CFD Analysis of Heat Transfer Performance of Graphene Oxide Nanofluid in Radiators

Srinivas Sulegao[1] Dr. Pravin V Honguntikar[2]
[1]M.Tech Student [2]Professor& Head of Department
[1,2]Department of Mechanical Engineering
[1,2]PDACEG, Kalaburagi, India

Abstract— For Improved performance of an automobile engine, Cooling systems are one of the critical systems that need attention. With increased capacity to carry away large amounts of wasted heat, performance of an engine is increased. Current research on Nano-fluids suggests that they offer higher heat transfer rate compared to that of conventional coolants. Hence this project seeks to investigate the use of hybrid nanofluids in radiators so as to increase its heat transfer performance. Aluminum Oxide and Graphene Oxide based nanoparticles were selected due to the very highthermal conductivity. System Analysis of the radiator was performed by considering a small part of the whole automobile radiator modelled using Solid works. CFD analysis was conducted using ANSYS FLUENT® for the nanofluid defined and the increase in effectiveness was compared to that of conventional coolants. Usage of such nanofluids for a fixed cooling requirement in the future can lead to significant downsizing of the radiator A Comparison between water Aluminum Oxide and Graphene Oxide is done.

Keywords: CFD, Radiator, Nano Fluid, Heat Transfer Rate, Graphene Oxide

I. Introduction

A. *Overview*

Continuous technological development in automotive industries has increased the demand for high efficiency engines. A high efficiency engine is not only based on its performance but also for better fuel economy and less emission. There are many systems which influence the engine performance like fuel ignition system, emission system, cooling system, etc. one of the parameters which affects the performance of engine is the cooling rate of radiator in engine cooling system. Addition of fins is one of the approaches to increase the heat transfer rate of the radiator. It provides greater heat transfer area and enhances the air convective heat transfer coefficient. However, traditional approach of increasing the cooling rate by using fins has already reached to their limit. As a result, there is a need of new and innovative heat transfer fluids for approving heat transfer rate in an automotive car radiator.

1) *Radiator –*

The radiator is type of heat exchanger. the radiator cools the coolant so that it can be reused. The main component of radiator's cooling system is Air blower, Cooling Fans, Radiator Pressure Caps, Water Pipes, Coolant Hoses, Radiator Parts, Radiators and Water Pumps. The radiator functions like this:

– The water pump pumps the coolant through the system via water pipes,
– The Air blower draws air through the radiator to achieve the cooling action.
– A water cooling system accomplishes the cooling action with the help of water.

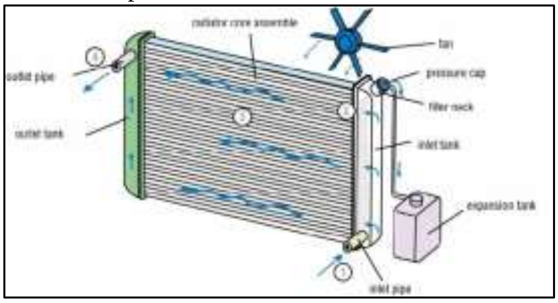


Fig. 1-1: Radiator Components

2) *Engine Coolant –*

Thermal fluid systems have number of applications including automotive cooling system. A coolant is a fluid which flows through engine and prevents it from overheating. Base fluids like water, ethylene glycol, and glycerol are known as conventional coolants and used in automobile radiator for many years. Ideal coolants have low viscosity, high thermal capacity, chemically inert and are low-cost. Further, it should be non-corrosive as well as have corrosion-resistant properties. In diesel engine radiator 50-50 ratio of water and coolant is added for cooling purpose. The most common and readily available coolant is water which has high heat capacity. It is usually used with additives, like corrosion inhibitors and antifreeze. The water-based coolant has to withstand temperatures below 0^0C, or when its boiling point has to be raised. But these fluids have low thermal conductivity which prompted the research to search for the fluids with higher thermal conductivities than conventional coolants. Hence an emerging new class of coolant is Nano fluids. In earlier investigations dispersion of millimeter or micrometer size particles into base fluid causes particle agglomeration and settling and their pumping power is increased. This leads to investigation on Nano Fluid. The concept of Nano Fluid was proposed by Choi. Nano fluid contains nanometer- sized particles, called nanoparticles. These fluids are suspensions of nanoparticles in a base fluid. The nanoparticles of Nano fluids are typically made of metals, carbides, oxides or carbon nanotubes. Common base fluids used are water and ethylene glycol. Nano fluid exhibit enhanced thermal conductivity and the convective heat transfer coefficient compared to the base fluid. They have micro channel cooling without clogging and also reduce the pumping power.

Nanotechnology is one of the emerging areas to be explored and attracting the attention of researchers and academicians round the globe. Nanotechnology has brought in many transformations in the fields of engineering and medicine and has influenced the life style of present day generation. The first half of the 20th century witnessed development of vacuum tube technology and the second half of this century belongs to the semiconductor technology. The 21st century is the era of Nanotechnology. Nanotechnology plays a vital role in the development of Nano robots, direct control of organic molecules, property manipulations of materials at Nano scale dimensions, etc.

The term Nanotechnology is oriented from the Greek word which means dwarf. Nanotechnology involves research and technology development at molecular levels. Materials anything smaller than 100nm in size and with novel properties are known as nanomaterial, according to US National Nanotechnology initiative. The thermos physical, mechanical, metallurgical properties and colorof the material at Nano dimension scale will be different from its bulk material. Nano materials give an opportunity to change its properties and can be used advantageously in many engineering applications.

B. Applications

In the recent years, Nano technology is profoundly influencing many facts of everyday life. Presence of Nano materials are seen in many fields which include food industries and information technology so on and so forth. In the field of energy Nano materials are used in solar cells, in fuel cells and for storage of hydrogen energy. Nano-coated surfaces are used to achieve higher transmissivity which increases the performance of solar collectors. Nanomaterial can also be used in medical diagnosis and drug delivery systems. Scratch free surfaces with wear and corrosion resistant properties are made by employing nanoparticles. The nanomaterial also used in the manufacture of Nano-composites for light weight structures in aerospace applications are manufactured using metallic powders in Nano size. Now days, Nano materials are manufactured using metallic powders in Nano size. Now days, Nano materials are being widely used in the manufacture of new generation sporting goods, computer chips, cosmetics and sun creams.

C. Nano fluids as Heat Transfer Liquids

Heat transfer is an important area of study in thermal engineering. Selection of a suitable heat transfer fluid for heat dissipation is an important consideration in the thermal design of heat exchangers. Heat transfer fluid is one of the critical parameters which affect the cost and size of heat exchangers systems. Conventional fluids like water and oils have limited heat transfer capabilities. The need for development of new kinds of fluids with improved heat transfer capabilities is being felt by different research groups across the world. The advances in Nanotechnology have made it possible to manufacture metal and metal oxides particles on Nano dimensional scale. Nano particles are considered to be new generation material having potential applications in the heat transfer area.

Any host liquid which contains nanoparticles in suspended state is known as Nano fluid. Nano fluids are two phase fluids of solid-liquid mixture and are considered to be new generation heat transfer fluids. In the recent past Nano fluids have emerged as promising thermo-fluids for heat transfer applications. The thermal conductivity property of Nano fluids is expected to be higher than that of the base liquids. The practice of adding micron size particles in the traditional heat transfer fluids was in existence since the time of Hamilton and Crosser (1962). But two phase suspensions micro particles produce sedimentation obstruct smooth fluid flow due to channel clogging and cause erosion on the tube material.

The Nano fluids on the other hand offer many advantages over the single phase pure fluids and suspensions with micro particle. The problems of the particle sedimentation due to gravity, clogging of micro channel passage, and erosion of tube material are minimized to a great extent when Nano fluids are used in heat exchangers. Besides, Nano fluids from stable suspensions with uniform dispersion of Nano particles in the host fluid. Thermo-physical properties of single phase heat transfer fluids such as water, oils and glycols are well established and are available in literature and hand books. The thermos-physical properties of two phase Nano fluids are not explored much. An accurate measurement of properties of Nano fluids is a prerequisite for determination of heat transfer coefficient of Nano fluids. The suitability of a particular Nano fluid in heat transfer application is then judged based on its heat transfer performance. Nano fluids are considered to be an alternate and new generation liquids for transport of heat energy and can be employed as heat transfer fluids in heat exchangers in place of pure single phase fluids. The applications of Nano fluid heat transfer include radiators in automotive, chemical engineering and process industries, solar water heater, refrigeration, cooling of electronics devices. The main objective of obtaining heat transfer enhancement using Nano fluids is to accommodate high heat fluxes and hence to reduce the cost and size of the heat exchangers which in turn results in conservation of energy and material.

D. Problem Statement

- Heat transfer fluids used in radiators have inherently low thermal conductivity and redesigning heat exchangers to increase radiators effectiveness limits.
- Nano-fluids can be employed to system requiring rapid heating and cooling. Due to Nano-size the powder is considered as the integral fluid.
- Lot of changes occurs in the properties of working fluid in radiator due to inserting Nanoparticles. Heat transfer increases due to large surface area of Nano particles in Nano fluids.
- Due to limitations of conventional coolants a specific need arises of using more effective coolants having high end cooling properties which increases efficiency of cooling and reduces the size of radiators.

E. Thesis Objective

For this particular topic related to enhancing heat transfer using Nano fluid lots of work has been done previously. Though this concept is not new but the work done on this subject is related with considering the factors which affects Heat transfer rate, like by changing volume concentration or by changing flow rate, at varied load condition on engine, change in inlet temperature of radiator, and comparing heat transfer rate enhancement by using different Nano fluids.

While studying this different research paper it has been observed that clogging of nanoparticles occur when volume concentration of nanoparticles increases in base fluid beyond certain limit. Also nanoparticles show tendency of settling down after some period.

