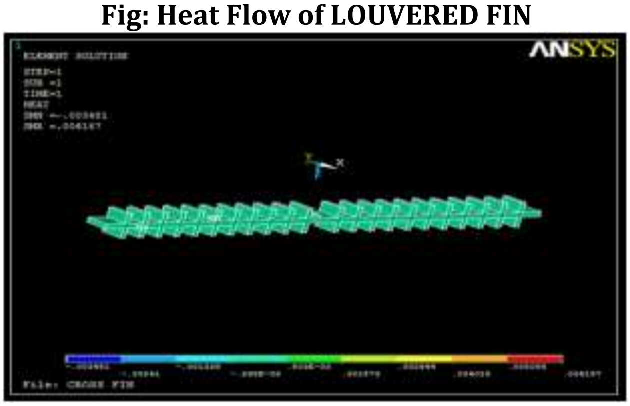


Fig: Heat Flow of CROSS FIN

VIII - CONCLUSION

In this project a radiator is designed, it has been modified by specifying louver fins. 3D model is designed in Catia.

Table for Results

S.No	Radiator Frame	Louvered Fin	Cross Fin	Louvered Fin Rod
Nodal Temperature	36.03	76.59	77.83	76.52
Temperature Gradient	2.04	0.278	0.216	0.89
Thermal Flux	0.28	0.038	0.030	0.12
Heat Flow	0.79	0.002	0.006	0.02

The analysis tool Ansys is used to perform thermal analysis on components of radiator at different areas. By observing the analysis results, the nodal temperature is increased by 76.5; temperature gradient is increased by 0.278 for the modified model of the radiator with louvered fins. Heat transfer analysis is performed to analyze the heat transfer rate to determine the thermal flux. The material taken is Aluminum alloy 6061 for thermal analysis. By observing the thermal analysis results, and thermal flux rate is 0.0389; the Heat flow rate is 0.0027 on the surface medium for the modified model of radiator.

REFERENCES

- Performance Improvement of a Automobile Radiator Using Conjugate Thermal CFD Analysis by Junjanna G.C
- Study on Performance Evaluation of Automotive Radiator by JP Yadav and Bharat Raj Singh
- Performance Investigation of an Automotive Car Radiator Operated With Nano fluid as a Coolant by Durgesh kumar Chavan and Ashok T. Pise Sahin
- R. Saidur, K.Y. Leong and H.A. Mohammad, A Review on Applications and Challenges of Nanofluids. Renewable and Sustainable Energy Reviews, 15, 3 (2011), 1646–1668.
- Pelaez, R.B., Ortega, J.C., Cejudo-Lopez, J.M., A three-dimensional numerical study and comparison between the air side model and the air/water side model of a plain fin and tube heat exchanger, Applied Thermal Engineering, 30 (2010), pp.1608-1615.
- Sahin, H.M., Dal, A.R., Baysal, E., 3-D Numerical study on correlation between variable inclined fin angles and thermal behavior in plate fin-tube heat exchanger, Applied Thermal Engineering, 27 (2007), pp.1806-1816.
- Wen, M.Y. Ho, C.Y., Heat transfer enhancement in fin and tube heat exchanger with improved fin design, Applied Thermal Engineering, 29(2009), pp.1050-1057.
- Yan, W.M., Sheen, P.J., Heat transfer and friction characteristics of fin and tube heat exchangers, International Journal of Heat and Mass Transfer, 43 (2000), pp.1651-1659.
- Wolf, I., Frankovic, B., Vilicic, I., A numerical and experimental analysis of neat transfer in a wavy fin and tube heat exchanger, Energy and the Environment (2006) pp.91-101.
- Tang, L.H., Zeng, M., Wang, Q.W., Experimental and numerical investigation on air side performance of fin and tube heat exchangers with various fin patterns, Experimental Thermal and Fluid science, 33(2009), pp.818-827.
- Wang, C.C., Lo, J, Lin, Y.T. Wei, C.S., Flow visualization of annular and delta winlet vortex generators in fin and tube heat exchanger application, International Journal of Heat and
- Mass Transfer, 45, (2002), pp.3803-3815.