INTERNATIONAL JOURNAL OF MERGING TECHNOLOGY AND ADVANCED RESEARCH IN COMPUTING

IJMTARC – VOLUME – IX – ISSUE – 34, APR - JUN, 2021 ISSN: 2320-1363

DESIGN AND COMPARISON OF THERMAL ANALYSIS BETWEEN LOUVERED FIN AND CROSS FIN AUTOMOTIVE RADIATORS USING CAE TOOLS

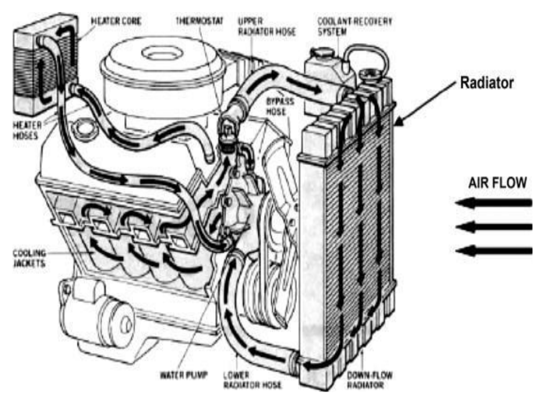
Polemoni Rakesh Yadav, Putnala Datta Krishna Reddy, Md.Shoaib Hussain, M Srikanth, M Venu Kumar, Rizwan Mohammed Khan, Mr. B.Narender Rao

Dept. of Mechanical Engineering, SVITS, Mahbubnagar, Telangana, India.

Abstract: *Radiators are used to transfer thermal energy from one medium to another for the purpose of cooling. Radiators are used for cooling internal combustion engines, mainly in automobiles but also in piston-engine aircraft, railway locomotives, motorcycles, stationary generating plant. The radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine. Research is being carried out for several decades now, in improving the performance of the heat exchangers, having high degree of surface compactness and higher heat transfer abilities in automotive industry. In this thesis, the compact heat exchangers have fins, louvered and cross type. Present study uses the analysis tool to perform a numerical study on a compact heat exchanger at different mass flow rates. The domain is identified from literature and validation of present numerical approach is established. The thermal analysis is done between louvered fin and cross fin, by working on the flow parameters like temperature, thermal flux, heat flow the results are compared. The material used for fins of radiator is Aluminum material. Modeling is performed in Catia and analysis is performed in Ansys. Recommendations have to be made on the optimal values and settings will be based on the variables tested, for the chosen compact heat 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 radiator is always a source of heat to its environment, although this may be for either the purpose of heating this environment, or for cooling the fluid or coolant supplied to it, as for engine cooling.

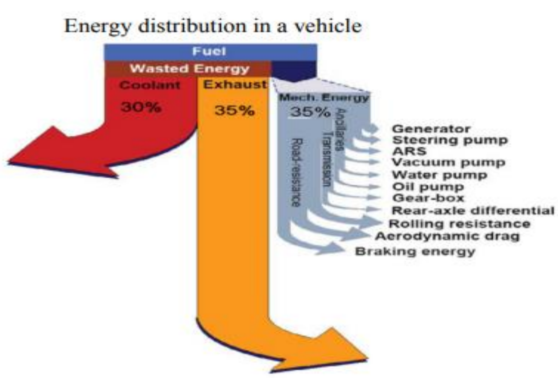


The radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine. A typical radiator used in automobile. Radiators are also often used to cool automatic transmission fluids, air conditioner refrigerant, intake air, and sometimes to cool motor oil or power steering fluid. Radiators are typically mounted in a position where they receive airflow from the forward

IJMTARC – VOLUME – IX – ISSUE – 34, APR - JUN, 2021 ISSN: 2320-1363

movement of the vehicle, such as behind a front grill.

Where engines are mid- or rear-mounted, it is common to mount the radiator behind a front grill to achieve sufficient airflow, even though this requires long coolant pipes. Alternatively, the radiator may draw air from the flow over the top of the vehicle or from a side-mounted grill. For long vehicles, such as buses, side airflow is most common for engine and transmission cooling and top airflow most common for air conditioner cooling. Radiators used in automotive applications fall under the category of compact heat exchangers.