Objective of this thesis work is not only to enhance heat transfer rate of radiator by using Al_2O_3, as Nano fluid but also try to avoid settling and clogging problem of nanoparticles when concentration of nanoparticles increased in base fluid. A novel methodology is suggested in this thesis work which helps to avoid settling and clogging problem as well as improve heat transfer rate.

II. LITERATURE REVIEW

A. *Overview*

Most of the literature studies available on enhancement in heat transfer rate have been carried out Analyticalally, by using different types of Nano fluids like CuO, Al_2O_3,SiO_2 and TiO_2 etc. with water or ethylene glycol as base fluid. Enhancement of heat transfer rate has been studied under different conditions like by varying load on engine, at different volume concentration of Nano fluid, for different types of flow like laminar or turbulent, for different particle size of Nano fluid. The results were analyzed using CFD.A numerical study presented by Vajjha et al. on laminar heat transfer using CuO and , Al_2O_3 – ethylene glycol and water inside a flat tube of a car radiator.

Naraki ET AL[7] investigated overall heat transfer coefficient Analyticalally for laminar regime. The results showed that overall heat transfer increases with enhancement in the Nano fluid concentration from 0 to 0.4%.Adnan M Husse, R.A.Bakar, K.Kadirgamap[1] Studied "forced convection Nano fluid heat transfer in the automotive cooling system. This article includes the friction factor and forced convection heat transfer of SiO_2 nanoparticle dispersed in water as a base fluid conducted in a car radiator Analyticalally and numerically. Four different concentrations of Nano fluid in the range of 1-2.5 vol% have been used. The flowrate changed in the range of 2-8 LPM. The result showed that the friction factor decreases with an increase in flowrate andincreases with increase in volume concentration. Nusselt number increases with increase in flow rate, volume concentration of Nano fluid and inlet temperature. The result revealed that application of SiO_2 Nano fluid with low concentration enhance heat transfer rate up to 50% as comparison with pure water. The simulation results with Analyticalal data shows good agreement.

Jaafer Albadr, Satinder Tayal, Mushtaq Alasadi[2] in "Elsevier" – case studies in thermal engineering studied "Heat transfer through heat exchanger using Al_2O_3 Nanofluid at different concentrations. This article reports an Analyticalal study on the forced convective heat transfer and flow characteristics of a Nano fluid consisting of water and different volume concentration of Al_2O_3 flowing in a horizontal shell and tube heat exchanger. The Al_2O_3 nanoparticles of about 30nm diameter are used in study. The results shows increase in heat transfer coefficient with increase in volume concentration as well as mass flow rate but at the same time friction factor also increases.

"B.Kirubadurai[4]"et al:in Interantional Journal of Research in Engineering and Technology (eISSN: 2319-1163 | pISSN: 2321-7308). In his paper shows varying factor affecting the thermal conductivity of Nano fluid at different conditions. They tried to increase the heat transfer rate by considering thermal conductivity Nano fluid. Thermal conductivity is increase with increasing concentration of metal particle within critical limit. They found out that thermal conductivity is affected by various parameters like shape, size, clustering, collision, melting point of nanoparticles etc. controlling these parameters to increase the thermal conductivity of Nano fluid.

"S.M. Peyghambarzadeh, S.H.Hashemabadi, M. Seifi Jamnani, S.M. Hoseini"[6]"Improving the cooling performance of automobile radiator with Al_2O_3/water Nano fluid". Applied Thermal Engineering 31 (2011) 1833e1838. In this paper, forced convective heat transfer in a water based Nano fluid has Analyticalally been compared to that of pure water in an automobile radiator. Five different concentrations of Nano fluids in the range of 0.1el vol% have been prepared by the addition of Al_2O_3 nano particles into the water. Results conclude that increasing the fluid circulating rate can improve the heat transfer performance while the fluid inlet temperature to the radiator has trivial effects. Meanwhile, application of Nano fluid with low concentrations can transfer efficiency up to 45% in comparison with pure water .

"Sidi El Becaye Maiga, Samy Joseph Palm, Cong Tam Nguyen, Gilles Roy, Nicolas Galanis"[5]"Heat transfer enhamcement by using Nano fluids in forced convection flows" They investigated thoroughly the laminar convective heat transfer for the two Nano fluids, as water-$Al_2O_3$and Ethylene Glycol-Al_2O_3.The presence of such particles induced drastic effect on the wall shear that increase appreciably with the particle loading. Among the two the mixture of Ethylene Glycol and Al_2O_3 has better heat transfer enhancement than that of water. The results have shown that, in general the heat transfer enhancement also increases with the augmentation of the flow Reynolds number.

Somchaiwongwiseset al[11]investigated heat transfer enhancement and flow characteristic of Al_2O_3-Water Nano fluid using micro channel heat sink. The test section dimension was 5x5mm and 50W heat was applied. Heat transfer enhanced at high Reynold number and high concentration of Nano fluid, because at high Reynolds number wall temperature decreases and pressure drop was increased.

Shuichi Torri[9] Investigated convective heat transfer coefficient of diamond based Nano fluid by using heat tube apparatus,. Tube Specification is 4.3mm outer and 4mm inner diameter and 100W power applied uniformly. The heat transfer coefficientincreases with increasing concentration and Reynolds number of Nano fluid, but at the same time it increases the pressure drop with increasing concentration of Nano particle.

Anil Kumar et al[10] studied the heat transfer enhancement of fin; utilizing Al_2O_3-Water Nanofluid analyzed using CFD. Rayleigh number increases due to Brownian motion, ballistic phonon transport, clustering and dispersion effect of nanoparticle. At high Rayleigh number flow rate at Centre of the circulation is increasing, so temperature is drop from center of fin. Volume of the circulation increases the velocity at center is increases as the result of increasing the solid-fluid heat transportation. Low aspect ratio fin is suitable for heat transfer enhancement, because heat affected zone is less.

Mohamed hadi et al: In this paper enhancement of heat transfer rate by considering clustering effect of nanoparticle has been studied. The heat transfer rate is increases with increasing the concentration of nanoparticle, but due to high concentration clustering is occur. Clustering increased the heat transfer rate of Nano fluid at certain contact time of particle, but cluster causes the problem of agglomeration. Agglomeration can reduce to use micro sized particle.

"Ravi Adwani"[3]et al. in international journal of Innovative science, Engineering & Technology,Vol. I, Issue 6, Aug-2014 (ISSN2348-7968)Analyticalally investigated heat transfer rate in Automobile Radiator usingNano Fluid Author had made a research by adding more thermally conductive solids into liquids. The suspension of nanoparticles in a base fluid which is termed as "Nano fluid", exhibits a substantially higher thermal conductivity than base fluids. By using different proportions of Al_2O_3 by weight have been added to conventional fluid and based on that the enhancement in heat transfer has been found out. They came to conclusion that the heat transfer rate in automobile radiator increases by using nanoparticles of Al_2O_3 in volume fraction in base fluid like water for particular load and constant flow rate of coolant. It is observed that the heat transfer rate increases with increase in volume fraction of Al_2O_3 in water at constant flow rate. When volume fraction increases beyond 6% the nanoparticles were settled at the bottom.

Sadollah Ebrahimi et. Al studied about application of Nanofluid as coolants. Today more than ever ultrahigh-performance cooling plays an important role in the development of energy-efficient heat transfer fluids which are required in many industries and commercial applications. However, conventional coolants are inherently poor heat transfer fluids. Nano fluid a term coined by Choi in 1995 is a new class of heat transfer fluids which is developed by suspending nanoparticles such as small amounts of metal, non-metal or nanotubes in the fluids. The goal of Nano fluids is to achieve the highest possible thermal properties at the smallest possible concentrations (preferably <1% by volume) by uniform dispersion and stable suspension of nanoparticles (preferably <10nm)in host fluids. We have divided this chapter to four sections. Section 1 has focused on the two methods of synthesizing Nano fluids and different methods for dispersing spherical and cylindrical nanoparticles such as Ag, Cu in a host fluid and also the common methods for measuring the thermal conductivity of Nano fluids respect to pure fluids to explain the effective thermal conductivity of Nano fluids.

Bhagat UK et. Al did a work on Heat Transfer Applications of zinc oxide Nano fluids. This paper describes preparation of zinc oxide (ZnO) based Nano fluids in polymer matrix. The rheological properties of Nano fluid were studied and were studied and were applied in heat transfer application. Heat transfer application of aqueous based ZnO Nano fluid was tested and it was observed that, the presence of ZnO Nano fluid effectively reduces the temperature propagation ina sono-chemically heated system. It is observed that the heat absorption capacity was increased by about 30p40% for the ZnO containing Nano fluid. For the preparation of Nano fluids, as synthesized ZnO nanoparticles were utilized after characterization by various modern tools such as UV-visible, Raman spectroscopy, XRD, SEM, Particle size analysis, and TGA studies. The average particle size of as prepared ZnO nanoparticle was in the range of 19 to 30 nm and XRD analysis revealed hexagonal crystal structure. Nano fluids (NFs) are new class of fluids comprising of base fluid i.e. water, ethylene glycols, oils, bio-fluids, polymer solutions or other common fluids with nanoparticles (1-100 nm) suspended in them.

R J Bhatt et. Al did work on the thermal abilities of Nanofluids as coolants. Today, the demand of automobile vehicles is on peak. So, it is a great challenge for automotive industries to provide an efficient and economical engine. The performance of an engine affects by various systems like fuel supply system, lubrication system, transmission system, cooling system etc. So, it becomes essential to account them while designing an engine for improves the engines performance. Cooling system is one of the important systems amongst all. It is responsible to carry large amount of heat waste to surroundings for efficient working of an engine. It enhances heat transfer and fuel economy which leads to maximize the performance of an engine. Most internal combustion engines are fluid cooled using either air or a liquid coolant run through a heat exchanger (radiator) cooled by air. The heat transfer through radiator can be improved by maximizing the heat transfer area and increasing the heat transfer coefficient. The heat transfer coefficient can be increased by using more efficient heat transfer methods or by improving the thermos-physical properties of the heat transfer material i.e. coolant.

