IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022 ISSN: 2320-1363

DESIGN AND ANALYSIS OF NORMAL FINS USING CAE TOOLS

Dasari Anil kumar, Nayakanti Bharath, Wakonthem Bikash Singh, Chakali Bhaskar, Akula Ramudu, Mr.K.Subahan

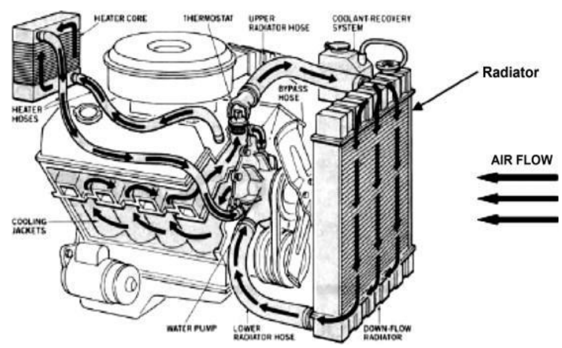
Dept. of Mechanical Engineering, Brindavan Institute of Technology and Science, Kurnool, Andhra Pradesh, India

ABSTRACT:

Extending fins improve the rate of heat transfer or decrease convection. A fin is only for the reason of increasing the surface area in order to maximize heat transmission like a motor, heat exchanger, CPU, or to a heat-generating surface area. The benefit of performing thermal analysis on a flat fin is that it will tell how far heat is dissipated. This paper's primary purpose is to design and evaluate the thermal properties of rectangular fin with varying geometry and content. In this thesis, the compact heat exchangers have rectangular/cross fin plate and louvered fin plates. Present study uses the analysis tool to perform a numerical study on a compact heat exchanger at different mass flow rates. The computational domain is identified from literature and validation of present numerical approach is established first. Later the numerical analysis is extended by modifying chosen geometrical and flow parameters like louver pitch, air flow rate, water flow rate, fin and louver thickness, by varying one parameter at a time and the results are compared. Modeling is performed in CATIA and material used for fins of radiator is Aluminum alloy 6061 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. Despite the name, radiators generally transfer the bulk of their heat via convection, not by thermal radiation, though the term "convector" is used more narrowly; see radiation and convection, below.

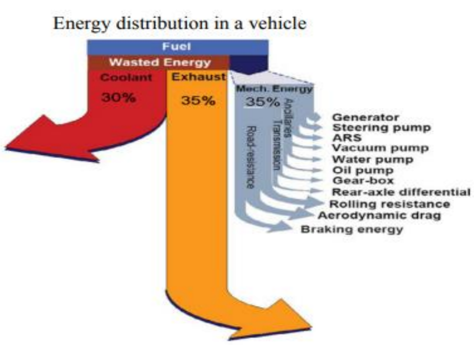


The radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine. A typical radiator used in automobile. Radiators are also often used to cool automatic transmission fluids, air conditioner refrigerant, intake air, and sometimes to cool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

ISSN: 2320-1363

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

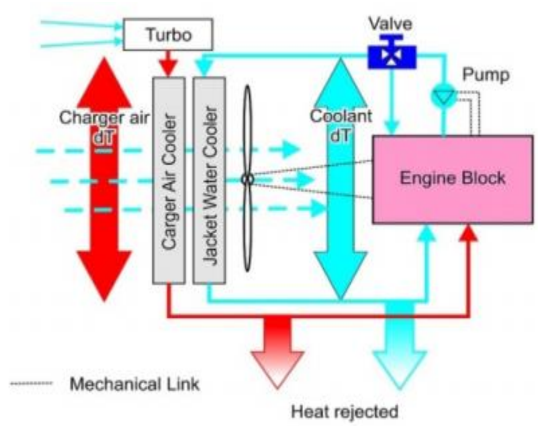
Where engines are mid- or rear-mounted, it is common to mount the radiator behind a front grill to achieve sufficient airflow, even though this requires long coolant pipes. Alternatively, the radiator may draw air from the flow over the top of the vehicle or from a side-mounted grill. For long vehicles, 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.