There are two primary classifications of heat exchangers according to their flow arrangement, parallel flow and counter flow. In parallel-flow heat exchangers, the two fluids enter the exchanger at the same end, and travel in parallel to one another to the other side. In counter-flow heat exchangers the fluids enter the exchanger from opposite ends.

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.

Materials used in Radiators

From the early years of the automotive industry up until today, various materials have been used to construct radiators. As the technology of car production progressed, so did the choice of substances increased. This was done in order to provide maximum cooling effect, ensuring the integrity of both the device and car. Here are the different kinds of car radiators.

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

 ISSN: 2320-1363

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.

3.1 Summary of capabilities

Like any software it is continually being developed to include new functionality. The details below aim to outline the scope of capabilities to give an overview rather than giving specific details on the individual functionality of the product.

Catia Elements is a software application within the CAID/CAD/CAM/CAE category, along with other similar products currently on the market.

Catia Elements is a parametric, feature-based modeling architecture incorporated into a single database philosophy with advanced rule-based design capabilities.

The capabilities of the product can be split into the three main heading of Engineering Design, Analysis and Manufacturing. This data is then documented in a standard 2D production drawing or the 3D drawing standard ASME Y14.41-2003.

3.2 Engineering Design

Catia Elements offers a range of tools to enable the generation of a complete digital representation of the product being designed. In addition to the general geometry tools there is also the ability to generate geometry of other integrated design disciplines such as industrial and standard pipe work and complete wiring

ISSN: 2320-1363

definitions. Tools are also available to support collaborative development.

A number of concept design tools that provide up-front Industrial Design concepts can then be used in the downstream process of engineering the product.

These range from conceptual Industrial design sketches, reverse engineering with point cloud data and comprehensive freeform surface tools.

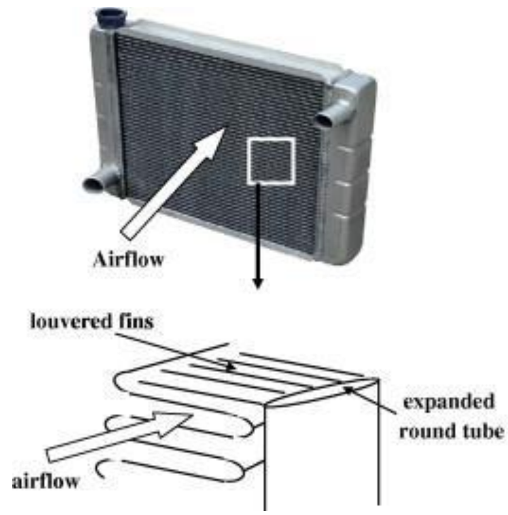
3.3 Analysis

Ansys Elements has numerous analysis tools available and covers thermal, static, dynamic and fatigue FEA analysis along with other tools all designed to help with the development of the product. These tools include human factors, manufacturing tolerance, mould flow and design optimization. The design optimization can be used at a geometry level to obtain the optimum design dimensions and in conjunction with the FEA analysis.

IV - WORKING MECHANISM

4.1 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.

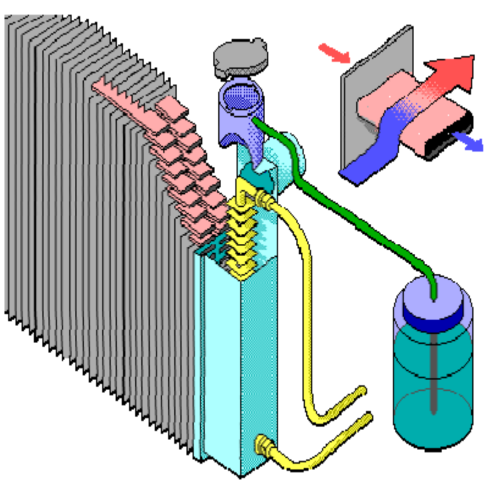


The tubes sometimes have a type of fin inserted into them called a tabulator, which increases the turbulence of the fluid flowing through the tubes. If the fluid flowed very smoothly through the tubes, only the fluid actually touching the tubes would be cooled directly. The amount of heat transferred 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.