As per cited papers it is observed that if the concentration of fluid increases beyond certain volume fraction heat transfer rate increases, however it becomes difficult to prevent settling of Nanoparticles at such high concentration. In this paper we are trying to reduce the settling problem of Nano particles (Al_2O_3) by providing stirring mechanism. Further details are explained in following article by providing adequate data.

B. Proposed Work

The Literature review above clearly shows that using Nano fluid can effectively improve the heat convective performance, but will also increase the pipeline pressure drop and pumping energy, clogging and particle settlement problem. Using Nano fluid with a high heat convective performance for heat exchange can help reduce the volume of the heat exchanger (radiator). In addition, using Nano fluid with higher heat transfer performance instead of the traditional working fluid for cooling can reduce the demand and cost of cooling system modifications. Most of the Nano fluids used in previous studies were used in single pipe heat transfer, micro channel heat sinks, plate heat exchangers, double-tube heat exchangers, or heated enclosures, and seldom used in air-cooled heat exchangers. Since the ultimate goal of radiators is to discharge heat into the atmosphere, and the air-cooled heat exchanger is widely used in automotive, air conditioning, proton exchange membrane fuel cell (PEMFC) and electronic chip cooling, and is therefore a worthy research direction.

This study uses a two-step synthesis method to make Al_2O_3/water Nano fluid, which can be used as coolant in diesel engine radiator to heat dissipation. Since the temperature in diesel engine is higher, by using Nano fluid with base fluid can increase heat transfer there by reducing size of diesel engine radiator. Identifying the differences in Nano fluid weight fractions, Mass flow rates, and temperature effects on heat exchange performance and pressure drop of the air –cooled heat exchanger makes it possible to evaluate the feasibility of applying Al_2O_3/water Nano fluid to PEMFC heat dissipation or electronic chip cooling in the future. Further the clogging and particle settlement problem which occurs beyond certain limit of volume concentration of nanoparticles can be overcome by stirring methodology suggested in this thesis.

III. COMPUTATIONAL METHODOLOGY

A. *Principle of Working*

According to Oleg Zikanov CFD can be defined as: “CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer.” According to this definition, CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (heat and mass transfer). In retrospect, it is integrating not only the disciplines of fluid mechanics with mathematics but also with computer science as illustrated in Figure 2.1. The physical characteristics of the fluid motion can usually be described through fundamental mathematical equations, usually in partial differential form, which govern a process of interest and are often called governing equations in CFD. JiyuanTu, Guan HengYeoh and Chaoqun Liu have discussed how to solve mathematical equations with using CFD:

“In order to solve mathematical equations, computer scientists convert them by using high-level computer programming languages into computer programs or software packages. The computational part simply means the study of the fluid flow through numerical simulations, which involves employing computer programs or software packages performed on high-speed digital computers to attain the numerical solutions. Another question arises "Do we actually require the expertise of three specific people from each discipline -fluids engineering, mathematics, and computer science- to come together for the development of CFD programs or even to conduct CFD simulations?” The answer is obviously no, and more likely it is expected that this field demands a person who will proficiently obtain some subsets of the knowledge from each discipline.”

B. *Modelling Equations*

1) *Governing Equations*

Simulations of fluids are based on the Navier Stokes Equations. In principle the Navier Stokes Equations (NSE) are only the momentum equations, but often when one refers to the NSE also the continuity and energy equations are meant. In conservative differential form the continuity, momentum and energy equations are respectively.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{u} = 0 ,$$

$$\frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p + \nabla \cdot (\mu \nabla \vec{u}) + \vec{f}$$

and

$$\frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho \vec{u} e) = -\nabla p \vec{u} + \nabla \cdot (\mu \vec{u} \nabla \vec{u}) + \nabla q .$$

These five equations contain seven unknowns; three velocity components of ~u, pressure p, density ρ, internal energy e and viscosity μ. When the flow is assumed to be an ideal gas, the energyequation can be rewritten so that the internal energy can be replaced by the temperature T. Since both the density and viscosity are a function of the temperature, it turns out that only five unknowns are left. Moreover it is expected that the flow is isothermal and thus the internal energy will remain constant over time. Next to these external forces such as body and gravity forces are absent as well. This leads to a simplified version of the NSE in which the energy equation candiscarded. Now the NSE for continuity and momentum read,

$$\nabla \cdot \vec{u} = 0$$

and

$$\frac{\partial \vec{u}}{\partial t} + \nabla \cdot (\vec{u} \vec{u}) = -\frac{\nabla p}{\rho} + \nabla \cdot (\nu \nabla \vec{u}) ,$$

In which the last term of the above Equation represents the viscous stress ∇τ. The above presented set of equations allows the flow to experience turbulent behaviour. Hence the flow contains turbulent kinetic energy (E) that is distributed over Eddies with varying sizes. When all the turbulence would be resolved, the mesh should be fine enough to capture even the smallest eddies. This technique is called Direct Numerical Simulation (DNS). However with the current available hardware this is only possible with very simple geometries and low Reynolds number flows. In order to get a clearer view of which Eddies are resolved using various methods, a logarithmic plot of this energy with respect to the reciprocal of the Eddy size, the wavenumber (K), is given in Figure 3.1.

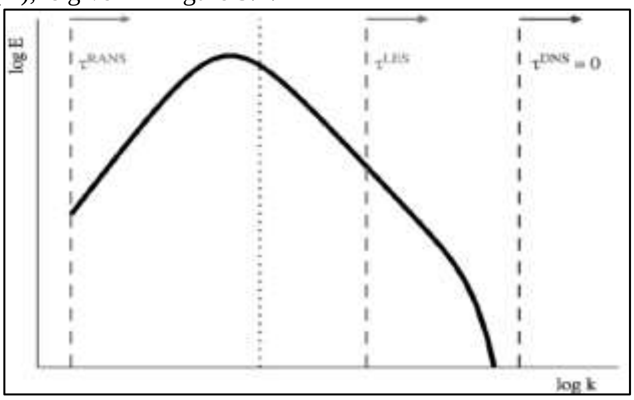


Fig. 3.1: The turbulent kinetic energy distributed over Eddies of different sizes.

From this figure it can be seen that DNS resolves all the turbulent kinetic length scales and thus all the turbulence. On the other hand RANS simulations model all the turbulence and thus none of the Eddies are resolved. It can be seen that most of the kinetic energy is located in the large Eddies. LES is based on this fact and resolves the large scales while the small scales are approximated using mathematical models. In some regions of a flow field, like the boundary layer, the large Eddies become very small as well, requiring very small cell sizes. By applying hybrid LES/RANS approaches the total number of cells required can be kept lower by using RANS in those regions. For the current study, the aerodynamics of road vehicles is considered. Since high Reynolds numbers are present and complex flow phenomena occur, DNS is not an option solving for the flow. LES is also discarded as an option due to its high computational costs. Since RANS uses the least computational resources while still providing sufficiently accurate results, the main focus lies on this technique. The downside of this technique originates from the fact that the highly vertical motion occurring in bluff body flows is only superficially resolved. It is expected that a hybrid RANS/LES method will provide more accurate results. Therefore this technique will be used to resolve the flow field around the vehicles as well. The next two sections go into more detail about both methods.

2) Reynolds Averaged Navier Stokes simulations

In 1984 Reynolds derived a statistical approach to approximate the NSE. By time averaging the equations, only the mean properties of the flow remain. According to Reynolds, for most turbulent flows only these averages characteristics are of interest. The quantities present in the NSE are split into a mean and a fluctuating component. For the velocity this yields in tensor notation the following,

$$u_i(x_i,t) = \overline{u}_i(x_i) + u_i'(x_i,t) \ ,$$

for which holds that,

$$\overline{u_i'}(x_i,t) = 0 \ .$$

Replacing all time varying quantities in the simplified NSE, as derived in the previous section, with this definition and rearranging the terms yields for the continuity and momentum equation in tensor notation,

$$\frac{\delta \overline{u}_j}{\delta x_j} = 0$$

$$\frac{\delta \overline{u}_i}{\delta t} + \overline{u}_j \frac{\delta(\rho \overline{u}_i)}{\delta x_j} = -\frac{1}{\rho}\frac{\delta \overline{p}}{\delta x_i} + \frac{1}{\rho}\frac{\delta}{\delta x_j}\left(2\mu S_{ij} - \frac{2}{3}\mu \frac{\delta u_l}{\delta x_l}\delta_{ij}\right) + \frac{\delta}{\delta x_j}\left(-\rho \overline{u'_i u'_j}\right) + S_i \ .$$

From the equations above it can be seen that for three-dimensional flow, four equations and ten unknowns are present. Besides three unknown velocity components and the pressure, the later equation contains an additional six unknowns. These are included in the Reynolds stress tensor,

$$\tau_{ij} = -\overline{u'_i u'_j} \ .$$

The Reynolds stress tensor has to be modelled in order to solve the RANS equations, which is also known as turbulence modelling. The quality of solution depends largely on the quality of the applied turbulence model.

3) Hybrid LES/RANS methods

As stated before, a hybrid LES/RANS method simulates the flow using a LES technique in the regions where the mesh density allows for large Eddies to be represented on the grid correctly. Regions where the size of the large Eddies becomes so small that they cannot be resolved correctly, the method switches to a RANS technique. Before combining both techniques it is interesting to look at LES simulations separately. After that the hybrid method itself is elaborated on.