The experimental work identified from the open literature to validate the present computational methodology is that will be discussed in detail in this chapter. The authors tested air-side heat



transfer and pressure drop characteristics of flow over louvered fins in compact heat exchangers experimentally. The test samples consist of two types of fin configurations. A series of tests were conducted to examine the geometrical parameters of louver pitch, louver arrangement (symmetrical and asymmetrical) and number of louver regions. Their calculated results indicate that a symmetrical arrangement of louvered fins provides a 9.3% increase in heat transfer performance and a 18.2% decrease in pressure drop than the asymmetrical arrangement of louvered fin. Also, for a constant rate of heat transfer and pressure drop, a 17.6% decrease of fin weight is observed for the symmetrical arrangement of fins and this is following by considerable decrease in total weight and cost of the heat exchanger. The results from this investigation indicate that the configuration of the louvered fins has the dominant influence on the heat transfer and pressure drop.

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.

2.1. 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

ISSN: 2320-1363

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.

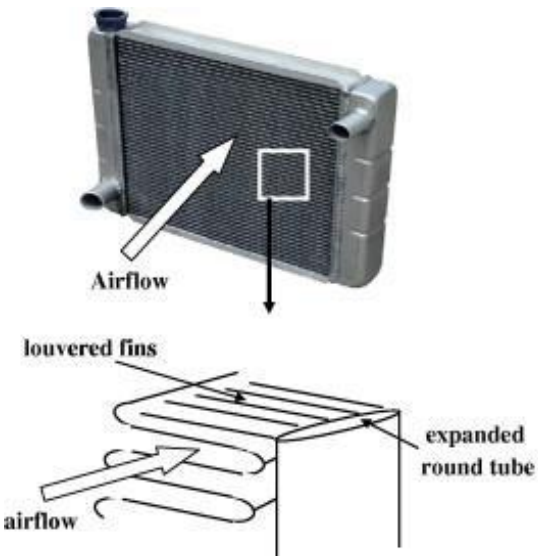
Due to these disadvantages, the power screw design was dropped and a fully new design was defined. The software to be used in design is Catia V5 and testing of design is Ansys.

IV - WORKING MECHANISM

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.



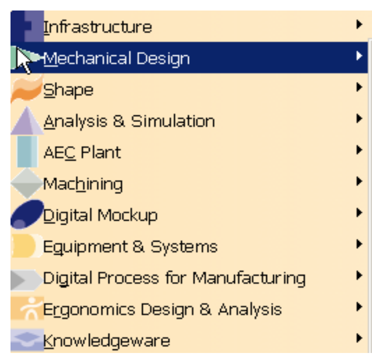
The louver arrangement in a fin used in an automotive radiator. Although lot of work has been done so far in the computational analysis for the compact heat exchangers, validation of an experimentally tested domain and conducting analysis of modified designs to optimize the design and improve performance 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.

V - DESIGN METHODLOGY OF AUTOMOBILE RADIATOR FINS

5.1 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).



The 3D CAD system CATIA V5 was introduced in 1999 by Dassault Systems. Replacing CATIA V4, it represented a completely new design tool showing fundamental differences to its predecessor. The user interface, now featuring MS Windows layout, allows for the easy integration of common software packages such as MS Office, several graphic programs or SAPR3 products (depending on the IT environment).

The concept of CATIA V5 is to digitally include the complete process of product development, comprising the first draft, the Design, the layout and at last the production and the assembly. The workbench Mechanical Design is to be addressed in the Context of this CAE training course. Sets of workbenches can be composed according to the user's preferences. Therefore Dassault Systems offers three different software installation versions. The platform P1 contains the basic features and is used for training courses or when reduced functionality is needed. For process orientated work the platform P2 is the appropriate one. It enables, apart from the basic design features, analysis tools and production related functions. P3 comprises specific advanced scopes such as the implementation of external software packages.

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.

