CFD Analysis of Helical Tube Automobile Radiator Considering Different Nano Fluid

Pradeep Pandey[1] Neeraj Pandey[2] Prof. Devendra Singh Sikarwar[3]
[1,2]PG Scholar, [3]Associate Professor & Head
[1,2,3]Department of Mechanical Engineering
[1,2,3]PCST Indore, India

Abstract— Increasing the rate of heat dissipation in automotive radiators is in great demand due to the use of more powerful engines in relatively small space. Insufficient rate of heat dissipation can result in overheating of the engine, and that causes corrosion and metal weakening of engine parts. The aim of this project is to design radiator for thermal stability of engine with increasing the rate of heat transfer and decreases the computational cost dramatically and leads the way to obtain hydrodynamic and thermal performance of elliptical and helical tube type radiator by using CFD analysis. By adopting this methodology, the thermal performance of a complete radiator design can be obtained within a reasonable computational time and a CFD model with the proposed methodology can be implemented as a design tool for the radiator design which would lead to more optimized radiator designs. Overall heat transfer is calculated for each design of elliptical & helical tube radiator. Different type of nano fluids Ethanol, Methanol, Al_2O_3/water2%, Al_2O_3/water3% and CuO/water and other coolants are taken into consideration while running the analysis. By Varying turbulence model and surface Nusselt number, the contours of temperature and pressure drop were calculated and plotted using software ANSYS. The purpose of this study is to minimize the heat load on the engine and lighten automotive component because of heat generation, and to make automotive radiators more compact while maintaining important level of heat performance The cooling capacity of Wagon-R radiator is computed with the proposed computational model and using different type of nano fluid to analyse thermal performance of both elliptical & helical tube type radiator and finally found that helical tube radiator with ethanol as coolant gives better result as compare to elliptical tube.

Keywords: CFD, Nano Fluid, Cooling Systems

I. INTRODUCTION

Thermal radiator is an important part of an automobile engine, and it has the functions of preventing engine from overheating and regulating engine temperature. High temperatures in an engine can cause oil to thin, engine parts to expand, lubrication to break down, and moving parts to be damaged. A more effective radiator will reduce engine temperature in the hot environments, while consume less power from the engine, therefore, the engine thermal efficiency can be increased. Today’s engine require higher output with decreased space available for cooling air circulation which necessitates a better understanding of the complex cooling fluid flow characteristics and thermal performance of the radiator is necessary as the performance, safety and life of engine depends on effective engine cooling . About 30% of the thermal energy generated is dissipated to the coolant circulating in the engine-cooling jacket. The hot coolant coming out of engine jacket is to be cooled in a radiator and circulated again.

The radiator plays a very important role in an automobile. It rejects the waste heat generated after the ignition procedure and useful work has been done. The effectiveness with which waste heat is transferred from the engine walls to the surrounding is crucial in preserving the material integrity of the engine and enhancing the performance of the engine. Design and research have been done on engine radiators focusing primarily on optimizing their performance. The utilization of Computational Fluid Dynamics (CFD) demonstrating reproduction of mass stream rate of air going over the containers of a car radiator was done. Concentrates on the utilization of Nano liquids in minimal heat exchangers were completed. Numerical investigation of warmth exchange and pressure drop in a heat exchanger that is planned with various shape pin fins was done.

Cooling is one of the top technical challenges to obtain the best automotive design in multiple aspects (performance, fuel consumption, etc.). Automotive radiator is an important part of the engine cooling system. Heat is exchanged from hot coolant to outside air. Radiator gathering comprises of three primary parts centre, intake tank and outlet tank as appeared in fig.1.1Centre has two arrangements of entry, an arrangement of tubes and an arrangement of fins. Coolant courses through tubes and wind streams between fins. The hot coolant sends warm through tubes to balances. Outside air going between balances pickups and diverts warm as appeared in fig 1.2. Because of constrained space at the front of the motor, the extent of the radiator is limited and can't be basically expanded. In this manner, it is important to expand the warmth exchange abilities of working liquids, for example, water and ethylene glycol in radiators on account of their low warm conductivity and furthermore the territory of warmth exchange is expanded by using the broadened surfaces as balances joined to dividers and surfaces.

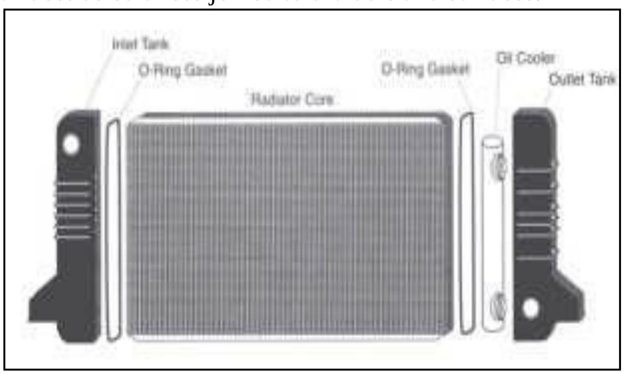


Fig. 1.1: Components of Automotive Radiator

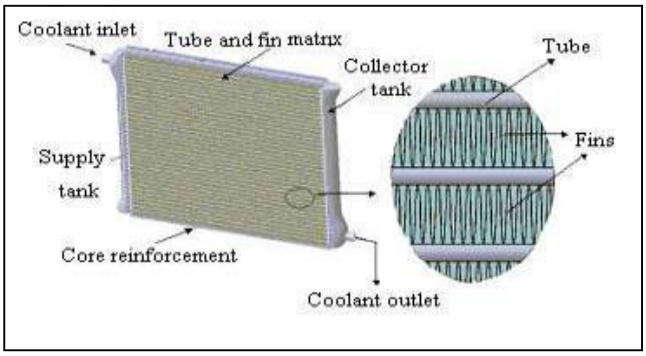


Fig. 1.2: Automotive Radiator fluid flow Circuit

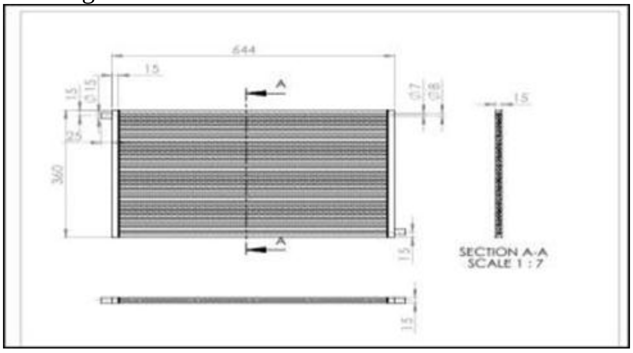


Fig. 1.3: Dimension Drawing Sheet of Automotive Radiator

A. Cooling Systems:

The internal combustion engine used today is far from being the most efficient, efficiency being between 25-35%. Out of 1 litter of fuel just 3dl is utilized to really move the car, rest of the fuel goes to waste. There are different losses in the engine but the biggest loss comes in terms of heat. Roughly 60% is used as heat. From this 60% minimal the greater part is exhaust gases, yet the rest is left for the cooling framework In a normal traveler car that produces around 100 kW, 30 kW is utilized to move the auto, 35 kW is put out however the fumes and 28 kW is left for the cooling framework to dispose of. 28 kW is a ton of heat to deal with, so the cooling framework has a hard assignment. For the engine itself it's indispensable to have relentless working temperature. To get the best efficiency out of the engine, the engine should be working between 85-95 °C. At the point when temperatures go over that, the wear of the engine increments drastically, parts begin to extend more, and so forth. Furthermore, if the temperature goes sufficiently high the engine will stop up.

In a conventional traveller auto the cooling framework comprises of 11 sections. These parts are depicted in the accompanying picture. In a dashing auto the cooling framework is somewhat more convoluted. In a normal traveller auto the weight, the space, the cost, and so forth are essential as well, however in a racing car these things are significantly more featured. The most serious issue by a wide margin is the space. In an open-wheel race-auto the space has a tendency to be extremely tight, however it is as yet required to fit the sufficiently huge radiator so as to have required cooling power. Then the next problem is the coolant pump. To get the most out of the engine with minimal measure of waste power, the direct should be sufficiently great to have the required coolant stream. So larger than average coolant pump isn't best.

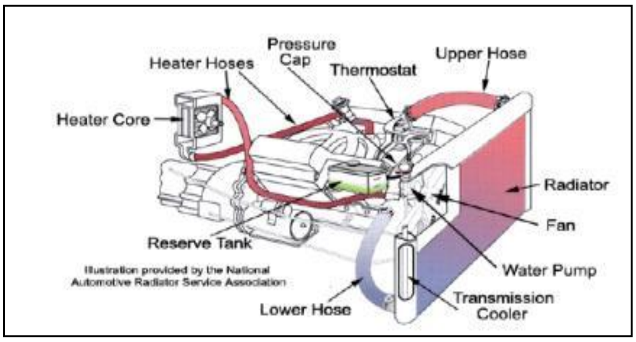


Fig. 1.4: Automotive Radiator Cooling System

The cooling system looks pretty simple. There are not too many parts and the functions of all the parts are understandable. The radiator is the part where the water is cooled. The majority of the radiators are air-cooled, that implies that the air experiencing the radiator removes the heat from the water. So the wind current through the radiator is extremely essential. The principle wellspring of air flow is shaped by the development of the car, yet when the auto is stopping the fan makes up the wind current.

Estimating methods to evaluate the cooling system and to check whether any changes were accomplished, it's essential to do a few estimations. Cooling performance wise, two things matter the most. Right off the bat the air side, so the genuine wind current through the radiator and after that the mass stream of fluid inside the cooling system. In this proposal the emphasis is broadcasting live side so the methods for estimating the wind stream and the air mass stream will be displayed.

There are diverse methods for estimating the wind current and the air mass stream. In the accompanying parts the most valuable ones are recorded. There are also 2 performance parameters that are useful in evaluating the cooling system performance there are a few problems that make the measuring more complicated in real-life situations:

- The air flow is not laminar
- The diverse Heat sources influence the outcomes
- The conditions are not steady

B. Thermal based Estimations:

Heat based estimation essentially comprises just of the Hot-Wire-Anemometer (HWA) or now and again alluded as the Mass-Air-Flow meter. The Mass-Air-Flow meter is more widely known in the automotive industry. The HWA comprises of a thin wire that is situated in the air flow and a little current is put through the wire. The wire heats up and the air flow around the wire chills it off. The cooling power is reliant on the speed of the wind stream. By measuring the resistance in the wire it’s possible to determine the mass flow rate. Typically these sorts of sensors are little, they have fast reaction to changes in the speed and even at moderate rates they are exact. In any case, this sort of estimation technique has a couple of real defects that keep its utilization in out and about testing and in wind burrows: "- The hot wire itself is extremely delicate. Dust, dirt, or gravel pieces can easily destroy the hot wire.

- Utilizing a HWA without knowing the connection of velocity, temperature and electric obstruction would cause a mind boggling set-up alignment.

- Due to the way that the estimating strategy is a thermally based framework it needs a steady domain under stable conditions. Putting the Hot-Wire-Anemometer before or behind the radiator suggests that few heat sources impact the estimation.
- Using a Hot-Wire-Anemometer requires a laminar wind stream in the cooling channel for solid outcomes. Turbulent wind current would misrepresent the test outcomes.

