ANALYSIS OF AUTOMOBILE RADIATOR USING COMPUTATIONAL FLUID DYNAMICS

A Project report submitted

in partial fulfillment of the requirement for the award of the degree of

BACHELOR OF ENGINEERING

In

MECHANICAL ENGINEERING

By

K KARTHIK PREETHAM	**318126520148**
KANDI HARISH	**318126520146**
ARAJALLA SOMANADH	**318126520118**
B YESHWANT BALVADA	**318126520119**
GANGARAPU MANJIT	**318126520137**

Under the guidance of

Mr. B G CHANDRA SEKHAR M.Tech(Ph.D)

Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES

Autonomous status accorded by UGC and Andhra University (Approved by AICTE,
Permanently Affiliated to A.U., & Accredited by NBA, NAAC) Sangivalasa, Bheemunipatnam
Mandal Visakhapatnam(District)-531162

2022

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES

Autonomous status accorded by UGC and Andhra University (Approved by AICTE, Permanently Affiliated to A.U., & Accredited by NBA, NAAC) Sangivalasa, Bheemunipatnam Mandal Visakhapatnam(District)-531162

CERTIFICATE

This is to certify that the Project Report entitled **"ANALYSIS OF AUTOMOBILE RADIATOR USING COMPUTATIONAL FLUID DYNAMICS"** has been conducted by **K KARTHIK PREETHAM (318126520148), KANDI HARISH (318126520146), ARAJALLA SOMANADH (318126520118),B YESHWANT BALVADA (318126520119), GANGARAPU MANJIT (318126520137)** in the partial fulfilment of the requirements for the award of the Degree of **Bachelor of Technology** in **Mechanical Engineering**. It is the work of bonafide, carried out under the guidance and supervision of Mr. **B G CHANDRA SEKHAR M.Tech (Ph.D)**, Assistant Professor, Department of Mechanical Engineering during the academic year of 2018-2022.

PROJECT GUIDE

Mr. B G CHANDRA SEKHAR M.Tech (Ph.D)
Assistant Professor
Dept. of Mechanical Eng.
ANITS, Sangivalasa
Visakhapatnam.

18.5.22

HEAD OF THE DEPARTMENT

Dr. B. Naga Raju
Professor
Dept. of Mechanical Eng.
ANITS, Sangivalasa
Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa-531 162 VISAKHAPATNAM Dist A.P.

THIS PROJECT WORK IS APPROVED BY THE FOLLOWING BOARD OF EXAMINERS

INTERNAL EXAMINER. 28.5.22

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa-531 162 VISAKHAPATNAM Dist A

EXTERNAL EXAMINER. 28/5/22

ACKNOWLEDGEMENT

We express immensely our deep sense of gratitude to **Mr. B G CHANDRA SEKHAR**, Assistant Professor Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, Sangivalasa, Bheemunipatnam (Mandal), Visakhapatnam (District) for his valuable guidance and encouragement at every stage of the work for the successful fulfilment of students.

We are here thankful to **Prof. T.V Hanumantha Rao,** Principal and **Prof. B.Nagaraju,** Head of the Department, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for their valuable suggestions.

We express our sincere thanks to the non-teaching staff of Mechanical Engineering for their kind co-operation and support to continue work.

Last but not least, we like to convey our thanks to all who have contributed either directly or indirectly for the completion of our work.

K KARTHIK PREETHAM	**(318126520148)**
KANDI HARISH	**(318126520146)**
ARAJALLA SOMANADH	**(318126520118)**
B YESHWANT BALAVADA	**(318126520119)**
GANGARAPU MANJIT	**(318126520137)**

ABSTRACT

Radiators are a type of heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. Upwards of 33% of the energy generated by the engine through combustion is lost in heat. Insufficient heat dissipation can result in the overheating of the engine, which leads to the breakdown of the lubricating oil, metal weakening of engine parts, and significant wear between engine parts. To minimize the stress on the engine because of heat generation, automotive radiators must be redesigned to be more compact while still maintaining an elevated level of heat transfer components. This led to the increased demand for power-packed radiators, which can dissipate the maximum amount of heat or any given space. This project aims to do the analysis of a straight tube radiator for different velocities. The modeling is done using Creo Parametric, fluid flow analysis is done by using Ansys Fluent.

Keywords: ANSYS, CFD, Velocity, Heat Exchanger, Fluent, Car Radiator

CONTENTS

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER-1

INTRODUCTION

1 INTRODUCTION

1.1 DEFINITIONS:

1.1.1 HEAT EXCHANGER:

A heat exchanger is a system used to transfer heat between two or more fluids. Heat exchangers are used in both cooling and heating processes. The fluids may be separated by a solid wall to prevent mixing, or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air. Another example is the heat sink, which is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often air or a liquid coolant. There are three primary classifications of heat exchangers according to their flow arrangement. In parallel-flow heat exchangers, the two fluids enter the exchanger at the same end, and travel in parallel to one another to the other side. In counterflow heat exchangers the fluids enter the exchanger from opposite ends. The counter current design is the most efficient, in that it can transfer the most heat from the heat (transfer) medium per unit mass due to the fact that the average temperature difference along any unit length is higher. For efficiency, heat exchangers are designed to maximize the surface area of the wall between the two fluids, while minimizing resistance to fluid flow through the exchanger.

1.1.2 RADIATOR:

The radiators are heat exchangers that are used to transfer thermal energy from one medium to another for the purpose of cooling and heating. A radiator is a device consisting of a large amount of cooling surface which contains enormous amounts of air so that it spreads through the water to cool efficiently. The radiator has a wide range of applications in automobile industries there are used to cool the internal combustion engine. They also used in piston-engine aircraft, railways, locomotives, motorcycles, stationary generating plants and other places where such engines are used. Radiators are usually made of copper and brass because of their high heat conductivity. The various sections of the radiators are joined by soldering. The radiator will do the cooling purposes because the temperature of the burning gases in the engine cylinder reaches up to 1500^{O}C to 2000°C. Radiators are classified according to the direction of the water flow through them. In some, the water flows from top to bottom-down flow type radiator.

1.2 RADIATOR TERMINOLOGY:

Radiators transfer most of their heat via convection rather than thermal radiation. If there are large temperature differences, it can cause distortion of the engine components. Its composition, there are water room, water chamber, and the radiator core of the three parts. The coolant is flowing in the radiator core, and the air is outside the radiator, and through the coolant, the stifling air into chilly air, so as to achieve the purpose of absorbing heat and cooling.

Upper tank: Due to absorbing heat from the engine coolant get hot, the liquid expands and creates pressure in the radiator additionally. The pressure causes the coolant to get higher than the pressure cap, in order to prevent leakage excess coolant needs to be captured somewhere the excess fluid flows into the pipe and goes into the overflow tank.

Lower tank: just after it has passed through the heat radiating tubes and fins in the body of the radiator the bottom tank receives the cooling water.

Tubes: On its way to the opposite tank, as the coolant passes through the radiator tubes, it transfers heat to the tubes that transfer the heat to the fins that are attached between the rows. The fins head the heat flow to the ambient air.

Filler cap: Since the coolant expands the high coolant temperature leads to an increase in pressure in the cooling system. Coolant is press in the tank that will increase the pressure in the tank.

Fins: Fins are surfaces that are used to increase the rate of heat transfer to or from the environment and they extend from the surface by increasing convection. Fins increase the surface area and can be an economical solution to heat transfer problems.

Fan: The fan is mounted behind the radiator on the water pump shaft. When the engine is running at the low speed it certainly insufficient to produce the desired cooling from nature. Here the fan serves the purpose.

Thermostat: Now what the thermostat does is as you have got coolant running through engine it looks at the temperature of it. Once the temperature gets too high and your engine is warmed up then thermostat detects and will pass the coolant back through the radiator and then the coolant will come cool through the radiator.

Water Pump: The water pump is used to increase the velocity of the circulating water. When a low-temperature coolant passes through the water pump it pumps the coolant back into the engine.

1.3 HISTORY:

The heating radiator was invented by Franz San Galli in 1855, a Kingdom of Prussia-born Russian businessman living in St. Petersburg. In the late 1800s, companies, such as the American Radiator Company, promoted cast iron radiators over previous fabricated steel designs in order to lower costs and expand the market. Car radiators grew with the invention of cars. In the history of the world's automobile development, the German engineer Karl Benz made the first gasoline-powered tricycle. The car uses a two-stroke gasoline engine and has some basic features of modern cars, including a water-cooling system. But there is no radiator, relying on a long and curved pipe to circulate water to ensure the cooling of the internal combustion engine. This type of coiled radiator has low efficiency, consumes a lot of water, and requires a bulky cooling circuit, which is far from satisfying the demand.

The first real car radiator in the world was the honeycomb radiator invented by Wilhelm Maybach in 1901. He was a German engine designer and industrialist who designed the first honeycomb radiator for the Mercedes 35hp. Later, he improved the design and changed the square tube into a round tube. The two ends of the tube were pierced into a hexagon, and the ends of the hexagon were welded together by soldering. Save copper wire. However, the internal water flow speed of the honeycomb radiator is slow, which is easy to accumulate; the welding seam is long, which is easy to leak; and the cost is high, and the maintenance is complicated. Therefore, it was later replaced by a tube-fin radiator.

At present there are two, one is aluminium, used in the general passenger car; the second is copper material, used in large commercial vehicles.

Fig 1.1.1 Car Radiator

1.4 TYPES OF HEAT EXCHANGERS:

1.4.1 DOUBLE PIPE HEAT EXCHANGER:

Double-pipe heat exchangers are devices that provide the transfer of thermal energy between two fluids at different temperatures. The major use of these heat exchangers is the sensible heating or cooling process of fluids where small heat transfer areas are required. The oil cooler is an example of these processes. In this study, fouled finned, clean finned, fouled unfinned, and clean unfinned double-pipe heat exchangers used as an oil cooler in ships have been compared. In thermal design, suggested different Nusselt number equations have been used, and the results of these equations have been shown in tables and figures. As a result, it is evaluated that using fouled finned double-pipe heat exchanger as oil cooler in ships is the most appropriate selection.

1.4.2 SHELL TUBE HEAT EXCHANGER:

A shell and tube heat exchanger are a class of heat exchanger designs. It is the most common type of heat exchanger in oil refineries and other large chemical processes and is suited for higher-pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to transfer heat between the two fluids. The set of tubes is called a tube bundle, and may be composed of several types of tubes: plain, longitudinally finned, etc.

1.4.3 PLATE HEAT EXCHANGER:

A plate heat exchanger is a type of heat exchanger that uses metal plates to transfer heat between two fluids. This has a major advantage over a conventional heat exchanger in that the fluids are exposed to a much larger surface area because the fluids are spread out over the plates. This facilitates the transfer of heat, and greatly increases the speed of the temperature change. Plate heat exchangers are now common and very small brazed versions are used in the hot-water sections of millions of combination boilers. The high heat transfer efficiency for such a small physical size has increased the domestic hot water (DHW) flowrate of combination boilers. The small plate heat exchanger has made a significant impact in domestic heating and hot water. Larger commercial versions use gaskets between the plates, whereas smaller versions tend to be brazed.

1.4.4 CONDENSERS AND BOILERS AS HEAT EXCHANGERS:

A condenser is a heat exchanger used to condense a gaseous substance into a liquid state through cooling. In so doing, the latent heat is released by the substance and transferred to the surrounding environment. Condensers are used for efficient heat rejection in many industrial systems. Condensers can be made according to numerous designs and come in many sizes ranging from small (hand-held) to exceptionally large (industrial-scale units used in plant processes). For example, a refrigerator uses a condenser to get rid of heat extracted from the interior of the unit to the outside air.

Condensers are used in air conditioning, industrial chemical processes such as distillation, steam power plants and other heat-exchange systems. Use of cooling water or surrounding air as the coolant is common in many condensers.

In a boiler heat exchanger, heat is transferred from the hot gases of a combustion process to water moving through the exchanger’s internal piping system. As a result, the water heats up as the gas cools down. Other appliances that utilize a heat exchanger include refrigerators and air conditioners, though the direction of heat transfer in these systems is reverse to that of a boiler.

1.5 CLASSIFICATION OF RADIATOR:

ACCORDING TO THE RADIATOR CORE STRUCTURE:

- CELLULAR RADIATOR
- TUBULAR RADIATOR

ACCORDING TO THE DIRECTION OF FLOW OF MOVEMENT:

- CROSS FLOW
- DOWN FLOW

1.5.1 ACCORDING TO THE RADIATOR CORE STRUCTURE:

1.5.1.1 CELLULAR RADIATOR:

The core is composed of many individual air cells which are surrounded by water. Because of its appearance, the cellular type usually is known as a honeycomb radiator, especially when the cells in front are hexagonal in form. In a cellular. radiator, the clogging of any passage results in a loss but of a small part of the total cooling surface. Air passes through the tubes and the water flows in the spaces between them in cellular type core. The core contains a large number of air cells that are surrounded by the radiator. It is known as a honeycomb radiator because of its appearance as the cells in front are hexagonal in form.

Fig 1.1.2 Cellular Radiator

1.5.1.2 TUBULAR RADIATOR:

In tubular type core, the upper and lower tanks are connected by a series of tubes through which water passes. Fins are placed around the tubes to improve heat transfer. Air passes around the outside of the tubes, between the fins, absorbing heat from the water in passing. In a tubular radiator, because the water passes through all the tubes, if one tube becomes clogged, the cooling effect of the entire tube is lost. In a cellular. radiator, the clogging of any passage results in a loss but of a small part of the total cooling surface.

Fig 1.1.3 Tubular Radiator

1.5.2 ACCORDING TO THE DIRECTION OF FLOW OF MOVEMENT:

1.5.2.1 CROSS FLOW RADIATOR:

A cross flow radiator is a radiator that has its tanks on the sides of the radiator core. The coolant moves across the core from the right side to the left side. This type of radiator is a good option for cars with a low hood line or a modern smaller car.

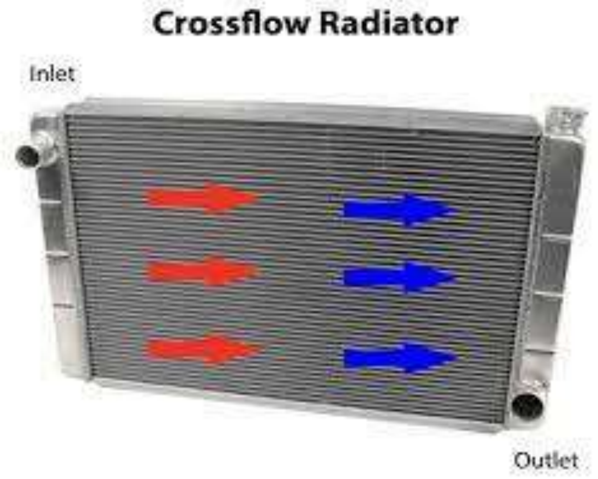


Fig 1.1.4 Cross flow Radiator

1.5.2.2 DOWN FLOW RADIATOR:

In a down-flow radiator you have a tank attached to the top and bottom of the radiator core. The coolant enters the top tank and flows down to the bottom tank. As you have guessed by now, a crossflow radiator has tanks on the left and right side. Coolant enters one side and flows across to the other.

Fig 1.1.5 Down flow Radiator

1.6 WORKING PRINCIPLE:

The radiator is simple device. Nowadays most modern cars use aluminium radiators. Radiators usually have a tank on each side, and inside the tank is a transmission.In this type of radiator, we are going to have an aluminum mesh. In this aluminum/copper device, it consists of one inlet and one outlet. Inside the radiator, there are tubes mounted in a parallel arrangement. And the aluminum/copper fins are attached to all the tubes. The Radiator working is quite simple. In the radiator, the coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. The hot water enters the radiator through the inlet port. And a fan is attached behind the radiator to cool down the hot water in the tubes. The fan blows the air and cools down the water. So, the water is going to come out cooler than it entered before and then go back to the engine. we have come to see the great purpose of a radiator in the automobile engine cooling system. Well, working is less complex and easy to understand. In a radiator, there is a tank on each side, and inside contained a transmission cooler. There are inlet and outlet ports, which from the inlet port coolant flows to the tubes where they are exposed to cooling. The tubes are in a parallel arrangement, where they come in contact with cooling fins to draw away heat from the core.

1.7 APPLICATIONS OF RADIATOR:

- To cool motor oil or power steering fluid.
- Automatic transmission fluid.
- Air conditioner and Automobiles.
- Cooling internal combustion engines.
- Piston-engine aircrafts.
- Railway locomotives where heat engines are used

CHAPTER-2

LITERATURE REVIEW

2 LITERATURE REVIEW

1 Adnan M. Hussain studied the forced convection heat transfer enhancement by TiO2 and SiO2 suspended in water as a base fluid inside the flat copper tubes of an automotive cooling system . Maximum Nusselt number enhancements of up to 11% and 22.5% were obtained for TiO2 and SiO2 nanoparticles, respectively, in water. The experimental results showed that the Nusselt number behaviours of the nanofluids highly depended on the volume flow rate, inlet temperature and nanofluid volume concentration. The results showed that the SiO2 nanofluid produces a higher heat transfer enhancement than the TiO2 nanofluid; likewise, TiO2 nanofluid enhanced heat transfer more than pure water. The results also proved that TiO2 and SiO2 nanofluid have a high potential for heat transfer enhancement and are highly appropriate for industrial and practical applications. The study succeeded in analysing the importance of nanofluid which in fact proves to be beneficial in many of the other industrial application.

2 A.H. Mamun, K.Y. Leong, S.N. Kazi, and R. Saidur: This paper outlines the effect on cooling capacity while using nanofluid based coolant in engine cooling system. Through this, it was discovered that nano-fluid increases heat transfer which has higher thermal conductivity than base coolant which blends of 50%/50% ethylene glycol and water. When same heat transfer occurs compared to base one of the radiator, its core area can be reduced. Ethylene glycol as a coolant in the radiator when used compared to the nanofluid, the thermal performance of a radiator is raised by increasing power of pumping.

3 Alhassan S. Tijani conducted research work with a goal to replace conventional coolants in radiators . For that he used Al2O3/CuO based nanofluid with water and ethylene glycol as base fluid . He conducted his experimental work to determine the thermos-physical properties and heat transfer characteristics of the used nanofluid. He also simulated the model using ANSYS Fluent software. The heat transfer characteristics were measured in terms of thermal conductivity , heat transfer coefficient ,Nusselt number and rate of heat transfer. Density plays an especially key role in determining the thermo-physical behaviour of nanofluids due to its influence on Reynolds number, Nusselt number and thermal diffusivity. As described in the paper the lowest thermal conductivity of the coolant is at 0.415 W/m K for the base fluid which is in fact conventionally used and the highest thermal conductivity observed is at 1.287 W/m K for 0.3%concentration of Al2O3 nanofluid. The nanofluids have a tremendous increase in thermal conductivity as compared with the base fluid and this is expected to increase heat transfer enhancement in the radiator. As the concentration of nanoparticles added to the nanofluid increases from0.05% to 0.3%, the thermal conductivity for both nano fluids increase as well. Comparing both nanofluids, CuO nanofluid had lower thermal conductivity than that of Al2O3, with thermal conductivity of 1.241 W/m K and 1.287 W/m K for CuO and Al2O3nanofluidrespectively.

4 A. Sing Nano fluid is the suspensions of nano particle in base fluid. Nano fluids are the unique feature which is different from conventional liquid solid mixture in which nm or μm sized particle are added in the base fluid to enhance the heat transfer rate. Most system/process whose performance is affected by the heat transfer disceptation nano fluid provides especially key role in such case. It is evident that the effects of viscosity and thermal conductivity should be considered together.

5 Chavan & Tasgaonkar: Low-lying areas and high-temperature areas (regions with low heat transfer areas) are identified in the corners. We see that the velocity increases with the rpm of the radiator fan. For optimal performance, it eliminates corners and improves the radius of the Circular mode. The low power consumption of the fan works well because the cost savings of the equipment is 24%, the cost savings on the production of large scale will be about 20% when the operation is done. Chavan et al., Hydraulic actuators such as pumps and motors were selected to obtain the required force of gravity. The calculation and temperature of the same temperature are calculated. To eliminate temperature growth, a cooling radiator is performed using the standard LMTD method. The wings have been shown solely to give a sense of proportion. The design has been successfully adapted to the available environment. Radiator analysis was performed using CFD in ANSYS Fluent.

6 Devireddy Sandhya conducted work for improving the performance of radiator with ethylene glycol water based TiO2 nanofluids . The paper focuses on measuring the overall heat transfer coefficient of the two working fluids that include 40:60% EG/W and 40:60% EG/W and mixed with TiO2 nanofluid which when worked experimentally is found that the presence of TiO2 enhances the heat transfer rate. They varied various concentrations of the nanofluid to research optimum condition for obtaining the maximum performance of the radiator. It was found that at 0.5% concentration the heat transfer rate is enhanced to about 35% of the normal base fluid . The paper also concludes that on the increasing the flow rate of nanofluids through the radiator tubes increases the heat transfer coefficient.

7 D.H. Lee, L.D. Tijing, B.C. Pak and B.J. Baek: By the utilization of twisted and straight internal fin inserts the heat transfer enhancement was researched under this article and the pressure drop and characteristics of heat transfer results on horizontal two tubes with coil-wire insert were concluded. The hot and cold-water mass flow rates effect directly the heat transfer rate and coefficient of heat transfer. It is seen that as Re raises the coil-wire insert effect on the magnification of heat transfer leads to reduce.

8 Hardik Kumar Patel & Deepu Dinesen: Using CFD were identified by comparing the heat transfer and pressure reduction of the heat exchanger with different performance parameters. Reduction of Vishwa Deepak Dwivedi and Ranjeet Rai in the cooling capacity of the incoming air temperature while the cooling capacity rises with the rise of the incoming cooling temperature. Decreased pressure also increases with increasing air pressure and with a decrease in the ratio of weight to the radiator. Approximately 6% increase in cooling capacity using a hot-duty louver fin with Nanofluid compared to a standard cooler with the same model.

9 Jama et al.: The air flow distribution and non-uniformity across the radiator of full-size Australian made ford falcon was evaluated in industrial wind tunnel. The cooling air intake of the vehicle were shielded by a quarter, one half and three quarter and fully blocked. The best method to shield front end is to employ horizontal method. This shielding method produces the more uniform cooling airflow distribution compared to other methods. Non uniformity index increased significantly as the front-end air intake area was shielded. It is reduced the cooling capacity of the vehicle. These shielding methods also produced higher average velocity across the radiator which is analogous to better cooling.

10 John Vetrovec: By use of passive heat load accumulator which normalizes out peak heat loads of the engine cooling system with heat load averaging capacity was evaluated through this paper. During reduced heat load condition, the phase change material will store the heat generated during peak which is called heat load accumulator. During a cold engine start the reduced emissions of harmful pollutants and faster engine warm-up was done by translating heat load to a smaller coolant inventory which by averaging allows relaxation of cooling system and substantial reduction of system size and weight, which was done by losing phase change of PCM (Phase Change Material) from liquid to solid or vice versa, which would result in same heat rejection and reduction in load on cooling system for compact heat exchanger. The accumulator can be adjusted to replace a portion of ECS (Engine Cooling System) coolant lines, when faster warmups and high transient loads at cold engine start was handled by the system which results in concept of down-sizing of ECS volume and weight when 1) handling of high-transient loads by surpassing a full-size ECS. 2) Under typical heat load conditions, comparable to a full-size ECS offering a heat load-handling performance.

11 JP Yadav and Bharat Raj Singh: This article reviews the variation of parameters like inlet coolant temperature, its mass flow rate; etc. and installation of the radiator into a test-setup to review a differential analysis between different coolants. The coolant used here is composed of water and propylene glycol in a ratio of 40:60 and other is water alone. It resulted that due to the corrosiveness of water and containment of dissolved salts decreased the quality of coolant flow passage and its limitation occurs to water independent of its results in the best coolant. From the tests that have been followed in this paper, it is concluded that: one. The cooling capacity and effectiveness will raise with raise in the rate of coolant flow in radiator more for water than the mixture. 2. Cooling Capacity and Output Temperature is more for water and mixture respectively for given Input Temperature of Radiator for both with and without Fan.

12 Komalangan Krishnakumar, Navid Bozorgan and Nariman Bozorgan:

A numerical study on coolant i.e., CuO-water nanofluid with a given pumping power and heat exchange of copper oxide water capacity was described in this article. Under turbulent flow conditions, the overall convective local coefficients of heat transfer of nanofluid at distinct volume fractions (2% to 0.1%) and effects of speed of automobile on the radiator's efficiency were researched. The nanofluid's total coefficient of heat transfer is much better than the coefficient of heat transfer of water was described in the numerical analysis. Here and hence, it is concluded that the total area of heat transfer of the radiator can be decreased.

13 M. Seifi Jamnani, S.M. Peyghambarzadeh, S.M. Hoseini and S.H. Hashem Abadi: This paper research about the experimental comparison of heat transfer by forced convection of Nano-fluid which is water based to that of pure water in the radiator of the automobile. Where on the addition of Al2O3 nanoparticles into the water, five distinct level of quantity magnitude of Nano-fluids in the domain of 0.2-1 vol. % was made, where the sample functional liquid goes into the radiator consisting of 34 vertical tubes which have the elliptical cross-section and with constant speed inside the tube bank the air creates a cross flow. To have the fully turbulent regime the flow rate of the liquid has been modified in the domain of 2- 5 lit/min. On the other hand, by fluctuating the temperature in the domain of 37-490oC the effect of fluids inlet temperature to that of radiator on coefficient of heat transfer has been studied, which results the improvement in heat transfer effectiveness by increasing the circulating rate of fluid while the inlet temperatures of fluid to that of radiator has adverse effects. In correspondence with pure water, the efficiency of heat transfer can be enhanced by the application of Nano-fluid at low concentrations, up to 45%.

14 N. Galanis, G. Roy, C.T. Nguyen, and C. Gauthier: Here the magnification of heat transfer delivered by a specific nanofluid i.e., water and Al2O3 composition have been experimentally studied for the cooling of microprocessors and other heated electronic components which is the work of water closed system. The use of Nanofluid was addressed to be advantageous from the data obtained for Nano-fluid and distilled water with various component concentrations about 2.2% and 0.95%. with respect to distilled water, the heat transfer improvement was observed for a 4.5% concentration.

15 Nor AzwadiCheSidik studied various experimental projects carried out regarding various nanofluids and the recent technological advancements of nano fluids in engine cooling system. He studied the works of Vajja et al., Eastman et al. , Liu et al. and derived the summary regarding various use of nanofluids as coolants in automobile system their preparation methods, their influence on the efficiency of the system , various stabilizing agents, and also respective thermal conductivities of various nanofluids. The research paper helped in recognizing the performance of each nanofluid including the accurate experimental results that can be used for further experimental validation.

16 Oliet et al. Studied varied factors which influences the radiator performance. It includes air, fin density, coolant flow and air inlet temperature. The radiator performance depends upon air and coolant mass flow rate. When air and coolant flow rates increase the efficiency of radiator also increases. When inlet air temperature increases the cooling capacity decreases. Smaller fin spacing and greater louver fin angle have higher heat transfer. Fin density may be increased till it blocks the air flow and heat transfer rate reduced.

17 Sadik Kakac, et al In his literature survey showed that nanofluids significantly improve the heat transfer capability of conventional heat transfer fluids such as oil or water by suspending nanoparticles in these base liquids. The understanding of the fundamentals of heat transfer and wall friction is prime importance for developing nanofluids for a wide range of heat transfer application. He concluded that although there are recent developments in the study of heat transfer with nanofluids, more experimental results and the theoretical understanding of the mechanisms of the particle movements are needed to understand heat transfer and fluid flow behaviour of nanofluids.

18 S. Cheong, Y. Kwon, Y. Hwang, D. Kim, and Y. Cho (2009): A straightforward circular tube which is having turbulent and laminar flow with a constant heat flux of convective heat transfer coefficient was studied, whose performances are affected by nanofluids in this paper. The coefficient of heat transfer by convection of alumina nanofluid is improved in correspondence to base sample fluid by 20% & 15% in turbulent and laminar flow correspondingly. Through their research, they also saw that in turbulent flow, thermal conductivity plays a key role and in the laminar flow, the thermal boundary layer plays a key role. There is no enhancement in the coefficient of heat transfer by convection for amorphous molecules of nanofluids.

19 S. Heris They study the effect of water ethylene glycol mixture base nanofluid in a car radiator. Significant enhancements in heat transfer rate are observed using this mixture. The highest Nu number enhancement up to 55% was obtained in 0.8 % volume concentration of CuO and water ethylene glycol mixture. As increase in inlet temperature the Nu number is increased.

20 Siraj Ali Ahmed too carried out work on use of nanofluids in radiator which too gives brief idea about the use of TiO2 nanofluid in radiator and helps us in reducing the focus of our work to the use of TiO2. The team concluded that overall heat transfer coefficient of TiO2 nanofluid can be experimentally used for measuring as a function of concentration and temperature. The paper concludes heat transfer coefficient improves for 0.2% nanoparticle concentration as compared to pure water which happens due to TiO2’s grater thermal conductivity, aspect ratio, lower specific gravity, thermal resistance and thermal resistance, geometry of particles and larger specific area as to compared to pure water.