4) LES

Smagorinsky (1963) came up with the basic principles of Large Eddy Simulations (LES). From the theory of Kolmogorov, the larger scale Eddies are mainly responsible for the turbulent kinetic energy in the flow. This can clearly be seen in Figure 3.2, which is a more detailed version of Figure 3.1. Three distinct regions can be observed:

– Energetic scales where the large Eddies are present. This region, as stated before, contains by far the largest share of turbulent kinetic energy.
– Inertial sub-range where medium sized Eddies is present. These scales are independent of the forcing scales and obey Kolmogorov's law,

$$E(k) = C_k \epsilon^{\frac{2}{3}} k^{-\frac{5}{3}}$$

– These eddies contain only a very small amount of turbulent kinetic energy and are almost not dissipative. Their main function is to transfer energy from the larger to the smaller scales.
– Dissipative scales where the smallest Eddies are present. In here viscous effects strongly damp the motion and hence cause the Eddies to be highly dissipative.

Together with the inertial sub-range, the dissipative range forms the universal equilibrium range which is free of any forcing. Without forcing, the behaviour of the eddies within the range is in general similar for all type of flows and hence this range can be modelled easily.

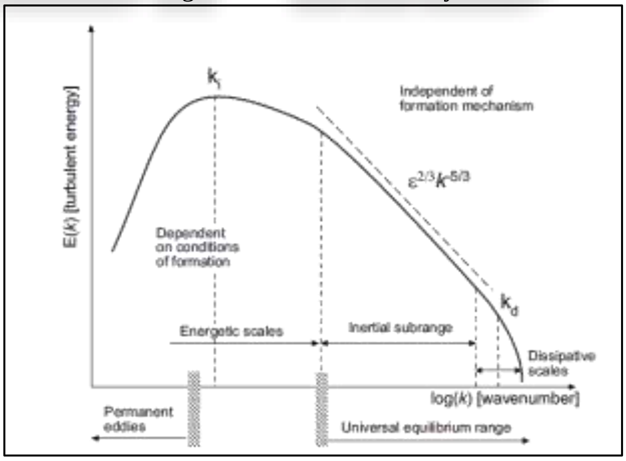


Fig. 3.2: The turbulent kinetic energy distributed over Eddies of different sizes.

The cut-off wavenumber K_c would represent the limiting wavenumber up until which the Eddies are still resolved. Its location would be slightly before the universal equilibrium range starts.

In order to separate the larger from the smaller scales a LES filter is applied. The filter width Δ is important in this context. Eddies with sizes larger than the width should be resolved and Eddies smaller should be modelled. Eddies with length scales below this width are also referred to as Sub Grid Scales (SGS). In fact, filter width Δ is the practical application of the cut-off wavenumber K_c.The contribution of the resolved scales and sub grid scales can be written as,

$$\bar{u}_i = u_i - u_i' \ ,$$

Where $\overline{u}_i$are the resolved scales and u_i' are the sub grid scales. The filtering process with filter function $G(\chi,\chi',\Delta)$ is formulated as,

$$u_i = \int u_i(x')\, G(x, x', \Delta)\, dx' \ .$$

The software package employed for this study uses an implicit top hat filter. This filter is define as

$$G(x, \Delta) = \begin{cases} \frac{1}{\Delta}, & \text{if } |x'| \leqslant \frac{\Delta}{2} \\ 0, & \text{otherwise} \end{cases} \ .$$

In the filtered Navier-Stokes equations, the transported variable is not just the velocity, but is the large scale filtered velocity, $\overline{u}_i$. The filtered NSE are as follows,

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \ ,$$

$$\frac{D\overline{u_i}}{Dt} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + -\frac{1}{\rho}\frac{\partial \tau_{ij}^R}{\partial x_j} + \nu \nabla^2 \overline{u_i} \ ,$$

Where the subgrid-stress tensor is equal to,

$$\tau_{ij}^R = \rho\left(\overline{u_i u_j} - \overline{u}_i \overline{u}_j\right) \ .$$

In order to define the unresolved scales, the Eddy viscosity model is applied,

$$\tau_{ij}^R = 2\rho\nu_t \overline{S_{ij}} - \frac{1}{3}\delta_{ij}\tau_{kk}^R \ ,$$

Where $\nu_t$is the kinetic energy viscosity which needs a turbulence model to close it.

5) Combining LES and RANS

Various methods exist in order to combine LES and RANS. An excellent review of the available methods is published by Fröhlich and von Terzi (2008). They devised a classification system that can be seen. Combining LES and RANS can be done in a segregated or in a unified manner.

When using a unified coupling of LES/RANS, it means that the same transport equation is used for the velocity. Every term in this equation is similar in LES and RANS, except for $\mathcal{T}_{ij}$. This term varies depending on how the flow should be solved in a specific region of the flow field. It can be modelled with a blending function,

$$\tau_{ij}^{model} = f^{RANS}\tau_{ij}^{RANS} + f^{LES}\tau_{ij}^{LES} \ ,$$

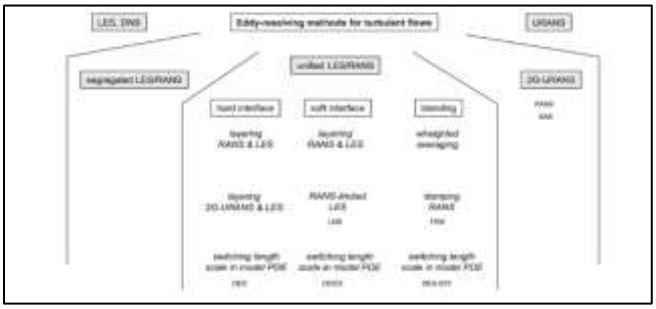


Fig. 3.3: A classification of the hybrid LES/RANS methods available.

Where f represents local blending coefficients that depend on the local value of a given criterion. Another way is to create an interface, which means that one part of the flow is computed purely with LES while the other part is computed purely with RANS. When a hard interface is used the regions are user defined and do not vary in time. On the other hand, when a soft interface is used, the regions are separated based on the actual values inside a cell and will thus vary in time.

The fact that many LES models are inspired on RANS models, simplifies the implementation of this technique. While for RANS models τ_{ij}^{model} depends on physical quantities describing the entire turbulence, $\tau_{ij}^{model}$for LES models depends on grid parameters such as step size. The step size of the grid determines which Eddy size can still be resolved and which have to be modelled. Often $\tau_{ij}^{LES}$is called the subgrid-scale model, of which the Smagorinsky model is one of the most well-known.

The other way of combining LES and RANS is by using segregated coupling. One region of the flow is resolved using LES, while the other uses RANS. A difference to the unified interface technique is that the flow quantities are not any more continuous throughout the interface. The computations are almost standalone and then coupled via boundary conditions. This method is unfavourable for most simulations.

6) Turbulence Models

The Reynolds stress tensor, τ_{ij}, is often approximated by the Boussinesq hypothesis in order to link the Reynolds stresses with the mean velocity gradients Nguyen (2005). This approximation is formulated as follows,

$$\tau_{ij} = 2\nu_t S_{ij} - \frac{2}{3}k\delta_{ij} \ ,$$

Where $S_{ij}$is the mean strain-rate tensor,

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \ .$$

In this equation both the kinetic Eddy viscosity, $\upsilon_t$and the turbulent kinetic energy, k, are unknown. Turbulence models provide a means to compute these extra quantities by introducing extra equations. The models can be classified into four different types; algebraic models, one equation models, two equation models and second order closure models. Thereof, according to Wilcox (1994), for RANS simulations the two equation models are the most popular. On the other hand, when a DES simulation is performed, often a one equation model is employed. The term one or two equation turbulence modelimplies that one or two extra transport variables are used for the formulation that are related to υ_t and k.

For RANS simulations the best choice is to use the $k-\epsilon$or the $k-\omega$ turbulence model. These models solve two extra transport equations. One for k and one for ϵ or ω. These extra variables relate to υ_t for the $k-\epsilon$model as,

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \ .$$

And for the $k-\omega$as,

$$\nu_t = \frac{k}{\omega} .$$

When running a DES simulation often the SpalartAllmaras turbulence model is applied. Then the second term of the Boussinesq approximation is left out,

$$\tau_{ij} = 2\nu_t S_{ij} .$$

And υ_t is solved directly using only one transport equation. All three turbulence models will be discussed in the following three subsections respectively.

7) $k-\epsilon$ turbulence model

The most famous model of the two is the $k-\epsilon$model of which the standard is set by Jones and Launder (1972). In this formulation the second transport variable used is the turbulent kinetic energy dissipation ϵ. In literature the method is often referred to as the Standard $k-\epsilon$ model. When applying the Standard $k-\epsilon$model, usually the closure coefficients published in Launder and Sharma (1974) are used. A disadvantage of this method is that it cannot cope very well with large adverse pressure gradients.

There exist two well-known improvements on this first formulation; the Re-Normalization Group (RNG) $k-\epsilon$ model of Yakhot et al. (1992) and the Realizable $k-\epsilon$model by Shih et al. (1995). RNG appears to perform better in predicting turbulence of swirling flow in cavities. Moreover it is said to be more effective for low Reynold number flows (Fluent Inc., 2006). As the name suggests it is based on a mathematical technique named the renormalization group method.

The Realizable variant of the $k-\epsilon$model is most often applied since it performs better in simulating flows involving rotation, boundary layers with large adverse pressure gradients, separation and recirculation (Fluent Inc., 2006). The method is called realizable since it satisfies certain mathematical constraints on the Reynolds stresses. This is in correspondence with the physical behaviour of turbulent flows. For the current study this model is employed.

Based on the work of Cable (2008), the formulation of the Realizable $k-\epsilon$model is present below. The first transport equation deals with the turbulent kinetic energy k and is formulated as,

$$\frac{D\rho k}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k ,$$

where $G_k$and G_b are respectively the generation of turbulent kinetic energy due to mean velocity gradients and buoyancy, Y_M the fluctuating dilation in compressible turbulence contributing to the overall dissipation rate and S_k is a user defined source term.