C. Performance Parameters:

The next two ways of assessing the cooling power rely completely in math and are not directly measured. Both of these lone require temperature estimations, so they are perfect for out and about and static tests. Also it makes them really cheap to use, so they are perfect for us to analysis the system performance.

1) Air-To-Boil (ATB):

The Air-To-Boil temperature is characterized as the surrounding temperature at which the coolant temperature at the radiator delta achieves the boiling point. Actually ATB gives a measure of how far the cooling framework is from boiling. "ATB is defined as

$$ATB = T_{BP} - (T_{CI} - T_{AI})$$

T_{BP} = coolant boiling point
T_{CI} = coolant radiator inlet temperature
T_{AI} = air radiator inlet temperature

The ATB-parameter must be estimated/figured amid vehicle tests when the entire cooling framework has accomplished a stable working point. It is an expensive and tedious process, since it requires stable encompassing and engine load condition."

ATB does not straightforwardly give the cooling wind stream rate but rather it can show the impact of changes in the wind current on the cooling execution.

2) The Specific Dissipation (SD):

"Rather than the ATB, the Specific Dissipation (SD) is moderately obtuse to changes in the surrounding and coolant temperatures."

$$SD = \varepsilon \times C_{min} = \frac{Q_c}{Q_{max}} \times C_{min}$$

Where:
ε = heat exchanger effectiveness
Cmin = minimum capacity rate

$$Q_c = m_c \times c_{pc} \times (T_{ci} - T_{co})$$
$$Q_{max} = m_a \times c_{pa} \times (T_{ci} - T_{ai})$$
$$C_{min} = m_a \times c_{pa}$$

Where:
T_{ci} = radiator coolant inlet temperature
T_{co} = radiator coolant outlet temperature
T_{ai} = radiator air inlet temperature
T_{ao} = radiator air outlet temperature
c_{pc} = coolant specific heat
c_{pa} = air specific heat

"SD is defined as the heat transfer rate of a heat exchanger divided through the maximum temperature difference across the heat exchanger

$$SD = [m_c \times c_{pc} \times (T_{ci} - T_{co})]/(T_{ci} - T_{ai})$$

Ignoring losses or thermal radiation the mass airflow can be approximated by stating:

$$m_c \times c_{pc} \times (T_{ci} - T_{co}) = m_a \times c_{pa} \times (T_{ao} - T_{ai})$$
$$m_a = [m_c \times c_{pc} \times (T_{ci} - T_{co})]/c_{pa} \times (T_{ao} - T_{ai})$$

Stream isn't estimated specifically yet figured. So there is dependably a greater blunder incorporated into the outcomes. Anyway this approach has additionally a few preferences:

- Both the money related and the time exertions for the test setup are fragmentary.

ATB requires two temperature sensors, SD four

- Both are appropriate for out and about tests SD parameter is significantly more heartless to the progressions of nature.

D. Technological Other Heat Transfer Enhancement Techniques:

These upgrade strategies can be ordered into dynamic and aloof procedures. Aloof systems don't require any sort of outer power for the warmth exchange enlargement, though, the dynamic strategies require some power remotely, for example, electric or acoustic fields and surface vibration.

1) Dynamic Heat Transfer Enhancement Techniques:

a) Mechanical Aids:

These guides comprise of mixing the liquid or turning the surfaces by mechanical means. Mechanical surface scrappers might be connected to the conduits of gases for the improvement of warmth exchange. Pivoting heat exchanger channels are monetarily used to enlarge the warmth exchange.

b) Surface Vibration:

Low or high recurrence surface vibrations are utilized to advance single stage warm exchange expansion. A piezoelectric gadget may likewise be utilized to vibrate a warmth exchange surface. Heffington et al. [2001] utilized a piezoelectric transducer to vibrate a plate at around 2.5 kHz, which delivers a shower of little distance across drops on the bubbling surface. The idea utilized by Heffington et al. [2001] is called Vibration Induced Droplet Atomization (VIDA).

c) Liquid Vibration:

The liquid vibration is the more viable sort of vibration upgrade because of the mass of the warmth exchangers. The liquid vibrations go from throbs of around 1 Hz to ultrasound and for the most part connected for the single stage liquids.

2) Passive Heat Transfer Enhancement Techniques:

a) Coating of the Surfaces

Condensation occurs on the surface whose temperature is less than the vapour saturation temperature. This condensed liquid on the surface exists either as a wetted film or in droplets. Droplets are formed if the condensate does not wet the surface. Drop wise condensation yields a high heat transfer coefficient but it cannot be sustained permanently. Non-wetting coating, such as Teflon, enhance the drop-wise condensation. A hydrophilic coating promotes the condensate drainage on evaporator fins by reducing the wet air pressure drop. Nucleate boiling can be enhanced by a fine scale porous coating. A porous coating on the base surface is an effective enhancement method for film condensation. Condensate drainage is assisted by capillary flow within the porous coating, resulting in a thinning of the condensate film thickness. The temperature drop across a laminar condensate film depends on the condensation thermal resistance and such capillary assisted film thinning reduces the condensate thermal resistance.

b) Rough surfaces

Surfaces may be made rough by machining or restructuring the base surface or by placing some "roughness" adjacent to the surface e.g. a wire coil insert. So many possible roughness geometries are studied by the researchers and the three dimensional roughness geometries like "cross-rifled" tubes of Nakamura and Tanaka [1973], three dimensional ribs by Liao et al. [2000] etc. offer higher enhancement level. For single stage stream, blending in the limit layer is advanced close to the surface as opposed to expand the heat exchange surface zone. A wire coil insert classified as wall attached roughness is shown in Figure 1.6. The knurled roughness on a vertical surface promotes the mixing in the condensate film.

c) Extended Surfaces

It is a most common approach to enhance the heat transfer by using the extended surfaces. A plain blade may expand the surface zone yet an uncommon shape broadened surface may build warm move coefficient notwithstanding the territory of heat exchanger. Enhanced extended surfaces used for the gases are already discussed in section 1.2. The broadened surfaces for fluids regularly utilize substantially littler blade statures than that utilized for gases due to the higher warmth exchange coefficient for fluids. Use of high fins with liquids would result in low fin efficiency and result in poor material utilization. Externally finned tube and internally finned tube are the examples of extended surfaces for liquids.

d) Swirl flow devices

These devices (Figure 1.8) include a number of geometrical arrangements or tube inserts for forced flow that create rotating or secondary flow. Full length bent tape supplements or inlet vortex generator and hub loop embeds with a screw compose winding are a few cases of swirl flow devices.

e) Surface tension devices

The local film thickness is determined by the force that drains the condensate. The use of surface tension forces to affect condensate drainage is an effective enhancement technique. Vertical fluted tubes are frequently used for vertical tube condensers used in desalination and are commercially available. Loosely attached axial wires (poor thermal contact) on vertical smooth tubes also provide surface tension condensation enhancement. The surface tension devices strictly do not increase the surface area of the base surface. Heat pipes typically use capillary wicking to transport liquid from the condenser section to evaporator section.

f) Additives for liquids

Additives for single phase liquids may be solid particles or gas bubbles. Kafanov [1964] performed a detailed study of solid particles additives such as water-chalk, water-coal, water-sand, water-aluminium etc in a circular tube for varying Reynolds number. Now-a-days, however, nano-sized metallic particles are of considerable interest with regard to increase in thermal conductivity of the flowing medium. Li and Xuan [2002] reported a 24% higher heat transfer coefficient attributed to 100 nm copper particles with 1% concentration in water.

Bubbling a gas through a stationary liquid stimulates the conditions for nucleate boiling because of the liquid agitation on the surface, caused by the vapor bubbles. Additives may also be such suspensions e.g. a dilute polymer-water solution, which reduce the fluid friction. Suspensions in dilute polymer and surfactants solutions reduce both heat transfer and fluid friction. The use of a rough surface recovers some of the heat transfer reduction, however, at the expense of increased friction

E. Objectives of the Present Work:

Performance of engine cooling system is influenced by factors like air and coolant mass flow rate, air inlet temperature, coolant fluid, fin type, fin pitch, tube type and tube pitch etc.

While designing cooling system main aim remains that the size of the cooling system should be less but three factors does not allow the size to decrease. The factors are

1) High altitude: At high altitude, air density becomes low and hence affects air mass flow rate.
2) Summer conditions: During summer surrounding air is hot i.e. air inlet temperature is more.
3) Maximum power: Engine condition producing maximum power like when vehicle is climbing uphill, maximum heat rejection is required during this condition.

The objective of the work:

1) To analyze and to compare heat dissipation of car radiator with existing elliptical tube radiator and proposed helical tube radiator with computational fluid dynamic results.
2) To analyze and to compare heat transfer of car radiator with different nano fluids with different base fluid, nano particles and concentration.

II. Literature Review

A. Research Contributions:

In the previous couple of decades, fast advancements of nanotechnology have prompted developing of new age of coolants called "Nano liquids". Nanofluids are defined as suspensions of Nanoparticles in a base fluid. The size of Nanoparticles are varied from 1 to 100 nm, with the commonly use base fluid of water, ethylene glycol or mixture of both.

Nanotechnology is being utilized or considered for use in numerous application focused to give cleaner, more effective energy supplies and employments. Most of the research have been done on Nanofluids are supporting the fact and theory of Nanofluids enhance the heat transfer of the system. By carefully selection of particles material, size, shape, concentration and base fluid type, Nanofluids will definitely perform much better coolant than conventional water coolant.

1) Brandon Fell et al (2007) Mentioned that Nanofluids increase conduction and convection coefficients, allowing for more heat transfer out of the coolant. General Motor is involving in development of Nanofluids use in automobile industry. The benchmark for heat exchange of current radiators is 140kW of heat at a inlet temperature of 95ºC. The essential radiator has a width of 0.5 to 0.6 meter, a height of 0.4 to 0.7 meter and a depth of 0.025 to 0.038 meter, however, these dimension are greatly depends on the make and engine size of the vehicle.