21 Yiding Cao et al.: They introduce application of heat pipe in automobile industry. In this application heat pipe is introduced in the automotive radiator to enhance heat transfer. The use of heat pipe increases the automobile radiator efficiency and reduces cooling fan power consumption. Heat pipes are wickless heat pipes and two- phase closed thermosyphons. The working fluids inside the heat pipe are different than the engine coolant. The effectiveness of heat pipe are hundred times higher than the copper. The gravity is used to assist the return fluid. Air is evocated from container and container is sealed. Heat was applied to the evaporator section, which causes the liquid to vaporize. The vapor then flows from the hotter section due to the higher vapor pressure to the colder section of the heat pipe, where it was condensed. The liquid condensate then returns to the evaporator section from the condenser section under the assistance of gravity.

CHAPTER-3

INTRODUCTION TO SOFTWARE PACKAGES

3 INTRODUCTION TO SOFTWARE PACKAGES

3.1 INTRODUCTION TO CREO PARAMETRIC:

PTC’s developers created Creo Parametric as a sound foundation software that allows users the ability to expand deeper functionality with each component. As your products become more complex in its engineering, Creo offers expanded capabilities to meet your requirements. Every product isn’t made equal, and your 3D CAD solution shouldn’t be either.

Creo Parametric provides the broadest range of powerful yet flexible CAD 3D modeling software capabilities to accelerate the design of parts and assemblies. With Creo and its extensions, you’ll have access to technologies such as:

- Generative design
- Real-time simulation
- Additive manufacturing
- Augmented reality

Creo Parametric is what most people think of when they think of “Creo”. It's the standard solution for scalable 3D CAD design and is the direct descendent of Pro/Engineer Wildfire. Creo Direct is a separate application that allows users to make edits to 2D and 3D parametric designs in a direct modelling environment.

PART MODELLING:

In Part Modelling you can create apart from a conceptual sketch through solid feature-based modelling, as well as build and modify parts through direct and intuitive graphical manipulation.

The Part Modelling Help introduces you to the terminology, basic design concepts, and procedures that you must know before you start building a part. Part Modelling shows you how to draft a 2D conceptual layout, create precise geometry using basic geometric entities, and dimension and constrain your geometry. You can learn how to build a 3D parametric part from a 2D sketch by combining basic and advanced features, such as extrusions, sweeps, cuts, holes, slots, and rounds. Finally, the Part Modelling Help provides procedures for modifying part features and resolving failures.

PART MODELLING MODULES:

- Part Modelling
- Sketcher
- Feature Recognition Tool
- Creo Flexible Modelling

ASSEMBLY DESIGN:

The Assembly Help describes the processes you use to manage assemblies, their features, and their components. The Help for Assembly area is divided into four high-level sections covering general assembly functionality, configuring assemblies, using top-down design, and managing large assemblies.

Use the Assembly Help to learn how to create, manipulate, redefine, analyse, and reorient your assembly.

ASSEMBLY DESIGN MODULES:

- Assembly
- Creo Intelligent Fastener
- Assembly Process Planning

MODEL ANALYSIS:

Model analysis lets you perform four different types of model evaluation: behavioural modelling, model checking, tolerance analysis, and design editing. Behaviour modelling enables you to perform a wide variety of analyses on a model and incorporate the results into the model. Model checking runs transparently within Creo Parametric, making sure your model complies with company design standards and best modelling practices. Design editing provides you with a list of the major design steps and model parameters that you can use to create a program that changes the model according to new design specifications.

3.2 INTRODUCTION TO ANSYS:

ANSYS, Inc. is an American Computer-aided engineering software developer headquartered south of Pittsburgh in Cecil Township, Pennsylvania, and United States. ANSYS publishes engineering analysis software across a range of disciplines including finite element analysis, analysis, computational, explicit and implicit methods, and heat transfer.

STRUCTURAL ANALYSIS:

ANSYS AUTODYN:

ANSYS Autodyn is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions.

ANSYS MECHANICAL:

ANSYS Mechanical is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behaviour, and supports material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo-electric analysis.

FLUID DYNAMICS:

ANSYS Fluent, CFD, CFX:

ANSYS Fluent, CFD, CFX, and related software are Computational Fluid Dynamics software tools used by engineers for design and analysis. These tools can simulate fluid flows in a virtual environment — for example, the fluid dynamics of ship hulls; gas turbine engines (including the compressors, combustion chamber, turbines and afterburners); aircraft aerodynamics; pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners, etc.

3.3 INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS (CFD):

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy. Computational Fluid Dynamics (CFD) is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers. Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows.

By means of

- Mathematical modelling (partial differential equations)
- Numerical methods (discretization and solution techniques)
- Software tools (solvers, pre- and post-processing utilities)

CFD enables scientists and engineers to perform ‘numerical experiments’ (i.e., computer simulations) in a ‘virtual flow laboratory’.

ADVANTAGES OF CFD:

Numerical simulations of fluid flow (will) enable

- Architects to design comfortable and safe living environments
- Designers of vehicles to improve the aerodynamic characteristics
- Chemical engineers to maximize the yield from their equipment
- Petroleum engineers to devise optimal oil recovery strategies
- Surgeons to cure arterial diseases (computational hemo dynamics)
- Meteorologists to forecast the weather and warn of natural disasters
- Safety experts to reduce health risks from radiation and other hazards
- Military organizations to develop weapons and estimate the damage
- CFD practitioners to make big bucks by selling colourful pictures

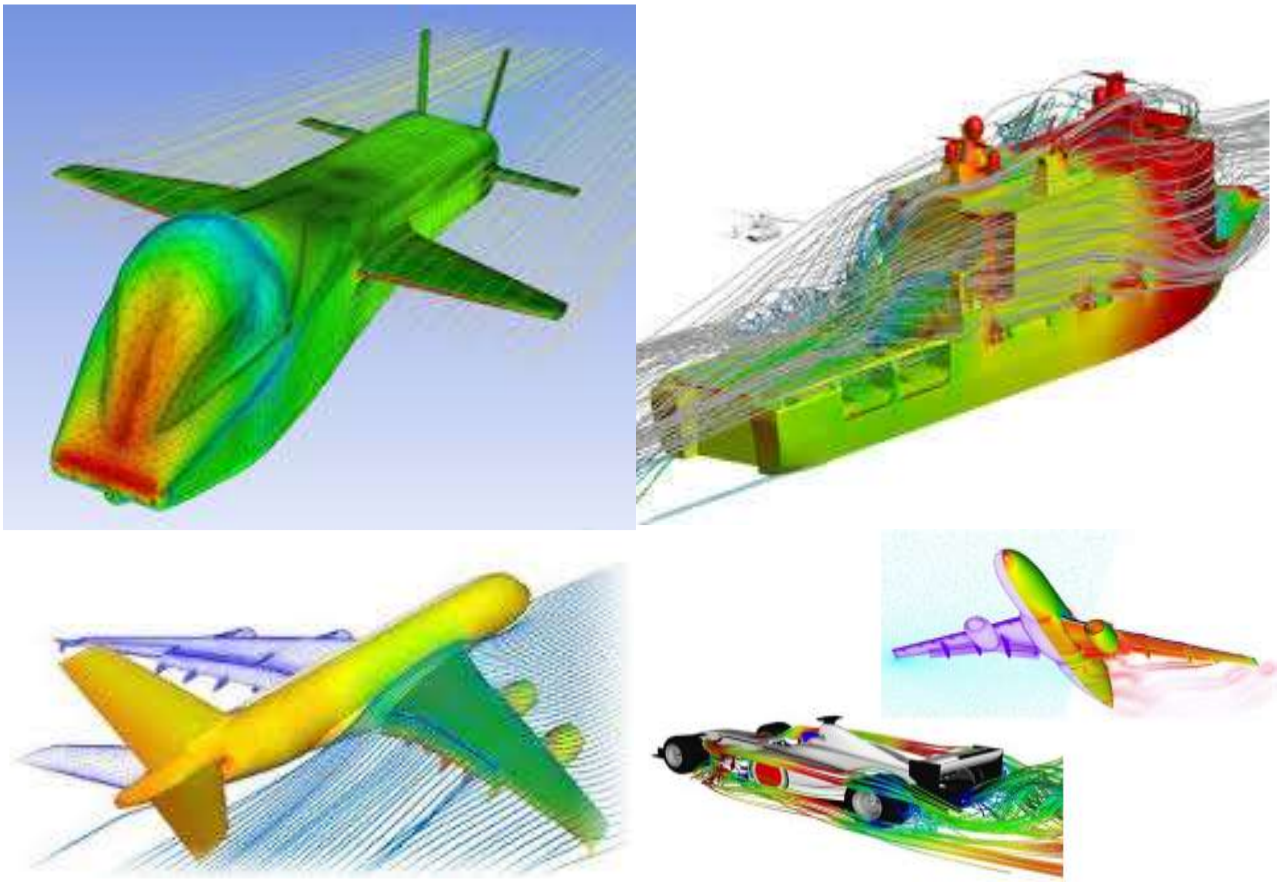


Fig 1.1.1 CFD Model applications and their simulations

EXPERIMENT VS SIMULATION:

CFD gives an insight into flow patterns that are difficult, expensive, or impossible to study using traditional (experimental) techniques.

Table 3.1 Comparison of Experiment and Simulation

Experiment	Simulation
Quantitative description of flow phenomena using measurements • For one quantity at a time • At a limited number of points and time instants • For a laboratory-scale model • For a limited range of problems and operating conditions Error sources: measurement errors flow disturbances by the probes	Quantitative prediction of flow phenomena using CFD software • For all desired quantities • with high resolution in space and time • For the actual flow domain • For any problem and realistic operating conditions Error sources: modelling, discretization implementation

As a rule, CFD does not replace the measurements completely but the amount of experimentation and the overall cost can be significantly reduced.

Table 3.2 General Comparisons

Experiments	Simulations
• Expensive • Slow • Sequential • Single purpose	• Cheaper • Faster • Parallel • Multiple purpose

Equipment and personnel are difficult to transport CFD software is portable, easy to use and modify

The results of a CFD simulation are never 100% reliable because

- The input data may involve too much guessing or imprecision
- The mathematical model of the problem at hand may be inadequate

CFD ANALYSIS PROCESS:

1. Problem statement	information about the flow
2. Mathematical model	IBVP = PDE + IC + BC
3. Mesh generation	nodes/cells, time instants
4. Space discretization	coupled ODE/DAE systems
5. Time discretization	algebraic system Ax = b
6. Iterative solver	discrete function values
7. CFD software	implementation, debugging
8. Simulation run	parameters, stopping criteria
9. Postprocessing	visualization, analysis of data
10. Verification model	validation / adjustment

MATHEMATICAL MODEL:

- Choose a suitable flow model (viewpoint) and reference frame.
- Identify the forces which cause and influence the fluid motion.
- Define the computational domain in which to solve the problem.
- Formulate conservation laws for the mass, momentum, and energy.
- Simplify the governing equations to reduce the computational effort:
  - Use available information about the prevailing flow regime
  - Check for symmetries and predominant flow directions (1D/2D)
  - Neglect the terms which have little or no influence on the results
  - Model the effect of small-scale fluctuations that cannot be captured
  - incorporate a priori knowledge (measurement data, CFD results)
- Add constitutive relations and specify initial/boundary conditions.

DISCRETISIZATION PROCESS:

The PDE system is transformed into a set of algebraic equations

- Mesh generation (decomposition into cells/elements)
  - Structured or unstructured, triangular or quadrilateral?
  - CAD tools + grid generators (Delaunay, advancing front)
  - mesh size, adaptive refinement in ‘interesting’ flow regions
- Space discretization (approximation of spatial derivatives)
  - Finite differences/volumes/elements

  - High- vs. low-order approximations
- Time discretization (approximation of temporal derivatives)
  - Explicit vs. implicit schemes, stability constraints
  - Local time-stepping, adaptive time step control

CFD SIMULATIONS:

The computing times for a flow simulation depend on

- The choice of numerical algorithms and data structures
- Linear algebra tools, stopping criteria for iterative solvers
- Discretization parameters (mesh quality, mesh size, time step)
- Cost per time step and convergence rates for outer iterations
- Programming language (most CFD codes are written in FORTRAN)
- Many other things (hardware, vectorization, parallelization etc.)

The quality of simulation results depends on

- The mathematical model and underlying assumptions
- Approximation type, stability of the numerical scheme
- Mesh, time step, error indicators, stopping criteria

POST PROCESSING AND ANALYSIS:

Post processing of the simulation results is performed in order to

Extract the desired information from the computed flow field

- Calculation of derived quantities (stream function, vortices)
- Calculation of integral parameters (lift, drag, total mass)
- Visualization (representation of numbers as images)
  - 1D data: function values connected by straight lines
  - 2D data: streamlines, contour levels, colour diagrams
  - 3D data: cutlines, cut planes, iso surfaces, iso volumes
  - Arrow plots, particle tracing, animations
- Systematic data analysis by means of statistical tools
- Debugging, verification, and validation of the CFD model

CHAPTER-4

MODELING PROCEDURE

4 MODELING PROCEDURE

4.1 GENERATION OF RADIATOR SHAPE USING CREO PARAMETRIC:

- Save the Radiator shape in ‘igs’ or ‘stp’ format.
- Start the sketch by drawing a end fin with the required dimensions and then mirror it to the end after creating the holes to insert the tubes.
- Now draw a tube with the required dimension and the tubes are replicated.
- Addition fins are added and the inlet and outlet is extended.

4.2 STEPS INVOLVED IN THE ANALYSIS AND SOLUTION:

4.2.1 ANSYS:

- Open the ANSYS workbench, open the tool box and then select the fluid flow (fluent) from the analysis system .
- Now import the ‘igs’ or ‘stp’ file into the ANSYS work bench.

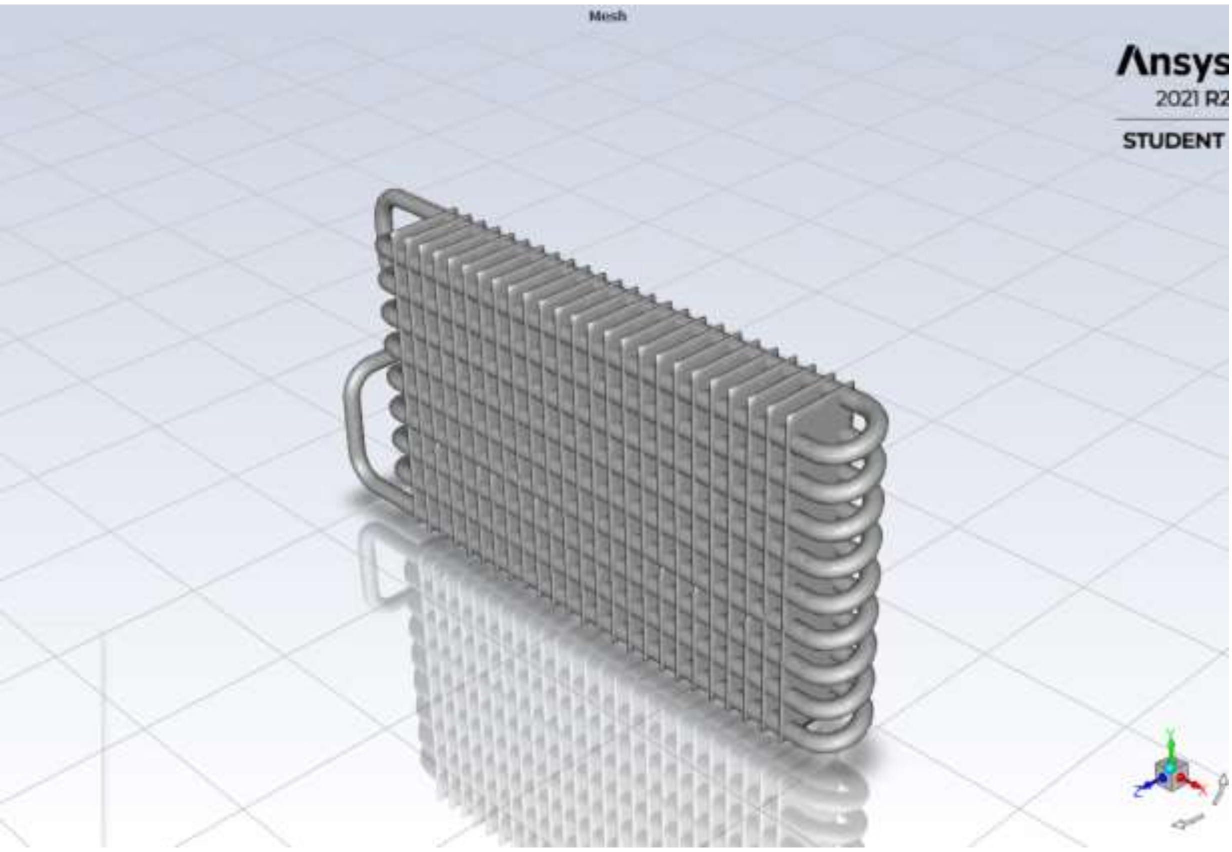


Fig 4.2.1 Car Radiator Design in Ansys

4.2.2 GEOMETRY:

Table 4.1 Dimensions table of the car radiator

	Units (mm)
Length of Fin plate	210
Width of Fin plate	44
Length of the tube	292
Diameter of each tube	5
Total no of Tubes	16
Inlet Diameter of Radiator tube	10
Outlet Diameter of Radiator tube	10

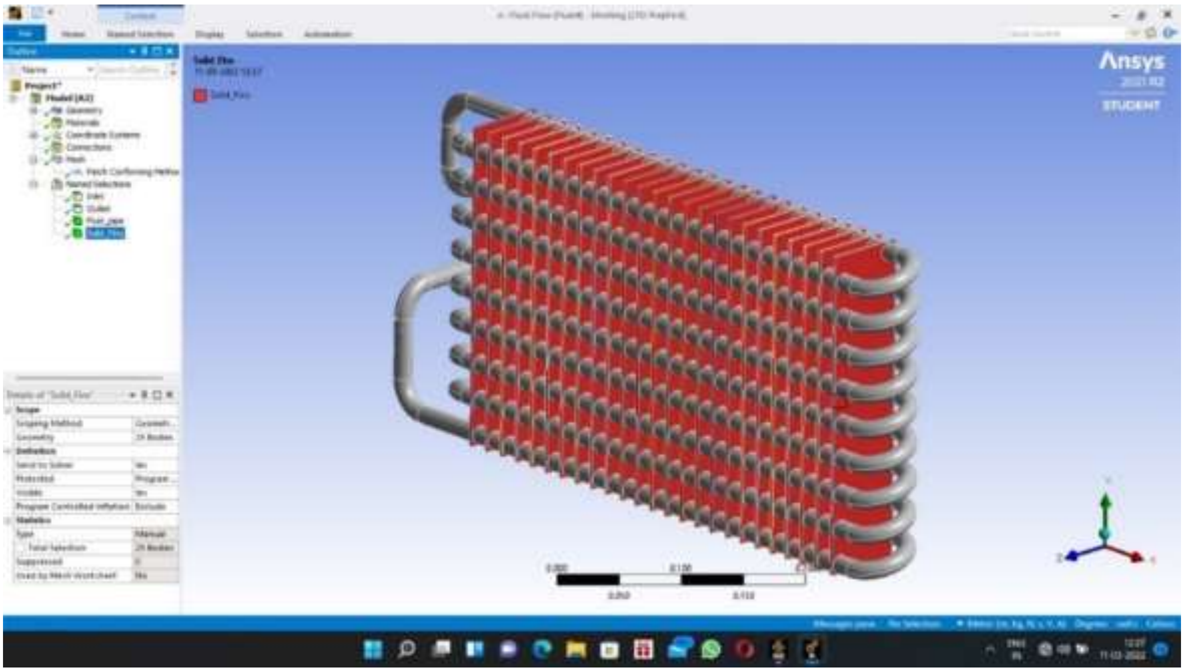


Fig 4.2.2 Car Radiator Fins

4.2.3 MESH:

- After the required profile is generated, enter the required data like body size and face size. Select the entities required for meshing.
- Now enter the number of divisions and element size.
- Now by using the inflation select the bodies and number of layers.

- The above-mentioned meshing structure has been accomplished by considering mapped meshing and applying various edge sizing criteria to carry out the task.
- The generated mesh have number of nodes is 58,882 and the number of elements is 1,97,973.

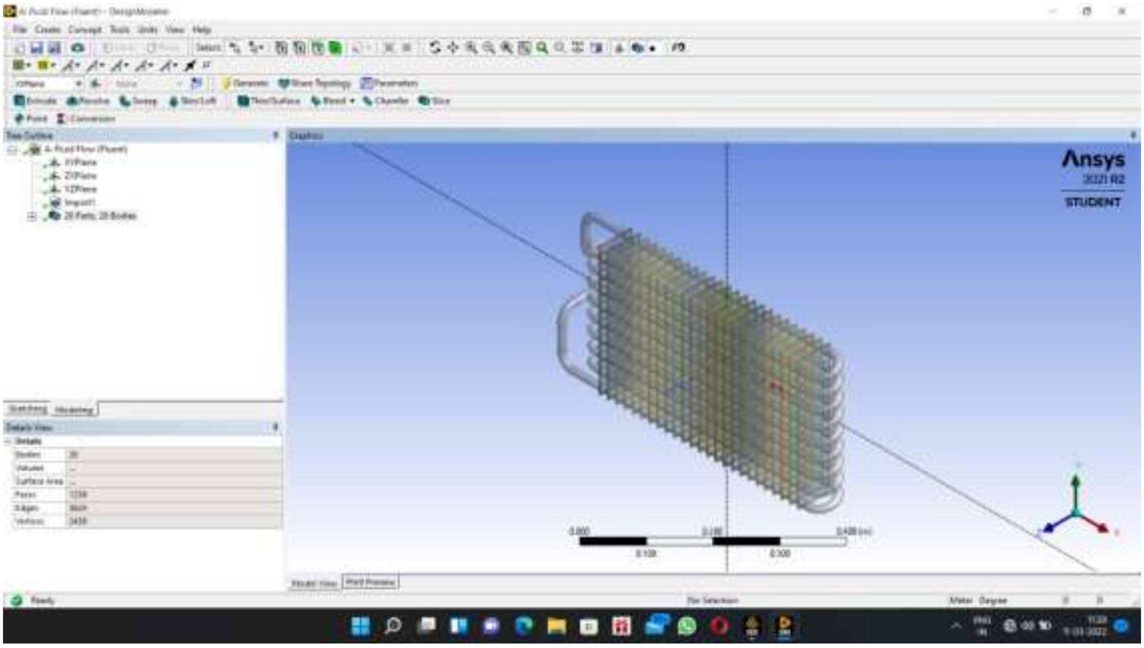


Fig 4.2.3 Values of Edges, faces of a Car Radiator Design in ANSYS

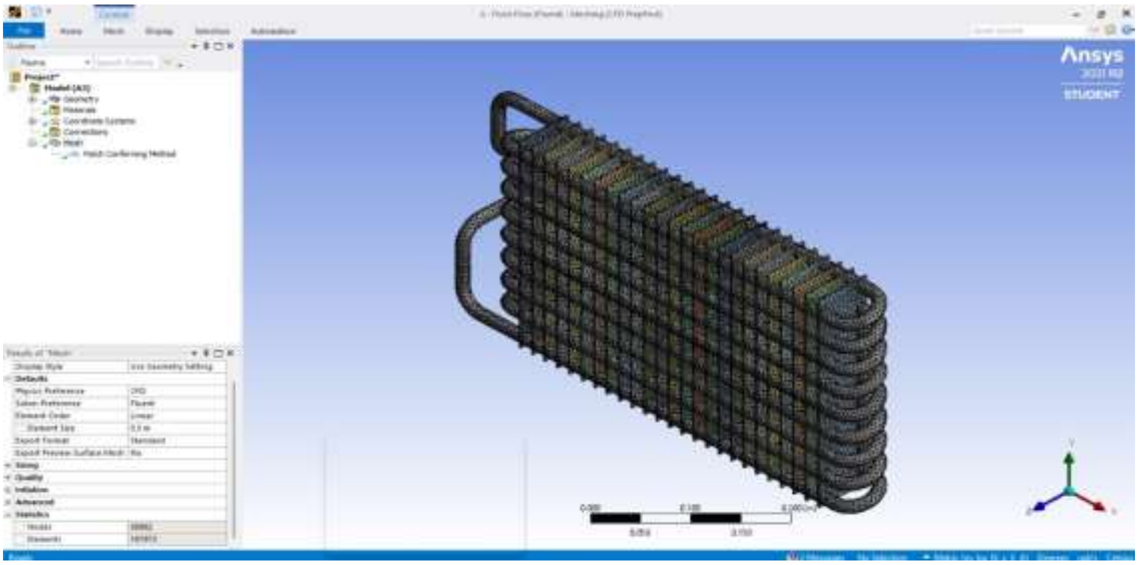


Fig 4.2.4 Values of nodes and elements of a Car Radiator Design in ANSYS

4.2.4 SETUP:

- Enter the boundary conditions and type of fluid flow.
- The atmospheric temperature is taken to be 373K and is fixed throughout the experiment.
- Select the type of material and enter the reference values.

4.2.5 SOLUTION:

- Enter the number of iterations value.

4.2.6 RESULTS:

- The Temperature contours are obtained for different velocities.