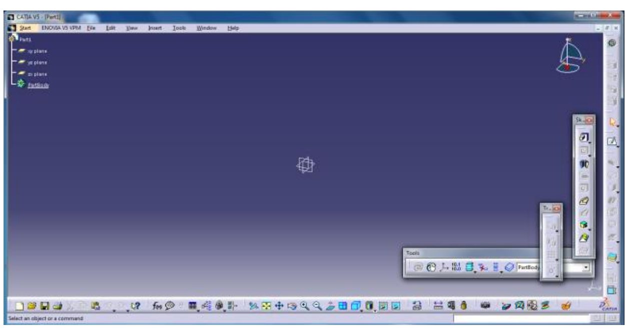


Fig: 5.1: Home Page of CatiaV5

The same CATIA V5 R20 3d model and 2d drawing model is shown below for reference. Dimensions are taken from. The design of 3d model is done in CATIA V5 software, and then to do test we are using below mentioned software's.

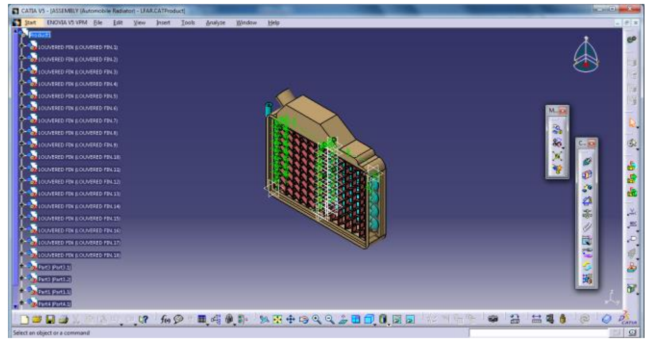


Fig: 5.2: Model design of Automobile Radiator in CATIA-V5

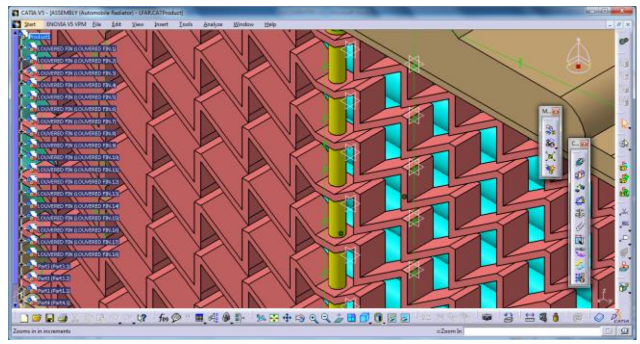


Fig: 5.3: Model arrangement in CATIA-V5

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

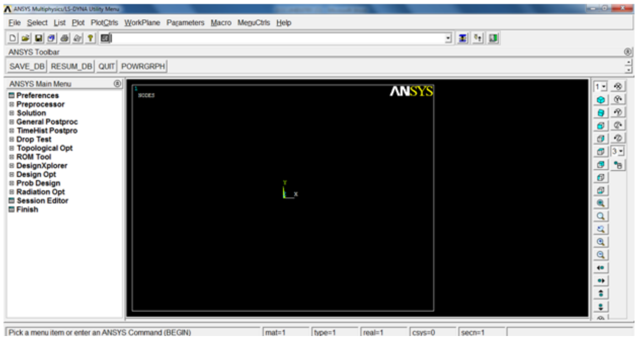


Fig.6.1: Import panel in Ansys.

6.2.1 Meshing:

Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is often used interchangeably. Typical uses are for rendering to a computer screen as finite element analysis or computational fluid dynamics. The input model form can vary greatly but common sources are CAD, NURBS, B-rep and STL (file format). The field is highly interdisciplinary, with contributions found in mathematics, computer science, and engineering.

Three-dimensional meshes created for finite element analysis need to consist of tetrahedral, pyramids, prisms or hexahedra. Those used for the finite volume method can consist of arbitrary polyhedral. Those used for finite difference methods usually need to consist of piecewise structured arrays of hexahedra known as multi-block structured meshes.