2) Dongsheng et al (2009) Highlighted the advantages of using Nanofluids compared to conventional solid liquid suspensions. The advantages are incorporated high particular surface region and subsequently more heat exchange surface among particles and liquids, high scattering solidness with overwhelming Brownian movement of particles, Reduced directing force when contrasted with unadulterated fluid to accomplish proportionate heat exchange increase, diminished molecule obstructing when contrasted with tradition slurries, hence advancing framework scaling down, and movable properties, including heat conductivity and surface wettability, by changing molecule fixations to suit diverse applications.
3) Sadik et al (2009) recorded the procedures of making metal Nano particles incorporate mechanical processing, inactive gas buildup system, concoction precipitation, compound vapor testimony, smaller scale emulsions, splash pyrolysis and thermal showering.
4) Clement et al (2011) Mentioned that the changes of viscosity, density and thermal conductivity are critical in Nano fluids. There are five classical models for effective thermal conductivity of Nano fluids, this also shows the aforementioned disadvantages, which is lack of agreement between results. However, Maxwell's model is widely used or modified such as Hamilton-Crosser's model.
5) Lee PohSeng (2003) assurance of certain thermo - physical properties of Nanoliquids.The properties of Nanofluids at desired temperature, particle size and concentration can be estimated using the regression equations developed to evaluate the viscosity and thermal conductivity for water based Nano fluids. The model which gives the lowest deviation is selected base on few aspects such as coefficient of equation, maximum and minimum deviation.
6) Nguyen et al. (2008) Has proposed a formula for calculating viscosity of Nanofluids for 47 and 36 n m particle -sizes and particle volume fractions of 1 and 4% with particle volume concentration, φ and temperature, T respectively.
7) Hilde Van DerVyer et al. (2003) conducted a CFD simulation of a 3-D tube-in-tube heat exchanger using Star-CD CFD software and made a validation test with the experimental work. The authors were fairly successful to simulate the heat transfer characteristics of the tube-in-tube heat exchanger. This has been used as the base for the procedures of CFD code validation of a heat exchanger.
8) Vikashkumar et.al (2003) done A three dimensional numerical simulation study has been carried out to predict air flow and temperature distribution in the tube type heat exchanger associated with large electrical motor. Because of symmetry in geometrical development, a segment of warmth exchanger has been considered for CFD examination by utilizing PHOENICS programming. The k - ε turbulence model has been used to solve the transport equations for turbulent flow energy and the dissipation rate.
9) Witryet. al.,(2003) carried out CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator, in which FLUENT 's segregated implicit 3-D steady solver with incompressible heat transfer is used as the tool. Here the shell side airflow pattern and tube side water flow pattern are studied to present the variation of overall heat transfer coefficients across the radiator ranging from 75 to 560 W/m2-K.
10) Chen et al, (2001) influenced a trial examination of the warmth to exchange attributes of a tube-and-balance radiator for vehicles utilizing a trial streamlining outline procedure on a breeze burrow test apparatus of the radiator. The creators have built up the relapse conditions of warmth dispersal rate, coolant weight drop and pneumatic stress drop.The influences of various parameters like the air velocity, inlet coolant temperature and volume flow rate of coolant on heat dissipation rate, coolant pressure drop and air pressure drop have been discussed in detail by means of the numerical analyses. The results provide a basis for the theoretical analysis of heat performances and structural refinement of the tube-and-fin radiator.
11) Sridhar Maddipatla, (2001) exhibited a technique to configuration vehicle radiator by coupling CFD with a shape advancement calculation on a streamlined 2D demonstrate. It incorporates mechanized work age utilizing Gambit, CFD examination utilizing Fluent and an in-house C-code executing a numerical shape advancement calculation.The flow simulations using FLUENT were performed using the classical simple algorithm with a k-ε turbulent model and second order upwind scheme. It involves calculating the overall pressure drop and mass flow rate distribution of the coolant and air in and around the single tube arrangement of an automotive radiator.
12) Yiding Cao et al. (1992) introduced heat pipe in radiator. Heat pipes including two-phase closed thermosyphons are two-phase heat transfer devices with an effective thermal conductance hundreds of times higher than that of copper. For the terrestrial applications, gravity is often used to assistant the return of the liquid condensate and no wick structure is needed inside the heat pipe, and this type of heat pipes is often referred to as two-phase closed thermosyphons. Using heat pipes in automotive radiator have benefits like higher effectiveness of heat exchange due to the counter-flow mode, increasing the reliability of radiators, increasing the overall heat transfer coefficient between air flow and coolant.

III. THEORETICAL CONSIDERATION

There are many new inventions that are icons of modern life. Advances like the computer, the automobile, and the harnessing of electricity are integrally linked by the need of a heat sink for the ability to function. Heat sink innovation is connected in different courses for these distinctive creations; however it appears as a radiator in automobiles. Radiators are more technically known as heat exchangers, but the basic principles are very similar. To keep engines cool enough to function properly, water is pumped through the internal combustion engine and heated by the combustion process. The water at that point streams into the radiator and through various littler tubes, which are associated with the numerous fins. The columns of fins extraordinarily increment the surface zone to improve the cooling limit of the air coursing through the radiator. The movement of the car, alongside the fan, powers air through the balances and around the containers of the radiator. The flowing air removes heat from the water flowing through the pipes to dissipate it to the atmosphere.

A. Mathematical Models:

The goal of the project is to use the analytical heat transfer process to determine outlet coolant and air temperatures and to compare this data with experimental results. To begin the theoretical process, it is important to distinguish the given, known information from the unknown information. All dimensions for both the fan and radiator are known from precise measurements. The mass flow rates of both the air and water are known based on the given specifications of the fan and radiator. The inlet temperature for air can be assumed to be nearly equal to the ambient temperature. The inlet temperature of water is equal to the temperature of the engine. The whole hypothetical process starts with Equation 1 below for the warmth exchange rate of the radiator. From this equation, it is clear that the only two unknowns for the system are the outlet temperatures for air and water. These two unknowns are initially assumed. The equations and calculations following are simply used to solve for both outlet temperatures. The initially first expected outlet temperatures and computed outlet temperatures are then iterated to illuminate for the real qualities for the outlet temperatures for air and water.

$$q = m_{air} c_{p,air} \left(T_{air,out} - T_{air,in}\right) = m_{water} c_{p,water} \left(T_{water,in} - T_{water,out}\right)$$

B. Internal Flow of Water:

The hot water from the engine travels through the tubes of the radiator.

1) Area of Tubes

$$\mathrm{D_{hydraulic}} = \frac{4\mathrm{A_{tube}}}{\mathrm{P_{tube}}}$$

The hydraulic diameter must be used because it is a non-circular cross section. The hydraulic diameter can then be used to estimate the Reynolds number. The equation for the hydraulic diameter calls for the wetted perimeter of the tubes. However, the difference in the outer and inner tube dimensions is so negligible that the outer perimeter is used for convenience.

2) Velocity

$$\mathrm{V_{water}} = \frac{\mathrm{Q_{water}}}{\mathrm{N_{tube} A_{tube}}}$$

The velocity of the water through each tube must be found to calculate the Reynolds number. The number of tubes is given by the chosen radiator.

3) Reynolds Number

$$\mathrm{Re_{water}} = \frac{\mathrm{\rho_{water} V_{water} D_{hydraulic}}}{\mathrm{\mu_{water}}}$$

4) Nusselt Number

The Nusselt number was found as a constant for a rectangular cross section from calculation for fully developed laminar flow. The ratio of width over height of the tube is used in this table to determine the Nusselt number. This leads to a Nusselt number of 3.96.

Convective Heat Transfer Coefficient for Water Flow

$$h_{water} = \frac{Nu_{water} K_{water}}{D_{hydraulic}}$$

At long last, the convective heat exchange coefficient for the interior flow can be discovered utilizing the condition.

C. External Flow of Air:

The wind currents from the fan over the radiator tubes and through the balances using convective warmth exchange. As a general rule, the flow of air over the tubes will be somewhat extraordinary because of the liquid streaming around the principal tube before achieving the second tube, so ascertaining the warmth exchange coefficient would be exceptionally troublesome. To rearrange the calculations, the stream is thought to be the same over the two tubes. Likewise, on the grounds that the tallness to width proportion of the tubes is so little, the air will be thought to stream on the two sides of a level plate.

1) Velocity

$$\mathrm{V_{air}} = \frac{\mathrm{Q_{air}}}{\mathrm{A_{radiator}} - (\mathrm{N_{tube} H_{tube} L_{radiator}})}$$

2) Reynolds Number

$$\mathrm{Re_{air}} = \frac{\mathrm{V_{air} W_{fin}}}{\mathrm{V_{air}}}$$

3) Nusselt Number

Taking a gander at the geometry of the tubes, it can be expected that the stream of air is like parallel stream over a level plate. Since the flow never reaches the critical Reynolds number for a flat plate, Re = 5x105, it is said to be laminar for the entire process.

$$\mathrm{Nu_{air}} = 0.664 \mathrm{Re_{air}^{\frac{1}{2}} Pr_{air}^{\frac{1}{3}}}$$

D. Fin Dimensions and Efficiency:

The geometry of the fins on the radiator is sinusoidal. The troughs of the fins contact the lower contiguous tube and the pinnacles of the fins contact the upper neighboring tube. The heat from the tubes exudes through the fins. The fins and tubes are then cooled by the air from the fan, which is bridging the radiator. To streamline the geometry for the simplicity of figurings, the fins are thought to be straight rather than sinusoidal. This can be found in Fig. 6. This is a minor progress in geometry since the shape and position of the genuine balances are so near the straight arrangement. The accompanying recipes are offered underneath to ascertain the fins productivity. The fins productivity condition considers the geometry of the fins and its measurements to discover the effectiveness the fins will have.

$$\mathrm{\eta_{fin}} = \frac{\tanh(\mathrm{mL_c})}{\mathrm{mL_c}}$$

1) Overall Surface Efficiency

The general surface effectiveness is required for the outer flow of air on the grounds that the defects of the flow around the balances must be considered.

$$\mathrm{\eta_o} = 1 - \frac{\mathrm{N_{fin} A_f}}{\mathrm{A_{fin,base}}} (1 - \mathrm{\eta_{fin}})$$

E. Effectiveness-NTU Method:

The Effectiveness-NTU technique is utilized to discover the effectiveness of the system. The general heat exchange coefficient is required. The surface productivity is required for the outer flow of air in light of the fact that the defects of the flow around the fins must be considered. Utilizing the convective heat exchange coefficients of both the interior and outer flow, the UA is figured. This esteem is then used to figure the NTU.

1) Generally Heat Transfer Coefficient

$$UA = \frac{1}{\left(\frac{1}{\eta_o h_{air} A_{external}} + \frac{1}{h_{water} A_{internal}}\right)}$$

2) Number of Transfer Units

$$NTU = \frac{UA}{C_{min}}$$

3) Effectiveness

The radiator uses a cross-stream single pass mastermind where the two liquids stay unmixed. This identifies with a specific condition to register viability.Be that as it may, this condition requires the heat limit proportion, Cr, to be equivalent to 1. The figured heat limit proportion is 0.455; consequently, the effectiveness is just a nearby estimate rather than the true value.

$$\varepsilon = 1 - \exp\left[\left(\frac{1}{Cr} NTU^{0.22}\right)\left(\exp^{(-Cr NTU^{0.78})} - 1\right)\right]$$

F. Heat Transfer Rate:

The most extreme heat exchange rate must be found to discover the predicted heat exchange rate. When this is known, the last yield temperature of both the hot and chilly liquid is ascertained utilizing a changed version of the initial thermal energy equation. These outlet temperatures must be iterated with the at first speculated outlet temperatures until the point when the numbers are comparable. The iterated outlet temperatures for both air and water are the hypothetical qualities, which are utilized to contrast and the exploratory outcomes.

1) Max Heat Transfer Rate

$$q_{max} = C_{min}\left(T_{water,in} - T_{air,in}\right)$$

2) Predict Heat Transfer

$$q_{predicted} = \varepsilon q_{max}$$

3) Temperature Out

$$T_{water,out} = T_{water,in} - \frac{q_{predicted}}{C_{water}}$$

$$T_{air,out} = T_{air,in} - \frac{q_{predicted}}{C_{air}}$$

IV. DESIGN AND METHOD

Modern automotive internal combustion engines generate a huge amount of heat. This heat is generated when the fuel and air mixture is ignited in the combustion chamber. This explosion causes the piston to be forced down inside the engine and creates power. Metal temperatures around the combustion chamber can exceed 200° F.