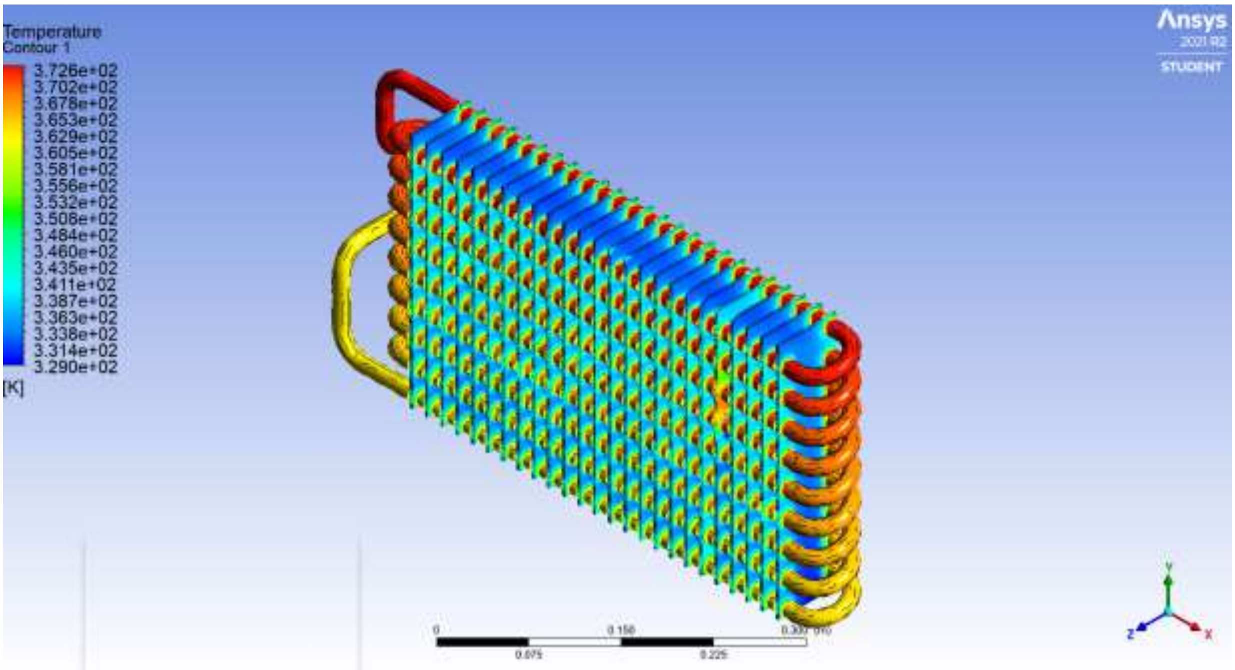


Fig 4.2.5 Temperature contour of a Car Radiator Design in ANSYS

4.3 CALCULATION:

Coolant (C): Water

Velocity (V): 1m/s

Diameter of the cross section (d): 10mm = 0.01m

Area (A): $\pi d^2/4 = (3.14*(0.01^2))/4$ = 7.85*10^-5m^2

Discharge (Q): Q=AV m^3/sec = (7.85*10^-5) * 1 = 7.85*10^-5 m^3/sec=7.85*10^-2 lit/sec

Density of water (ρ): kg/m^3 = 997 kg/m^3

Mass Flow rate (M): ρQ = (7.85*10^-5) * 997 = 0.078 kg/s

CHAPTER 5

RESULTS AND DISCUSSIONS

5 RESULTS AND DISCUSSIONS

FOR ALUMINIUM:

5.1 THE TEMPERATURE CONTOURS OF THE CAR RADIATOR:

Table 5.1 Table of Outlet temperature corresponding to the inlet velocity of the coolant for aluminium

VELOCITY (m/s)	OUTLET TEMPERATURE (K)
1	366.834
2	369.318
3	370.360
4	370.962
5	371.312
6	371.582
7	371.777
8	371.914
9	372.033
10	372.101