4.2 Radiator Cooling System Operation

In operation, water is pumped from the engine to the top (receiving) tank, where it spreads over the tops of the tubes. As the water passes down through the tubes, it loses its heat to the airstream which passes around the outside of the tubes. To help spread the heated water over the top of all the tubes, a baffle plate is often placed in the upper tank, directly under the inlet hose from the engine. Sooner or later, almost everyone has to deal with an overheating car. Since water is readily available, it is not beyond the ability of most people to add some to their radiator if it's low.



- Turn off the A/C. If the car is not seriously overheating, this will reduce the engine's temperature. The AC evaporator is located in front of the radiator, and it adds heat to the air going to your engine. The hotter the incoming air is, the less efficient the radiator will be.

- Turn on your heater (set on highest temperature setting, with blower on highest setting). This will be uncomfortable for you, but it will cool the engine by transferring the heat to the air. Roll down the windows, and remember how 'hot' you'll get if your engine needs replacement!

- If you're stuck in traffic, pull over and stop. Unless you're moving, very little cool air reaches the radiator. Open the hood and let the engine cool off. This takes time, so be patient. Use the time to go get a jug of water or antifreeze.

- Check the overflow tank coolant level. If it's empty, the radiator is probably low on coolant.

- Check the pressure of the system by wrapping a cloth around the upper radiator hose and squeezing it. If it's still under pressure (hot) it will not squeeze easily. Wait until it does.

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).

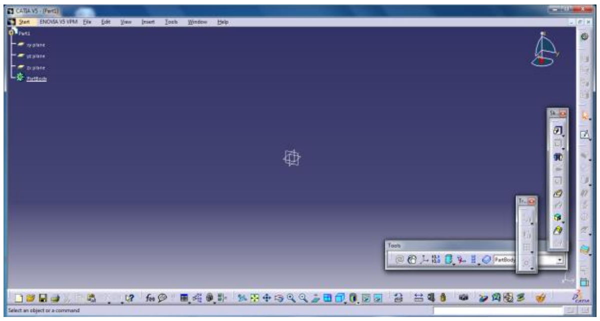


Fig: 5.1: Home Page of CatiaV5

Modeling of Automobile Radiator in CATIA V5

This Automobile Radiator is designed using CATIA V5 software. This software used in automobile, aerospace, consumer goods, heavy engineering etc. it is very powerful software for designing complicated 3d models, applications of CATIA Version 5 like part design, assembly design.

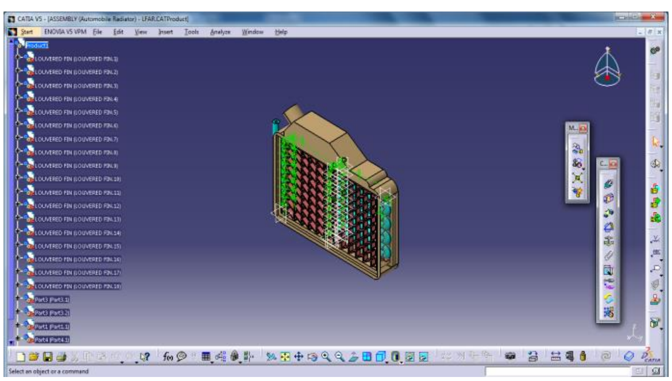


Fig: 5.2: Model design of Automobile Radiator in CATIA-V5

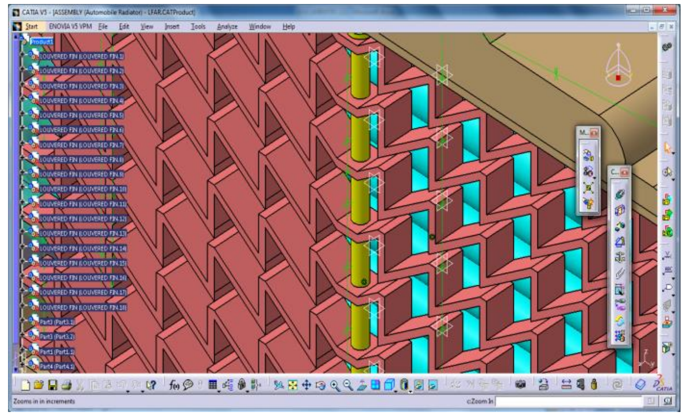


Fig: 5.3: Model arrangement in CATIA-V5

VI - ANALYSIS OF AUTOMOBILE RADIATOR FINS

6.1 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.