Out of this generated heat approximately 30-35% of the heat in combustion is converted into power to drive the vehicle and its accessories. Another 35-40% of the heat is carried off into the atmosphere through the exhaust system. The remaining 30-35% must be removed from the engine by the cooling system in order to prevent the overheating of the engine oil, cylinder walls, pistons, valves, and other components by these extreme temperatures, it is necessary to effectively dispose this heat. But sometimes this heat cannot be effectively removed due to inefficient design of radiator. This problem has been taken into consideration for our project work. Also excessive cooling system capacity can also be harmful, and may affect engine life and performance. So our ultimate aim is to design an effective cooling system which controls the engine temperature within a specific range so that engine stays within peak performance.

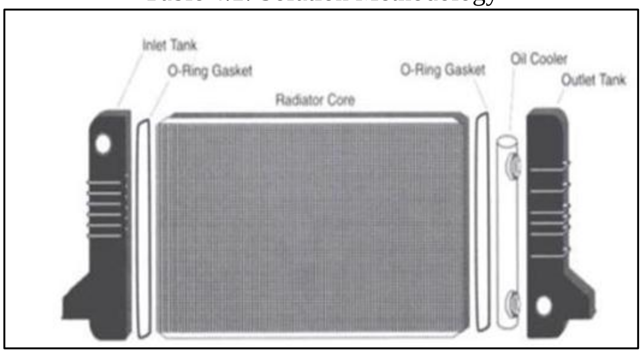
Computational Fluid Dynamics (CFD) is used in much of the automotive industry for cooling system design. CFD calculations are made on expensive supercomputers and the process can take weeks to yield results. When multiple designs are to be evaluated, especially early in the design phase of a new vehicle, long CFD calculations are costly and can delay a final design decision. The economic and marketing pressures to reduce time to market for new products and improve engineering productivity are always present. They require a faster, more efficient process for making product development decisions.

A. Problem Formulation and Solution:

Table 4.1 shows the details of solution methodology adopted for formulation and solution of the problem

S. No.	Step No.	Remark
1	Selection of a standard automobile engine radiator	The targeted automobile is wagon R
2	Calculation of outlet temperature	Calculated from heat transfer equations
3	Investigations on Nano fluid used in radiator and design of radiator tube	Elliptical and helical tube radiator used with different nano fluid like water, Ethanol and Methanol
4	Investigations on parameter for CFD analysis using software approach	The targeted software was ANSYS
5	Comparison of different nano fluid used in radiator	----
6	Remark of different Nano fluid and proposed helical tube	----

Table 4.1: Solution Methodology



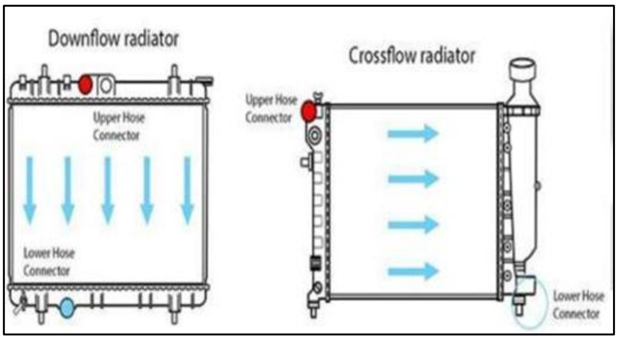


Fig. 4.1: Down flow & Cross flows Radiator

B. Design Calculations:

We have following data of Maruti Wagon-R car(Data available on Maruti wagon r user's manual& specifications):
Engine: - K10B
Displacement: - 998cc
Fuel Type: - Petrol (Gasoline)
Max. Power (P): - 67.04 BHP @ 6200 RPM
Max. Torque: 90Nm @3500 RPM
Amount of heat lost by radiator

$$\text{Brake Power, BP} = \frac{2 \times \pi \times N_{max} \times T_{max}}{60}$$

$$\text{Brake Power, BP} = \frac{2 \times 3.142 \times 6200 \times 90}{60}$$

BP = 58.43 KW

It is recommended that the mechanical efficiency (η_{mech}) of an internal combustion engine may vary from 75 to 90 percent.

Let, Mechanical efficiency, η_{mech} = 85% = 0.85
Indicated Power, IP = BP / η_{mech}
= 58.43 / 0.85
= 68.74 KW

The thermal efficiency (η_{ith}) of an internal combustion engine may vary from 30 to 35 percent.

Now, Let Indicated thermal efficiency, η_{ith} = 30% = 0.3
Therefore, Total Heat Produced, Q = IP / η_{ith}= 68.74 / 0.3
Q = 230 KW
Now, out of this total heat,
Heat Exhausted & Unaccounted, = 40% = 92 KW
Heat used for Power Generation, = 30 - 35% = 69 - 80.5 KW
Heat loss in Radiator, = 30 - 35% = 69 - 80.5 KW

So, we have to remove minimum 69 KW and maximum 80.5 KW of heat through the cooling system. We have done calculations to remove this amount of heat through from designed radiator.

S. No	Parameters	Specifications
1.	Inlet Temperature of Water (Th1)	85°C
2.	Inlet Temperature of Air (Tc1)	35°C
3.	Dimensions of inlet & outlet tank mm	50 × 60 × 354
4.	Shape of Tube	Elliptical
5.	Major & minor axis of tube resp.	8 mm & 4 mm
6.	Length of tube	400 mm
7.	Number of tubes	32
8.	Distance between two tubes	30 mm
9.	Length of the fin on tube	5 mm
10.	Thickness of fin on tube	1 mm
11.	Length of fin between two tubes	30 mm
12.	Thickness of fin	0.5 mm
13.	Surface Area	0.51m2
14.	Distance between two fins	2.85 mm
15.	Mass flow rate of water (mw)	0.746 Kg/sec
16.	Mass flow rate of air (ma)	2.27 Kg/sec
17.	Total heat transfer coefficient (U)	27.216 W/m2K

Table 4.2: Available data for Elliptical Tube Automotive Radiator (Wagon-R)[4]

C. Calculated data for Elliptical Tube Automotive Radiator (Wagon-R):

Max Heat Transfer Rate

$$Q_{max} = 80.5\ \text{KW}$$

Min Heat Transfer Rate

$$Q_{min} = 69\ \text{KW}$$

Predict Heat Transfer Effectiveness (ε)

$$Q_{predicted} = \varepsilon Q_{max}$$

$$\varepsilon = \frac{Q_{predicted}}{Q_{max}} \times 100 = \frac{69}{80.5} \times 100 = 85.71\%$$

Temperature Out

$$T_{water,out} = T_{water,in} - \frac{q_{predicted}}{C_{water}}$$

$$T_{water,out} = T_{water,in} - \frac{q_{predicted}}{m_w c_{pw}} = 85 - \frac{69}{3.11} = 62.81^{\circ}\text{C}$$

$$T_{air,out} = T_{air,in} + \frac{q_{predicted}}{C_{air}}$$

$$T_{air,out} = T_{air,in} + \frac{q_{predicted}}{m_a c_{pa}} = 35 + \frac{69}{2.28} = 65.26^{\circ}\text{C}$$

Heat Transfer Rate, $Q = U \times A \times \theta_m \times F$

$$U = 27.216\ \text{W/m}^2\text{K} = 7.43\text{KW/m}^2{}^{\circ}\text{C}$$

$$A = 0.51\text{m}^2$$

$$\Delta T_m = \frac{\Delta T_b - \Delta T_a}{\ln\left(\frac{\Delta T_b}{\Delta T_a}\right)}$$

Where ΔT_a And ΔT_b are defined as the difference between the hot fluid inlet temperature and cold fluid inlet temperature, and hot fluid outlet temperature and cold fluid inlet temperature respectively. The term ΔT_m denotes that this LMTD is for the case of a counter flow heat exchanger. For the other types of heat exchanger i.e. cross flow a correction factor F must be applied.

The Correction factor is obtained from the calculation of two ratios P and R associated with the inlet and outlet temperature of the two fluids (cross flow heat exchanger)

$$P = \frac{t_2 - t_1}{T_1 - t_1}$$

$$R = \frac{T_1 - T_2}{t_2 - t_1}$$

Correction factor, F = 0.87 From graph in HMT Data Book

$$Q = 7.43 \times 0.51 \times 22 \times 0.87$$

$$Q = 72.52\ \text{KW}$$

Calculated Heat Transfer Effectiveness (ε)

$$Q_{min} = \varepsilon Q_{max}$$

$$\varepsilon = \frac{72.52}{80.5} \times 100 = 90.08\%$$

S. No	Parameters	Specifications
1.	outlet temperature of water (th2)	62.81°c
2.	Outlet temperature of air (Tc2)	65.26°C
3.	Total Heat transfer (Q)	72.52 KW
4.	Effectiveness (ε)	90.08 %

Table 4.3: Calculated data for Elliptical Tube Automotive Radiator (Wagon-R):

Fig. 4.2: Actual Design of Automotive Radiator (Wagon-R)

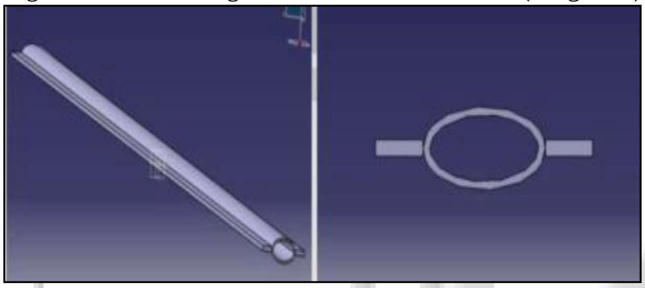


Fig. 4.3: 3D and Cross-sectional views of Radiator Tube

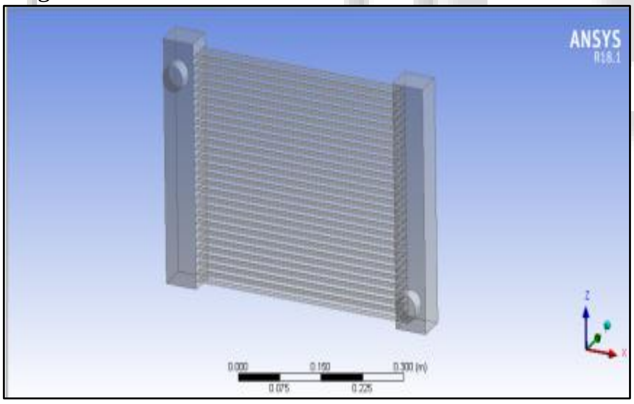


Fig. 4.4: ANSYS Actual Design of Elliptical Tube Automotive Radiator (Wagon-R)

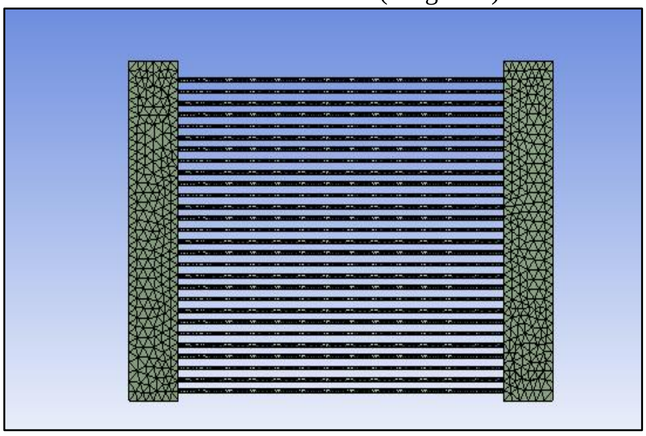


Fig. 4.5: ANSYS meshed model of Elliptical Tube Automotive Radiator (Wagon-R)

D. Proposed Design:

The proposed design of radiator is done as per the standard designing procedure for our project work.
It includes the design of radiator model on 3D modelling mechanical software (CREO),
Its Manual calculations, CFD analysis on ANSYS software and its results.