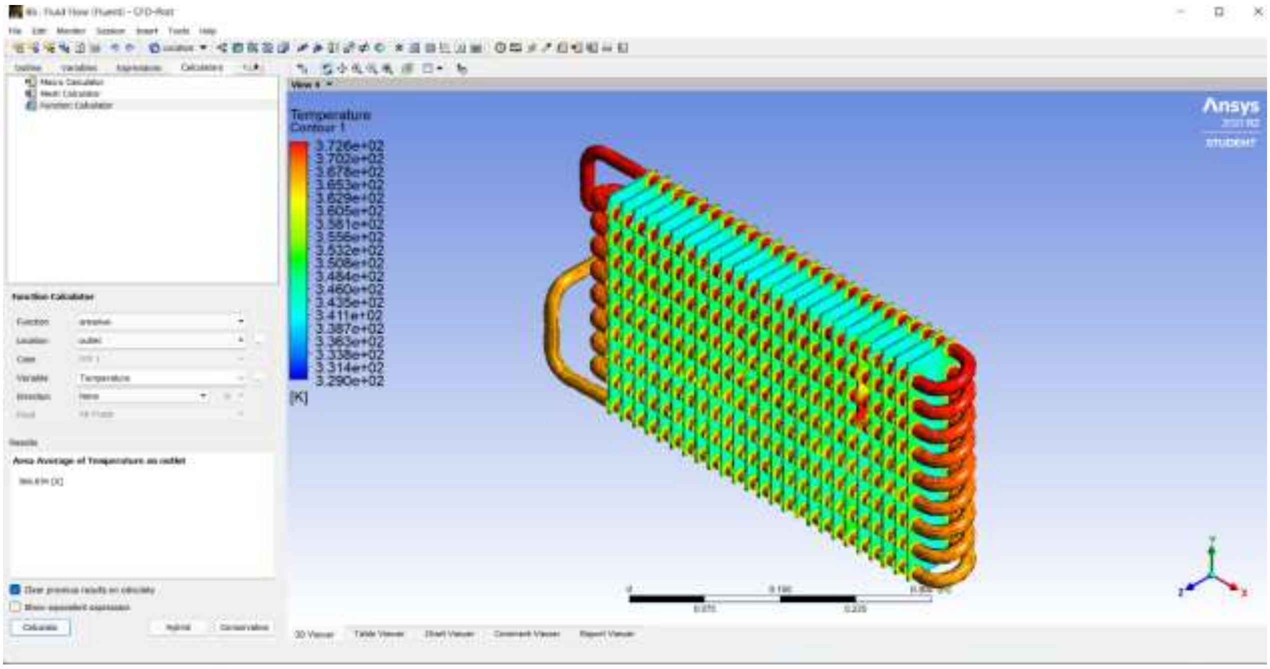


Fig 5.1.1 Temperature contour for the radiator at 1m/s

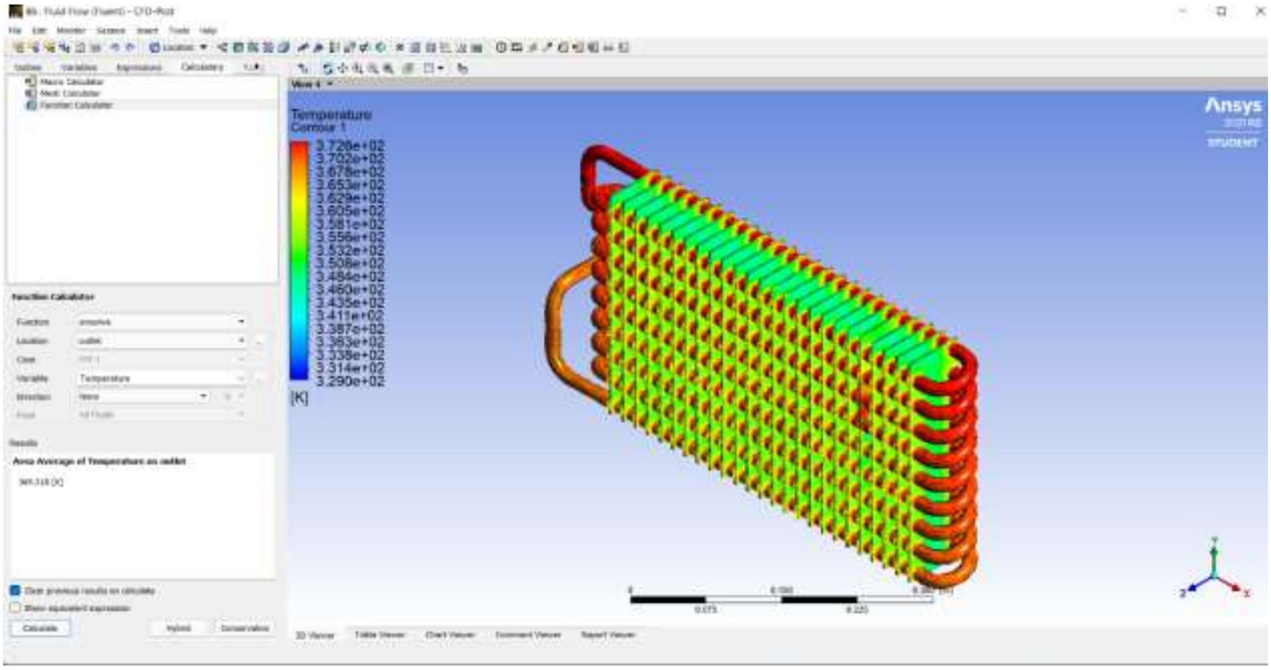


Fig 5.1.2 Temperature contour for the radiator at 2m/s

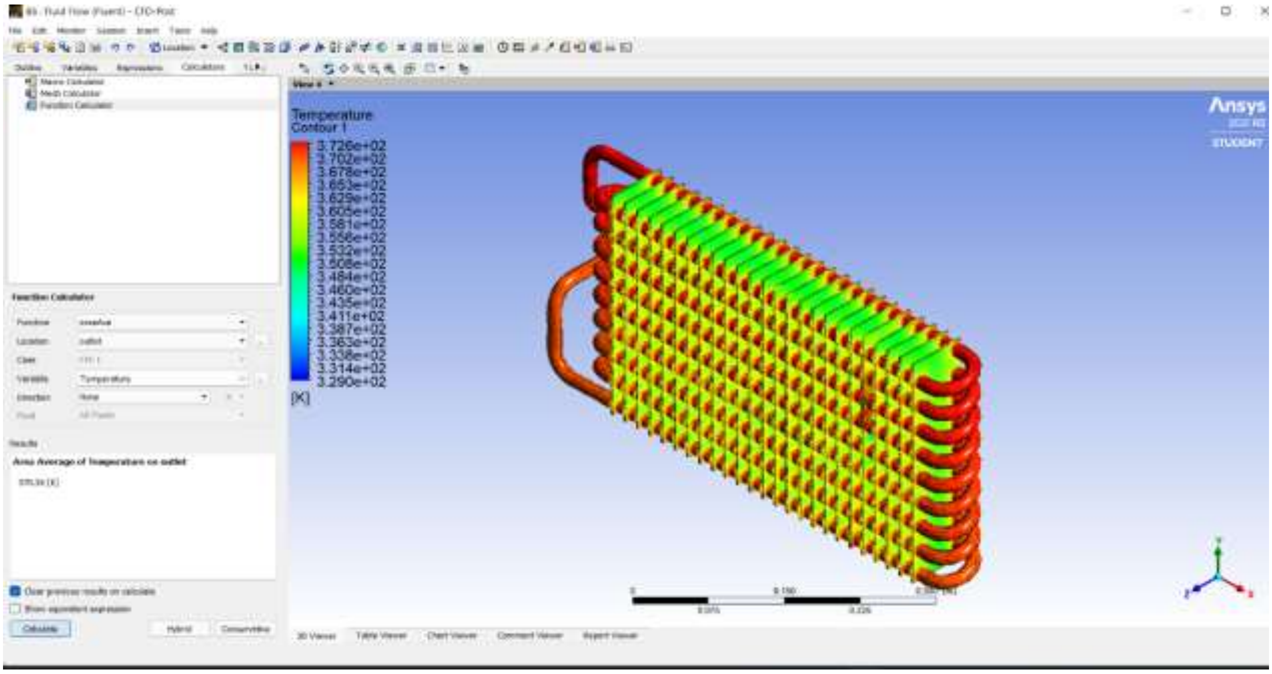


Fig 5.1.3 Temperature contour for the radiator at 3m/s

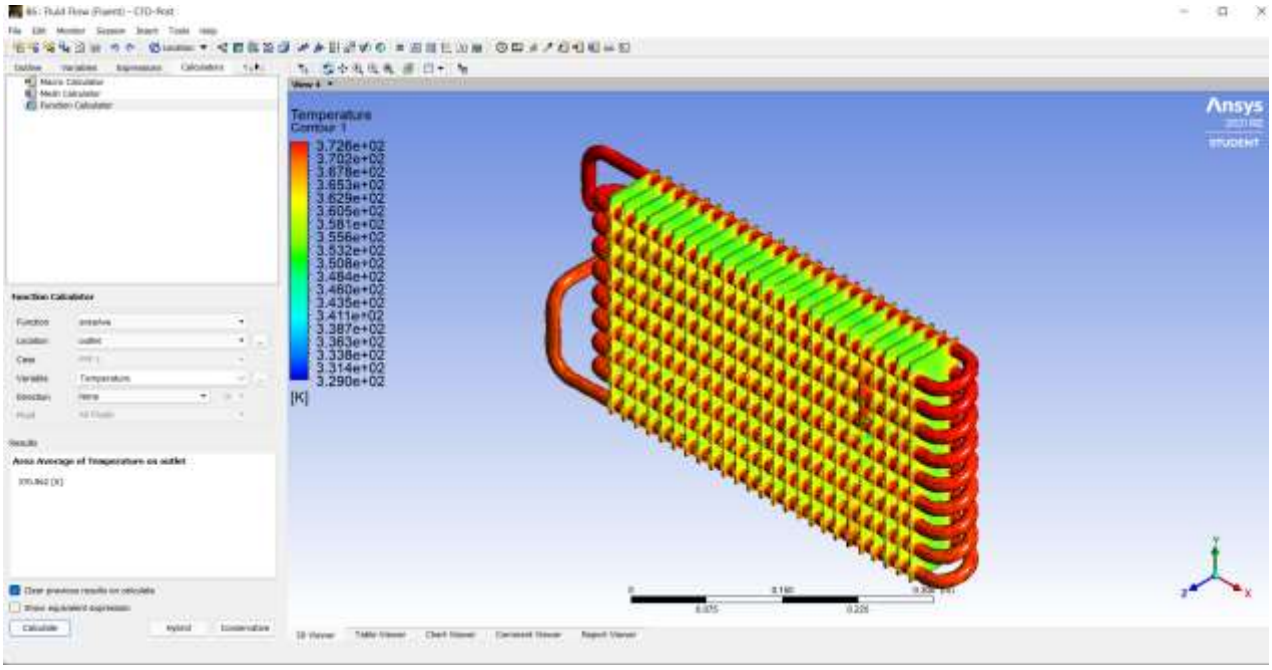


Fig 5.1.4 Temperature contour for the radiator at 4m/s

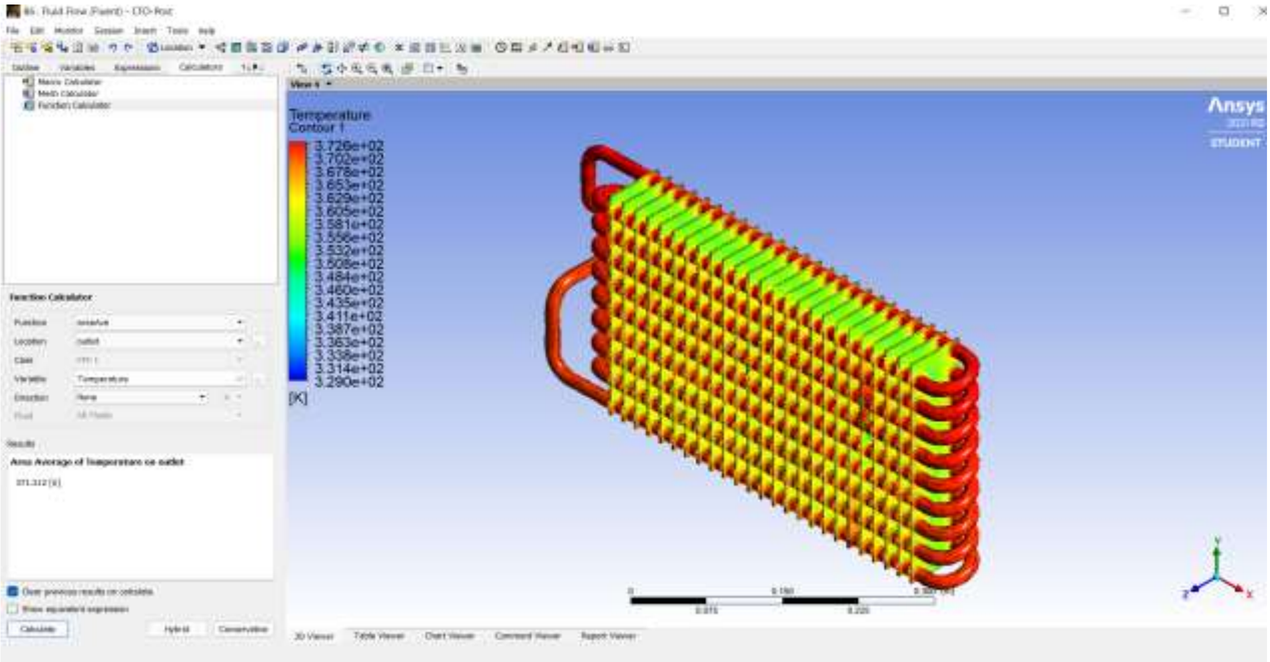


Fig 5.1.5 Temperature contour for the radiator at 5m/s

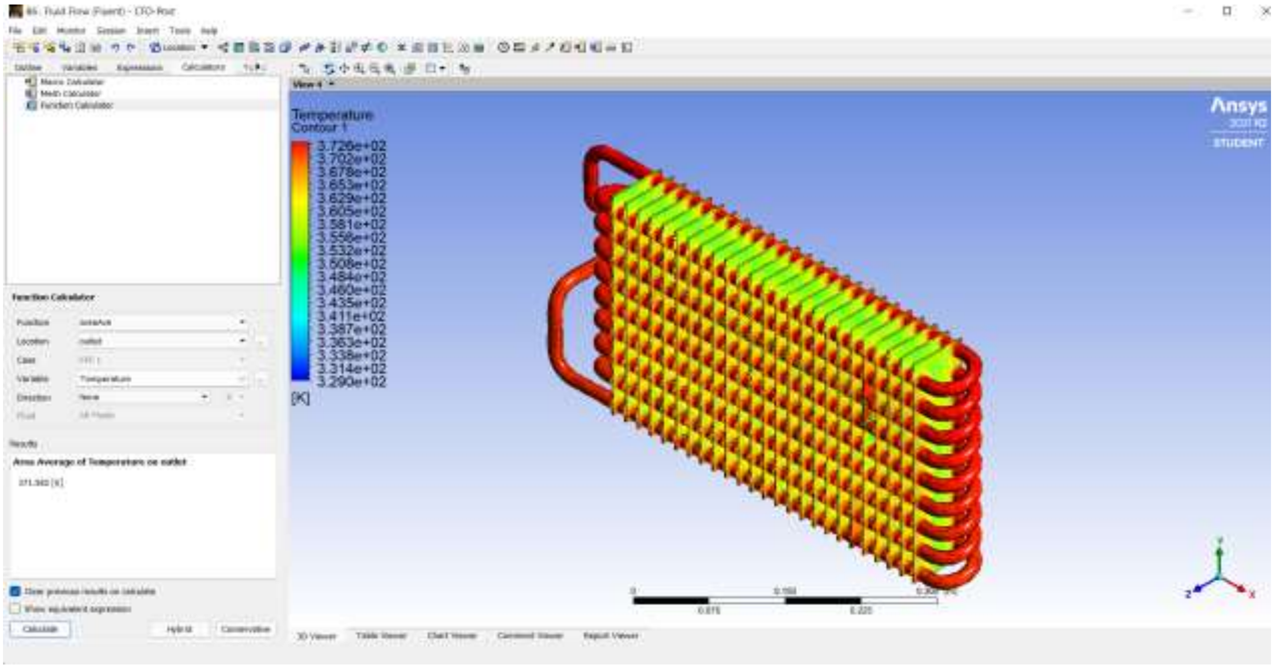


Fig 5.1.6 Temperature contour for the radiator at 6m/s

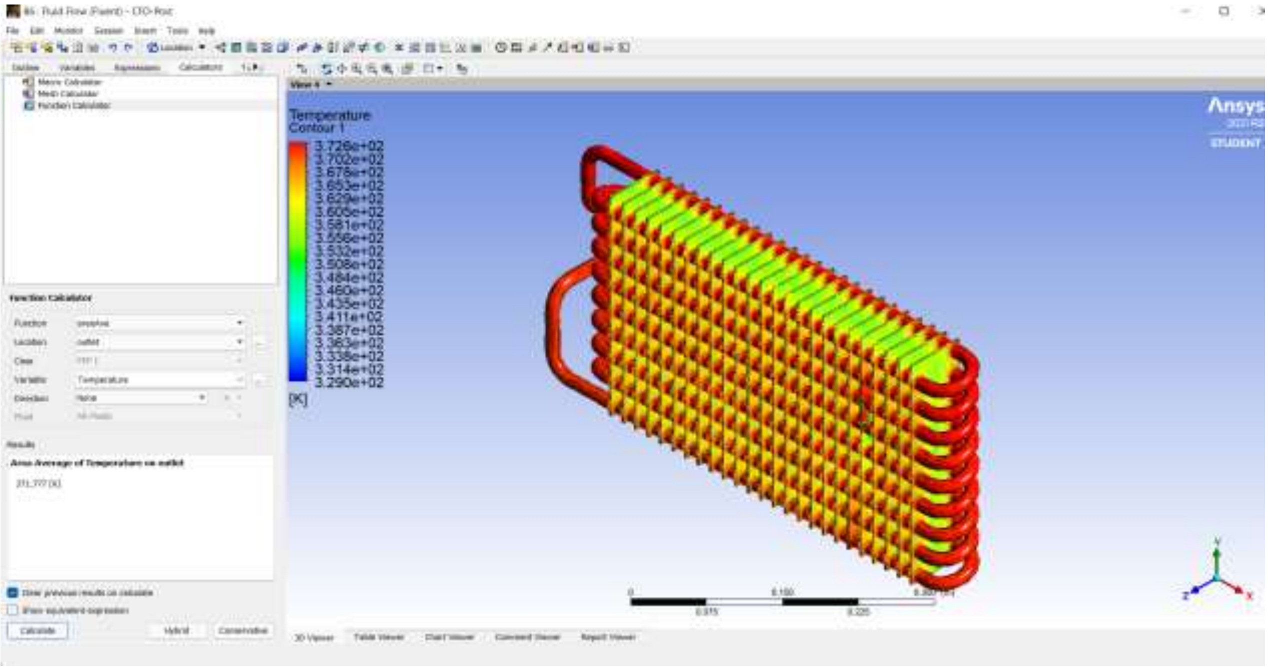


Fig 5.1.7 Temperature contour for the radiator at 7m/s

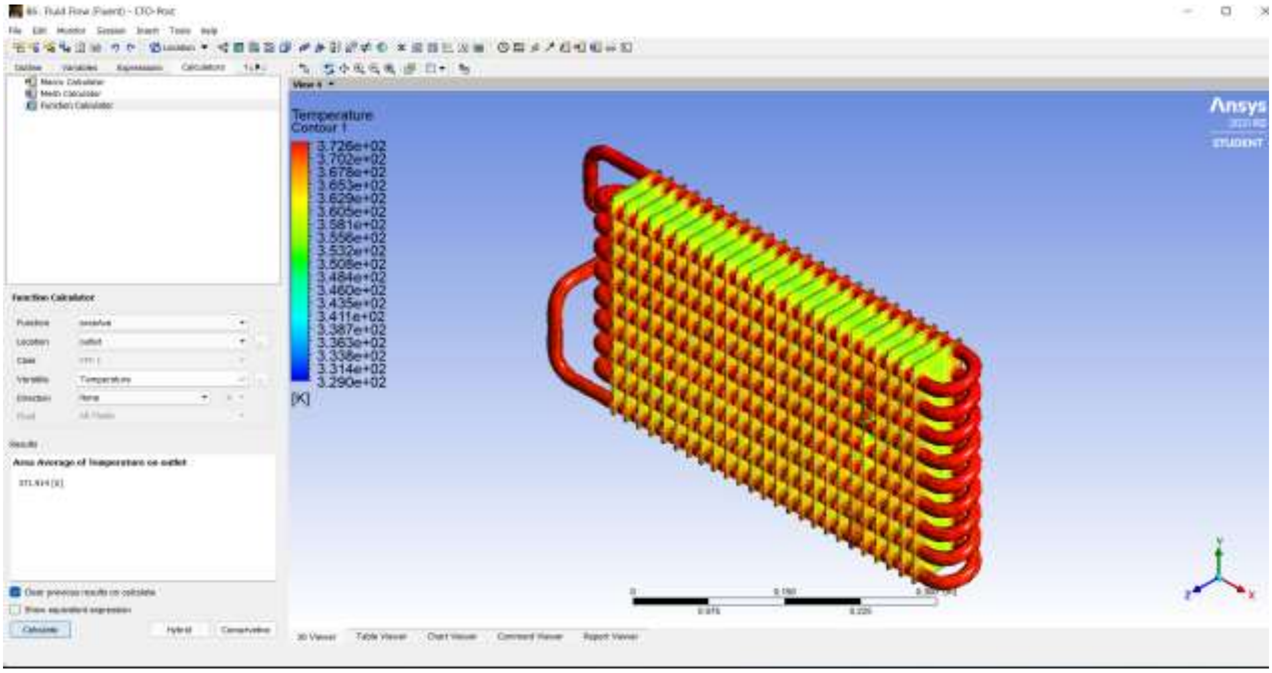


Fig 5.1.8 Temperature contour for the radiator at 8m/s

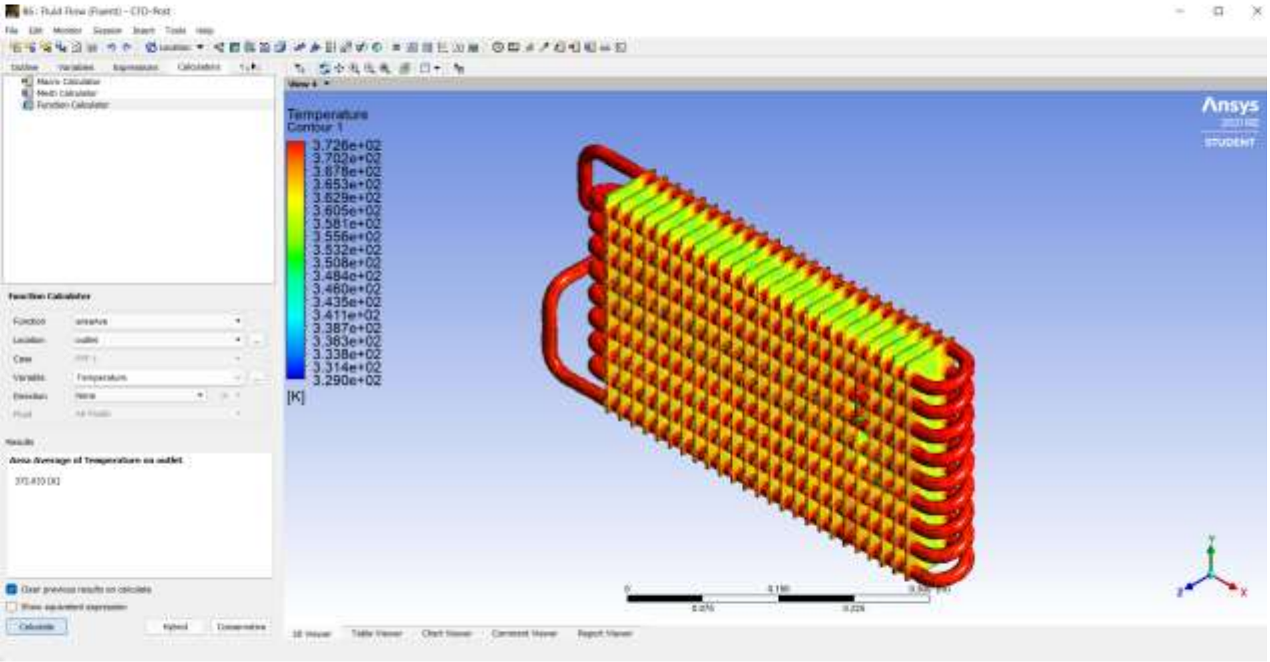


Fig 5.1.9 Temperature contour for the radiator at 9m/s

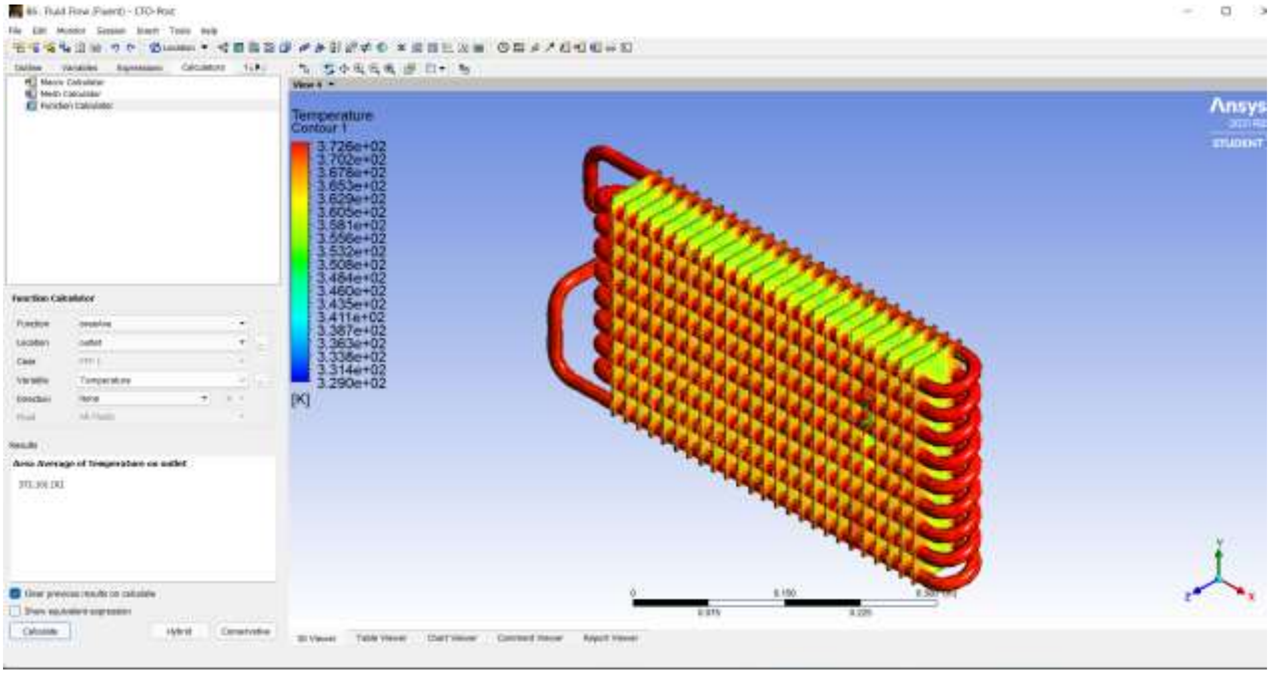


Fig 5.1.10 Temperature contour for the radiator at 10m/s

5.2 SCALED RESIDUALS:

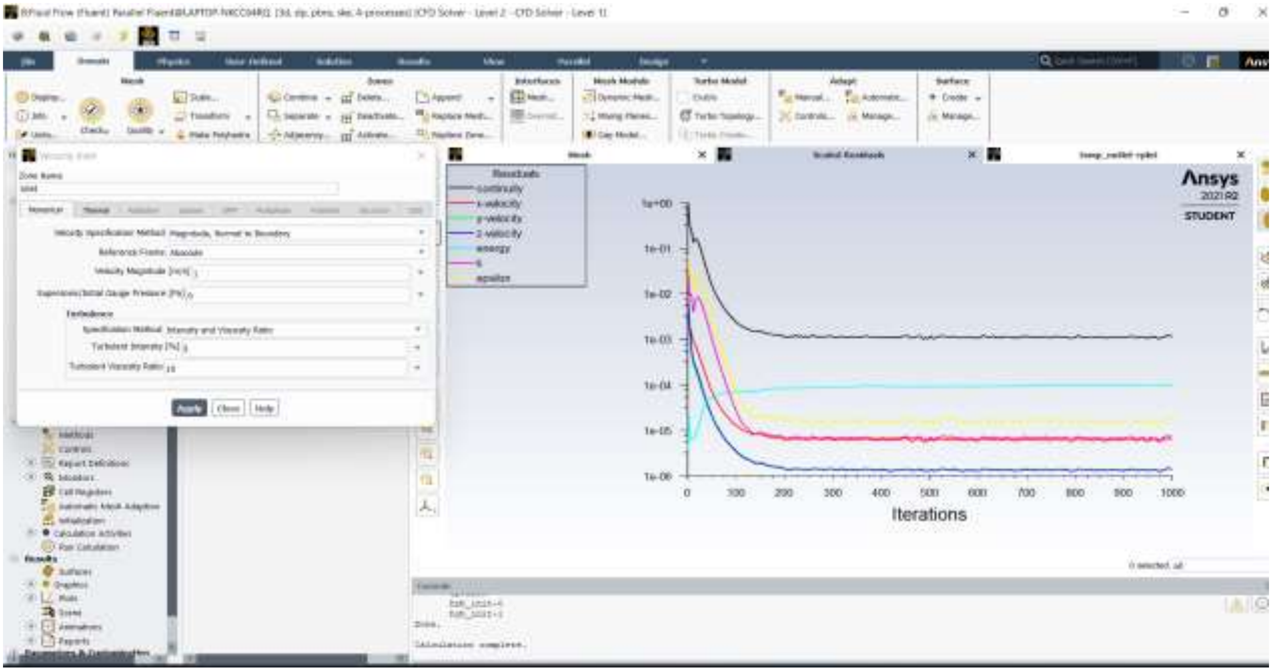


Fig 5.2.1 Scaled residuals for the radiator at 1m/s

Fig 5.2.2 Scaled residuals for the radiator at 2m/s

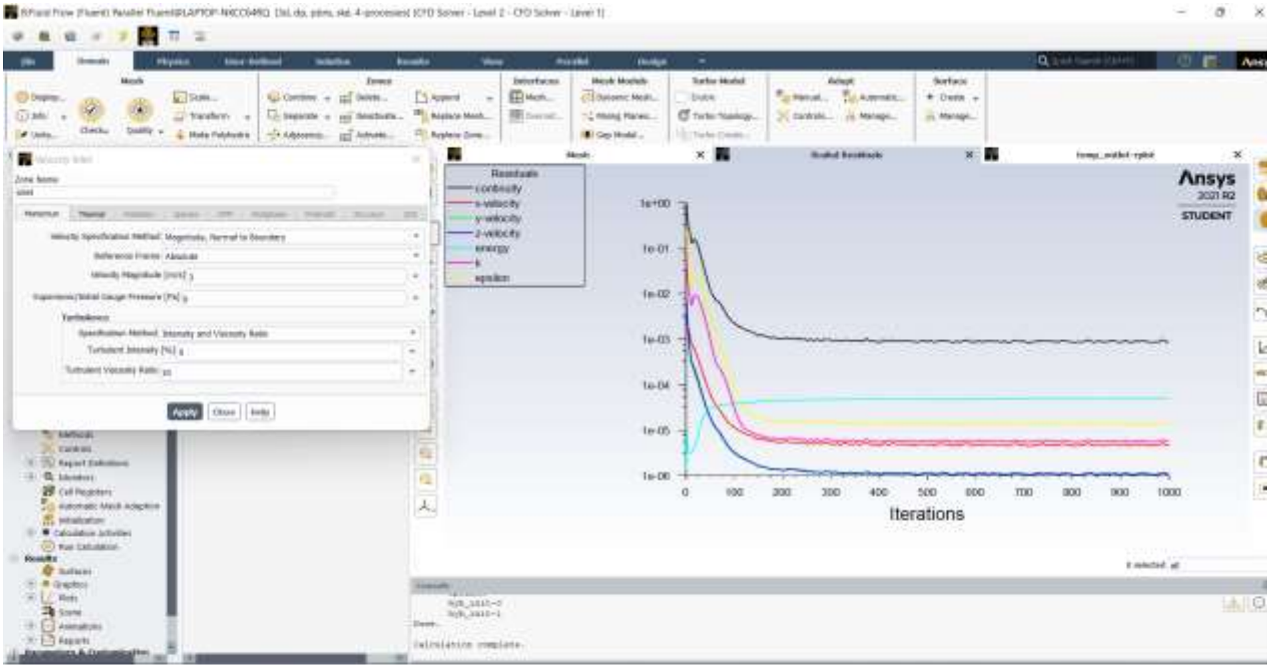


Fig 5.2.3 Scaled residuals for the radiator at 3m/s

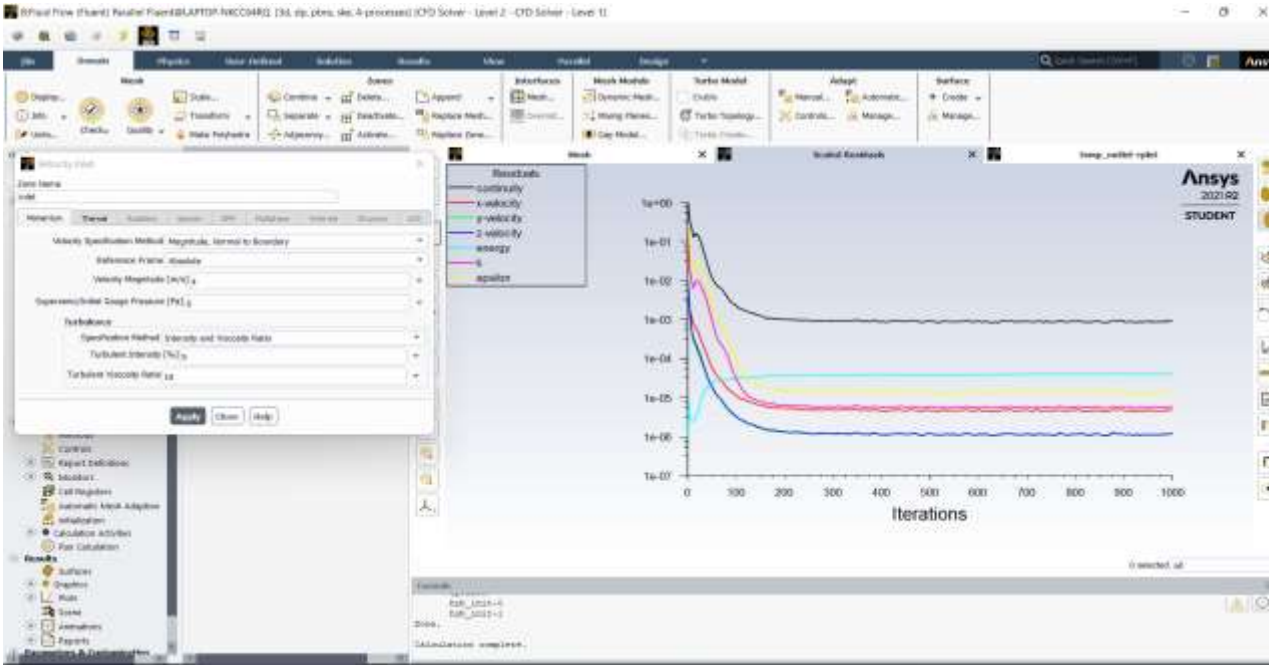
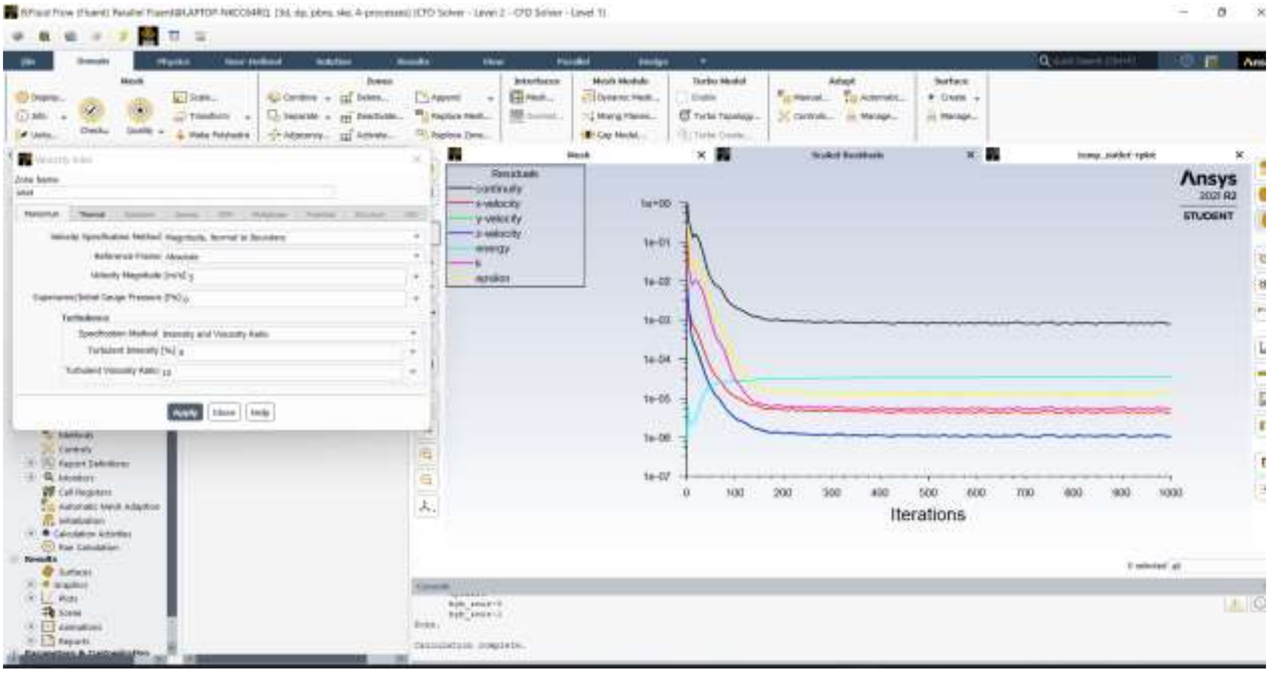


Fig 5.2.4 Scaled residuals for the radiator at 4m/s



Velocity Inlet

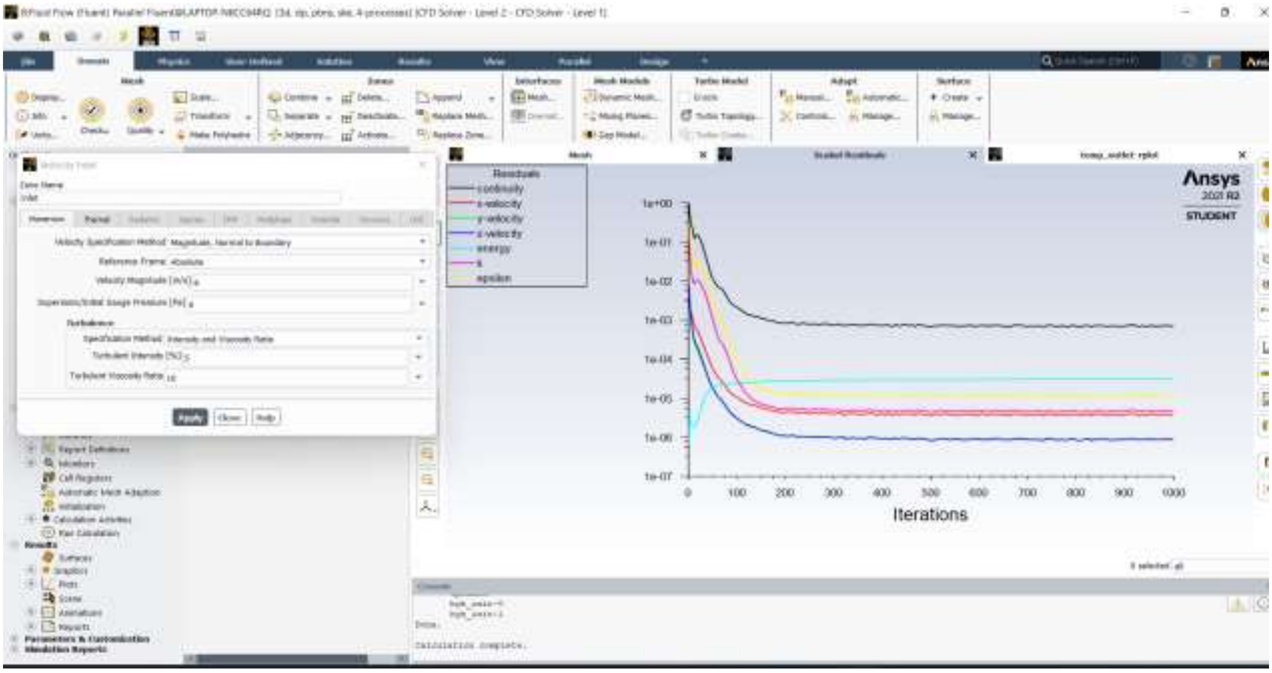
Zone Name
inlet

Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude [m/s]: 5
Supersonic/Initial Gauge Pressure [Pa]: 0
Turbulence
Specification Method: Intensity and Viscosity Ratio
Turbulent Intensity [%]: [illegible]
Turbulent Viscosity Ratio: 10

Done.

Calculation complete.

Fig 5.2.5 Scaled residuals for the radiator at 5m/s



Velocity Inlet

Zone Name
inlet

Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude [m/s]: 6
Supersonic/Initial Gauge Pressure [Pa]: 0
Turbulence
Specification Method: Intensity and Viscosity Ratio
Turbulent Intensity [%]: 5
Turbulent Viscosity Ratio: 10

Done.

Calculation complete.

Fig 5.2.6 Scaled residuals for the radiator at 6m/s

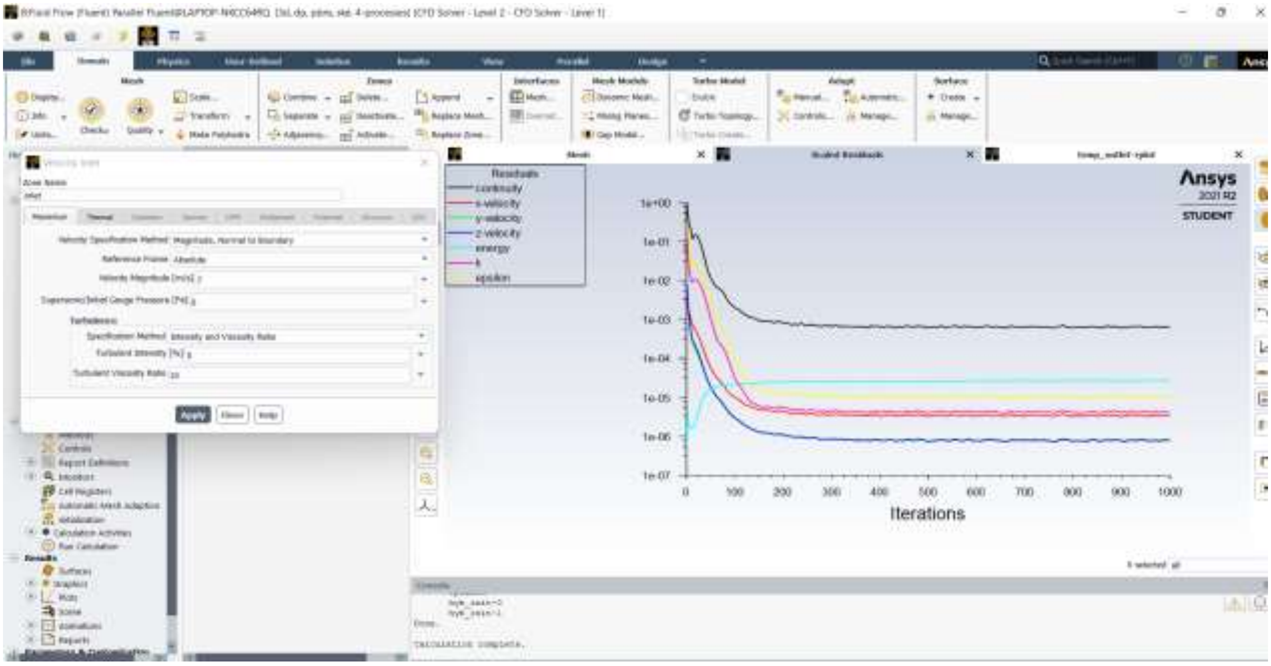


Fig 5.2.7 Scaled residuals for the radiator at 7m/s

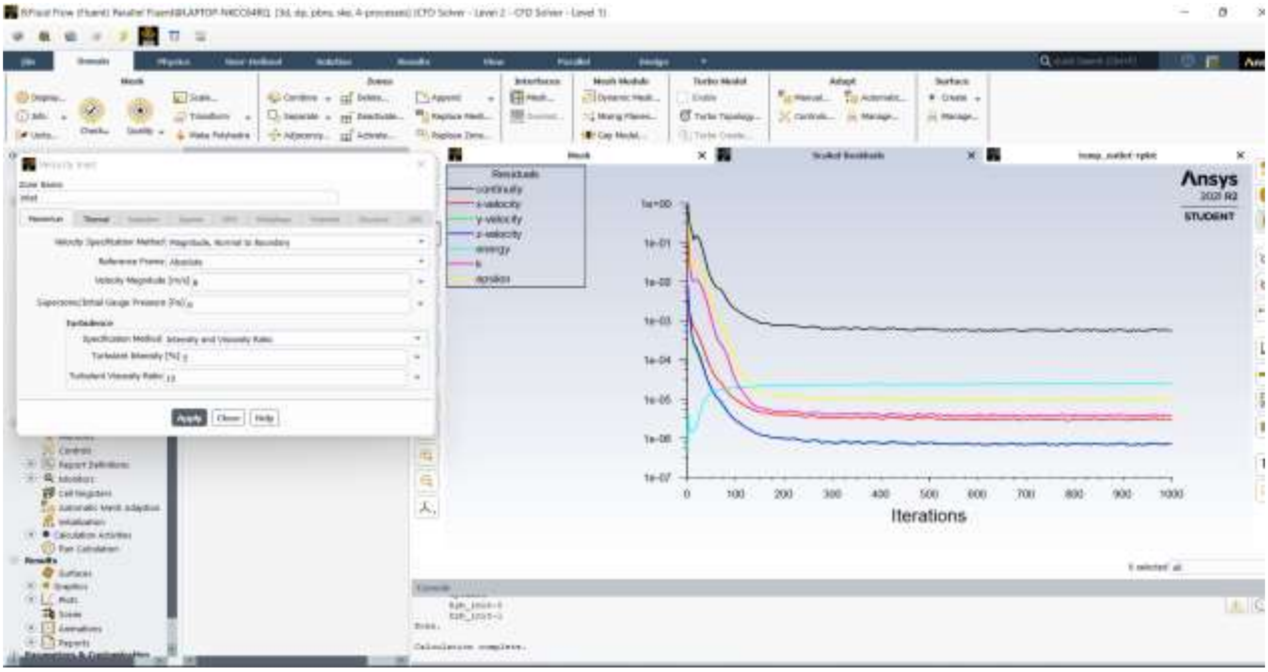
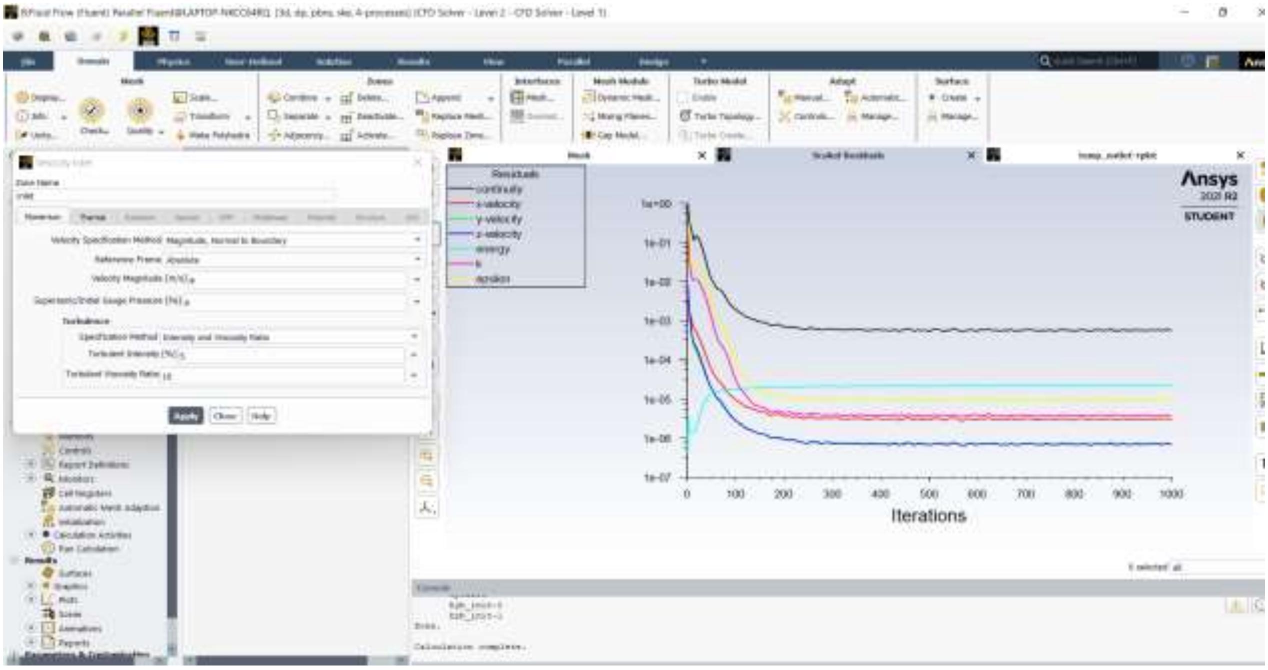


Fig 5.2.8 Scaled residuals for the radiator at 8m/s



Velocity Inlet

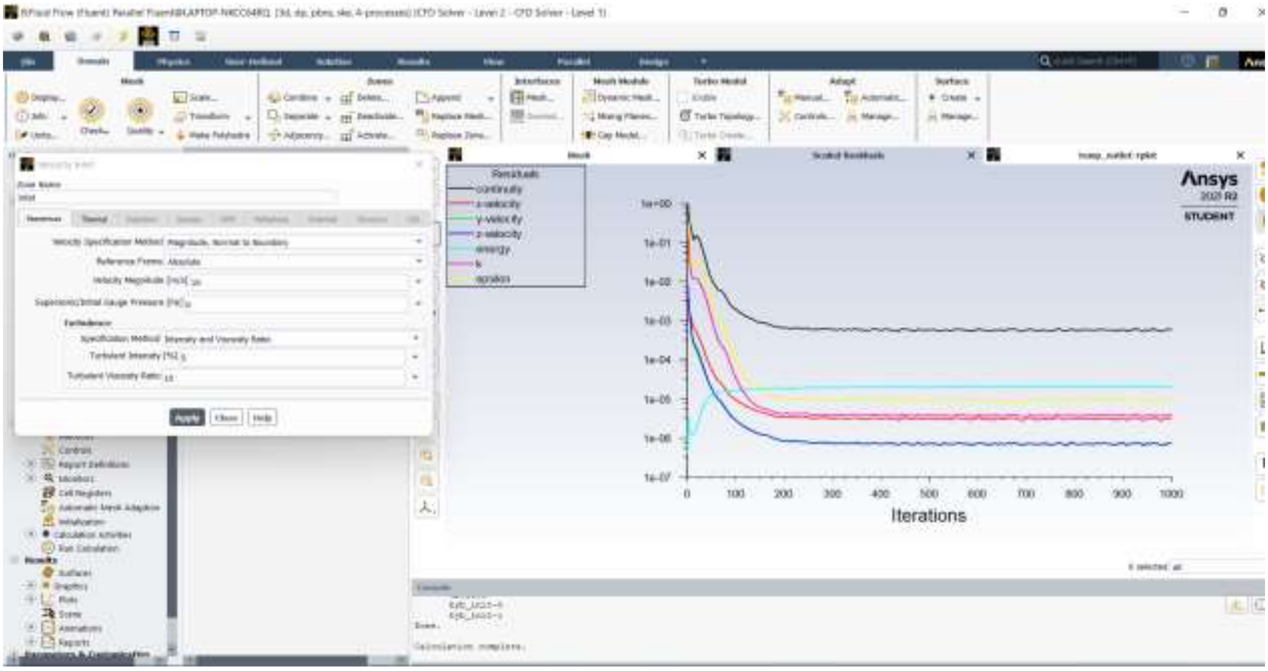
Zone Name
inlet

Momentum

Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude [m/s]: 9
Supersonic/Initial Gauge Pressure [Pa]: 0
Turbulence
Specification Method: Intensity and Viscosity Ratio
Turbulent Intensity [%]: 5
Turbulent Viscosity Ratio: 10

Calculation complete.

Fig 5.2.9 Scaled residuals for the radiator at 9m/s



Velocity Inlet

Zone Name
inlet

Momentum

Velocity Specification Method: Magnitude, Normal to Boundary
Reference Frame: Absolute
Velocity Magnitude [m/s]: 10
Supersonic/Initial Gauge Pressure [Pa]: 0
Turbulence
Specification Method: Intensity and Viscosity Ratio
Turbulent Intensity [%]: 5
Turbulent Viscosity Ratio: 10

Calculation complete.