Meshing is an integral part of the computer-aided engineering (CAE) simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution. From easy, automatic meshing to a highly crafted mesh, ANSYS provides the ultimate solution. Powerful automation capabilities ease the initial meshing of a new geometry by keying off physics preferences and using smart defaults so that a mesh can be obtained upon first try. Additionally, users are able to update immediately to a parameter change, making the handoff from CAD to CAE seamless and aiding in up-front design. Once the best design is found, meshing technologies from, ANSYS provide the flexibility to produce meshes that range in complexity from pure hex meshes to highly detailed Hybrid meshes. It has a range of meshing tools that cater to nearly all physics. While the meshing technologies were developed to meet specific needs in the areas of solid, fluid, electromagnetic, shell, 2-D and beam models, access to these technologies is available across all physics.

VII - DISCUSSION ON ANALYSYS RESULT

7.1 Results of Nodal Temperature analysis:

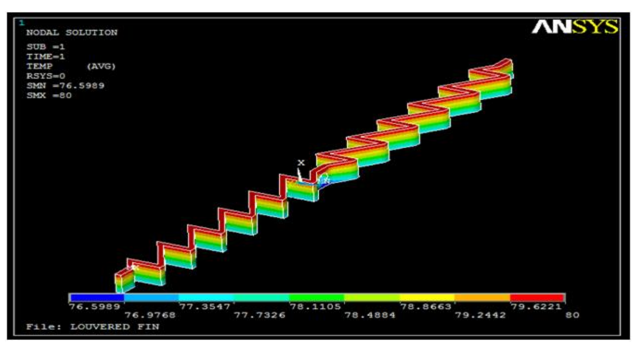


Fig: Nodal Temperature of LOUVERED FIN

ISSN: 2320-1363

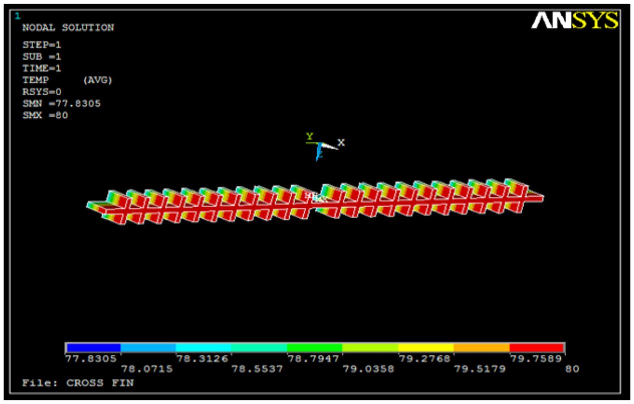


Fig: Nodal Temperature of CROSS FIN

7.2 Results of Thermal Gradient analysis:

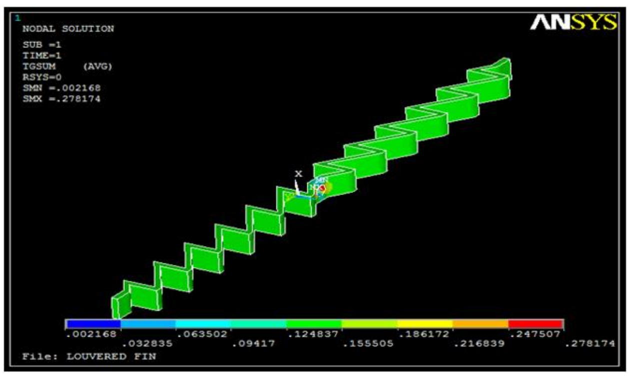


Fig: Thermal Gradient of LOUVERED FIN

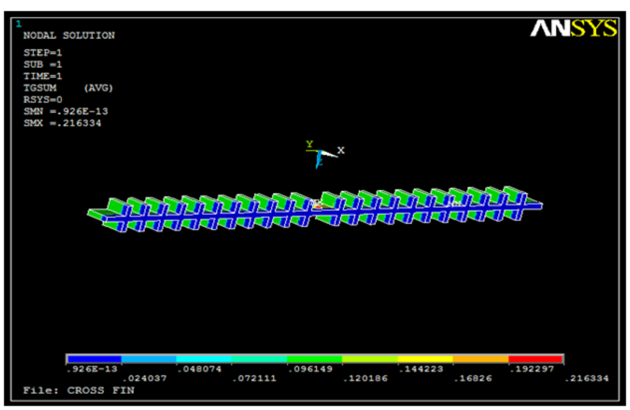


Fig: Thermal Gradient of CROSS FIN

6.3 Results of Thermal Flux analysis:

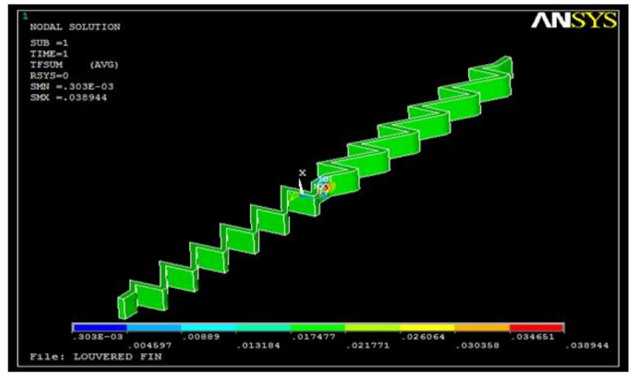


Fig: Thermal Flux of LOUVERED FIN

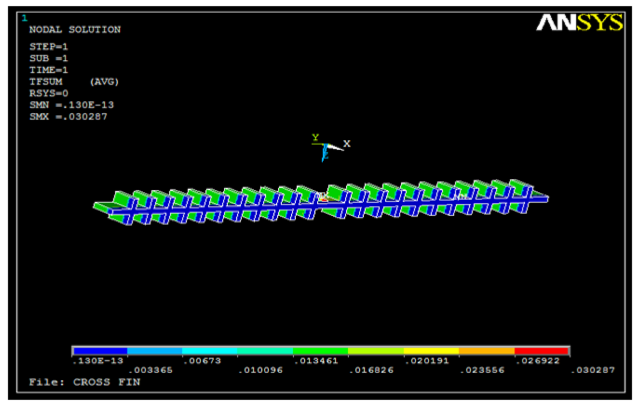


Fig: Thermal Flux of CROSS FIN