S. No	Specification	Value
1	Shape of Tube	Helical Type
2	Dimensions of inlet & outlet tanks mm	50 × 60 × 295
3	Number of tubes	19
4	Helical type tube mean diameter	8 mm
5	Distance between two tubes	30 mm
6	Pitch	16
7	Number of turns	25
8	Helix angle	32.48^0

Table 4.4: Specification of Helical Tube Radiator

1) CREO Model of Helical Tube Radiator:

The designed model of radiator is made with the help of CREO software as per dimensions and calculations carried out for our project work. Figure 2.1.1 shows the 3D model of radiator

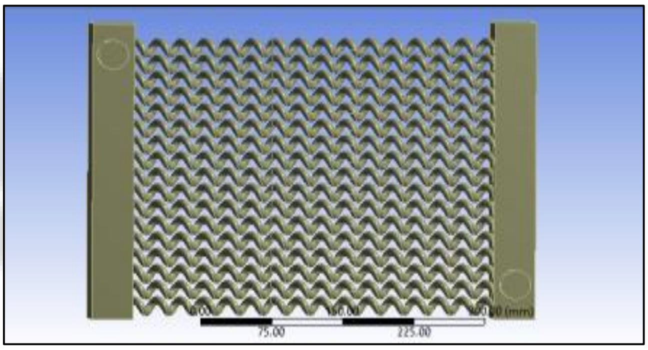


Fig. 4.6: CREO model of Helical tube Radiator

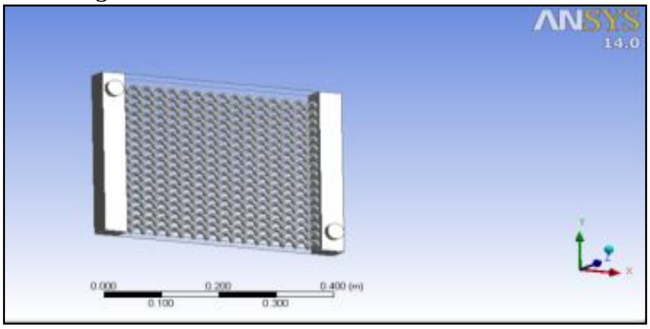


Fig. 4.7: ANSYS model of Helical tube Radiator

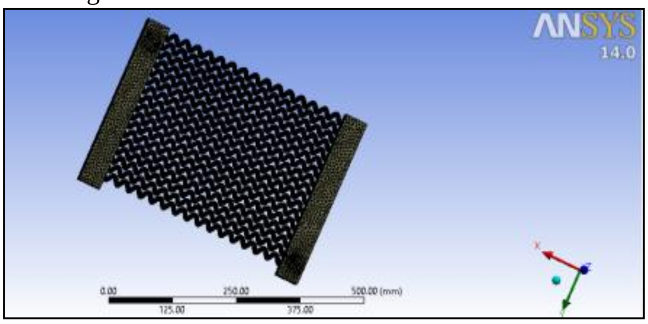


Fig. 4.8: ANSYS meshed model of Helical tube Radiator

2) Analysis of fin Arrays for Heat Transfer with ANSYS Workbench:

ANSYS is a broadly useful programming, used to reenact collaborations of all controls of material science, basic, vibration, liquid flow, warm exchange and electromagnetic for engineers. So ANSYS, which empowers to reenact tests or working conditions, empowers to test in virtual condition before assembling models of items. Moreover, deciding and enhancing frail focuses, registering life and predicting plausible issues are conceivable by 3D reproductions in virtual condition. ANSYS programming with its modular structure as found in the table below gives an opportunity for taking just required highlights. ANSYS can work incorporated with other utilized designing programming on work area by including CREO and FEA association modules. ANSYS can import CREO data and moreover engages to produce geometry with its "pre-dealing with" limits. Basically in the same pre-processor, finite element model (mesh) which is required for count is created. Subsequent to characterizing loadings and completing examinations, results can be seen as numerical and graphical. ANSYS can do propelled building investigations rapidly, securely and for all intents and purposes by its assortment of contact calculations, time based stacking highlights and nonlinear material models. ANSYS Workbench is a stage which integrates simulation advancements and parametric CREO frameworks with extraordinary mechanization and execution. The intensity of ANSYS Workbench originates from ANSYS solver algorithms with long periods of experience. Moreover, the protest of ANSYS Workbench is check and enhancing of the item in virtual condition.

ANSYS Workbench, which is composed for abnormal state similarity with particularly PC, is in excess of an interface and anyone who has an ANSYS permit can work with ANSYS Workbench. As same as ANSYS interface, limits of ANSYS Workbench are limited due to possessed license. Import the Geometry

After the creation of design the next process is to analysis the fin arrays for heat transfer by using software ANSYS Workbench. The important factor is while saving the model in CREO it should be in ".igs" file format. So it can access through the FEA software. Then, select the type of analysis as thermal analysis for Transient heat transfer process. Assign unit system as customization length in mm, temperature in °C. Import the models to ANSYS Workbench.

3) Generating the Mesh

A mesh independency study has been performed for a continuous fin case, with six different mesh sizes.

Sr. no.	Entity	Details
1	Element type	Medium
2	Number of nodes	1526453
3	Number of elements	8113361

Table 4.5: Elliptical Tube Radiator Meshing Attributes for CFD Analysis

S. no.	Entity	Details
1	Element type	Auto mesh
2	Number of nodes	1426453
3	Number of elements	8213361

Table 4.6: Helical Tube Radiator Meshing Attributes for CFD Analysis

a) Assigning Material and loads to the Meshed Model

Aluminium has selected for the model due to light in weight and high rate of heat transfer and heat dissipation in this material. The manufacturing process also simple in the aluminium and cost wise it is an economic.

The material having thermal conductivity, convection coefficient of heat transfer for air, temperature of surface and ambient temperature as:

Molar mass, 26.98 Kg K/mol

Density, 2702 Kg/m^3

Thermal conductivity, k = 237 W/m K

Specific Heat, 903 J/kg K

Heat flow through wall surface at which fin attached = 69 - 80.5 KW

Ambient temperature, t_a = 35 °C

V. RESULT AND DISCUSSION

A. Methodology & Theoretical Orientation:

Heat transfer increases as we redesign the tube of radiator and increase the surface area of the fins with the use of nano fluids. Modern design (Varying turbulence model) can dissipate the same and/or more heat by using vortex generators in tubs of radiator. Helical tube disturbs the flow field and provides swirling flow which causes high rate of heat exchange of core and wall fluid.

1) Findings:

Vortex generators in helical tube decrease the wake region size and increase the intensity of secondary flow. The fins with the helical tube increases the heat transfer performance of radiator about 55.8% compared to existing radiator.

Input Data: [Reference 4]

Air inlet temp : 308K

Coolant inlet temp : 358.15K

Coolant mass flow : 0.746 kg/sec.

Flow region : Laminar

Overall heat transfer co efficient across the radiator ranges from 25 to 560 W/ m^2-K

B. Assumptions:

In order to solve the analytical model, the following assumptions are made:

Coolant flow rate is constant and there is no phase change in the coolant. Heat conduction through the walls of the coolant tube is negligible. Heat loss by coolant was just exchanged to the cooling air, in this manner no other heat exchange mode, and for example, radiation was considered. Coolant liquid stream is in a completely created condition in each tube. All measurements are uniform all through the radiator and the heat exchange surface zone is predictable and distributed consistently. The heat conductivity of the radiator material is thought to be consistent. There are no heat sources and sinks inside the radiator. There is no liquid stratification, misfortunes and flow non distribution. Force condition: Tube divider is stationary.

Coolant	Density Kg/m^3	Specific Heat Capacity J/Kg K	Dynamic Viscosity Kg/m Sec	Thermal Conductivity W/m K

Water	997.0	4181.7	8.899E-4	0.6069
Al2o3 water nano fluid 2%	1033.6	4120.5	0.000627	0.68
Al2o3 water nano fluid 3%	1059.8	4086.2	0.000642	0.699
Methanol CH4O	791	2500	0.594E-03	0.201
Ethanol C2H6O	789	2500	1.197E-03	0.177
CuO/water	1061	4150.9	0.000612	0.662

Table 5.1: Property table of nano Fluid (ANSYS Library material Properties)

C. Analysis of Radiator with Elliptical Tubes (mass flow rate =0.746 kg/sec)

Elliptical Tube Using Coolant: Water
Elliptical Tube Using Coolant: Ethanol
Elliptical Tube Using Coolant: Methanol

S. No	Parameters	Specifications
1	Shape of Tube	Elliptical Type
2	Dimensions of inlet & outlet tanks (mm)	50 × 60 × 354
3	Number of tubes	32
4	Major & minor axis of tube respectively	8 mm & 4 mm
5	Distance between two tubes	30 mm

Table 5.2: Radiator Specification for Elliptical Type Tubes

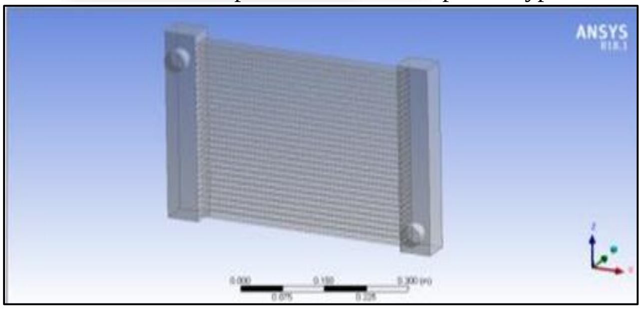


Fig. 5.1: Actual Design of Elliptical Tube Automotive Radiator (Wagon-R) in ANSYS

1) Case-I: Analysis of Radiator with Elliptical Tube Using Coolant: Water

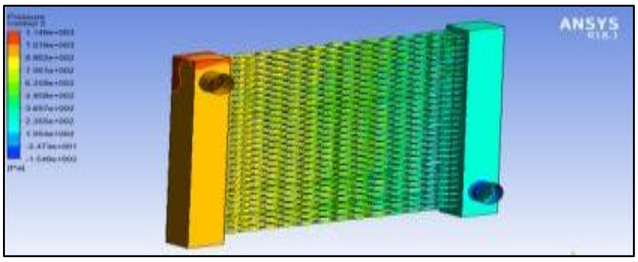


Fig. 5.2: Pressure Contour diagram of elliptical tubes used in Radiator. (Water)

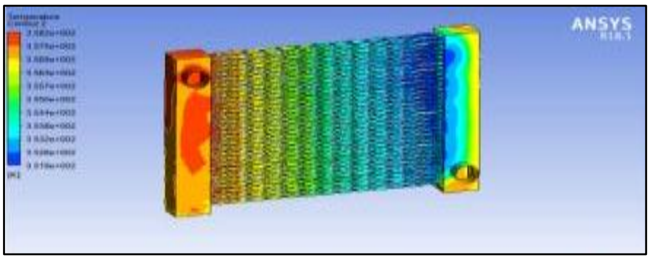


Fig. 5.3: Temperature diagram of elliptical tubes used in Radiator. (Water)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	350.19	7.96