Fig 5.2.10 Scaled residuals for the radiator at 10m/s

5.3 TEMPERATURE OUTLET PLOT:

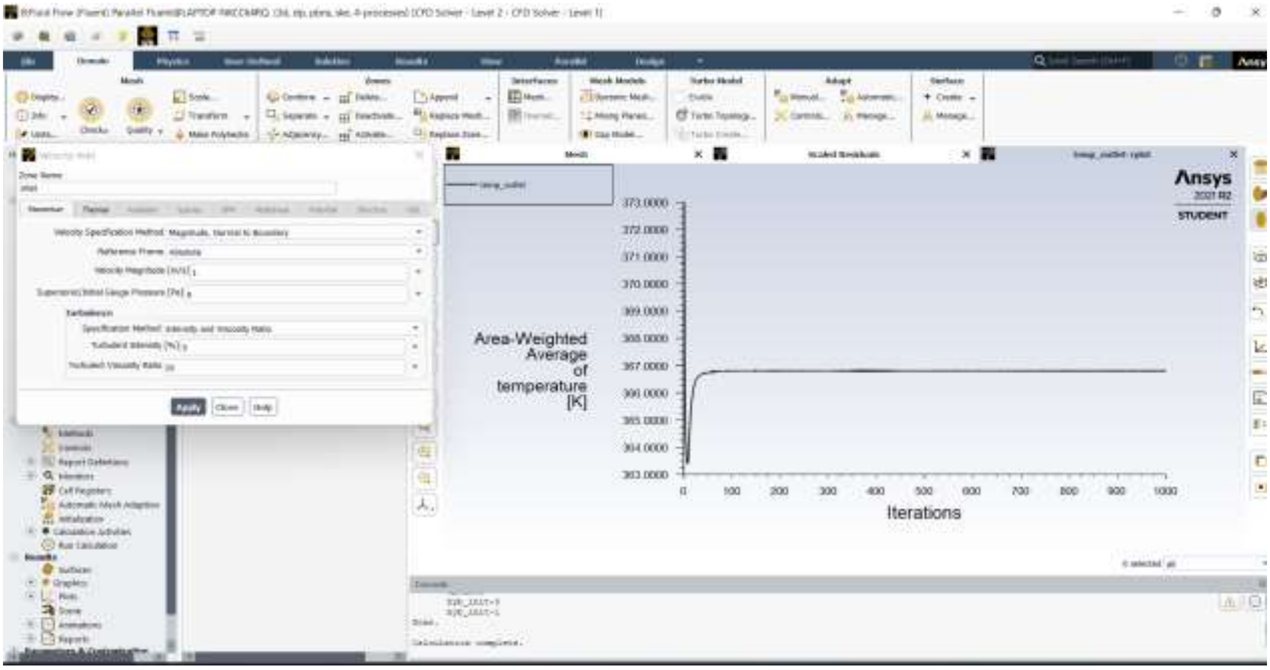


Fig 5.3.1 Temperature outlet for the radiator at 1m/s

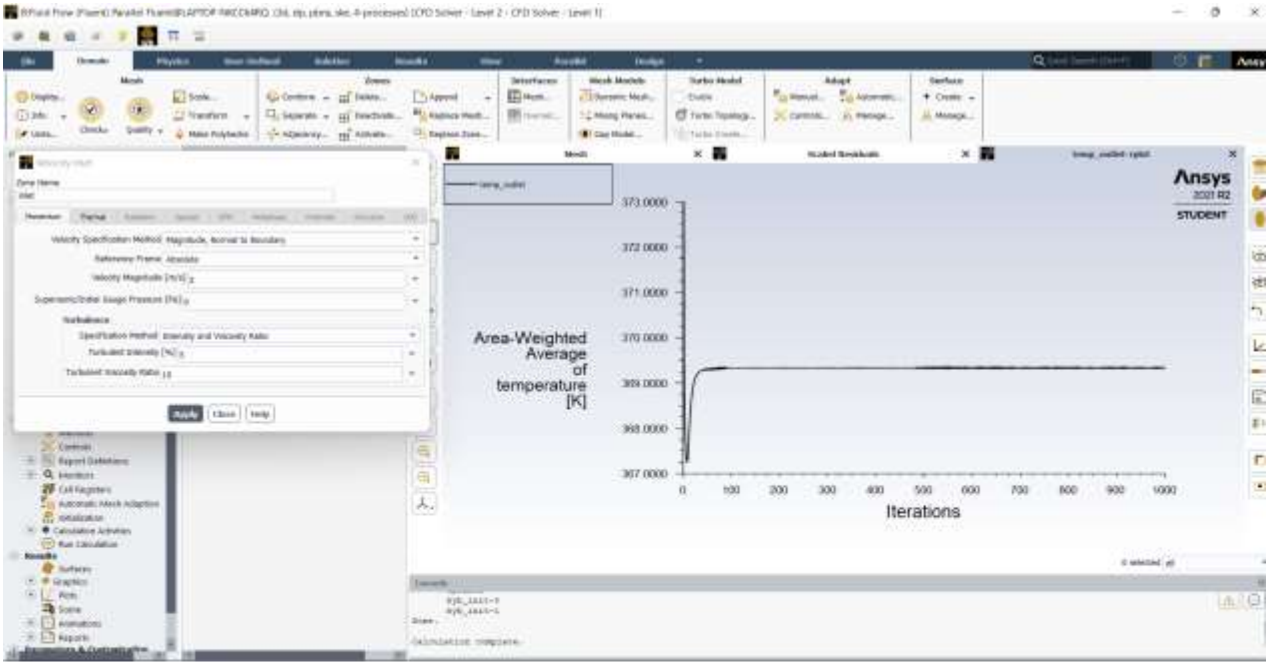
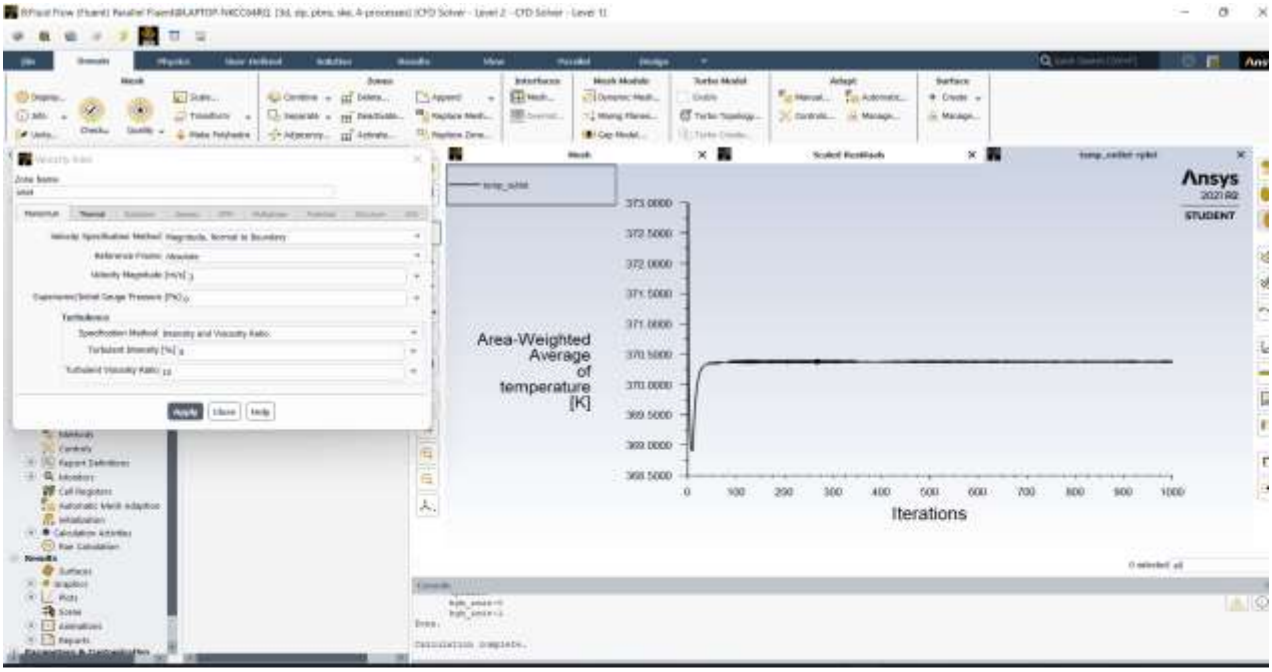


Fig 5.3.2 Temperature outlet for the radiator at 2m/s



Velocity Inlet

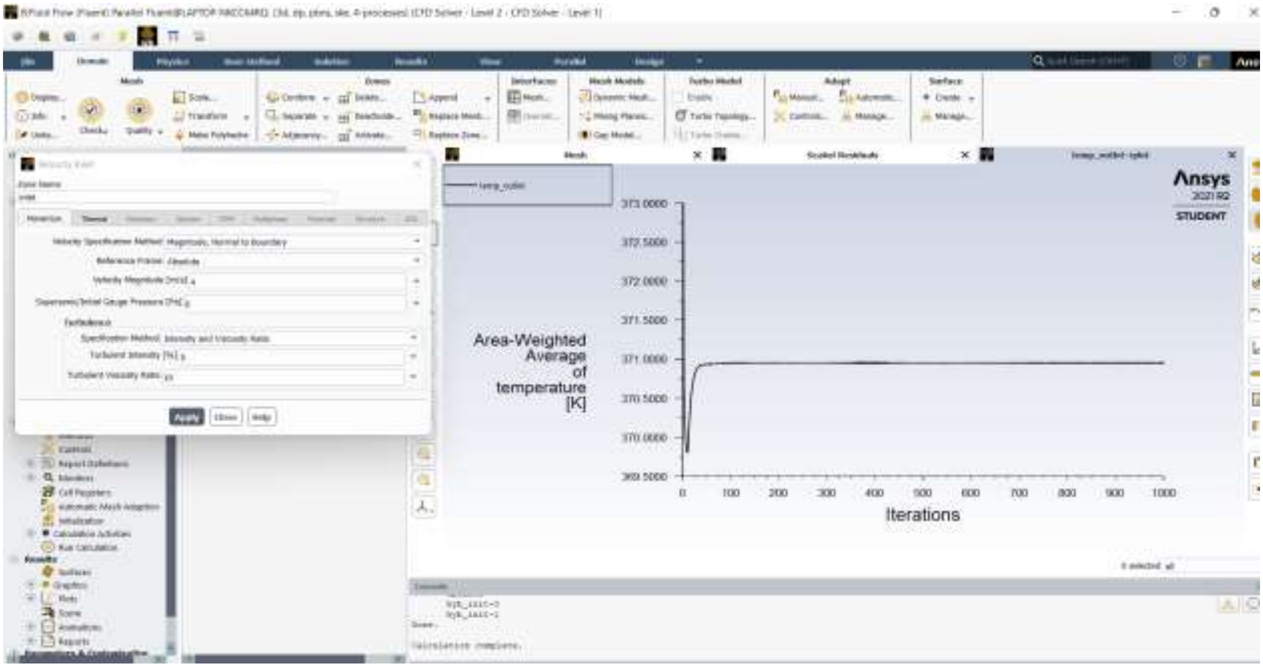
Zone Name
inlet

Velocity Specification Method Magnitude, Normal to Boundary
Reference Frame Absolute
Velocity Magnitude [m/s] 3
Supersonic/Initial Gauge Pressure [Pa] 0
Turbulence
Specification Method Intensity and Viscosity Ratio
Turbulent Intensity [%] 5
Turbulent Viscosity Ratio 10

```
Done.

Calculation complete.
```

Fig 5.3.3 Temperature outlet for the radiator at 3m/s



Velocity Inlet

Zone Name
inlet

Velocity Specification Method Magnitude, Normal to Boundary
Reference Frame Absolute
Velocity Magnitude [m/s] 4
Supersonic/Initial Gauge Pressure [Pa] 0
Turbulence
Specification Method Intensity and Viscosity Ratio
Turbulent Intensity [%] 5
Turbulent Viscosity Ratio 10

```
Done.

Calculation complete.
```

Fig 5.3.4 Temperature outlet for the radiator at 4m/s

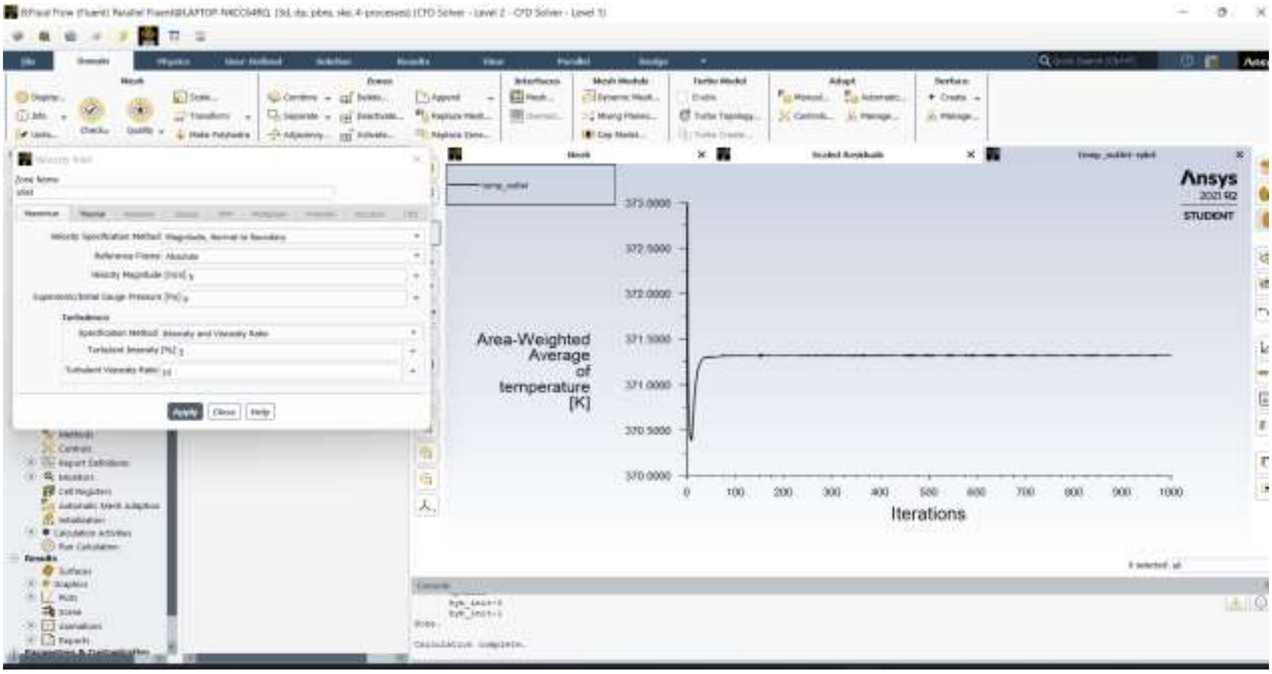


Fig 5.3.5 Temperature outlet for the radiator at 5m/s

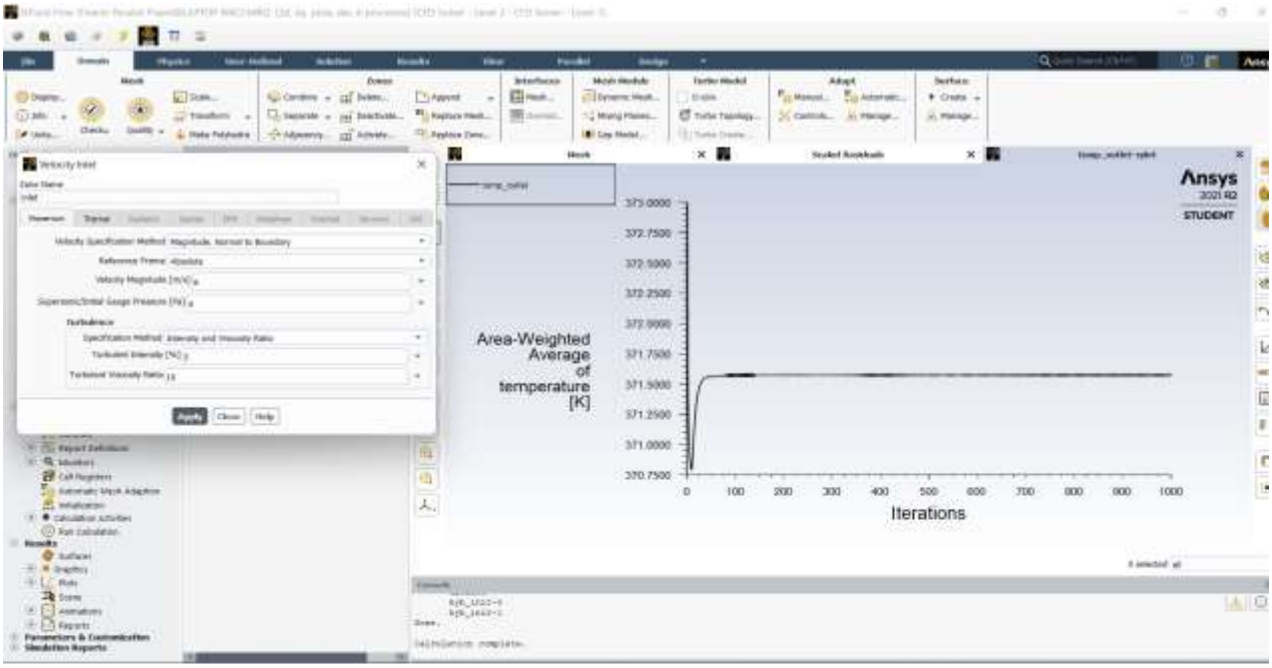
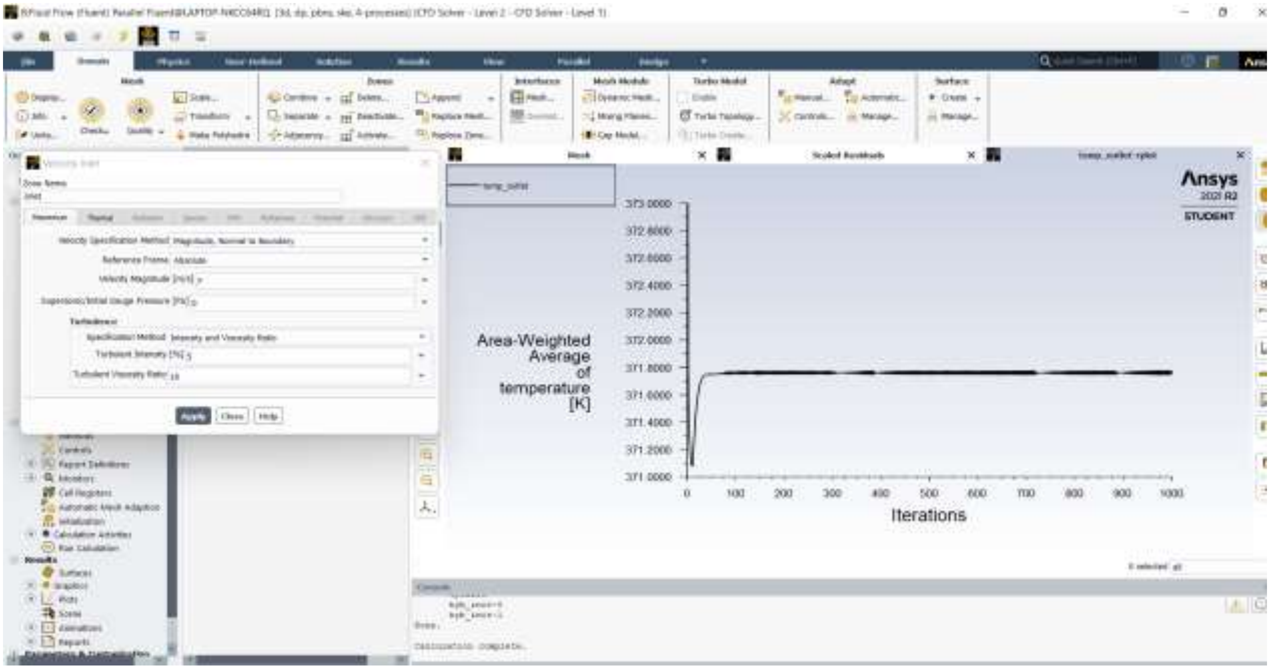


Fig 5.3.6 Temperature outlet for the radiator at 6m/s



Velocity Inlet

Zone Name: inlet

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude [m/s]: 7

Supersonic/Initial Gauge Pressure [Pa]: 0

Turbulence

Specification Method: Intensity and Viscosity Ratio

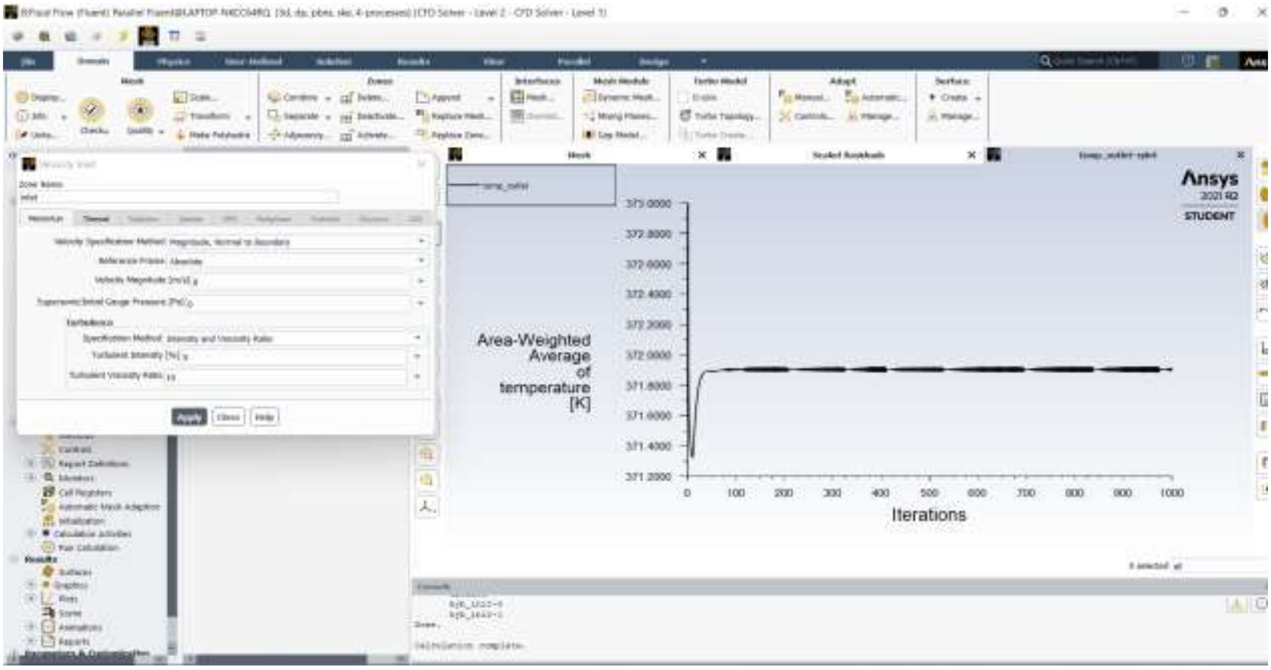
Turbulent Intensity [%]: 5

Turbulent Viscosity Ratio: 10

Done.

Calculation complete.

Fig 5.3.7 Temperature outlet for the radiator at 7m/s



Velocity Inlet

Zone Name: inlet

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude [m/s]: 8

Supersonic/Initial Gauge Pressure [Pa]: 0

Turbulence

Specification Method: Intensity and Viscosity Ratio

Turbulent Intensity [%]: 5

Turbulent Viscosity Ratio: 10

Done.

Calculation complete.

Fig 5.3.8 Temperature outlet for the radiator at 8m/s

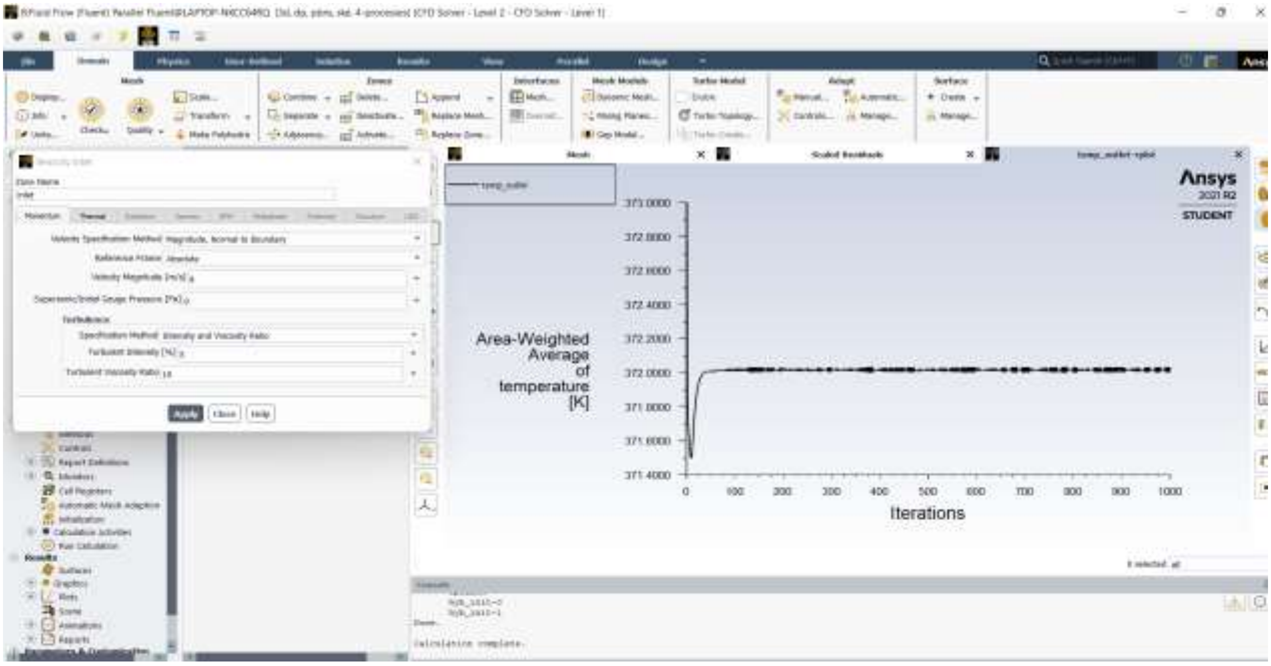


Fig 5.3.9 Temperature outlet for the radiator at 9m/s

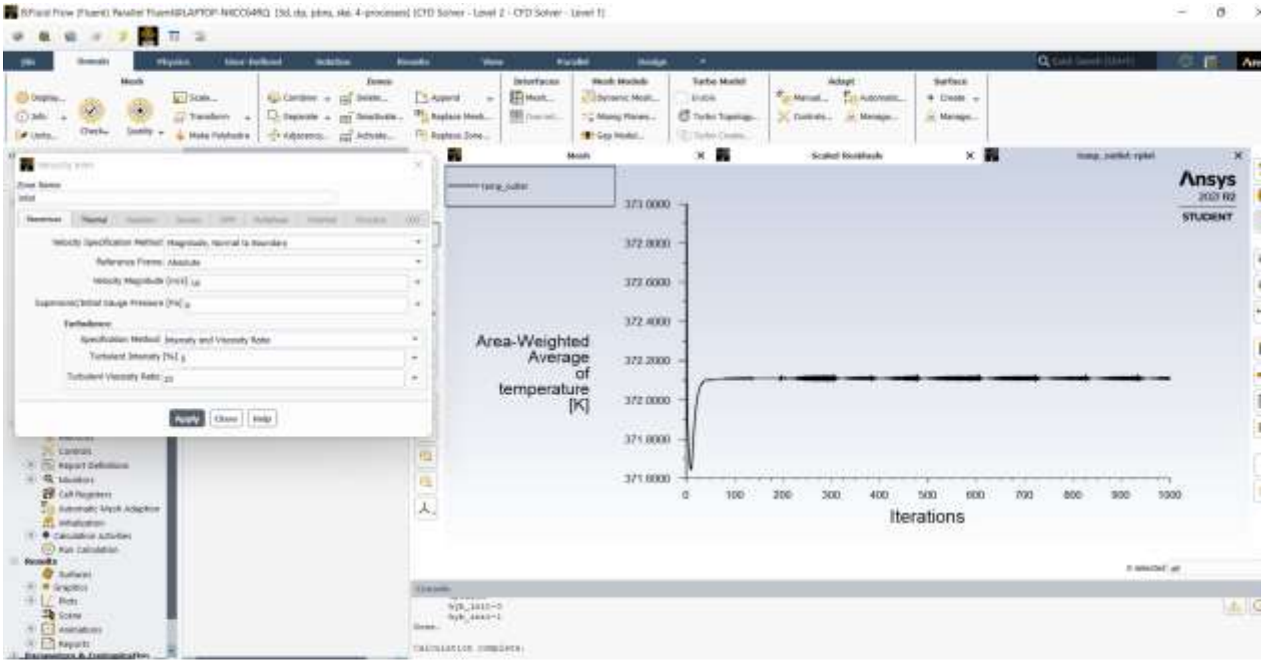


Fig 5.3.10 Temperature outlet for the radiator at 10m/s

FOR COPPER:

5.4 THE TEMPERATURE CONTOURS OF THE CAR RADIATOR:

Table 5.2 Table of Outlet temperature corresponding to the inlet velocity of the coolant for copper

VELOCITY (m/s)	OUTLET TEMPERATURE (K)
1	366.684
2	369.243
3	370.295
4	370.893
5	371.261
6	371.582
7	371.742
8	371.892
9	372.006
10	372.086