6.2 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

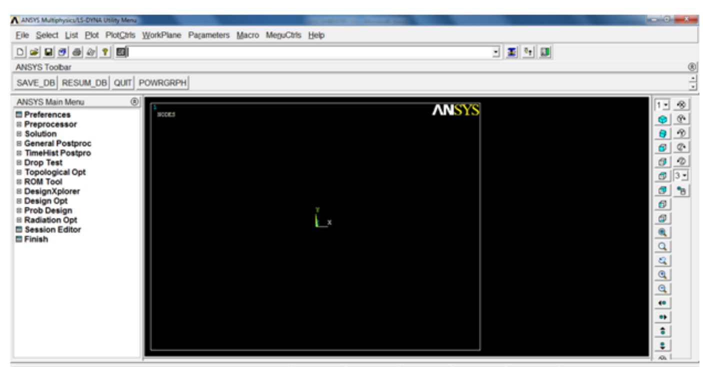


Fig.6.1: Import panel in Ansys.

Rod is modeled with 1d element and shown as above and assembled with adjacent components. Few components are solved using Thermal Analysis for checking the stress and displacements while flowing the fluid.

After completing the meshing of each assembly components next is to do analysis based on the OEM (Original Equipment of Manufacturer) application. So all the models which are analyzed, we need to mention in the Ansys software to get accurate results as per the original component. Some of the components are needed to be solved using thermal analysis.