2) Case-II: Analysis of Radiator with Elliptical Tube Using Coolant: Ethanol

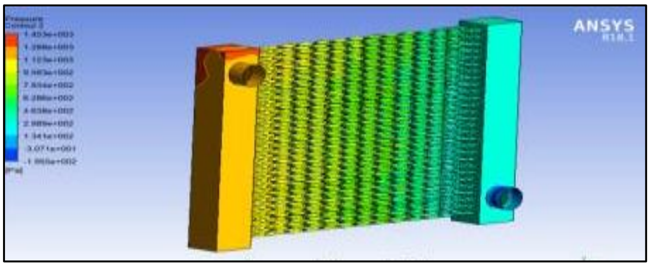


Fig. 5.4: Pressure Contour diagram of elliptical tubes used in Radiator. (Ethanol)

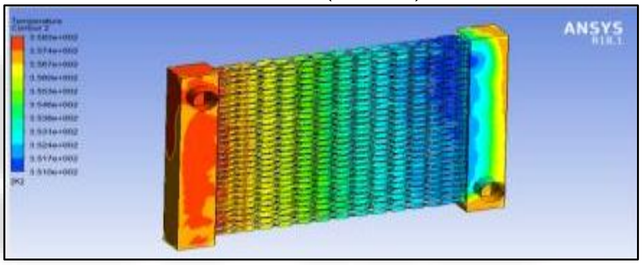


Fig. 5.5: Temperature diagram of elliptical tubes used in Radiator. (Ethanol)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	342.89	15.26

3) Case-III: Analysis of Radiator with Elliptical Tube Using Coolant: Methanol

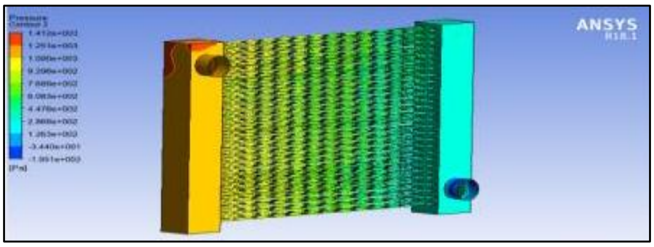


Fig. 5.6: Pressure Contour diagram of elliptical tubes used in Radiator. (Methanol)

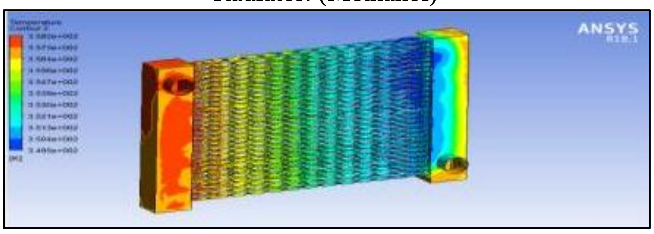
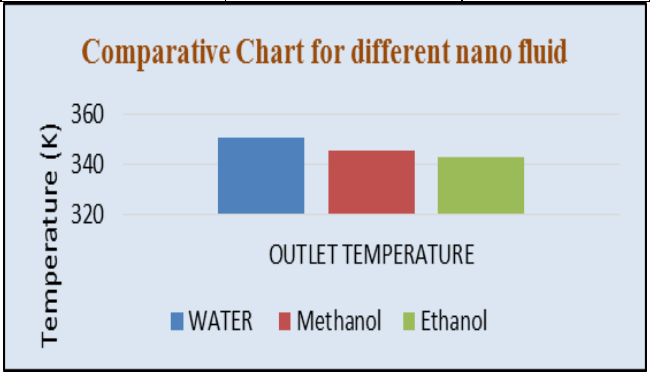


Fig. 5.7: Temperature diagram of elliptical tubes used in Radiator. (Methanol)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	345.34	12.81

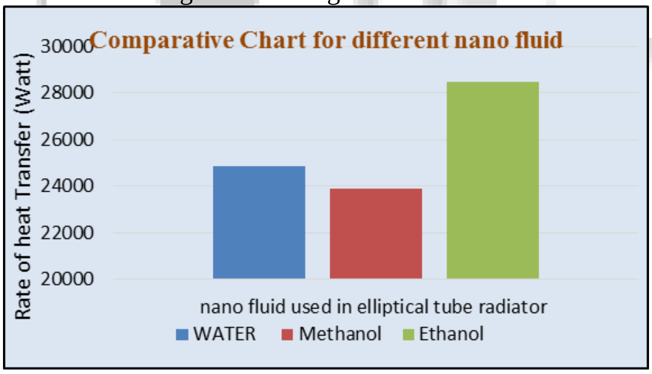


Graph 5.1: Outlet Temperature of different nano fluid used in elliptical tube Radiator

As discussed for the small scale radiator, the flow is not distributed among tubes as seen in Graph 5.1. This non-uniformity also contributes to the temperature gradient among the tubes in the x-direction. According to the simulation, the average outlet coolant temperature was found as a result, According to this temperature drop, the total heat capacity of the radiator can be calculated for different type of coolant;

$$Q = m\, C_p \Delta T$$

According to the Ansys CFD results for various nano fluid used in helical tube type radiator, the outlet temperature of nano fluid is determined and numerically determined outlet temperatures, which is quite acceptable. The heat capacity of the radiator is determined as by the calculations using the relation given above.



Graph 5.2: Rate of Heat dissipation through Elliptical tube Radiator

Coolant Used in Radiator	Mass flow rate m (Kg/Sec)	Specific Heat C_p (KJ/KgK)	Coolant in temp(k)	Coolant out temp(k)	Temperature difference ΔT (k)	Rate of Heat dissipation $Q = m\, C_p \Delta T$ (Watt)
Water	0.746	4181.7	358.15	350.19	7.96	24831.60
Ethanol	0.746	2500	358.15	342.89	15.26	28459.90
C2H6O						
Methanol (CH4O)	0.746	2500	358.15	345.34	12.81	23890.65

Table 5.3: Rate of Heat dissipation through elliptical tube radiator

D. Analysis of Radiator with Helical Tubes (mass flow rate =0.746 kg/sec)

- Helical Tube Using Coolant: Water
- Helical Tube Using Coolant: Al2o3 water nano fluid 2%
- Helical Tube Using Coolant: Al2o3 water nano fluid 3%
- Helical Tube Using Coolant: Methanol
- Helical Tube Using Coolant: Ethanol
- Helical Tube Using Coolant: CuO/water

S. No	Specification	Value
1	Shape of Tube	Helical Type
2	Dimensions of inlet & outlet tank (mm)	50 × 60 × 354
3	Number of tubes	19
4	Helical type tube mean diameter	8 mm
5	Distance between two tubes	30 mm

Table 5.4: Radiator Specification for Helical Type Tubes:

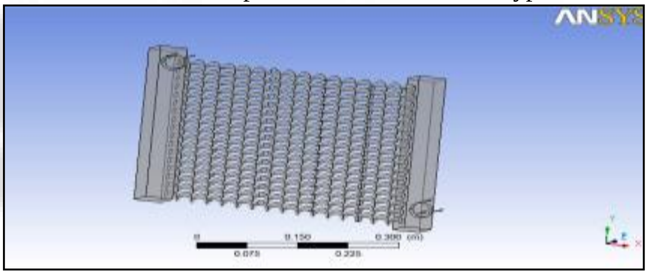


Fig. 5.8: Helical tube radiator in ANSYS

1) Case-I: Analysis of Radiator with Helical Tubes Using Coolant: Water

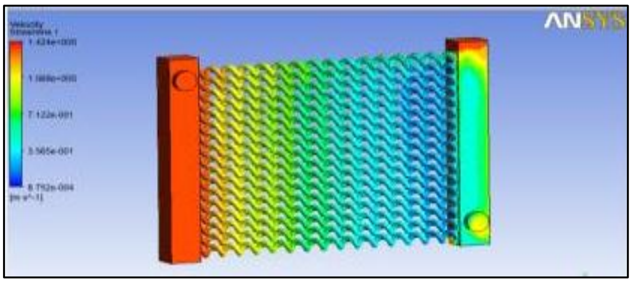


Fig. 5.9: Velocity Streamline diagram of helical tubes used in Radiator. (Water)

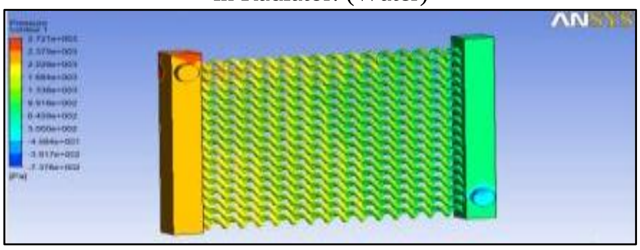


Fig. 5.10: Pressure Contour diagram of helical tubes used in Radiator. (Water)

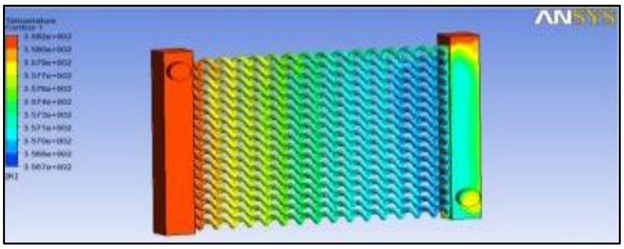


Fig. 5.11: Temperature diagram of helical tubes used in Radiator. (Water)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	349.16	8.99

2) Case-II: Analysis of Radiator with Helical Tubes Using Coolant:Al2o3 water nano fluid 2%

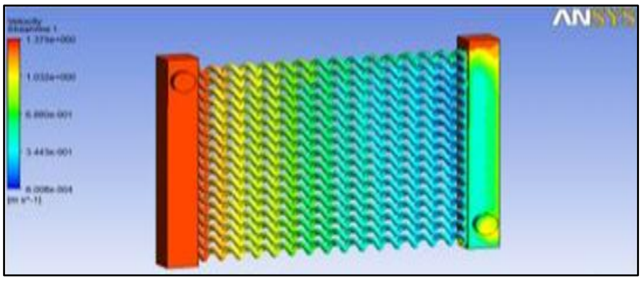


Fig. 5.12: Velocity Streamline diagram of helical tubes used in Radiator. (Al2o3 water nano fluid 2%)

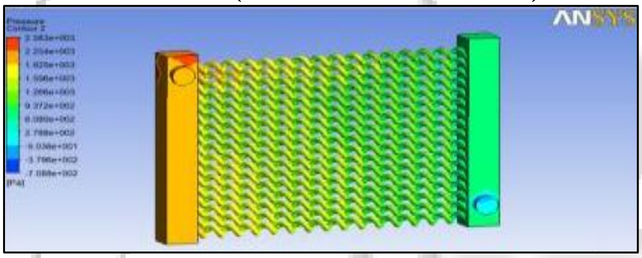


Fig. 5.13: Pressure Contour diagram of helical tubes used in Radiator. (Al2o3 water nano fluid 2%)