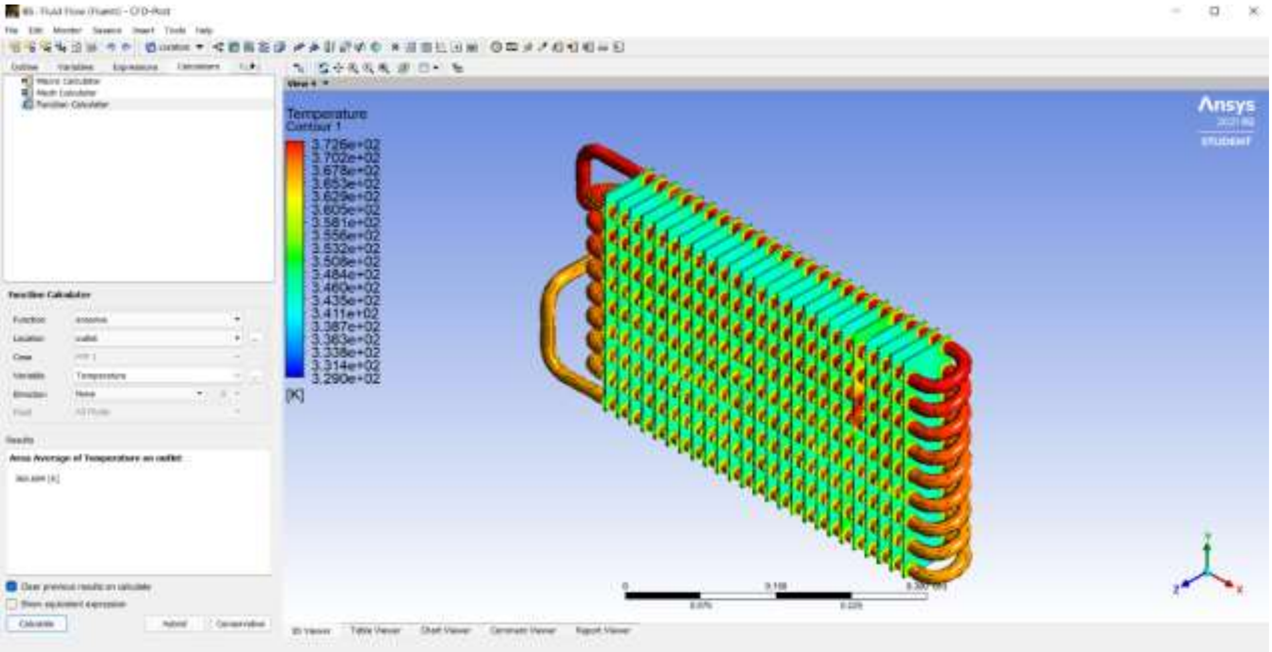


Fig 5.4.1 Temperature contour for the radiator at 1m/s

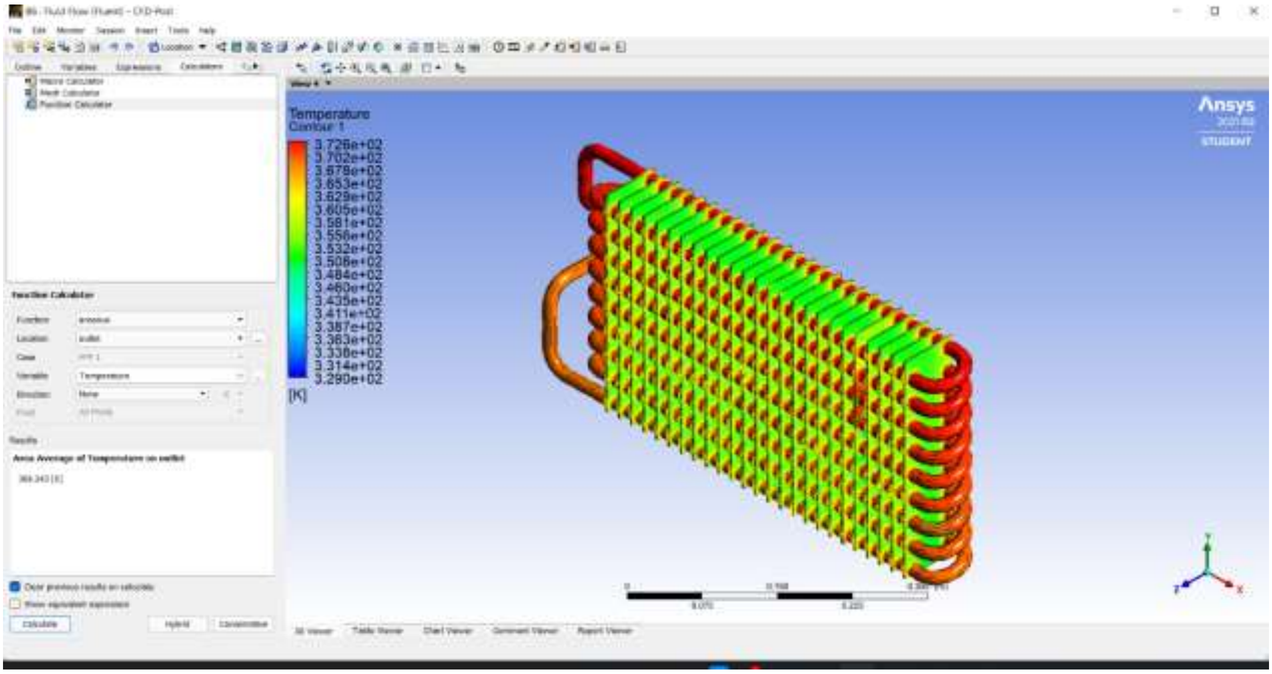


Fig 5.4.2 Temperature contour for the radiator at 2m/s

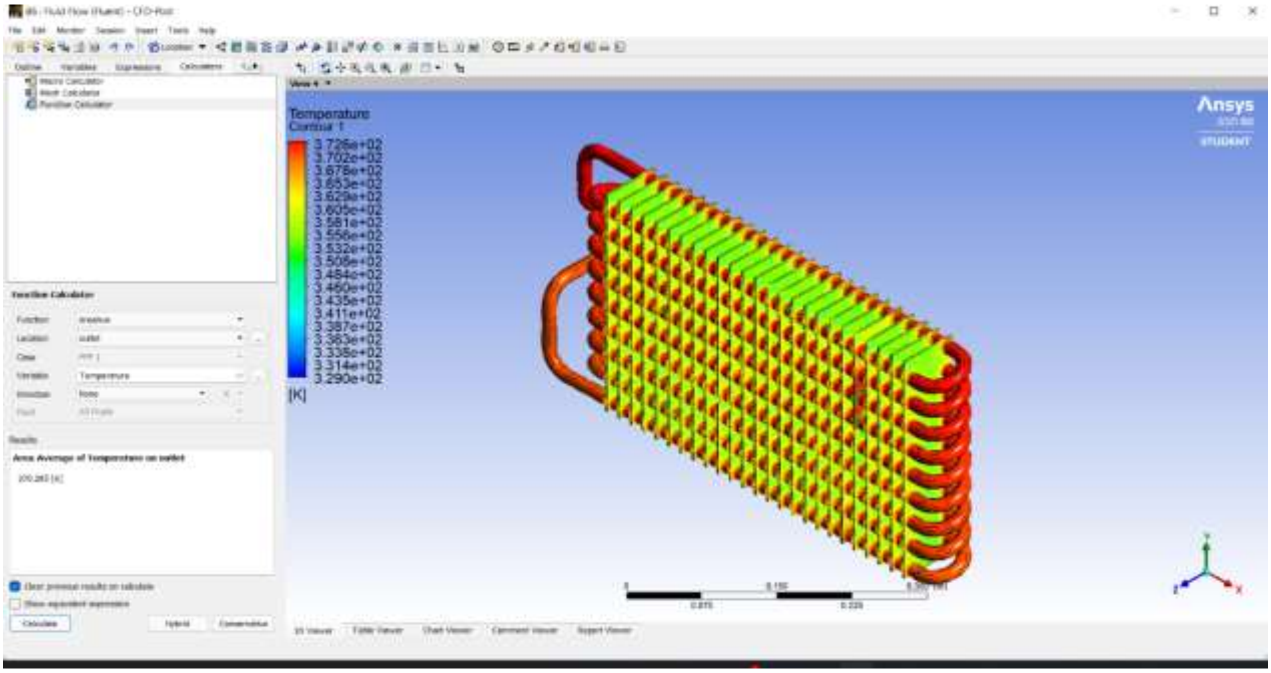


Fig 5.4.3 Temperature contour for the radiator at 3m/s

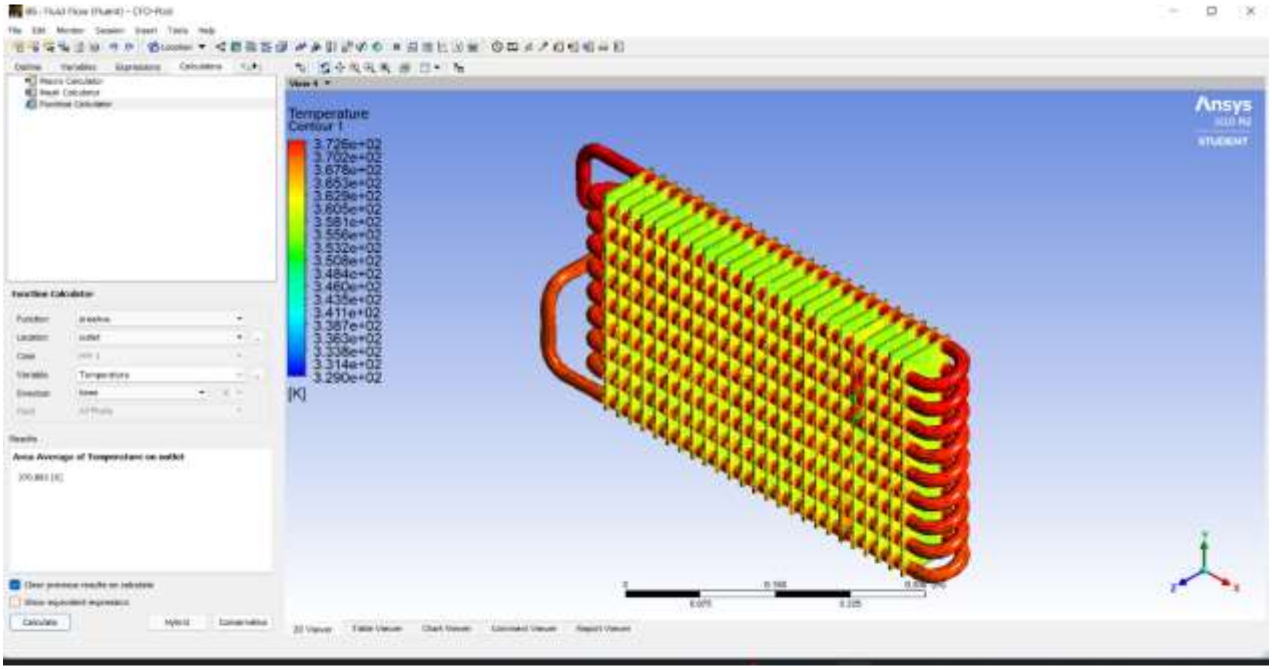
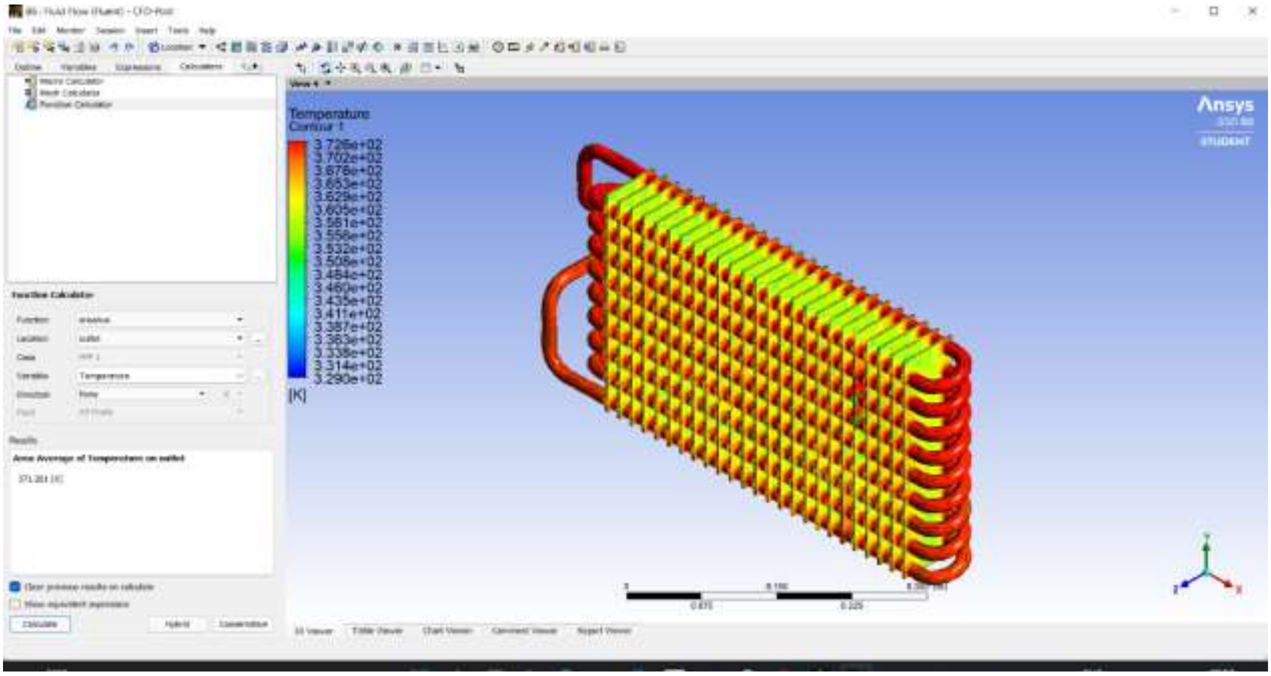


Fig 5.4.4 Temperature contour for the radiator at 4m/s



Function Calculator

Function: areaAve

Location: outlet

Variable: Temperature

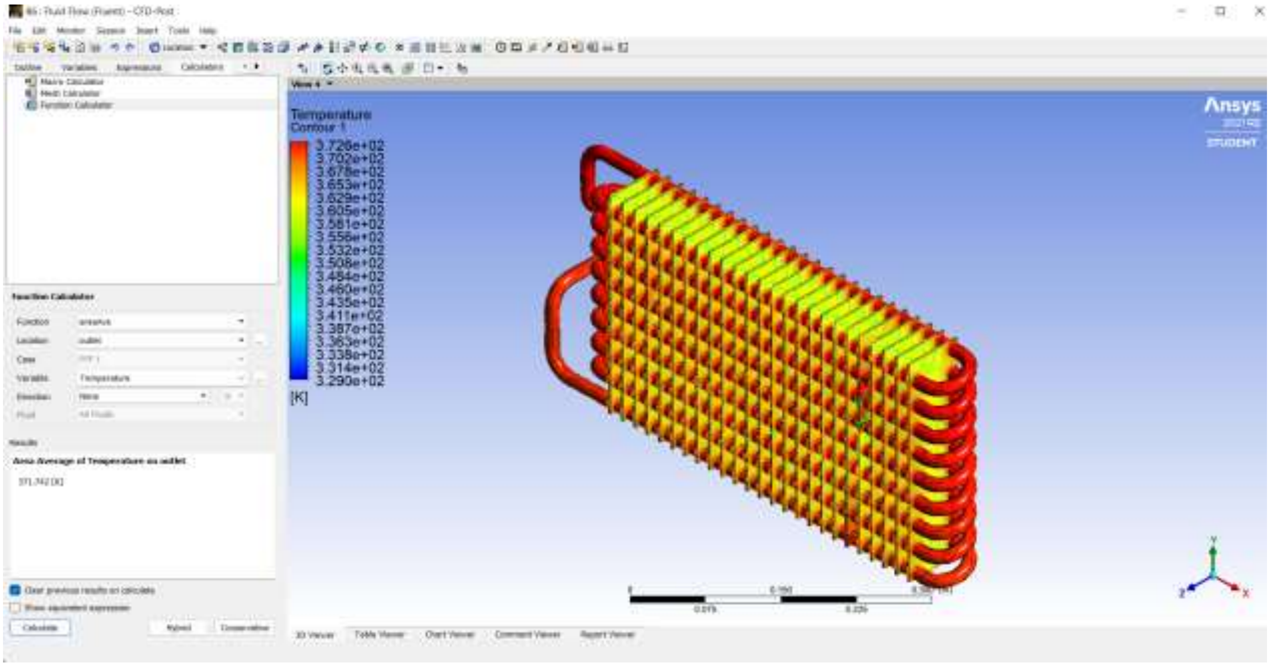
Direction: None

Results

Area Average of Temperature on outlet

371.281 [K]

Fig 5.4.5 Temperature contour for the radiator at 5m/s



Function Calculator

Function: areaAve

Location: outlet

Variable: Temperature

Direction: None

Results

Area Average of Temperature on outlet

371.742 [K]