The termsG_k, G_b and Y_M are defined as,

$$G_k = -\rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i} , \qquad G_b = \beta g_i \frac{\mu_t}{\mathrm{Pr}_t}\frac{\partial T}{\partial x_i} , \qquad Y_M = 2\rho\varepsilon M_t^2 ,$$

Where,

$$\beta = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_p .$$

The second transport equation deals with the turbulent kinetic energy dissipation ϵ and is formulated as,

$$\frac{D\rho\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon - \rho C_2\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon ,$$

Where $S_\epsilon$is again a user defined source term.

The turbulent viscosity and its constant are defined as,

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} , \qquad C_\mu = \frac{1}{A_0 + A_S\frac{kU^*}{\varepsilon}} ,$$

Where,

$$U^* = \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}} , \qquad \tilde{\Omega}_{ij} = \bar{\Omega}_{ij} - \varepsilon_{ijk}\omega_k - 2\varepsilon_{ijk}\omega_k .$$

Note that $\overline{\Omega}_{ij}$ is the mean rate of rotation tensor with respect to a moving reference frame having an angular velocity ω_k. The constant A_S is defined as,

$$A_S = \sqrt{6}\cos\phi , \qquad \phi = \frac{1}{3}\cos^{-1}\left(\sqrt{6}\frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}\right) ,$$

Where,

$$\tilde{S} = \sqrt{S_{ij}S_{ij}} .$$

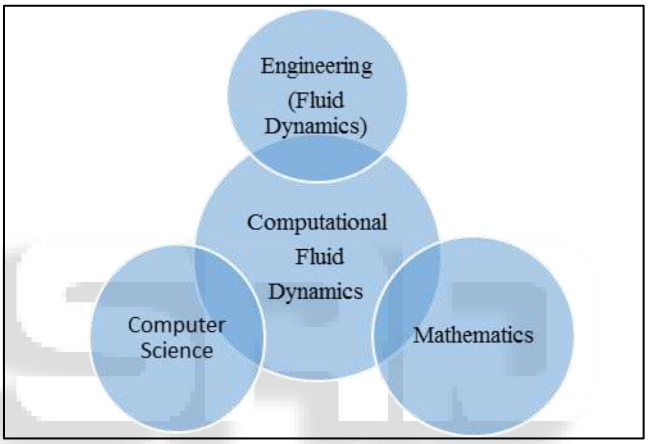


Fig. 3.4: The different disciplines contained within computational fluid dynamics

CFD has also become one of the three basic methods or approaches that can be employed to solve problems in fluid dynamics and heat transfer. As demonstrated in Figure 3.4, each approach is strongly interlinked and does not lie in isolation.

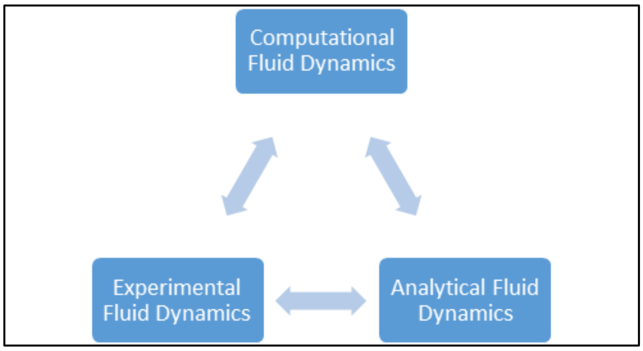


Fig. 3.5: The three basic approaches to solve problems in fluid dynamics and heat transfer.

C. Advantages of Computational Fluid Dynamics

With the rapid advancement of digital computers, CFD is poised to remain at the forefront of cutting edge research in the sciences of fluid dynamics and heat transfer. Also, the emergence of CFD as a practical tool in modern engineering practice is steadily attracting much interest and appeal.

There are many advantages in considering CFD. The theoretical development of the computational sciences focuses on the construction and solution of the governing equations and the study of various approximations to these equations. CFD complements experimental and analytical approaches by providing an alternative cost-effective means of simulating real fluid flows. Particularly, CFD substantially reduces lead times and costs in designs and production compared to experimental based approach and offers the ability to solve a range of complicated flow problems where the analytical approach is lacking. CFD has the capacity of simulating flow conditions that are not reproducible in experimental tests found in geophysical and biological fluid dynamics, such as nuclear accident scenarios or scenarios that are too huge or too remote to be simulated experimentally (e.g., Indonesian Tsunami of 2004). Furthermore, CFD can provide rather detailed, visualized, and comprehensive information when compared to analytical and experimental fluid dynamics. Although CFD is advantageous, it cannot easily replace experimental testing as a method to gather information for design purposes. Despite its many advantages, the researcher must consider the inherent limitations of applying CFD. Numerical errors occur during computations; therefore, there will be differences between the computed results and reality.

D. *Numerical Method*

CFD codes are structured around the numerical algorithms that can handle fluid flow problems. All the CFD commercial packages available in the market have three basic elements, which divide the complete analysis of the numerical experiment to be performed on the specific domain or geometry. The three basic elements are

1) Pre-processor
2) Solver
3) Post-Process

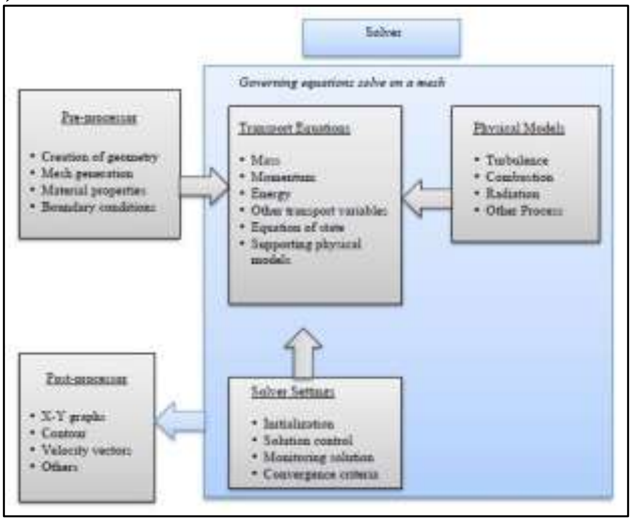


Fig. 3.6: The inter-connectivity functions of the three main elements within a CFD analysis framework.

1) *Pre-Processor*

Pre-processor consists of input of a flow problem by means of a user-friendly interface and subsequent transformation of this input into form of suitable for the use by the solver. The pre-processor is the link between the user and the solver.

2) *Creation of Geometry*

This process involves several computer aided design (CAD) software like CATIA, Solidworks, Pro-Eand many more. The help of CAD software defines the topology of the fluid flow region of interest. This software plays a major part of the design and optimization process in research analysis.

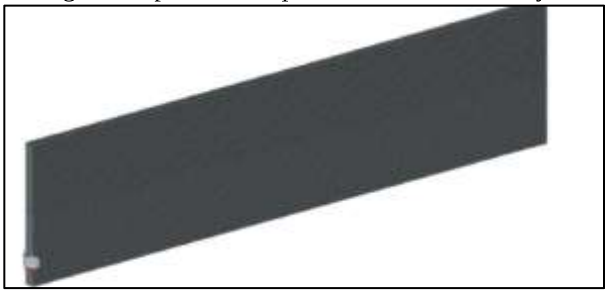


Fig. 3.7 (a): Full Scale Model of the Radiator Fins

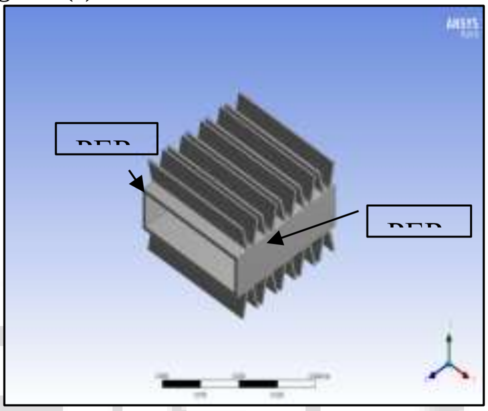


Fig. 3.7(b): Scaled Geometry imported for CFD simulation

The Above Fig represents the part of the Radiator which is considered for simulation as u can see all sides of the Walls are periodic that means the reduced Element will give the exact output.

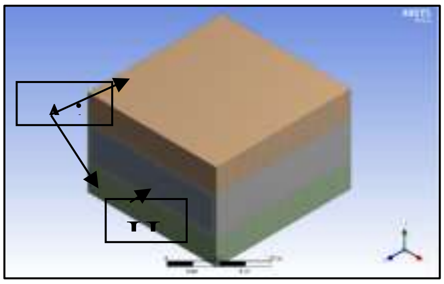


Fig. 3.8: Fluid Volumes Generated for CFD Simulation

3) *Mesh Generation*

Mesh generation constitutes one of the most important steps during the pre-process stage after the definition of the domain geometry. CFD requires the subdivision of the domain into a number of smaller, non-overlapping subdomains in order to solve the flow physics within the domain geometry that has been created; this results in the generation of a mesh (or grid) of cells (elements or control volumes) overlaying the whole domain geometry. The essential fluid flows that are described in each of these cells are usually solved numerically so that the discrete values of the flow properties such as the velocity, pressure, temperature, and other transport parameters of interest are determined. This yields the CFD solution to the flow problem that is being solved. The accuracy of a CFD solution is governed by the number of cells in the mesh within the computational domain. In general, the provision of a large number of cells leads to the attainment of an accurate solution. However, the accuracy of a solution is strongly dependent on the imposed limitations dominated by the computational costs and calculation turnover times.