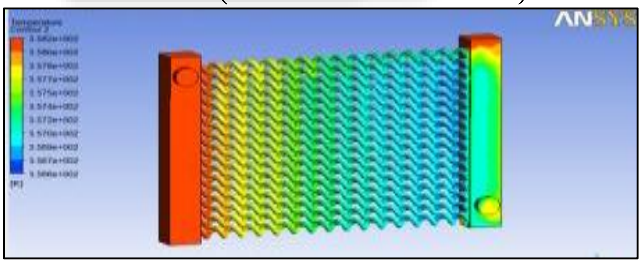


Fig. 5.14: Temperature diagram of helical tubes used in Radiator. (Al2o3 water nano fluid 2%)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	350.33	7.82

3) Case-III: Analysis of Radiator with Helical Tubes Using Coolant:Al2o3 water nano fluid 3%

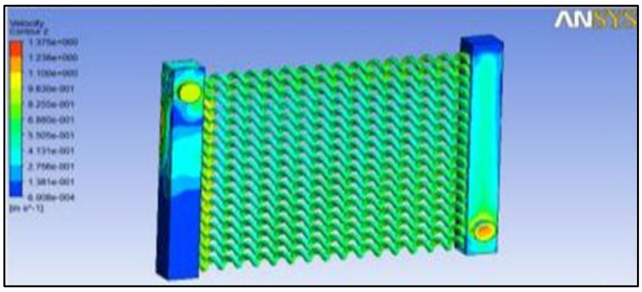


Fig. 5.15: Velocity Streamline diagram of helical tubes used in Radiator. (Al2o3 water nano fluid 3%)

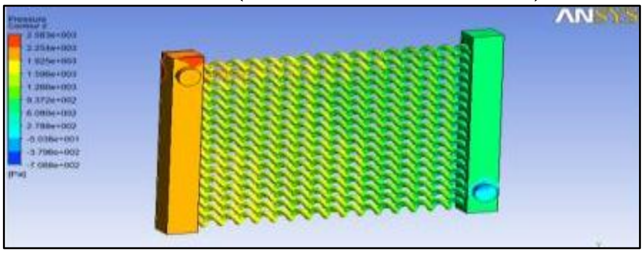


Fig. 5.16: Pressure Contour diagram of helical tubes used in Radiator. (Al2o3 water nano fluid 3%)

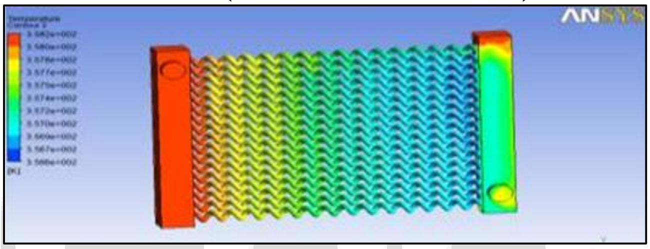


Fig. 5.17: Temperature diagram of helical tubes used in Radiator. (Al2o3 water nano fluid 3%)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	350.33	7.82

4) Case-IV: Analysis of Radiator with Helical Tubes Using Coolant: Methanol

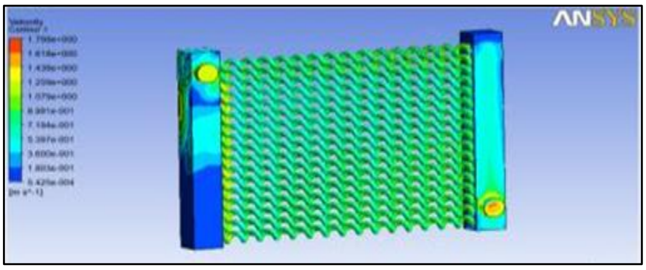


Fig. 5.18: Velocity Streamline diagram of helical tubes used in Radiator. (Methanol)

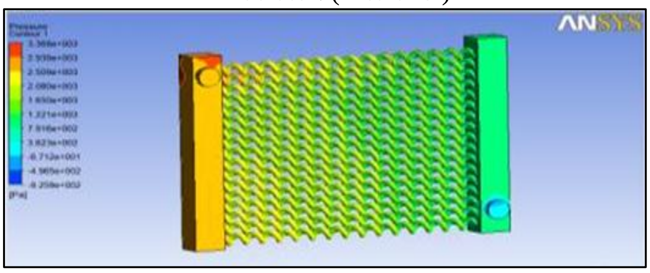


Fig. 5.19: Pressure Contour diagram of helical tubes used in Radiator. (Methanol)

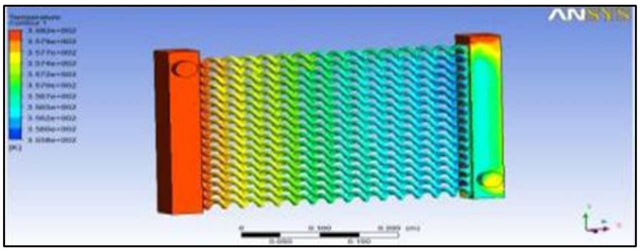


Fig. 5.20: Temperature diagram of helical tubes used in Radiator. (Methanol)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff.(k)
358.15	353.94	4.21

5) Case-V: Analysis of Radiator with Helical Tubes Using Coolant Ethanol: -

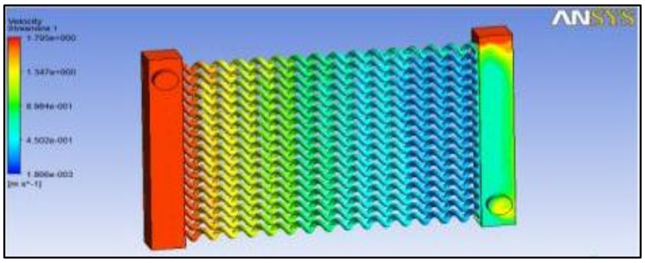


Fig. 5.21: Velocity Streamline diagram of helical tubes used in Radiator. (Ethanol)

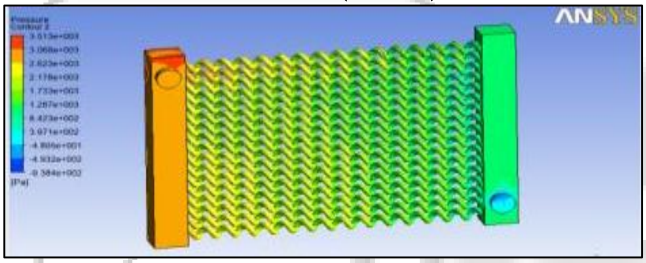


Fig. 5.22: Pressure Contour diagram of helical tubes used in Radiator. (Ethanol)

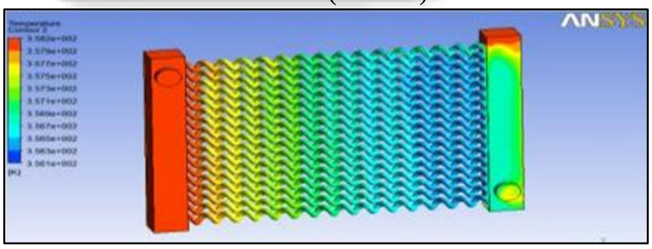


Fig. 5.23: Temperature diagram of helical tubes used in Radiator. (Ethanol)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	336.91	21.24

6) Case-VI: Analysis of Radiator with Helical Tubes Using Coolant CuO/water: -

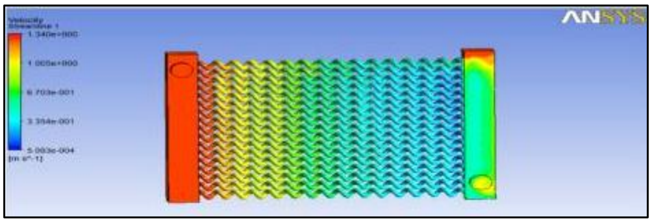


Fig. 5.24: Velocity Streamline diagram of helical tubes used in Radiator. (CuO/water)

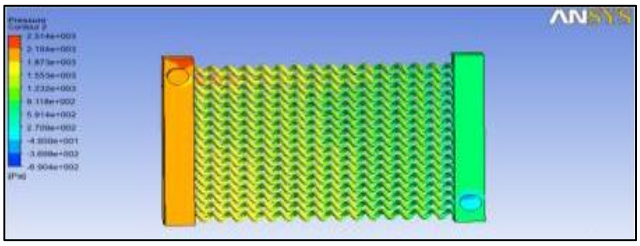


Fig. 5.25: Pressure Contour diagram of helical tubes used in Radiator. (CuO/water)

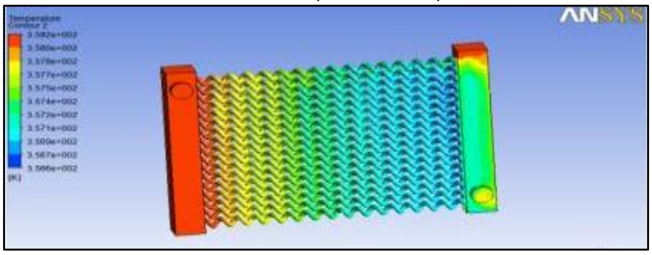
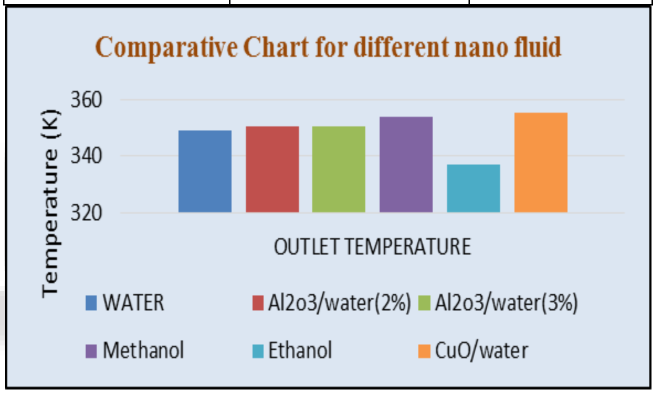


Fig. 5.26: Temperature diagram of helical tubes used in Radiator. (CuO/water)

Coolant in temp(k)	Coolant out temp(k)	Temp. diff. (k)
358.15	355.38	2.77



Graph 5.3: Outlet Temperature of different nano fluid used in Helical tube Radiator

We can see that by fixing the water proportion in elliptical tube radiator and taking the reading with different coolant (i.e. Like Ethanol, Methanol, Al_2O_3/water2%, Al_2O_3/water3% and CuO/water) in 30 % mixing ratio with the water, then the ethanol gives the least outlet temperature 342.89K among all the mixtures and in case of helical tube radiator ethanol gives the least outlet temperature 336.91K. So from the result, it is desirable to use ethanol with water, which gives better performance.

As discussed for the small scale radiator, again the flow is not distributed among tubes as seen in Graph 5.3. This non-uniformity also contributes to the temperature gradient among the tubes in the x-direction. According to the simulation, the average outlet coolant temperature was found as a result, According to this temperature drop, the total heat capacity of the radiator can be calculated for different types of coolant;

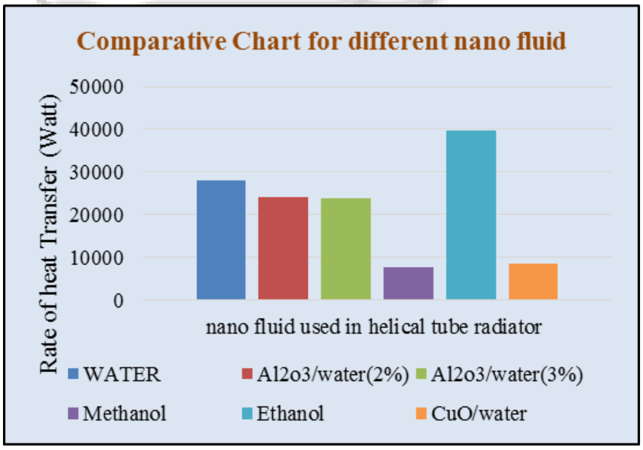
$$Q = m\, C_p \Delta T$$

According to the Ansys CFD results for various nano fluid used in helical tube type radiator, the outlet temperature of nano fluid is determined and numerically determined outlet temperatures, which is quite acceptable.