VII - DISCUSSION ON ANALYSYS RESULT

7.1 Results of Nodal Temperature analysis:

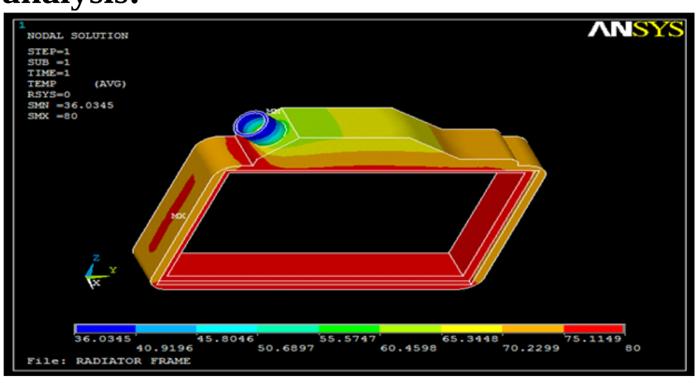


Fig: 7.1: Nodal Temperature of **RADIATOR FRAME**

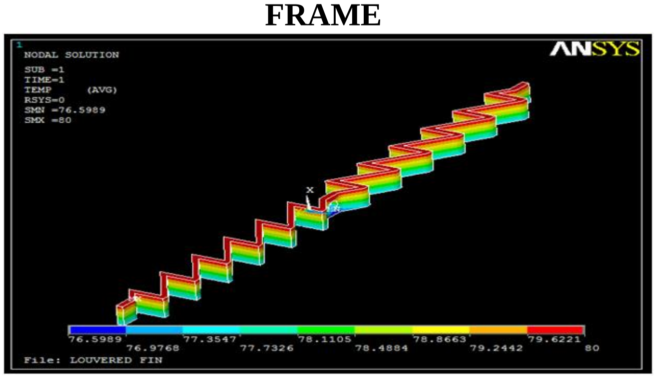


Fig: 7.2: Nodal Temperature of **LOUVERED FIN**

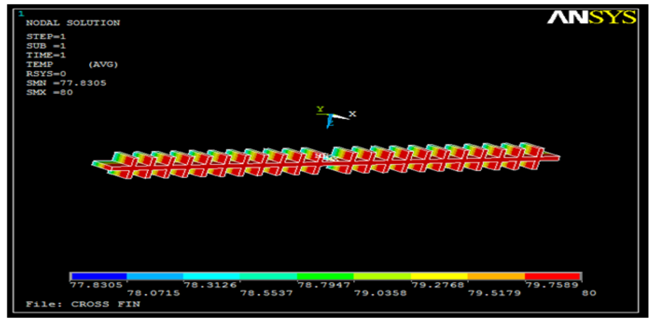


Fig: 7.3: Nodal Temperature of **CROSS FIN**

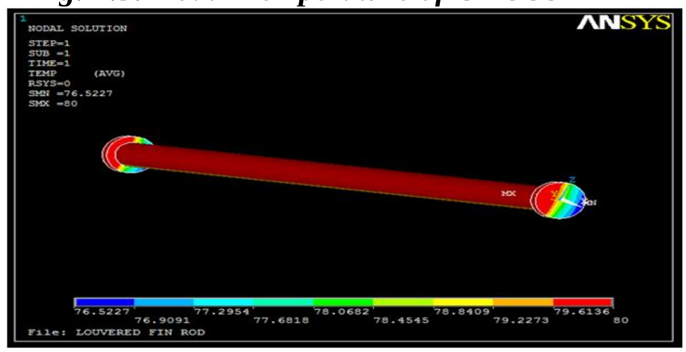


Fig: 7.4: Nodal Temperature of **LOUVERED FIN ROD**

7.2 Results of Thermal Gradient analysis:

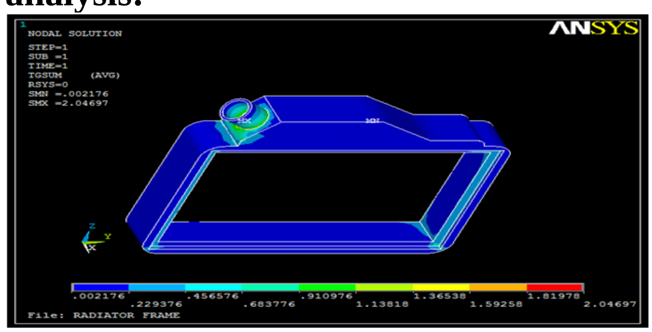
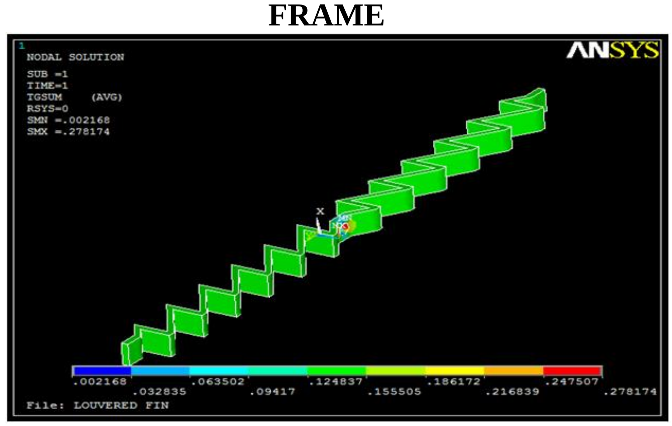