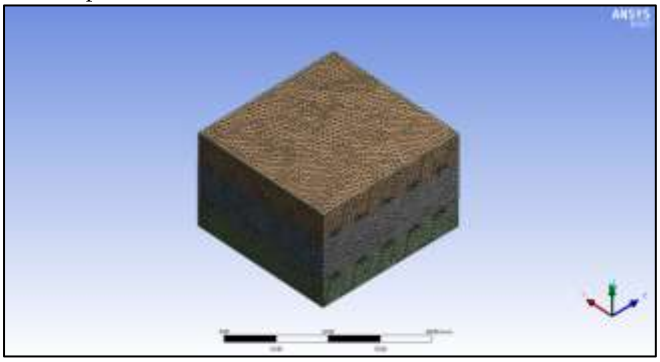


Fig. 3.9: Mesh Generated

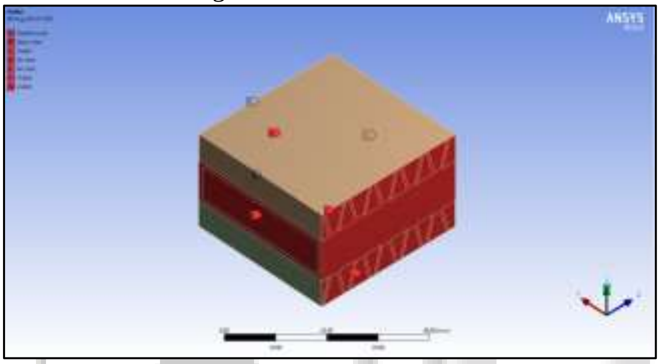


Fig. 3.10: Boundary Conditions

4) *Definition of Fluid Properties*

Every surface or fluid domain has its own distinct property. The properties of the fluid used in the CFD domain are defined at this stage of the CFD Process.

5) *Boundary Conditions*

The complex nature of many fluid flow behaviours has important implications in which boundary conditions are prescribed for the flow problem. A CFD user needs to define appropriate conditions that mimic the real physical representation of the fluid flow into a solvable CFD problem. Every different setup of the CFD domain needs to have an initialization, which is fulfilled by the boundary conditions input.

The CFD code usually has this facility to define the boundary conditions of the CFD problem, where each cells at specific boundary are given finite values.

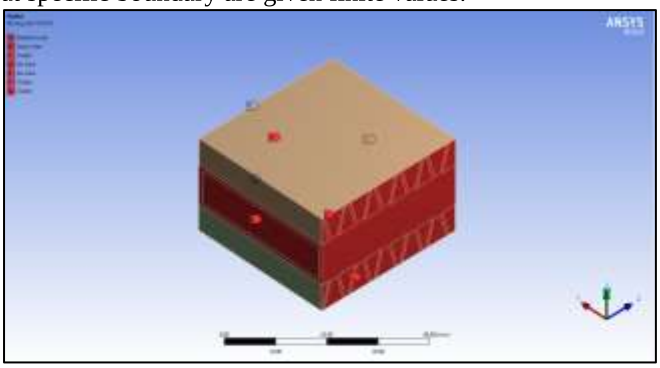


Fig. 3.11: Boundary Conditions

Volume Concentration:	1 %, 2%, 3% of each CG and GO
Mass Flow Rates (LPM):	4,5,6
Inlet Fluid Temperature (K):	323.15
Inlet Air Temperature (K):	298.15
Inlet Air Velocity (m/s):	1.5

Table 4: Input conditions considered

6) *Numerical Solver*

The appropriate usage of either an in-house or a commercial CFD code commands the core understanding of the underlying numerical aspects inside the CFD solver. This section focuses on the treatment of the solver element. A CFD solver can usually be described and envisaged by the solution procedure presented in Figure 2.4

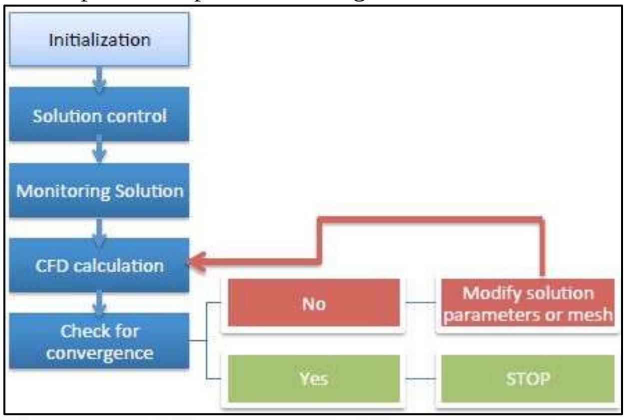


Fig. 3.15:An overview of the solution procedure

In the current market, the solvers usually use three distinct ways of calculating the solutions, namely, the finite difference method, finite element method and the finite volume method. The finite difference and finite element method are usually suitable for stress and structure analysis. On the other hand the finite volume method is the most suitable method for the CFD process. As the name implies, finite volume method is the numerical algorithm calculation process involving the use of finite volume cells.

The steps involved in this solving process are usually carried out in the following sequence:

- Formal integration of the governing equations of fluid flow over all theControl volumes or finite volumes of the solution domain.
- The conversion of the integral forms of the equations into a system ofAlgebraic equations.
- Calculations of the algebraic equations by an iterative method.

7) *Post Processor*

Commercial CFD codes such as ANSYSInc., CFX, ANSYS Fluent, STARCD, and others often incorporate impressive visualization tools within their user friendly GUIs to allow users to graphically view the results of a CFD calculation at the end of a computational simulation. Those data visualization tools of the CFD solver to observe the following results of the simulation:

1) Domain geometry and Grid display
2) Vector plots
3) Line and shaded contour plots
4) 2D and 3D surface plots
5) Particle tracking
6) XY plots and graphs of results

E. Preparation of Nano Fluid Two step Process:

This technique is also known as Cool-Aid method which is usually used for oxide nanoparticles. In this technique nanoparticles are obtained by different methods (in form of powders) and then are dispersed into the base fluid. The main problem in this technique is the nanoparticle agglomeration due to attractive Van der waals forces. One step process: In this process the dispersion of nanoparticles is obtained by direct evaporation of the nanoparticle metal and condensation of the nanoparticles in the base liquid and is the best technique foe metallic Nano fluids such as Cu Nano fluids. The main problems in this technique are low production capacity, low concentration of nanoparticles and high costs. While the advantage of this technique is that nanoparticle agglomeration is minimized. The suspensions obtained by either case should be well mixed, uniformly dispersed and stable in time. Also it should be noted that the heat transfer properties of Nano fluids could be controlled by the concentration of the nanoparticle and also by the shape of nanoparticles. The parameter which affects thermal conductivity is size of Nano particles. Overall we can say that the smaller the size the greater the stability of colloidal dispersion, the greater the stability of colloidal dispersion the greater the probability of interaction and collision among particles and collision among particles and fluid and the greater the effective heat energy transport inside the liquid (Xue 2003).

Thermal conductivity enhancement ratio:

$(K_{effective} = K_{nanofluid} / K_{basefluid})$

And the parameters that most affect the thermal conductivity of Nano fluids are:

1) Particle volume fraction
2) Particle material
3) Base fluids
4) Particle size
5) Temperature

Properties	Water	Al_2O_3	GO
Density (Kg/m^3)	9971	3970	121
Specific Heat (J/Kg K)	4179	765	2100
Thermal Conductivity (W/m K)	0.605	40	2720

Table 1: Properties of Nano fluid and base Fluid

Thermo-physical Properties of Nano-fluid:

Some important Thermo-physical properties of Nano fluid on which heat transfer coefficient of Nano particle depends are given below

1) Specific heat of Nano fluid is define as
   $Cp_{nf} = \Phi\ \rho_p\ Cp_p + (1-\Phi)\ \rho_{bf}\ Cp_{bf} / \rho_{nf}$
2) Density of Nano fluid is define as
   $\rho_{nf} = \Phi\ \rho_p + (1-\Phi)\ \rho_{bf}$
3) Viscosity at room temperature is calculated by using following relation
   $\mu_{nf} = \mu_{bf}(1+2.5\Phi)$
4) Thermal conductivity of Nano fluid for Al_2O_3 + Water is developed by
   $K_{nf} / K_{bf} = Re_{nf}^{0.175}\Phi^{0.5} (K_b / K_{bf})^{0.2324}$
5) Volume concentration or weight concentration of particle Φ is given by
   $\Phi = (w_p / \rho_p) / (w_{nf} / \rho_{nf})$

1) Case 1 Simulation of Radiator With Water

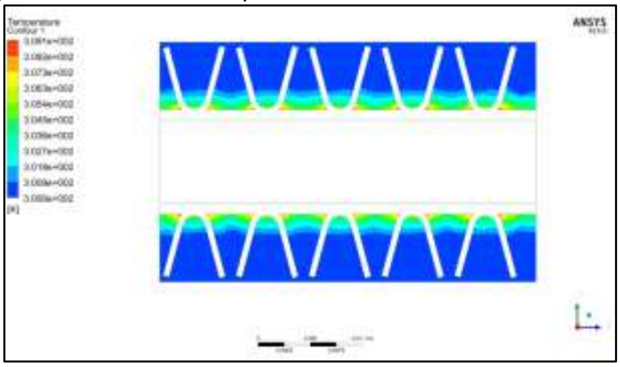


Fig. 4.1: Temperature Contour of Radiator Fluid Domain with 3 LPM.