6.4 Results of Heat Flow analysis:

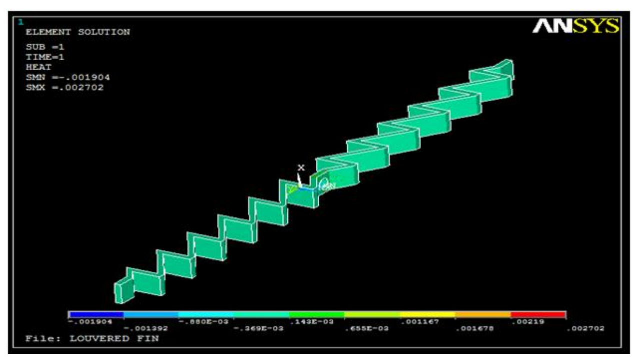


Fig: Heat Flow of LOUVERED FIN

IJMTARC – VOLUME – IX – ISSUE – 38, JUL - SEP, 2022

ISSN: 2320-1363

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.

Future Scope

So it can be concluded that modifying the radiator model with louver fins yields better results. It can be summarized that by providing louvers for the radiator and increasing the louver pitch helped in reducing the pumping power requirements with increase in heat transfer rate. This will help in increasing the power output per unit mass of the radiator. Hence it is recommended to increase the louver spacing for the geometry under consideration.

Scope for Future, in the present Analysis, constant temperature boundary conditions are considered along the walls. The model presents an idealized situation. A more accurate model could be to consider the finite thickness of the plates of the louvered fin exchanger and the interrupted louvered fin and solve the conjugate heat transfer problem. The solution of the conjugate heat transfer problem can be expected to yield predictions that are more exact. The computations can further be performed comparing different types of fin shapes.

The present work can be further extended for different geometries of the inserts like fins being used between the radiator frames of the compact heat exchanger.

The analysis can be performed assuming the flow regime to be turbulence model and force convection. And changes of geometry make in only rectangular fin may be similar type of geometry can change in different geometry shape.

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.