The heat capacity of the radiator is determined as by the calculations using the relation given above.

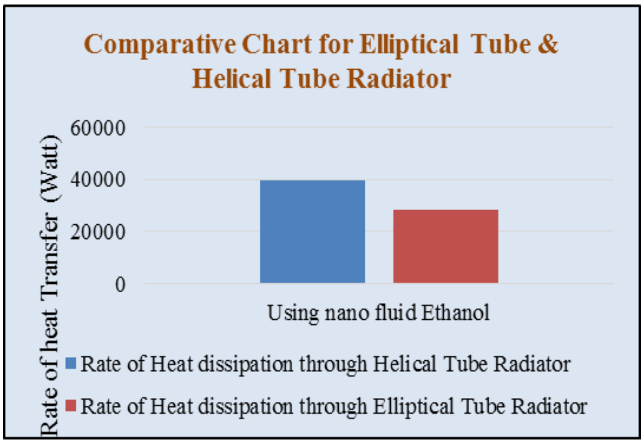
Coolant Used in Radiator	Mass flow rate m (Kg/Sec)	Specific Heat C_p (KJ/KgK)	Coolant in temp(k)	Coolant out temp(k)	Temperature difference ΔT (k)	Rate of Heat dissipation $Q = m C_p \Delta T$ (Watt)
Water	0.746	4181.7	358.15	349.16	8.99	28044.70
Al2o3 water nano fluid 2%:	0.746	4120.5	358.15	350.33	7.82	24037.80
Al2o3 water nano fluid 3%:	0.746	4086.2	358.15	350.33	7.82	23837.70
Methanol (CH4O)	0.746	2500	358.15	353.94	4.21	7851.65
Ethanol C2H6O	0.746	2500	358.15	336.91	21.24	39612.60
CuO/water	0.746	4150.9	358.15	355.38	2.77	8577.50

Table 5.5: Rate of Heat dissipation through helical tube radiator



Graph 5.4: Rate of Heat dissipation through Helical tube Radiator

We can see that by using different coolant (i.e. Like Ethanol, Methanol, Al2O3/water2%, Al2O3/water3% and CuO/water) in the elliptical type radiator ethanol gives the highest heat transfer 28459.9W among all the mixtures and in case of helical tube radiator ethanol gives the highest heat transfer 39612.6W. So from the result, it is desirable to use ethanol with water, which gives better performance



Graph 5.7: Rate of Heat dissipation through Elliptical tube & Helical tube Radiator

VI. SUMMARY AND CONCLUSION

Performance of engine cooling system is influenced by factors like air and coolant mass flow rate, air inlet temperature, coolant fluid, fin type, fin pitch, tube type and tube pitch etc.

While designing cooling system main aim remains that the size of the cooling system should be less. but three factors does not allow the size to decrease. The factors are

– High altitude: At high altitude, air density becomes low and hence affects air mass flow rate.
– Summer conditions: During summer surrounding air is hot i.e. air inlet temperature is more.
– Maximum power: Engine condition producing maximum power like when vehicle is climbing uphill, maximum heat rejection is required during this condition.

To remunerate every one of these elements radiator centre size required might be extensive. In this study approach has been made to increase the value of air flow rate which in turn takes care of the size of the radiator.

– We can conclude that the highest outlet temperature 350.19K for water as coolant among all the nano fluid as coolant and ethanol gives the least outlet temperature 342.89K in elliptical tube radiator with different coolant (i.e. Like Ethanol, Methanol and CuO/water). So from the result, it is desirable to use ethanol with water, which gives better performance in elliptical tube radiator used in Wagon-R car, it gives the low temperature at the outlet.
– We can conclude that by fixing the water proportion and taking the reading with different coolant (i.e. Like Ethanol, Methanol, Al_2O_3/water2%, Al_2O_3/water3% and CuO/water) CuO/water gives the highest outlet temperature 355.38K among all the mixtures and ethanol gives the least outlet temperature 336.91K in helical tube radiator. So from the result, it is desirable to use ethanol with water, which gives better performance.
– For Elliptical & Helical tubes Radiator; maximum temperature drop is due to Ethanol as coolant.
– In case of helical tube radiator compared to elliptical tube radiator gives better performance such as –

- Maximum temperature drop

- High rate of heat dissipation
- Compactness and high rate of heat dissipation

VII. SCOPE OF FUTURE WORK

An arrangement of numerical information on car radiator utilizing coolant working at high temperature has been displayed in the examination. By the writing study various proposals have been accommodated the advancement of a more effective and compact radiator. The same is expounded in the segment, future scope. In the execution assessment of the radiator, a radiator is introduced into a test set up and parameter of mass stream rate of air is changed its impact on the effectiveness and cooling limit is examined. Similar parameters were exhibited graphically and the derivations made.

As a future work, CFD studies can be performed with various kinds of radiator arrangements and approvals can be performed with experimentation Optimization of radiators as far as size and weight can be performed computationally for a scope of cooling limits. Also, fan of the radiator might be executed into the computational model for more sensible investigations particularly for other vehicle applications in which the fan is situated before the radiator. Together with the expanding computational intensity of the computers, the current computational model may likewise be executed for the under hood space reproductions of the entire cooling system.

REFERENCES AND WEB RESOURCES

[1] Prof. A. R. Khot, Prof. D.G. Thombare, An Overview of Radiator Performance Evaluation and Testing IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) ISSN: 2278-1684, PP: 07-14.

[2] Prof. Ghanbarali Sheikh zadeh, Prof. Mohammad hadiHajilou, Hamed Jafarian, Analysis of thermal performance of a car radiator, employing nanofluid International Journal of Mechanical Engineering and ApplicationsPublished online September 20, 2014, ISSN: 2330-023X (Print); ISSN: 2330-0248

[3] Prof. Krunal Suryaknt, Prof. Kayastha, Sharvil Dushyant Shah, Comparative Analysis of Helical tube Automobile Radiator considering different Coolants used CFD, IJAERD Volume 2,Issue 1, January -2015 e-ISSN(O): 2348-4470 p-ISSN(P): 2348-6406

[4] Prof. V. C. Pathade, Sagar R. Satpute, Design And Analysis Of Car Radiator By Finite Element Method, Dmietr,SawangiMeghe, Wardha(Mah.), India. Vol-3 Issue-2 2017IJARIIE-ISSN(O)-2395-4396

[5] Prof. Hardikkur Patel, Deepu Dinesan Performance Analysis of an Automobile Radiator using CFG, IJIRST Volume 1 | Issue 12 | May 2015 ISSN (online): 2349-6010

[6] Prof. Tonye. K. Jack, Prof. Mohammed M. Ojapah, Water-Cooled Petrol Engines: A Review Of Considerations In Cooling Systems Calculations With Variable Coolant Density And Specific Heat, May 2013. ©IJAET ISSN: 2231-1963

[7] Prof. AdityaChoure, RavibalaPatil, KishorDolare Performance Investigation Of Automobile Radiator Using Al2o3 As Base Nanofluid, International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 4, Issue-6, Jun.-2016

[8] Prof. DevendraVashist, Sunny Bhatia, Some Studies on the Performance of Automotive Radiator at Higher Coolant Temperature, Journal of Basic and Applied Engineering Research, ISSN: 2350-0077; Online ISSN: 2350-0255; Volume 1, Number 3; October, 2014

[9] Prof Bharat Raj Singh Study on Performance Evaluation of Automotive Radiator, Article June 2015 DOI: 10.18090/samriddhi.v2i2.1604

[10] Designing a More Effective Car Radiator Maplesoft, a division of Waterloo Maple Inc., 2008

[11] Lee PohSeng "Determination of Certain Thermo-Physical Properties of Nanofluids" Thesis, Universiti Malaysia Pahang, Malaysia, pp. 14 to 24, DECEMBER 2010.

[12] Vikashkumar et.al (2003) "Cfd Analysis of Cross Flow Air To Air Tube Type Heat Exchanger" Vol. 3, Hemisphere Publishing Corporation, pp.10 –16, 2003.

[13] Witry M.H. Al-Hajeri and Ali A. Bondac, "CFD analysis of fluid flow and heat transfer in patterned roll bonded aluminium radiator", 3rd International conference on CFD, CSIRO, Melborne, Australia, pp. 12-19, 2003.

[14] Hilde Van Der Vyer, JacoDirker and Jousoa P Meyer, "Validation of a CFD model of a three dimensional tube-in-tube heat exchanger", Third International Conference on CFD in the Minerals and Process Industry, CSIRO, Melborne, Australia. pp. 25-32, 2003.

[15] J A Chen, D F Wang and L Z Zheng, "Experimental study of operating performance of a tube-and-fin radiator for vehicles", Proceedings of Institution of Mechanical Engineers, Republic of China, 215: pp. 2-8, 2001.

[16] Changhua Lin and Jeffrey Saunders, "The Effect of Changes in Ambient and Coolant Radiator Inlet Temperatures and Coolant Flowrate on Specific Dissipation", SAE Technical Papers, 2000-01-0579, 2000.

[17] Sridhar Maddipatla, "Coupling of CFD and shape optimization for radiator design", Oakland University. Ph.D. thesis.

[18] J.P.Holman, Heat transfer, Tata-McGraw-Hill Publications, 2001.

[19] Changhua Lin and Jeffrey Saunders, "The Effect of Changes in Ambient and Coolant Radiator Inlet Temperatures and Coolant Flowrate on Specific Dissipation", SAE Technical Papers, 2000-01-0579, 2000.

[20] Seth Daniel Oduro, "Assessing the effect of dirt on the performance of an engine cooling system", Kwame Nkrumah University of Science and Technology, PG thesis, 2009.

[21] Beard, R. A. and Smith, G. J., "A Method of Calculating the Heat Dissipation from Radiators to Cool Vehicle Engines", SAE Technical Paper 710208, 1971.

[22] SalvioChacko, "Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization" Engineering Automation Group, Tata Technologies Limited, Pune, India, Technical paper, 2003.

[23] S.N Sridhara, S.R. Shankapal and V UmeshBabu, "CFD analysis of Fluid Flow & Heat Transfer in a Single Tube-Fin Arrangement of an Automotive Radiator"

International Conference on Mechanical Engineering, Dhaka, Bangladesh, Conference Paper, pp. 16-25, 2005.

[24] Yiding Cao and KhokiatKengskool, "An Automotive Radiator Employing Wickless Heat Pipes" Florida International University, Miami, Conference Paper, 1992.

[25] Krunal Suryaknt Kayastha, Sharvil Dushyant Shah, "Comparative Analysis of Helical tube Automobile Radiator considering different Coolants used CFD" IJAERD, Volume 2,Issue 1, January -2015