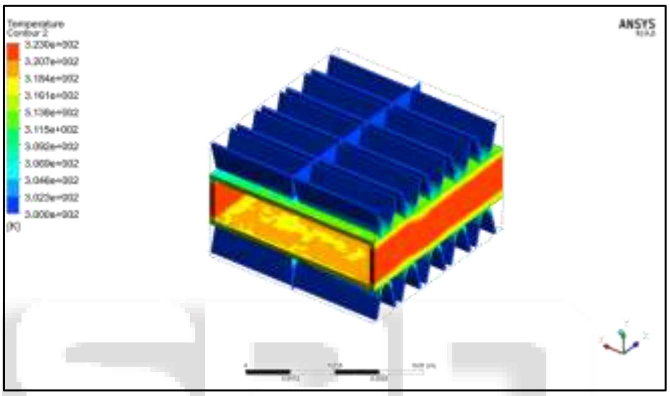
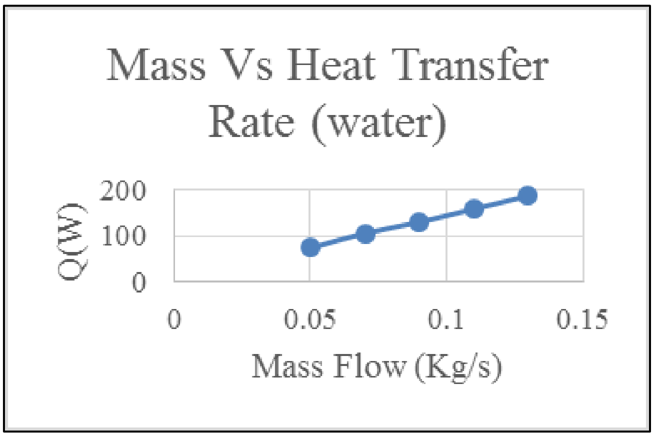


Fig. 4.2: Temperature Contour of Radiator Wall with 3 LPM.

M(kg/s)	Ti (K)	To(K)	Q(W)
0.05	300	301.451	73.0165
0.07	300	301.497	105.4638
0.09	300	301.435	129.9804
0.11	300	301.431	158.4221
0.13	300	301.432	187.357

Table 1: Simulation Results for Water as Base Fluid



Plot 1: Mass Vs Heat transfer Rate for Water

2) Case 2 Simulation of Radiator with Al2o3 Volume Concentration 1%

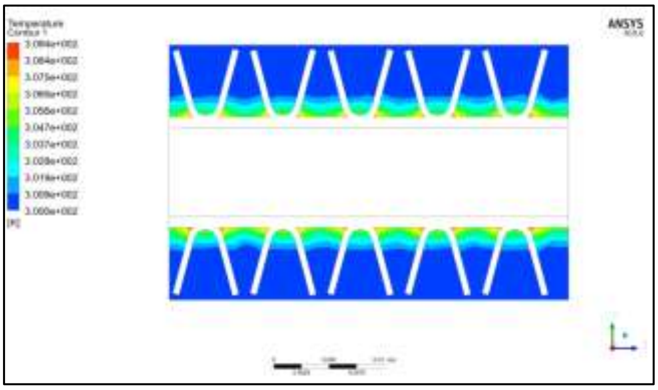


Fig. 4.3: Temperature Contour of Radiator Fluid Domain with Al2o3.

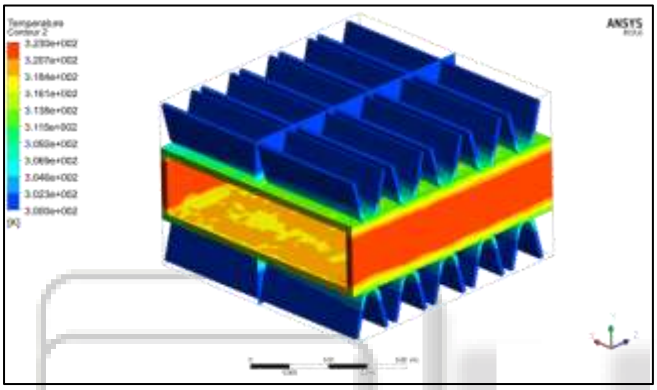
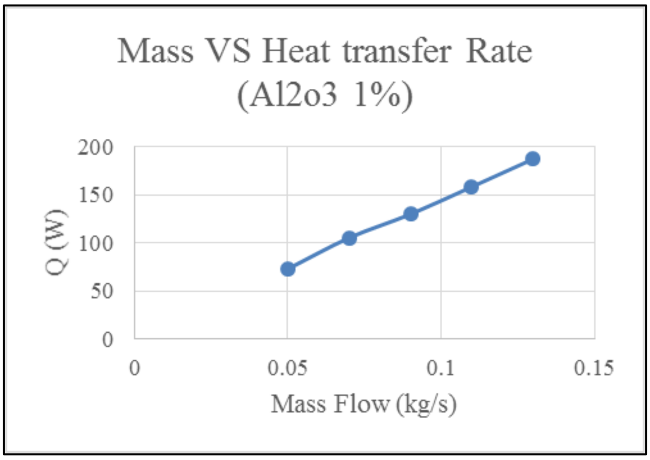


Fig. 4.4: Temperature Contour of Radiator Wall with Al2o3.

Al2o3 1 %				
M(kg/s)	cp	Ti (K)	To(K)	Q(W)
0.05	1006.43	300	301.557	78.3506
0.07	1006.43	300	301.509	106.309
0.09	1006.43	300	301.465	132.698
0.11	1006.43	300	301.433	158.644
0.13	1006.43	300	301.432	187.357

Table 2: Simulation Results for Nano Fluid with 1 % Al2o3 as Base Fluid



Plot 2: Mass Vs Heat transfer Rate for Al2o3 1%

3) Case 3 Simulation of Radiator with Al2o3 Volume Concentration 3 %

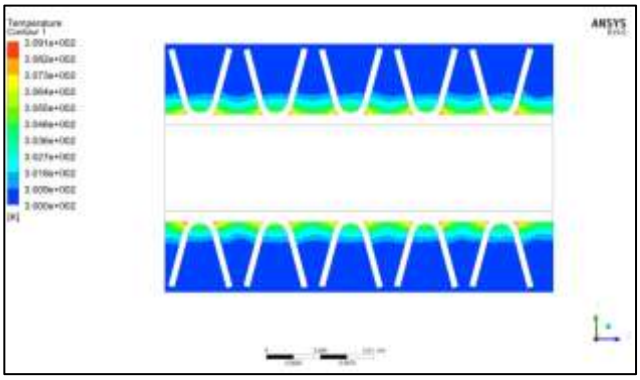


Fig. 4.5: Temperature Contour of Radiator Fluid Domain with Al2o3.

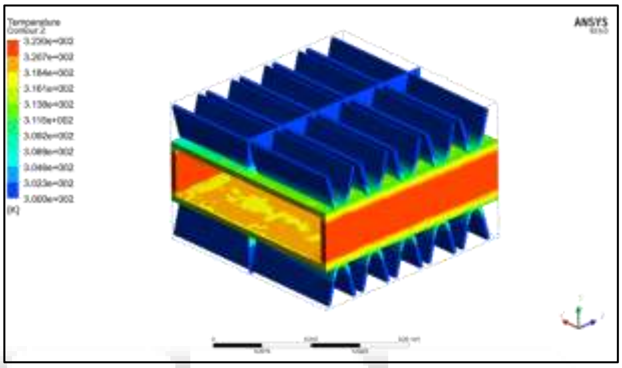
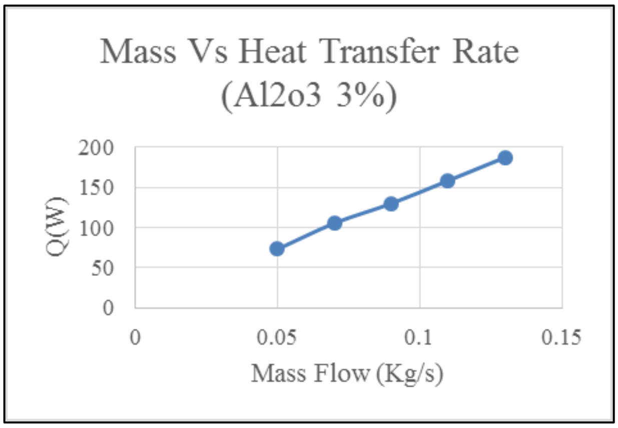


Fig. 4.6: Temperature Contour of Radiator Wall with Al2o3.

Al2o3 3%			
M(kg/s)	Ti (K)	To(K)	Q(W)
0.05	300	301.451	73.0165
0.07	300	301.497	105.4638
0.09	300	301.435	129.9804
0.11	300	301.431	158.4221
0.13	300	301.432	187.357

Table 3: Simulation Results for Nano Fluid with 3 % Al2o3 as Base Fluid



Plot 3: Mass Vs Heat transfer Rate for Al2o3 3%

4) Case 4 Simulation of Radiator with Graphene Oxide (GO) Volume Concentration 1%

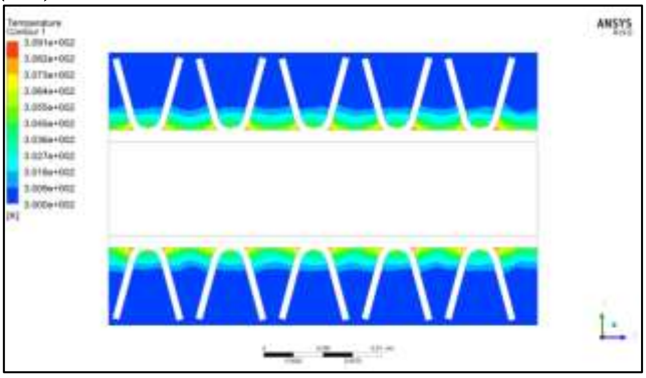


Fig. 4.7: Temperature Contour of Radiator Fluid Domain with GO

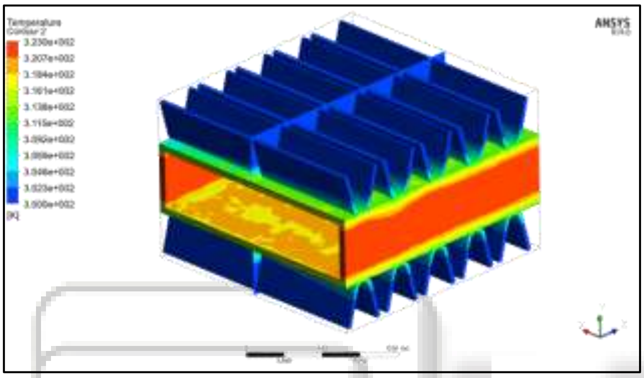
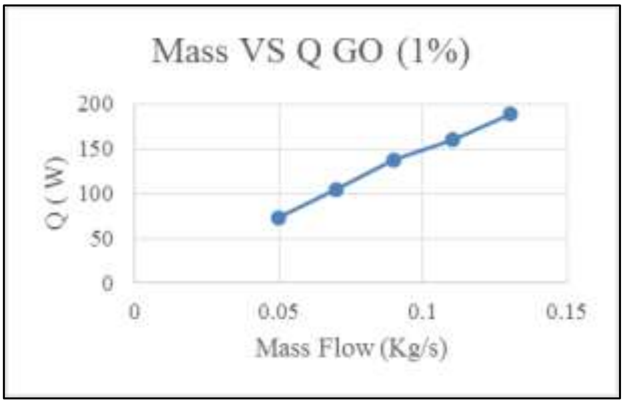


Fig. 4.8: Temperature Contour of Radiator Wall with GO.