Fig 5.4.6 Temperature contour for the radiator at 6m/s

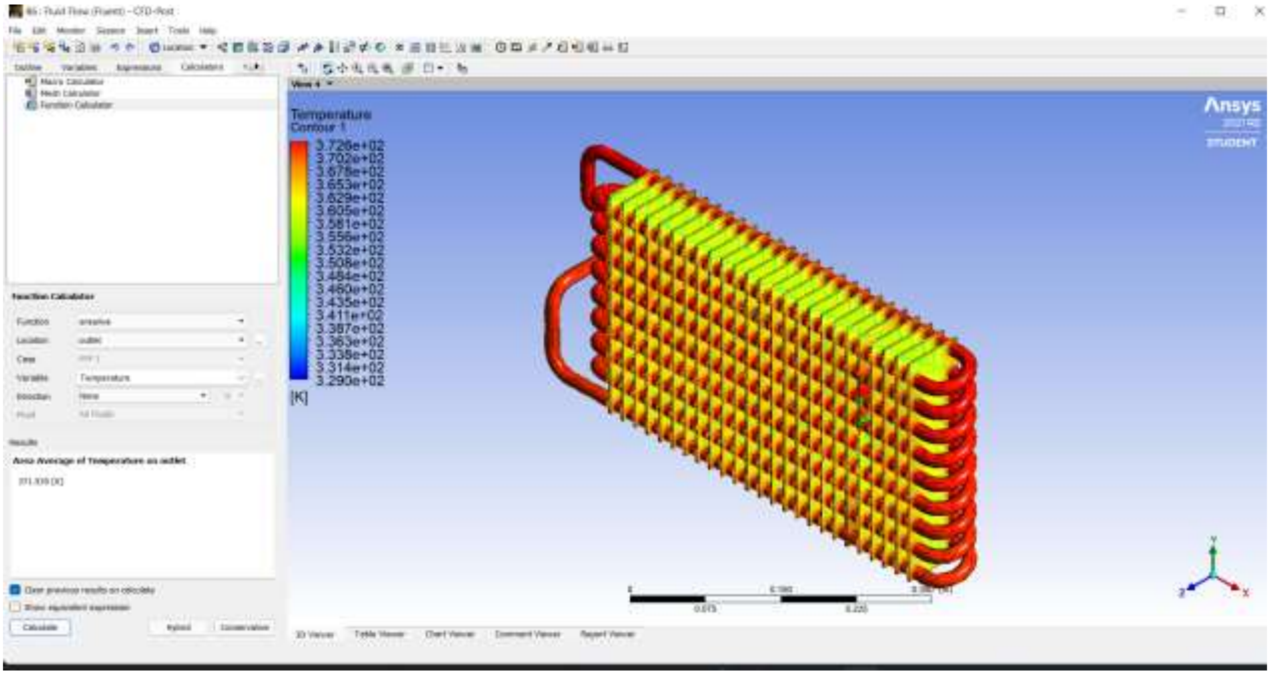


Fig 5.4.7 Temperature contour for the radiator at 7m/s

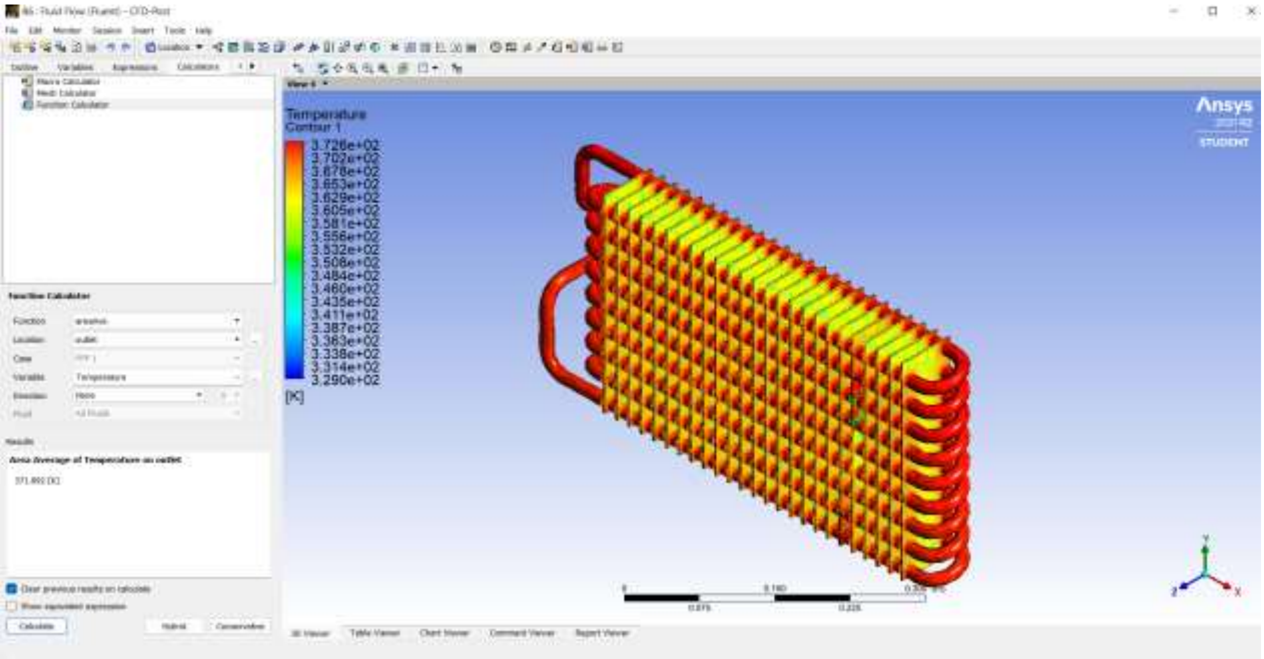


Fig 5.4.8 Temperature contour for the radiator at 8m/s

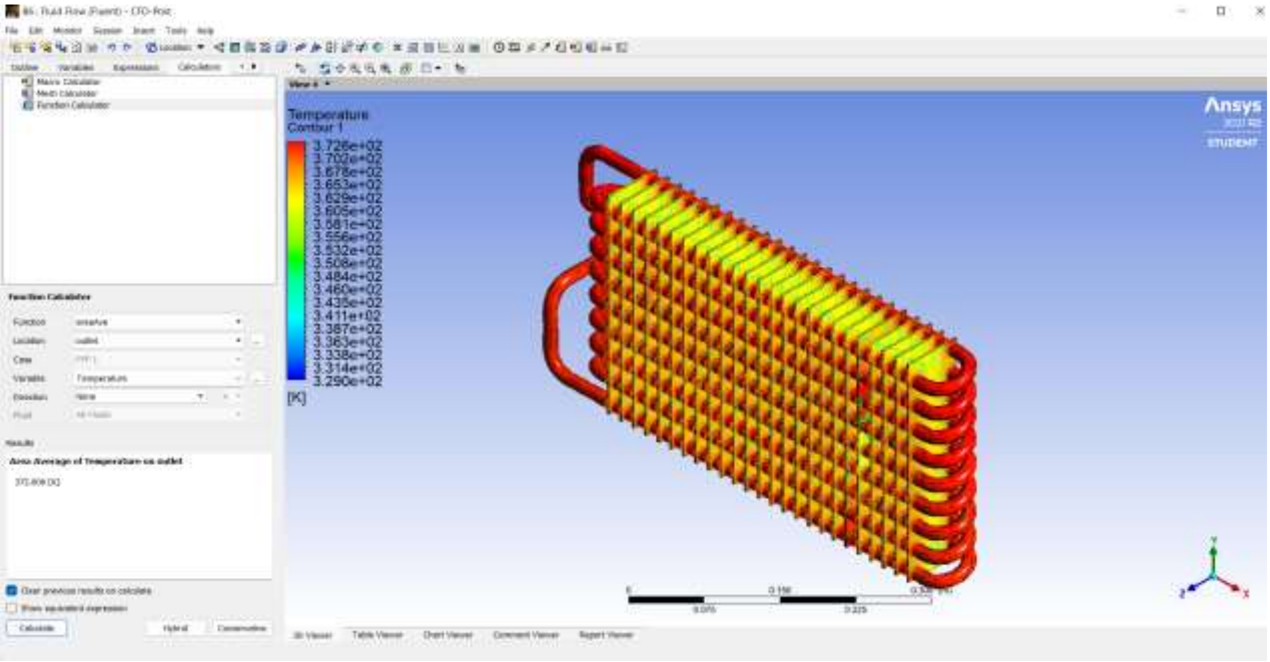


Fig 5.4.9 Temperature contour for the radiator at 9m/s

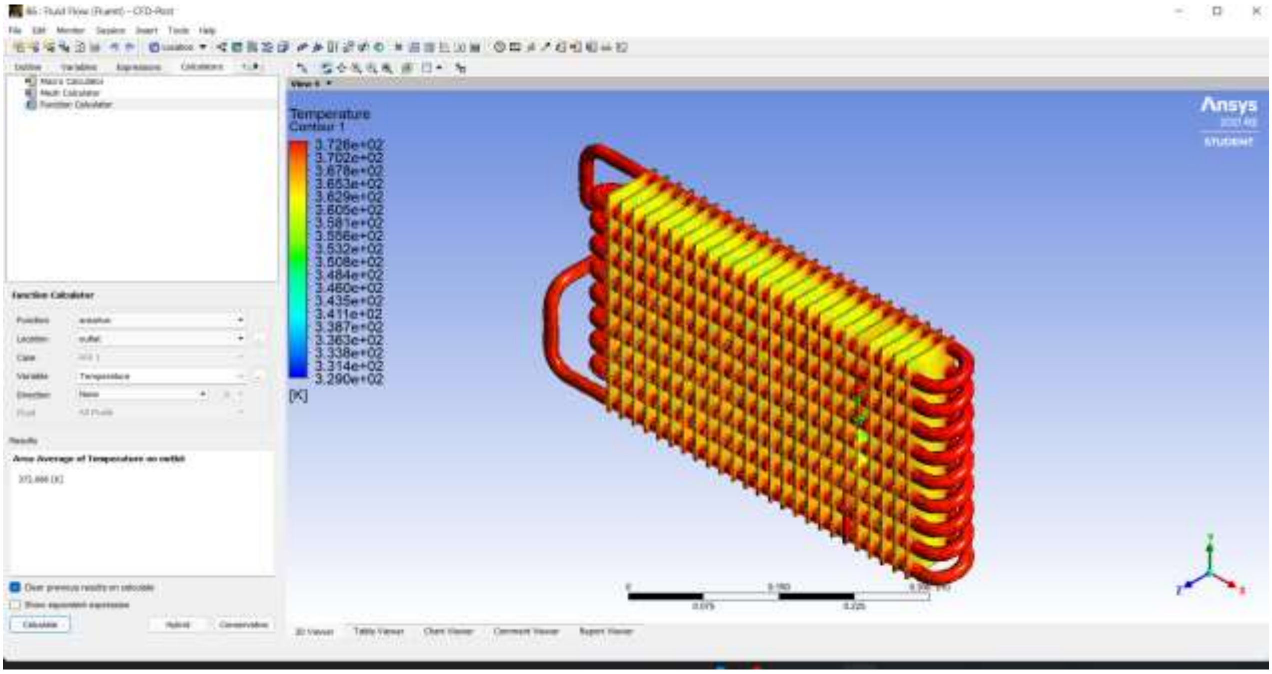


Fig 5.4.10 Temperature contour for the radiator at 10m/s

5.5 SCALED RESIDUALS:

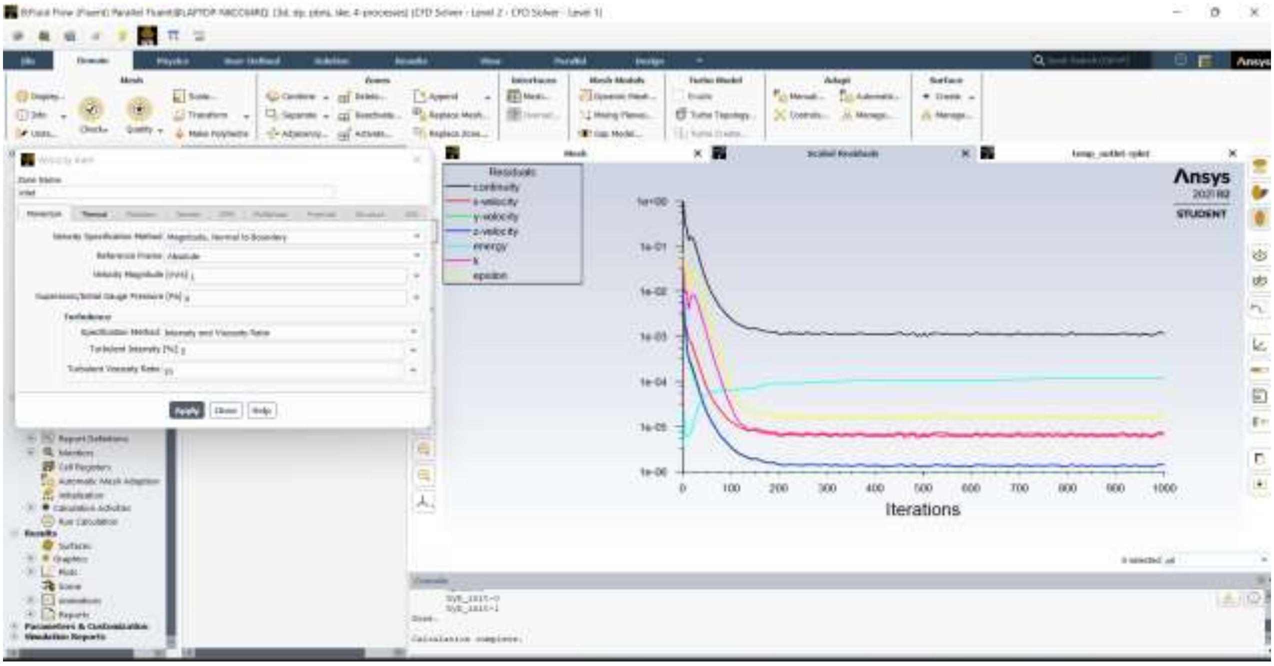


Fig 5.4.1 Scaled residuals for the radiator at 1m/s

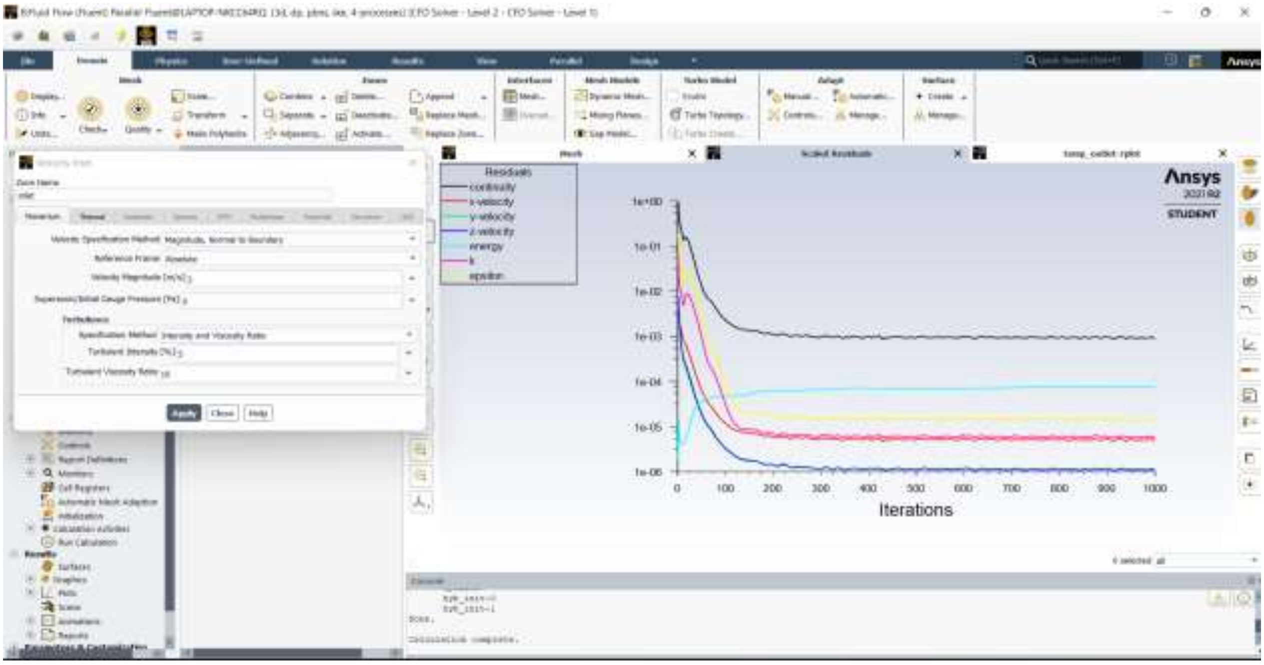


Fig 5.5.2 Scaled residuals for the radiator at 2m/s

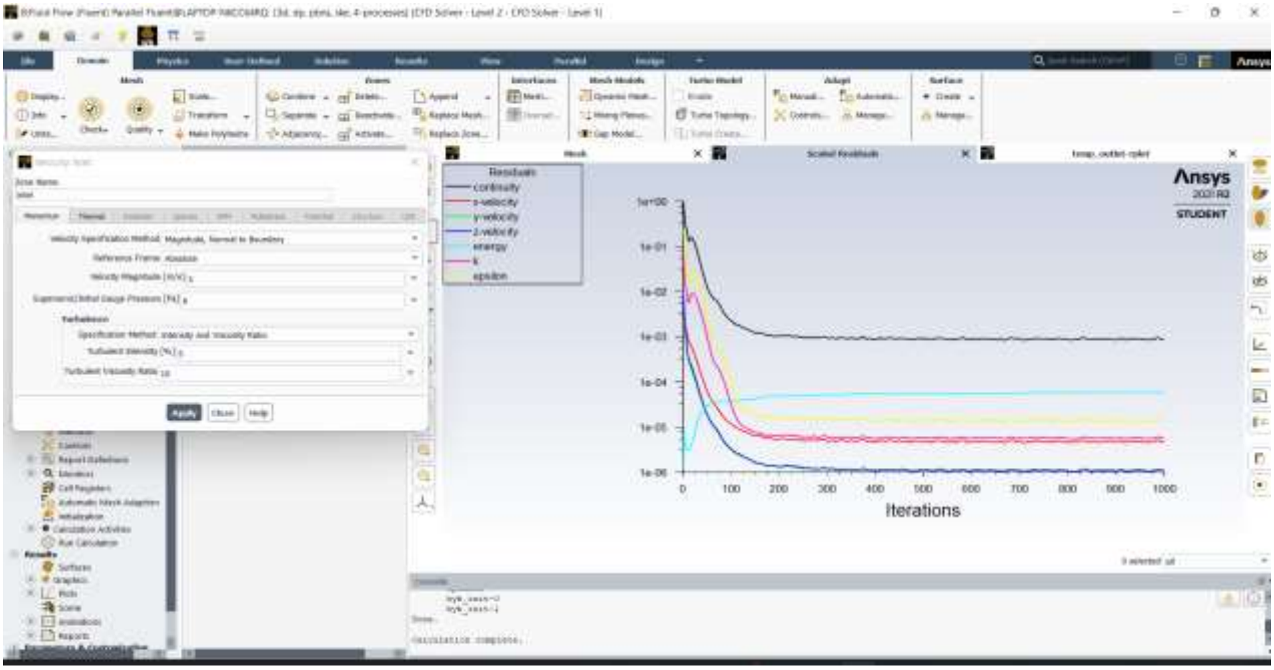


Fig 5.5.3 Scaled residuals for the radiator at 3m/s

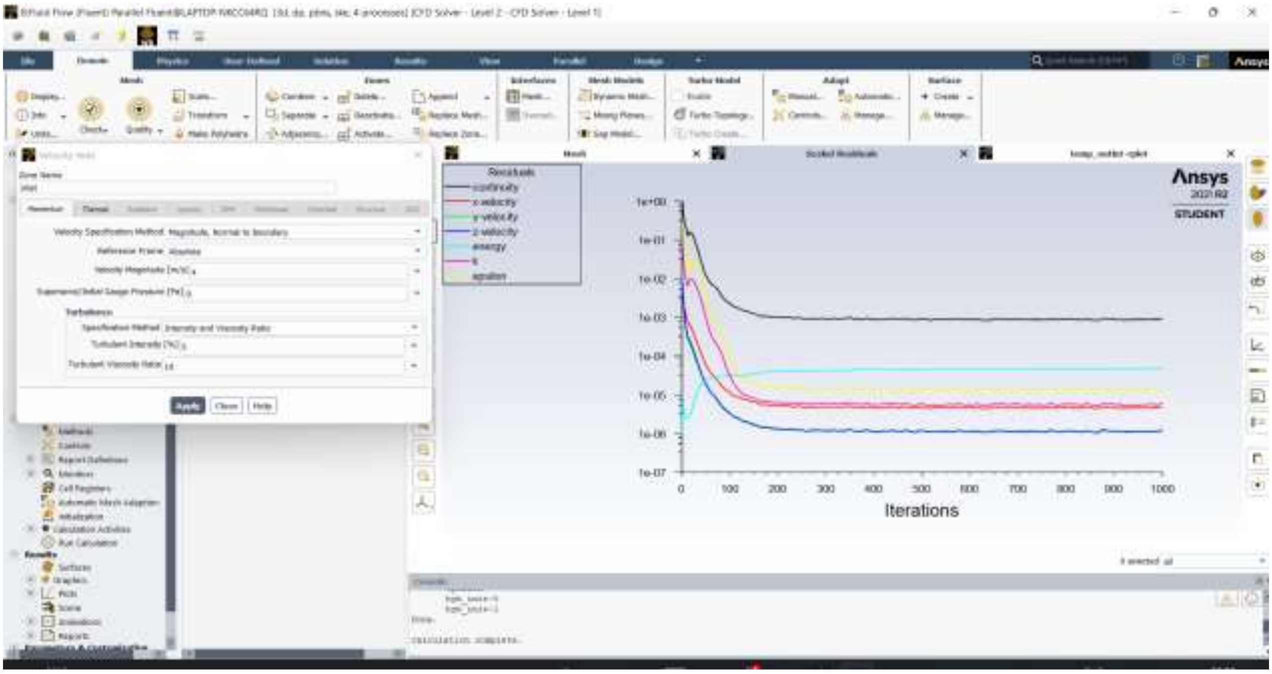
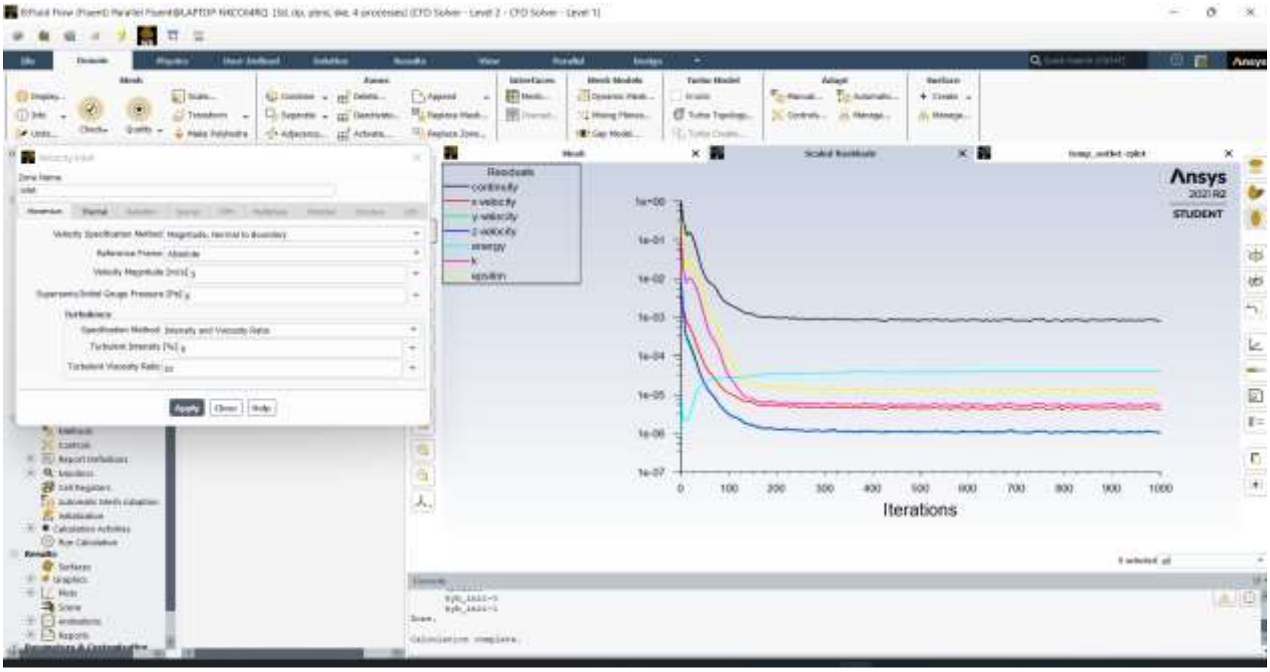


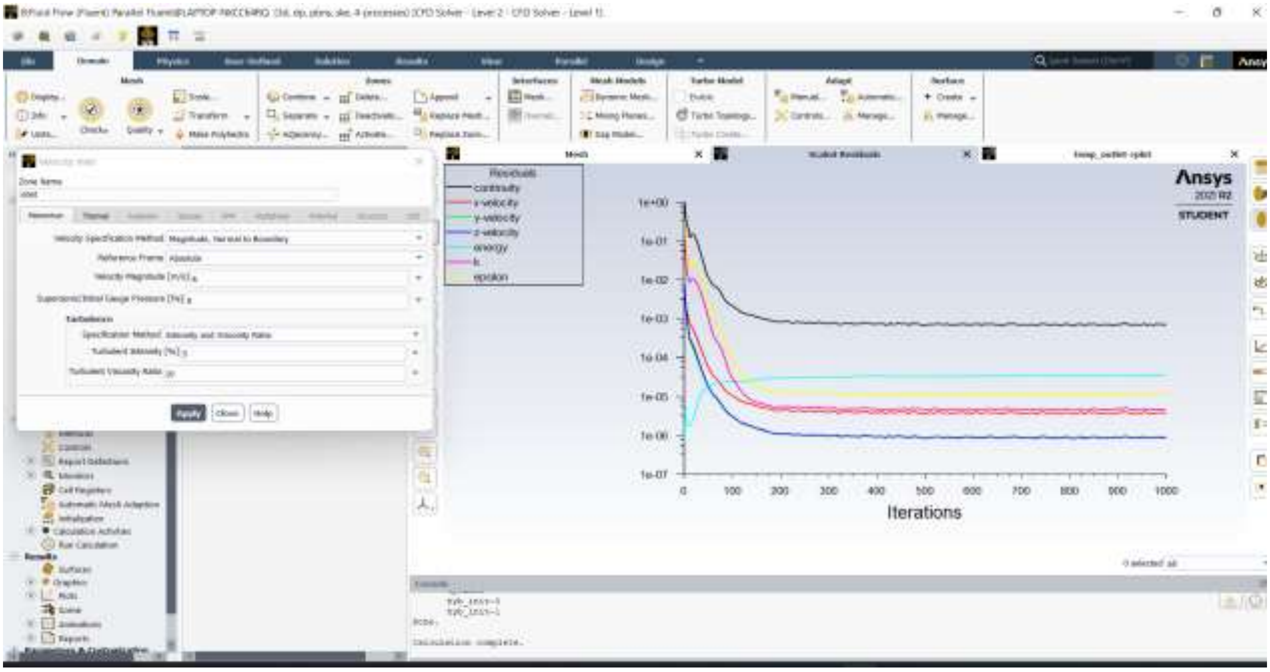
Fig 5.5.4 Scaled residuals for the radiator at 4m/s



```
Done.

Calculation complete.
```

Fig 5.5.5 Scaled residuals for the radiator at 5m/s



```
Done.

Calculation complete.
```

Fig 5.5.6 Scaled residuals for the radiator at 6m/s

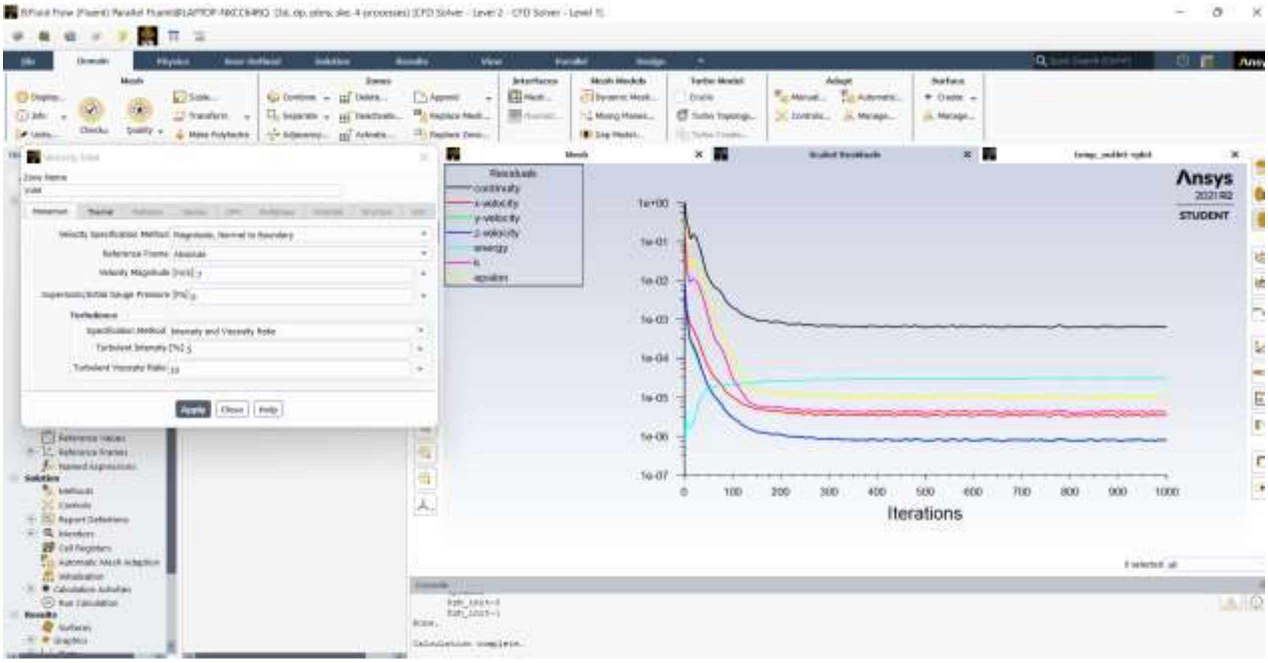


Fig 5.5.7 Scaled residuals for the radiator at 7m/s

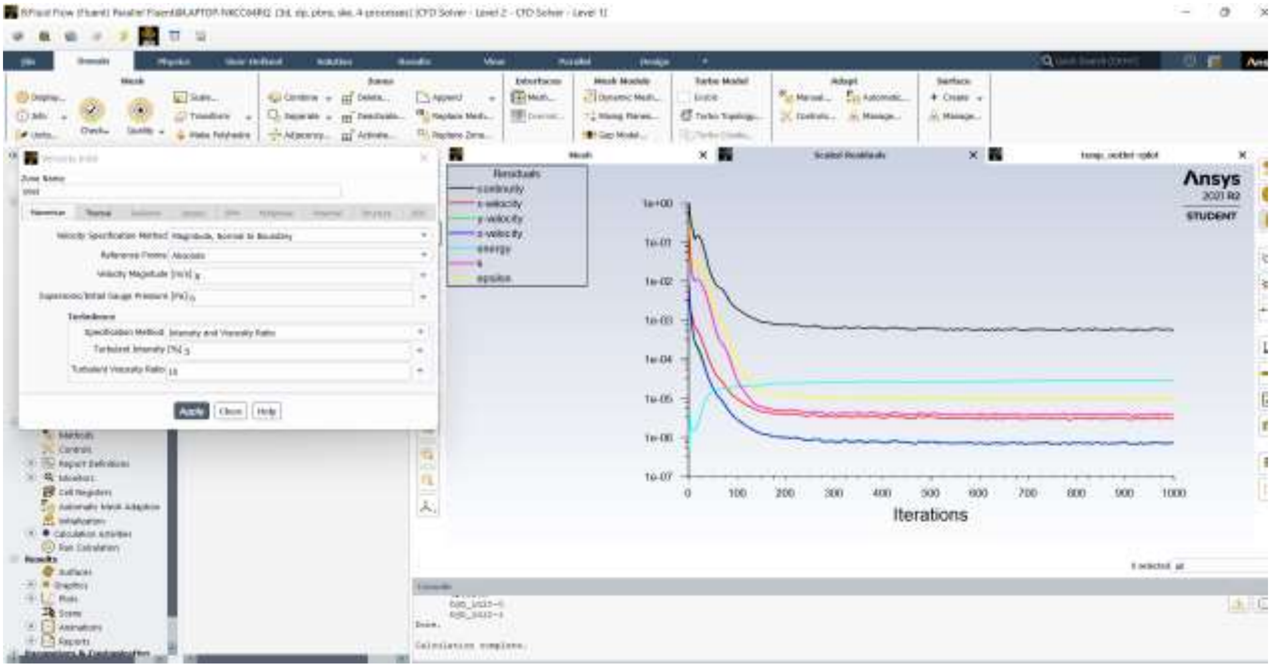
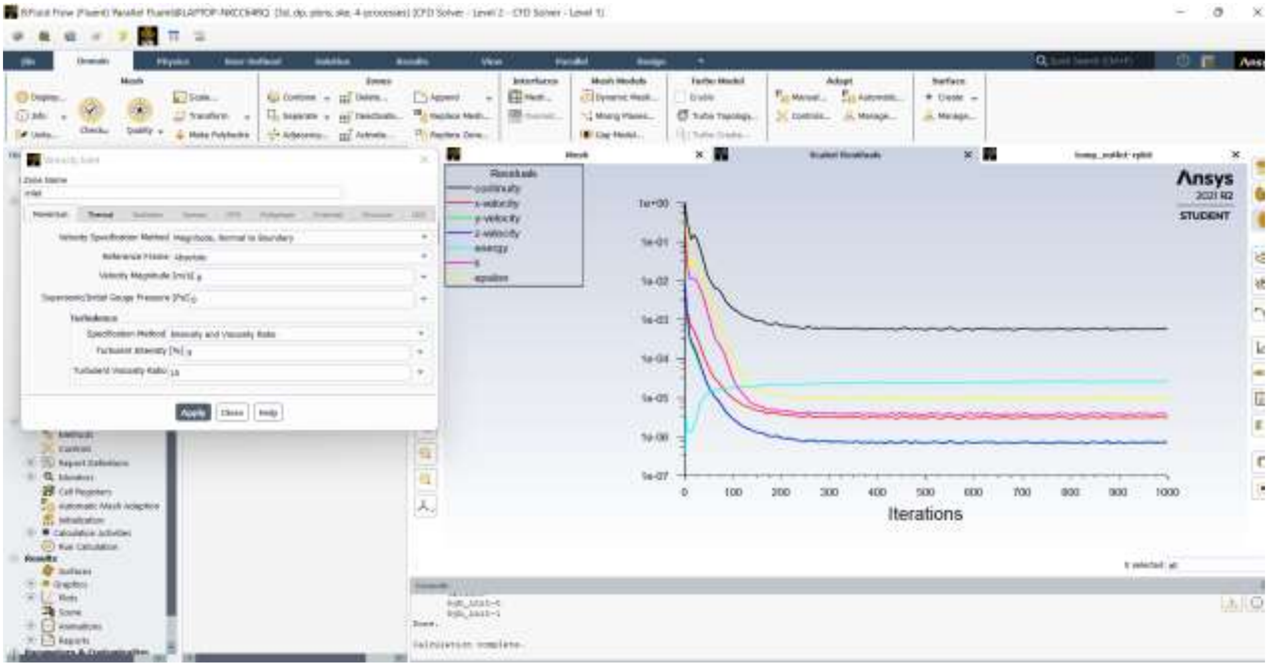
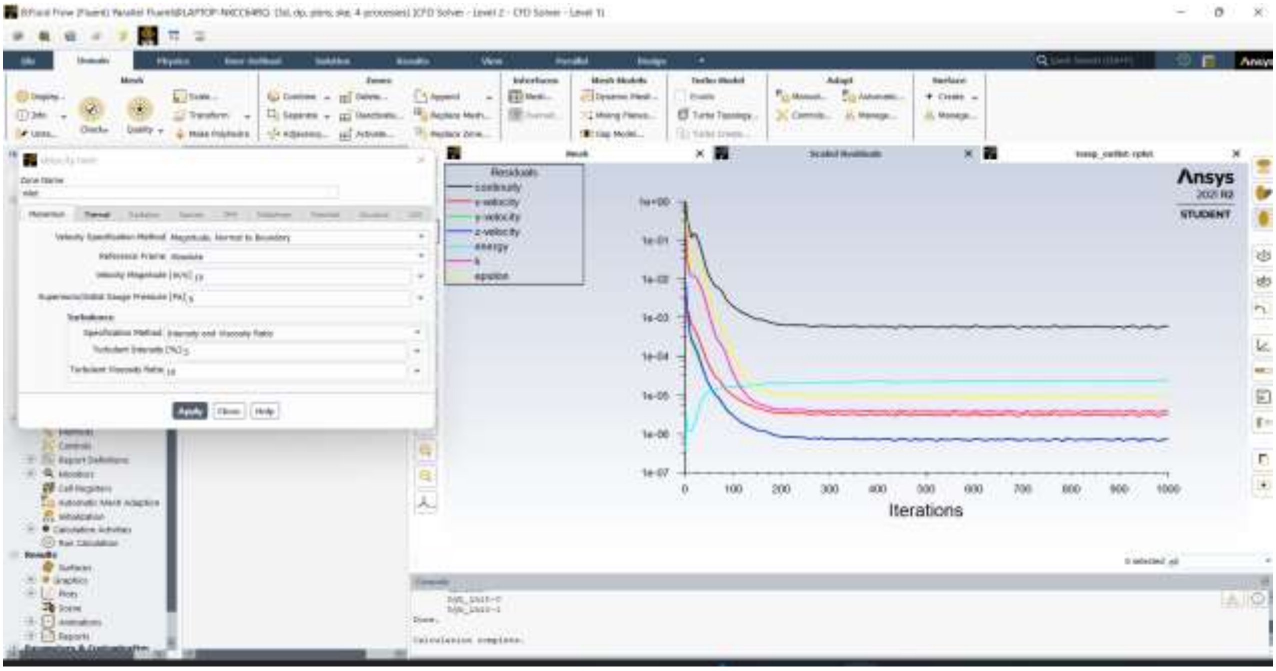


Fig 5.5.8 Scaled residuals for the radiator at 8m/s



Calculation complete.

Fig 5.5.9 Scaled residuals for the radiator at 9m/s



Calculation complete.