Fig: 7.5: Thermal Gradient of **RADIATOR FRAME**



ISSN: 2320-1363

Fig: 7.6: Thermal Gradient of **LOUVERED FIN**

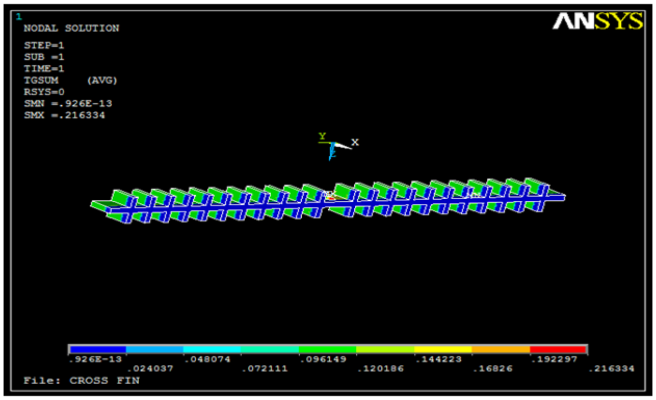


Fig: 7.7: Thermal Gradient of **CROSS FIN**

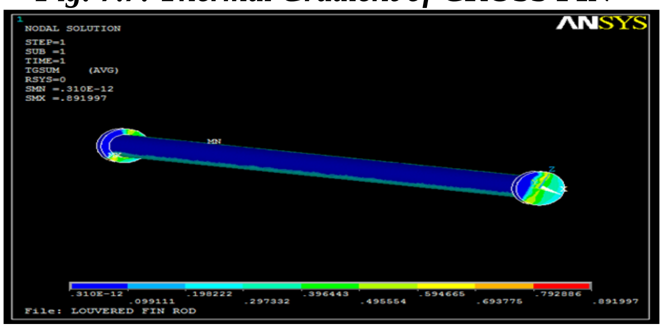


Fig: 7.8: Thermal Gradient of **LOUVERED FIN ROD**

6.3 Results of Thermal Flux analysis:

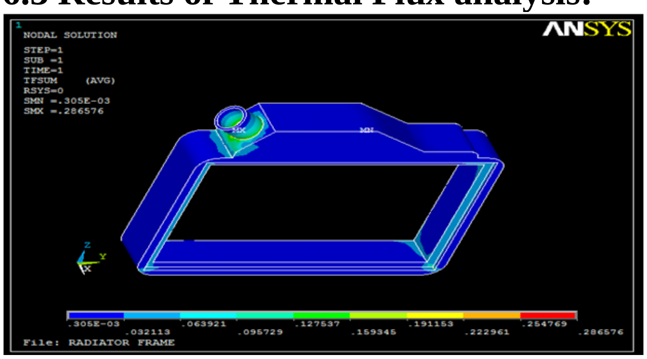


Fig: 7.9: Thermal Flux of **RADIATOR FRAME**

Fig: 7.10: Thermal Flux of **LOUVERED FIN**

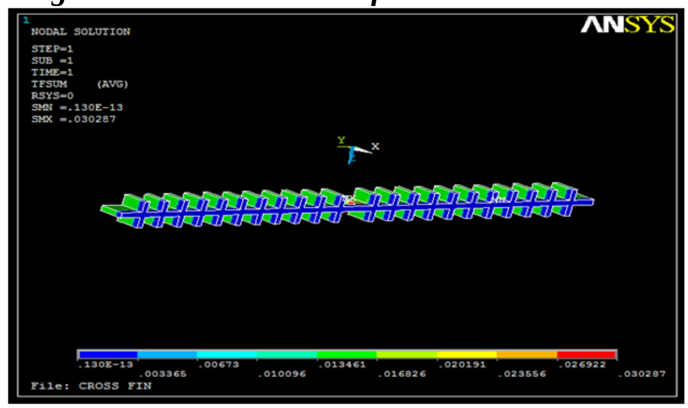


Fig: 7.11: Thermal Flux of **CROSS FIN**

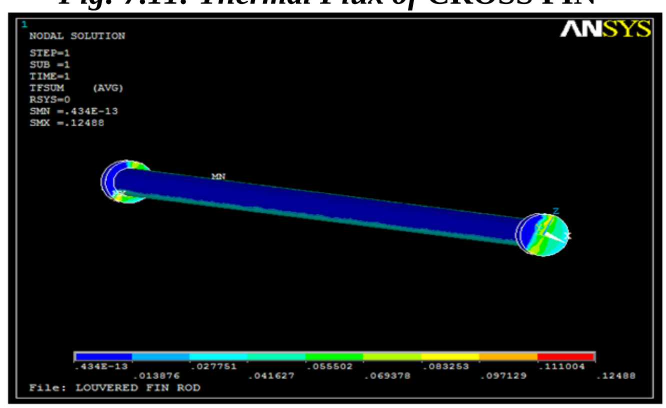


Fig: 7.12: Thermal Flux of **LOUVERED FIN ROD**

6.4 Results of Heat Flow analysis:

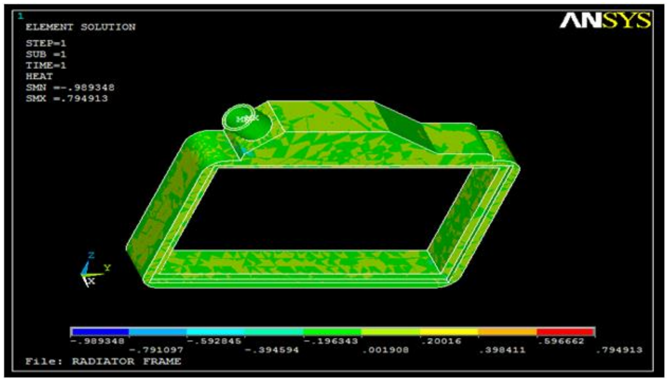


Fig: 7.13: Heat Flow of RADIATOR FRAME

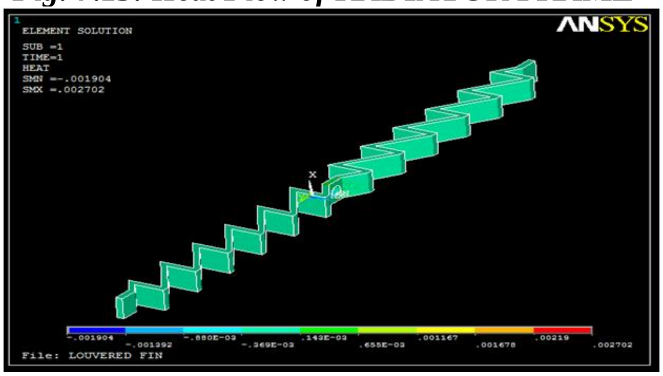


Fig: 7.14: Heat Flow of LOUVERED FIN

Fig: 7.15: Heat Flow of CROSS FIN

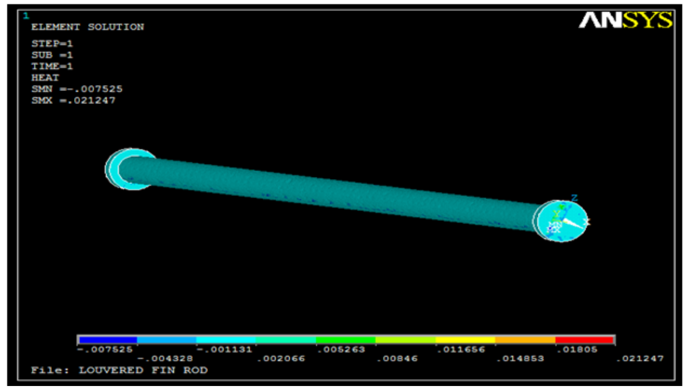


Fig: 7.16: Heat Flow of LOUVERED FIN ROD

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	Louvered Fin Rod
Nodal Temperature	36.03	76.59	76.52
Temperature Gradient	2.04	0.278	0.89
Thermal Flux	0.28	0.038	0.12
Heat Flow	0.79	0.002	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.

COMPARISION RESULTS

S.No	Louvered Fin	Cross Fin
Nodal Temperature	76.59	77.83
Temperature Gradient	0.278	0.216
Thermal Flux	0.038	0.030
Heat Flow	0.002	0.006

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.

IX - 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.
- Fiebig, M., Valencia, A., Mitra, N.K.., Local heat transfer and flow losses in fin and tube heat exchangers with vortex generators: A comparison of round and flat tubes, Experimental Thermal & Fluid Science, 8(1994), pp.35-45.

AUTHOR PROFILE

Polemoni Rakesh Yadav B.Tech student in the Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR.

Putnala Datta Krishna Reddy B.Tech student in the Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR

Md.Shoaib Hussain B.Tech student in the Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR.

M Srikanth B.Tech student in the Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR.

M Venu Kumar B.Tech student in the Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR

Rizwan Mohammed Khan B.Tech student in the Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR

Mr. B.Narender Rao, ASST. PROFESSOR Mechanical Engineering from Sri Visvesvaraya institute of technology and science, MBNR