M(kg/s)	Ti (K)	To(K)	Q(W)
0.05	300	301.459	73.4190685
0.07	300	301.485	104.6183985
0.09	300	301.52	137.679624
0.11	300	301.445	159.9720485
0.13	300	301.438	188.1420242

Table 4: Simulation Results for Nano Fluid with 1 % GO as Base Fluid



Plot 4: Mass Vs Heat transfer Rate for GO 1%

5) Case 5 Simulation of Radiator with Graphene Oxide (GO) Volume Concentration 3 %

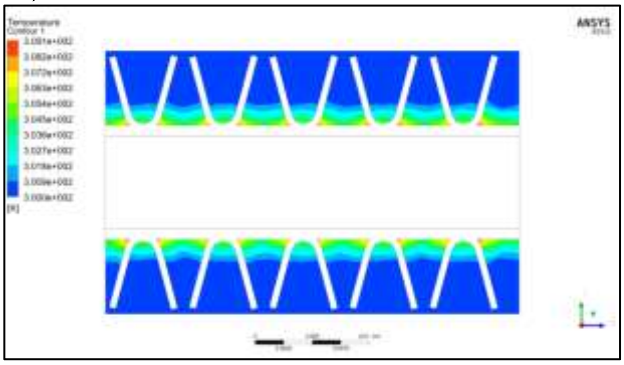


Fig. 4.9: Temperature Contour of Radiator Fluid Domain with Graphene Oxide (Go)

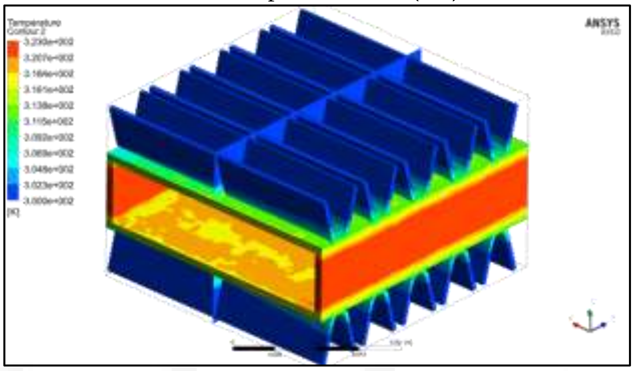
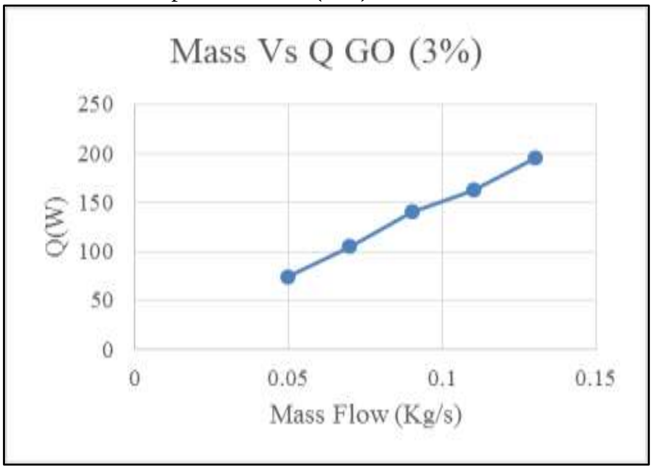


Fig. 4.10: Temperature Contour of Radiator Wall with Graphene Oxide (Go).

M(kg/s)	Ti (K)	To(K)	Q(W)
0.05	300	301.451	73.0165
0.07	300	301.497	105.4638
0.09	300	301.435	129.9804
0.11	300	301.431	158.4221
0.13	300	301.432	187.357

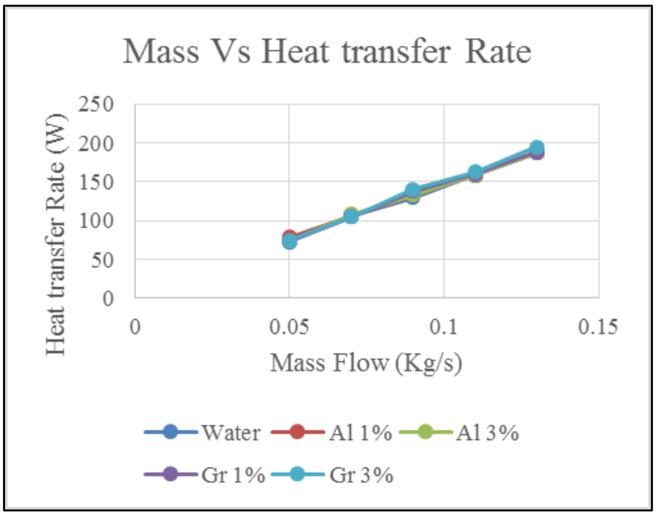
Table 5: Simulation Results for Nano Fluid with 3 % Graphene Oxide (GO) as Base Fluid



Plot 5: Mass Vs Heat transfer Rate for GO 3%

m	Water	Al 1%	Al 3%	Gr 1%	Gr 3%
0.05	73.0165	78.35058	73.87196	73.41907	74.52614
0.07	105.4638	106.3092	108.1409	104.6184	104.7593
0.09	129.9804	132.6978	133.1507	137.6796	140.397
0.11	158.4221	158.6436	158.865	159.972	162.5183
0.13	187.357	187.357	187.357	188.142	194.8147

Table 4: Comparisons between Water and Nano Fluid.



Plot 6: Comparison of Heat transfer Rate between Water and Graphene Nano Fluid

IV. Summary and Conclusion

The convective heat transfer performance of Aluminium Oxide and Graphene Oxide Nano fluid flowing on diesel engine radiator has been Analytical investigated. Analytical study have been carried out under turbulent conditions. The effect of particle concentration, inlet temperature and flow rate has been determined. Comparison of heat transfer rate in water and in Nano fluid is done from readings by plotting graph.

Important conclusions are:

1) Dispersion of nanoparticles into distilled water increases the thermal conductivity and viscosity of Nano Fluid.
2) This augmentation increases with increase in particle concentration.
3) Though increase in particle concentration increase friction factor. This problem can be overcome by using stirring mechanism which enhances heat transfer rate at low concentrations also.
4) It has been observed that using stirring mechanism clogging and settlement of particles problems can be effectively overcome.
5) Stirring mechanism improves heat transfer rate by 10-15% than without stirring.
6) The Comparison of Heat transfer between al2o3 & GO Clearly Shows that GO has Better Performance than other nano Fluids.

V. Future Scope

Analytical and studying Nano fluids validate their properties but focus should be to try to extract advantages and limitations of use of Nano fluid in cooling system. Focus must be on overcoming the limitations of Nano fluids and it will be worthwhile to work out with various permutations and combinations of the Nanoparticles and base fluids which might help to develop the science related to Nano fluids more extensively because in most part of India atmospheric temperature conditions make it essential to use water in cooling system.

References

[1] Adnan M. Hussein, R.A. Bakar, K. Kadirgama – "Study of forced convection Nanofluid heat transfer in the automotive cooling system".
[2] Jaafar Albadr , Satinder Tayal, Mushtaq Alasadi – "Elsevier" –case studies in thermal engineering studied "Heat transfer through heat exchanger using Al_2O_3 nanofluid at different concentrations" (eISSN:2319-1163 | pISSN: 2321-7308).
[3] Ravi Adwani, Shri Krishna Choudhary - "Analyticalal Investigation of Heat Transfer Rate In Automobile Radiator Using Nanofluid" at JISET – International Journal of Innovative Science, Engineering & Technology, Vol. 1 Issue 6, August 2014. (eISSN: 2319-1163 | pISSN: 2321-7308).
[4] Kirubadurai "et al: in International Journal of Research in Engineering and Technology (eISSN: 2319-1163 | pISSN: 2321-7308).
[5] Sidi El BecayeMaiga, Samy Joseph Palm, Cong Tam Nguyen, Gilles Roy, Nicolas Galanis – "Heat transfer enhancement by using Nano fluids in forced convection flows".
[6] S.M. Peyghambarzadeh, S.H. Hashemabadi, M. SeifiJamnani, S.M. Hoseini – "Improving the cooling performance of automobile radiator with Al_2O_3/water nano fluid". Applied Thermal Engineering (2011) 1833e1838.
[7] S.M. Peyghamberzadeh, S. H. Hashemabadi, M. Naraki, Y. Vermahmodi – "Analyticalal study of overall heat transfer coefficient in the application of dilute Nano fluids in the car radiator"
[8] Kiran bhusal, J.H.Bhangale – "Study and Investigation of Heat transfer enhancementof a car radiator by using Nano fluid-review".
[9] Shuichi Torii – "Analyticalal study on Convective heat transfer of aqueous suspension of Nano diamond particle", International symposium on Eco Topia science 07 (2007).
[10] S.H. Anil Kumar and G.Jilani – "Convective heat transfer enhancement in an enclosure with fin utilizing nano fluid", International journal of aerospace and mechanical engineering