Fig 5.5.10 Scaled residuals for the radiator at 10m/s

5.6 TEMPERATURE OUTLET PLOT:

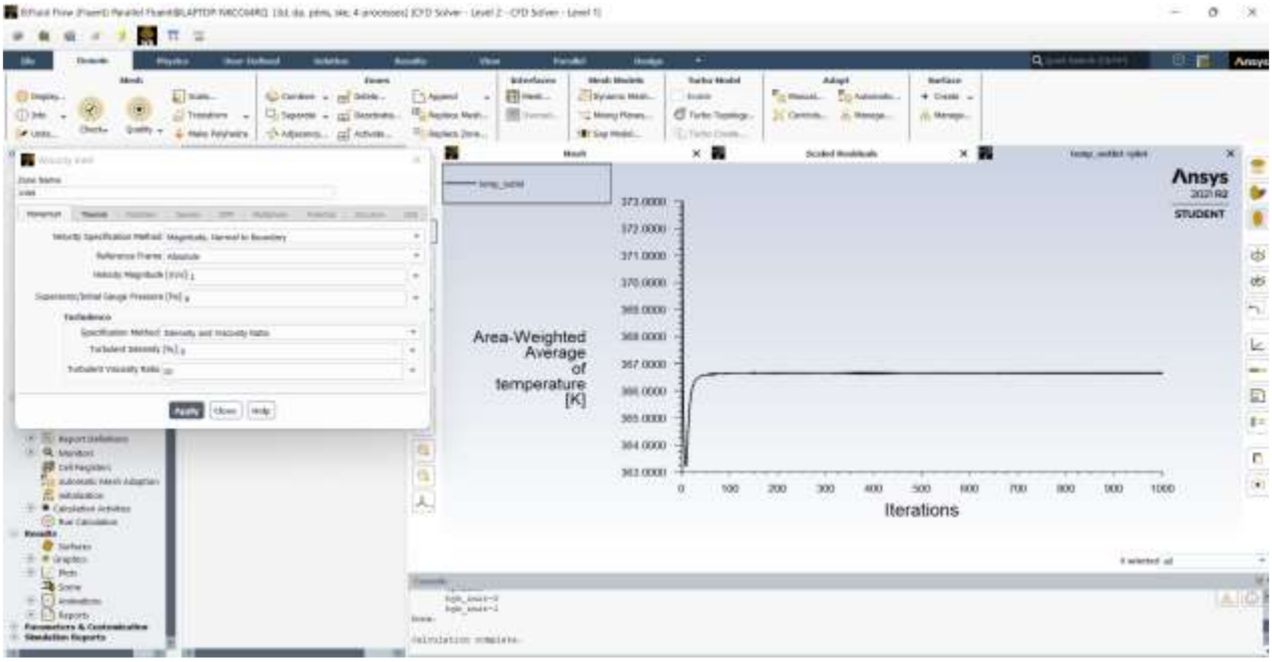


Fig 5.6.1 Temperature outlet for the radiator at 1m/s

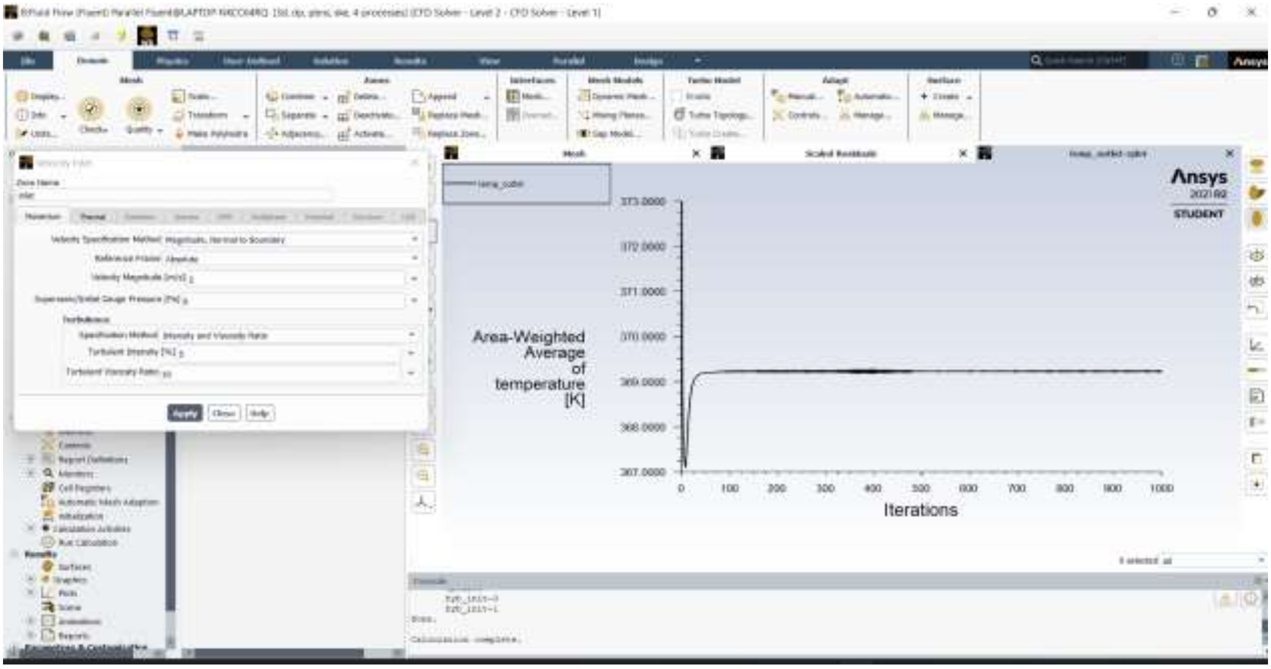


Fig 5.6.2 Temperature outlet for the radiator at 2m/s

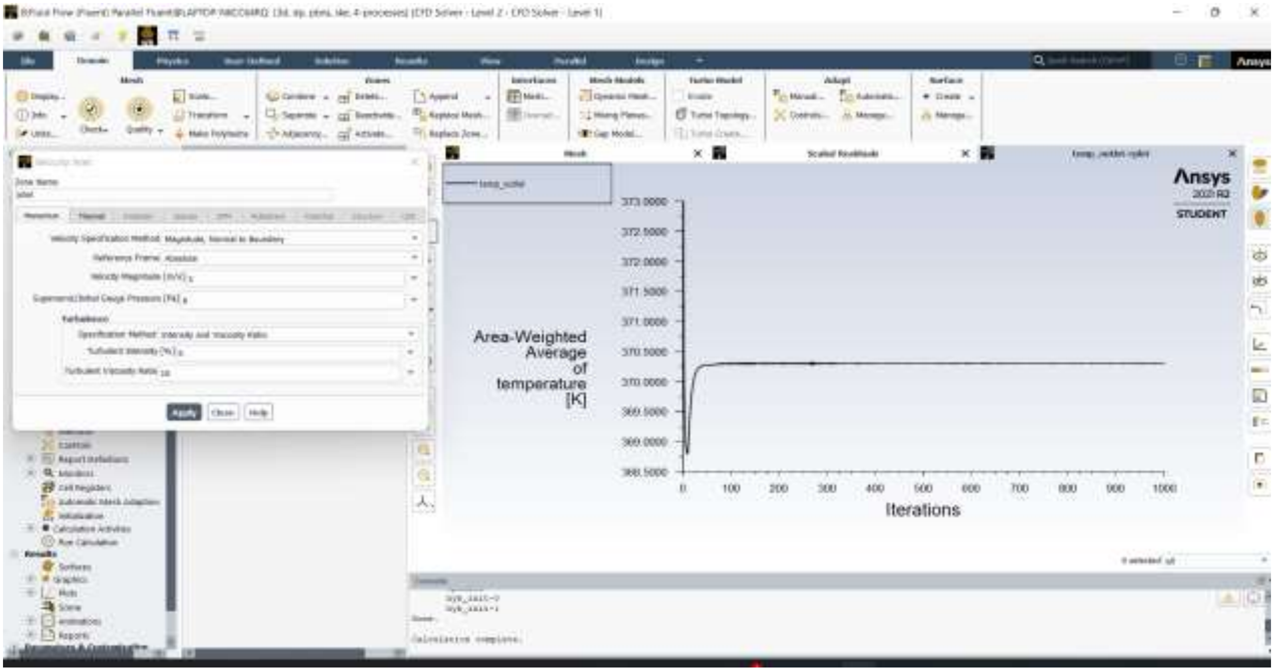


Fig 5.6.3 Temperature outlet for the radiator at 3m/s

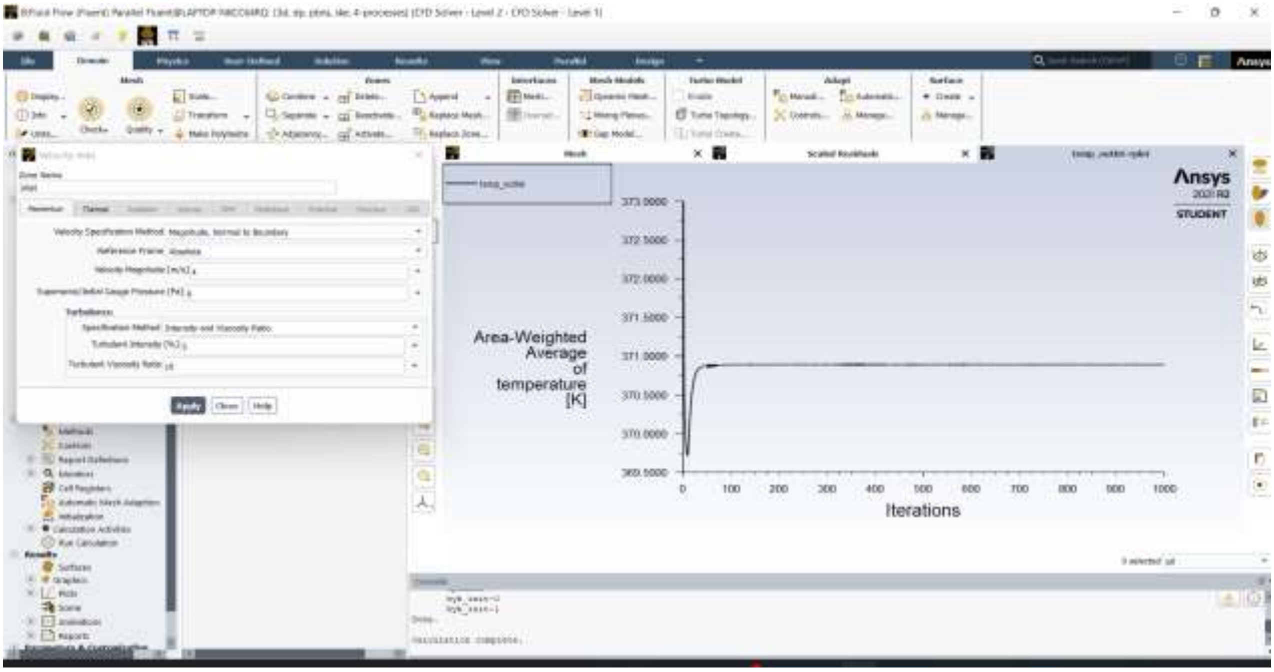


Fig 5.6.4 Temperature outlet for the radiator at 4m/s

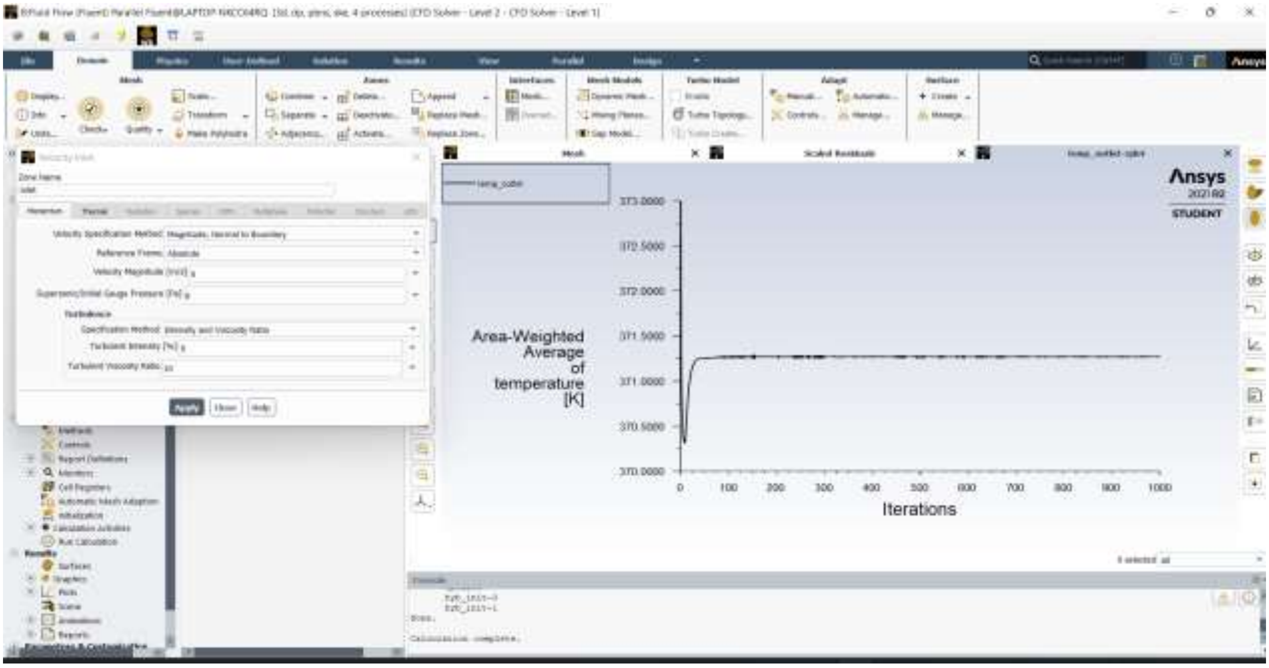


Fig 5.6.5 Temperature outlet for the radiator at 5m/s

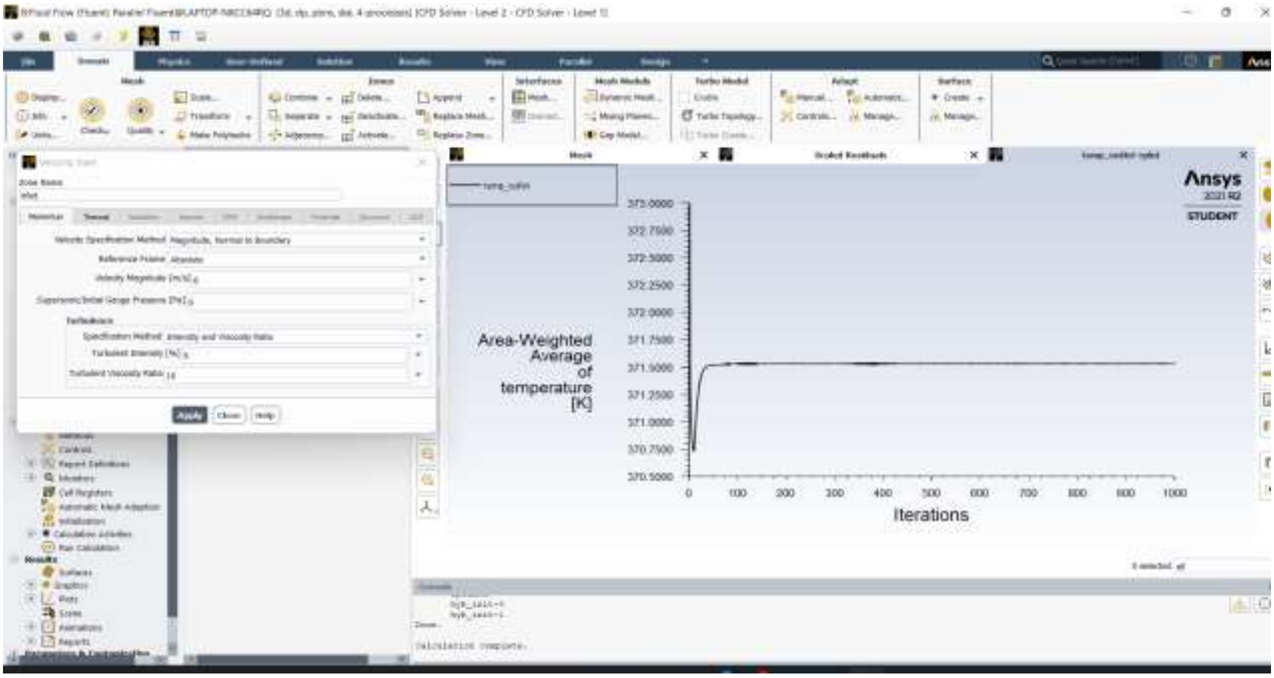


Fig 5.6.6 Temperature outlet for the radiator at 6m/s

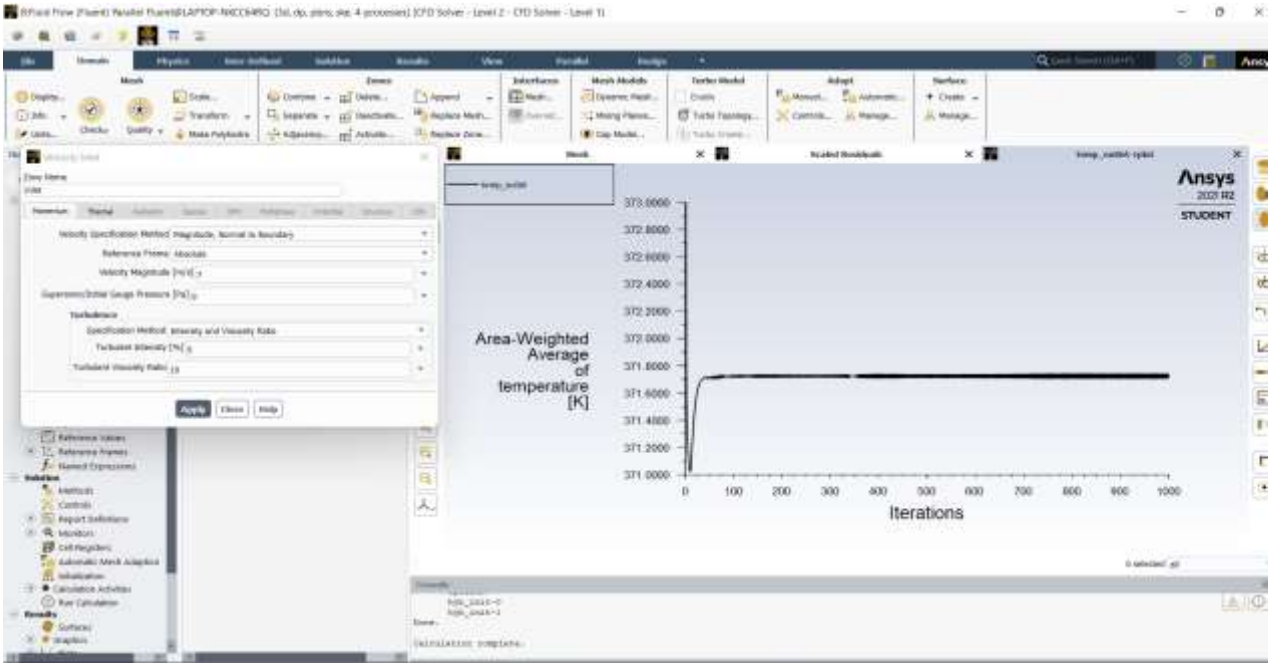


Fig 5.6.8 Temperature outlet for the radiator at 7m/s

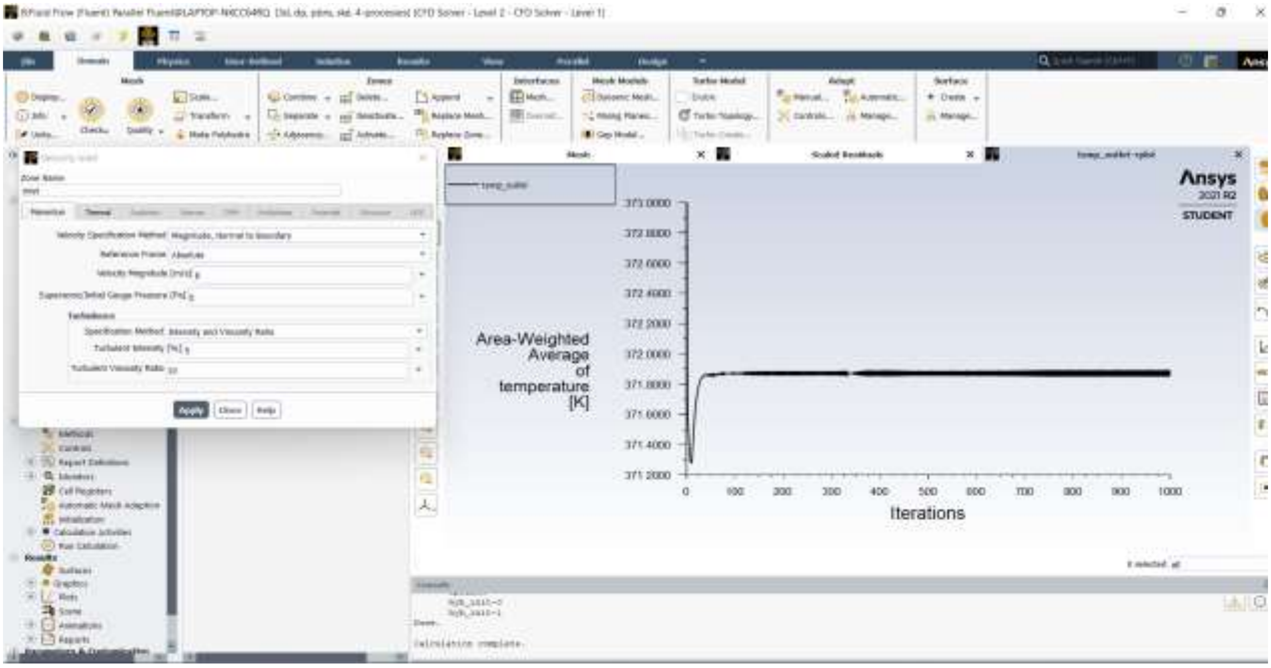


Fig 5.6.7 Temperature outlet for the radiator at 8m/s

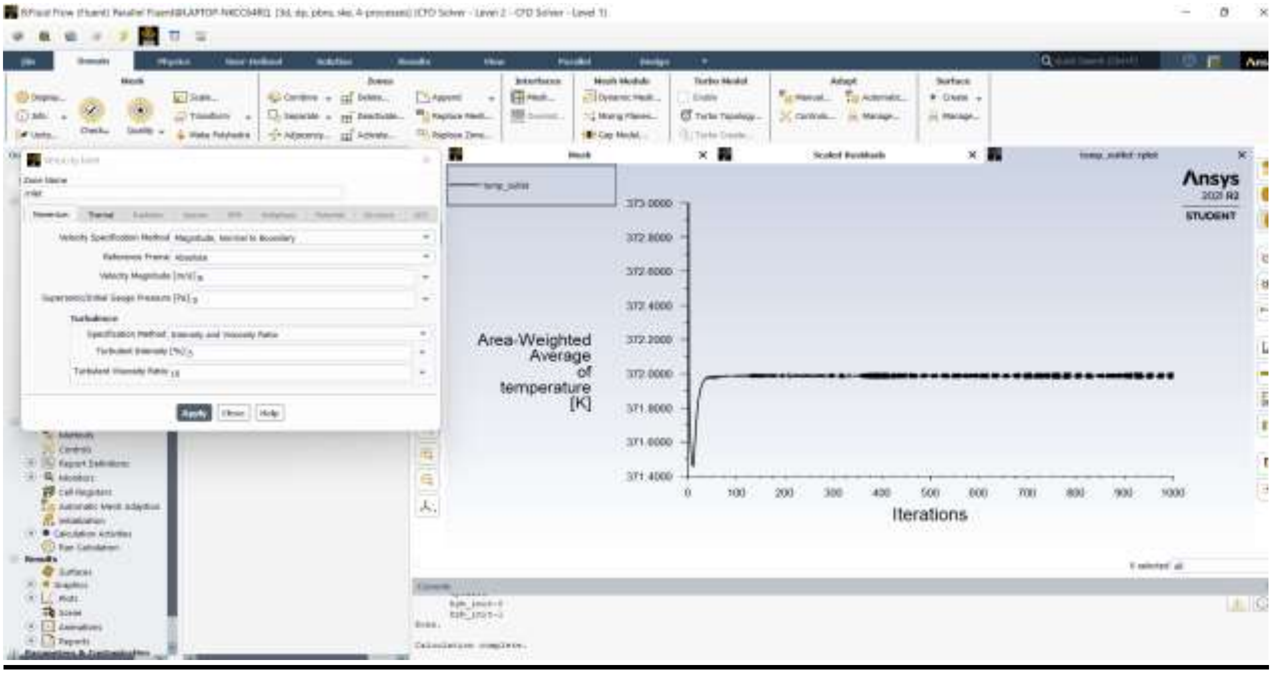


Fig 5.6.9 Temperature outlet for the radiator at 9m/s

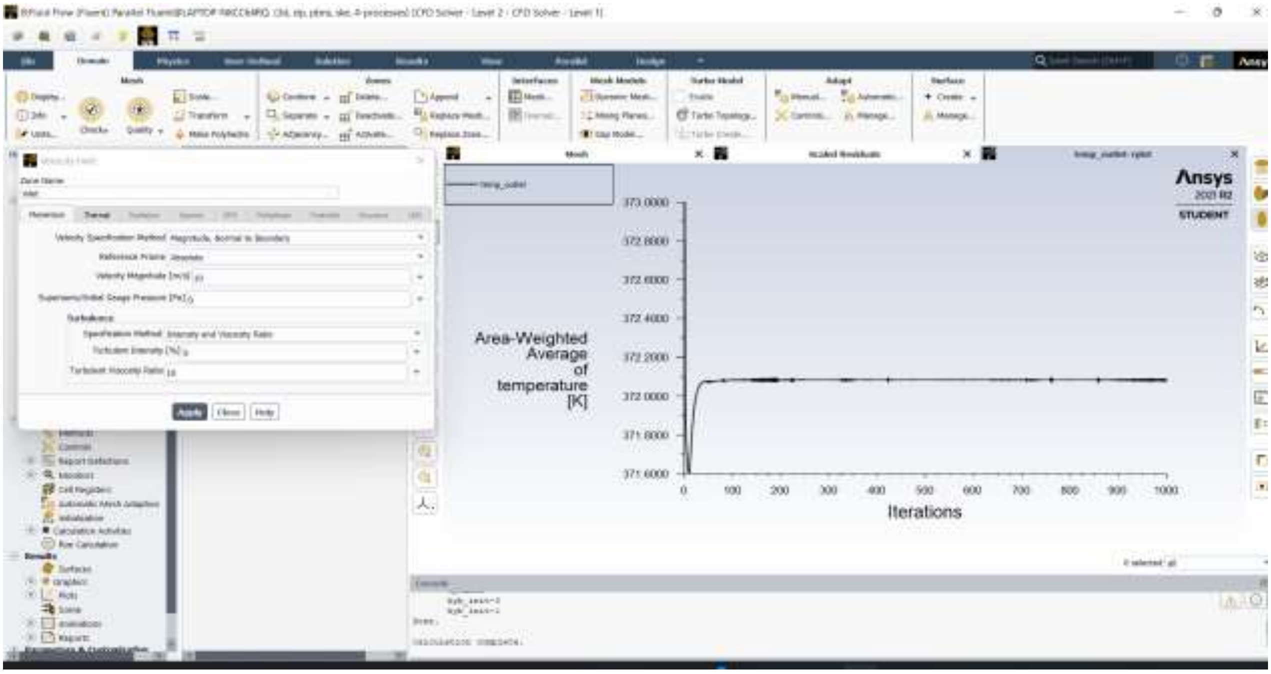


Fig 5.6.10 Temperature outlet for the radiator at 10m/s

CHAPTER 6

CONCLUSION

6 CONCLUSIONS:

- In this present work, CFD Analysis of Car Radiator is done for different velocities by using Aluminium and Copper and we have observed that as velocity of the coolant increases the cooling capacity of the radiator decreases.
- It is observed that for Copper as radiator material, at a velocity of 1m/s it is preferable to entry the coolant when compared to the remaining values from 1m/s to 10m/s.
- Similarly, it is observed that for Aluminium as radiator material, at a velocity of 1m/s it is preferable to entry the coolant when compared to the remaining values from 1m/s to 10m/s.
- When comparing the Copper and Aluminium as the materials of the radiator, the outlet temperature of coolant is less for copper at a Fluid velocity of 1m/s when compared to Aluminium material. For a velocity of 1m/s the calculated mass flow rate of the coolant is 0.078 kg/s and discharge is 7.85*10^-2 lit/sec. So, it is advisable to entry the fluid at a velocity as 1m/sec for copper material of the Radiator.

CHAPTER 7

REFERENCES

7 REFERENCES

[1] **Adnan M. Hussein, R.A. Bakar , K. Kadirgama** and **K.V. Sharma**, “Heat transfer enhancement using nanofluids in an automotive cooling system”www.elsevier.com/ locate/ichmt

[2] **A.H. Mamun, K.Y. Leong, R. Saidur** and **S.N. Kazi**, Performance Investigation of an Automotive Car Radiator Operated with Nanofluid-Based Coolants (Nanofluid as a Coolant in a Radiator), Applied Thermal Engineering, vol. 30, December 2010, pp.

[3] **Alhassan Salami Tijani** and **Ahmad Suhail bin Sudirman**, “Thermos-physical properties and heat transfer characteristics of water/anti-freezing and Al2O3/CuO based nanofluid as a coolant for car radiator”www.elsevier.com/locate/ijhmt

[4] **A. Sing**, Thermal conductivity of nano fluid“ defence science journal 58(2008)

[5] **Chavan & Tasgaonkar** , Study, Analysis & Design of Automobile Radiator Proposed with CAD Drawings & Geometrical Model of the Fan, Inter. Journal of Mech. & Prod. Engg. Research & Dev., 3(2), (2013) 137 to 146.

[6] **Devireddy Sandhya , M. Chandra Sekhar Reddy** and **Veeredhi Vasudeva Rao**, A paper on “Improving the cooling performance of automobile radiator with ethylene glycol water based TiO2 nanofluids”www.elsevier.com/ locate/ichmt

[7] **D.H. Lee, L.D. Tijing, B.C. Pak** and **B.J. Baekand**, A Study on Heat Transfer Enhancement Using Straight and Twisted Internal Fin Inserts, International Communications in Heat and Mass Transfer, vol. 33, March 2013, pp.

[8] **Hardik kumar Patel** and **Deepu Dinesan**, Optimization & Performance Analysis of an Automobile Radiator using CFD, Inter. Journal for Innov. Research in Sci. & Tech., 1(7) (2014)

[9] **Jama. H, S. Watkins, C. Dixon** and **E. Ng**, “Air flow distribution through the radiator of a typical Australian passenger car” 15 th Australian Fluid Mechanics Conference 15 (2004).

[10] **John Vetrovec**, Engine Cooling System with a Heat Load Averaging Capability, SAE International, April 2008, pp

[11] **JP Yadav** and **Bharat Raj Singh**, Study on Performance Evaluation of Automotive Radiator, SAMRIDDHI-A Journal of Physical Sciences, Engineering and Technology, vol. 2, June 2015, pp.

[12] **Komalangan Krishnakumar, Navid Bozorgan** and **Nariman Bozorgan**, Numerical Study on Application of Cuo-Water Nanofluid in Automotive Diesel Engine Radiator, Modern Mechanical Engineering, vol. 2, November 2012, pp.

[13] **M. Seifi Jamnani, S.M. Peyghambarzadeh, S.H. Hashemabadi** and **S.M. Hoseini**, Improving the Cooling Performance of Automobile Radiator with Al2O3/Water Nano Fluid, Applied Thermal Engineering, vol. 31, February 2011, pp.

[14**] N. Galanis, C.T Nguyen, G. Roy** and **C. Gauthier**, Heat Transfer Enhancement Using Al2o3-Water Nanofluid for an Electronic Liquid Cooling System, Applied Thermal Engineering, vol. 27, 2007, pp.

[15] **Nor Azwadi CheSidika,, Muhammad Noor Afiq WitriMohdYazida** and **Rizalman Mamatb**, "Recent advancement of nanofluids in engine cooling system".

[16] **Oliet. C, A. Oliva, J. Castro** and **C. Pérez-Segarra**, "Parametric studies on automotive radiators" Applied Thermal Engineering, 27 (2007)

[17] **S. Kakac** and **A. Pramuanjaroenkij**, "Review of convective heat transfer enhancement with nanofluids", International Journal of Heat and Mass Transfer 52 (2009)

[18] **S. Cheong, D. Kim,Y. Kwon, Y. Cho** and **Y. Hwang**, Convective Heat Transfer Characteristics of Nanofluids under Laminar and Turbulent Flow Conditions, Current Applied Physics, vol. 9, 2009, pp.

[19] **S. Zenial Heris, M. Shokrogzar, S.Poorpharhang, M. Shanbedi** and **S.H. Noie,** "Experimental study of heat transfer of a car radiator with CuO/Ethylene glycol-water as a coolant" Journal of Dispersion Science and Technology (2014).

[20] **Siraj Ali Ahmed, Mehmet Ozkaymak , Adnan Sözen , Tayfun Menlik** and **Abdul karim Fahed** "Improving car radiator performance by using TiO2-water nanofluid" www.elsevier.com/locate/jestch

[21] **Yiding Cao** and **Khokiat Kengskool**, "An automotive radiator employing wickless heat pipes" Florida International University, Miami, Conference Paper, 1992.