DESIGN, ANALYSIS AND FABRICATION OF F1 CAR FOR IMPROVING AERODYNAMIC EFFICIENCY

A PROJECT REPORT

Submitted By

D.VENKATA RAMANA REDDY	**(18104084)**
JS. KARTHIKEYAN	**(18104118)**
G. MONISH KUMAR	**(18104119)**
G. ANURAG PARIDA	**(18104123)**

in partial fulfilment for the award of the degree of

BACHELOR OF TECHNOLOGY
IN
AUTOMOBILE ENGINEERING

DEPARTMENT OF AUTOMOBILE ENGINEERING

SCHOOL OF MECHANICAL SCIENCES

HINDUSTAN INSTITUTE OF TECHNOLOGY& SCIENCE

PADUR, CHENNAI – 603103

MAY 2022

BONAFIDE CERTIFICATE

Certified that this project titled "**DESIGN, ANALYSIS AND FABRICATION OF F1 CAR FOR IMPROVING AERODYNAMIC EFFICIENCY"** is the bonafide work of **D. VENKATA RAMANA REDDY (18104084), JS. KARTHIKEYAN (18104118), G. MONISH KUMAR (18104119) G. ANURAG PARIDA (18104123)** who carried out the work under my supervision. Certified further that to the best of my knowledge the work reported herein does not form part of any other thesis or dissertation on the basis of which a degree or award was conferred on an earlier occasion on this or any other candidate.

HEAD OF THE DEPARTMENT	**SUPERVISOR**
Dr. M. Jaikumar	Dr. C.Prasad
Professor and Head	Assistant Professor (SG)
Dept. of Automobile Engineering	Dept. of Automobile Engineering
Hindustan Institute of Technology & Science	Hindustan Institute of Technology & Science
Padur, Chennai - 603103	Padur, Chennai – 603103

The Project Viva-Voce Examination is held on_______________

INTERNAL EXAMINER **EXTERNAL EXAMINER**

ACKNOWLEDGEMENT

we express our heartfelt gratitude to our Chancellor **Dr. Mrs. Elizabeth Verghese**, Pro- Chancellor **Dr**. **Anand Jacob Verghese** and Director **Mr. Ashok Verghese**, the Management of Hindustan Institute of Technology and Science for providing all the facilities to carry out this project. We also express my sincere thanks to our Vice Chancellor **Dr. S.N.Sridhara** for approving and allowing us to undertake this project.

We wish to express our heartfelt gratitude to **Dr. M. Jaikumar, Professor & Head, Department of Automobile Engineering** for much of his valuable support encouragement in carrying out this work.

We would like to thank our internal guide **Dr. C.Prasad, Assistant Professor (SG), Department of Automobile Engineering** for guiding and actively participating in our project, giving valuable suggestions to complete the project work.

We would like to thank our project coordinator **Dr. V. Ramanathan, Assistant Professor(SG), Department of Automobile Engineering** for guiding and actively participating in our project, giving valuable suggestions to complete the project work.

D.VENKATA RAMANA REDDY

JS. KARTHIKEYAN

G. MONISH KUMAR

G. ANURAG PARIDA

ABSTRACT

This paper discusses the capabilities of front and rear wings to improve the aerodynamic properties of a sports car exposed flow. In such conditions, the aerodynamic drag and lift both increase with the introduction of a side force and a yawing moment, which contribute to the decrease of the car’s handling properties and force the car to change its driving path. Formula vehicle becomes very known to almost anyone due to peoples having so much love and passion on Racing and Automobiles vehicles.

Aerodynamics plays an important role in efficiency of the vehicle and engine performance. To maximize the performance of the vehicle, the aerodynamics forces acting on the automobile vehicle and how to utilize those forces for increasing the performances and stability. A wing or car spoiler is accessory that generally attached to the front and rear end of the automobile’s vehicles like car, and normally mounted on top of a car's trunk or positioned under the front bumper.

The low-pressure zone at back end creates drag force on vehicle is overcome by using rear Spoiler. The different designs of front and rear spoiler used are based on the different type of the automobile vehicles used, therefore aerodynamic shape of the automobile bodies and the point of the front and rear spoiler is important in this analysis

The model has been scaled down to do aerodynamic analysis using the wind tunnel testing equipment. The prototype model has been 3D printed for the analysis and been tested in the wind tunnel testing equipment.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1AERODYNAMICS

It is "a branch of dynamics that deals with the motion of air and other gaseous fluids and with the forces acting on bodies in motion relative to such fluids.

aerodynamics doesn't do justice to its scope, when you consider that aerodynamics is a primary contributor to the thrill and excitement in Formula 1 racing. The six minutes of weightlessness in store for passengers on Virgin Galactic sub-orbital space flights is only possible because of its innovative aerodynamic design, which is based on the X-Prize winning White Knight and SpaceShipOne combination designed by Burt Rutan's Scaled Composites, LLC.

Aerodynamics is an integral part of everyday life too, from the mundane use of a hair dryer to the airplane carrying you to your vacation destination. So, aerodynamics is important and exciting, but what is it?

Aerodynamics is the study of objects (cars, airplanes, etc.) moving relative to a fluid, such as air. This also includes stationary objects where the fluid is moving, such as a hair dryer or wind tunnel. Lift and drag forces are arguably the two most important performance measures studied in aerodynamics. Lift is the force directed perpendicular to the direction of motion relative to the fluid. Drag is the force directed parallel and in opposition to the direction of motion.

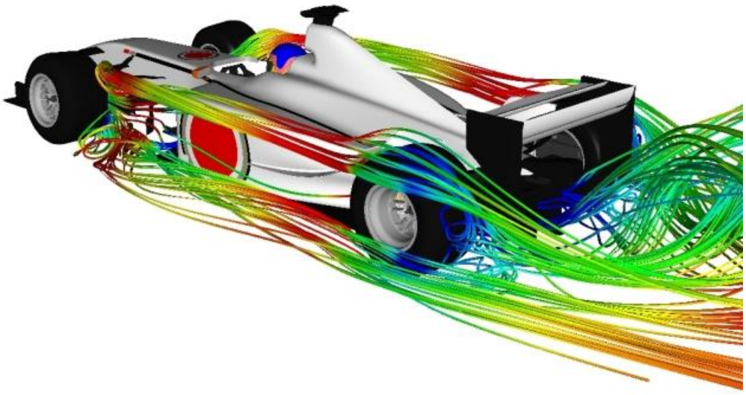

Figure 1.1 Flow trajectory (SOURCE: https://jamracingf1.wordpress.com/2013/04/02/f1-newbie-downforce-corner)

1.2 F1 AERODYNAMICS

A modern F1 race car is capable of reaching speeds of 360 kph, generating steady state cornering forces of 2.5g to 3.0g and straight-line braking forces of 3.5g to 4.0g. Such braking and cornering capabilities require the generation of a large amount of aerodynamic downforce. In order to generate sufficient levels of downforce, the cars are equipped with front and rear wings, and specially contoured bodywork and underbodies, all operating in close proximity to the ground. Due to the strong ground effect and wheels fully exposed to the airflow, the nature of the flow field around a F1 vehicle is highly complex. The presence of large body and tyre wakes, strong vortices and separated flow regions all contribute to create a complicated bluff body aerodynamic flow scenario.

SAUBER PETRONAS is currently building its own state-of-the-art wind tunnel to be in full operation by 2004.

With an unusually large test section, even full-size testing will be feasible. The team has acknowledged that such an investment must also include added CFD capabilities.

In fact, since 2000, the capacity of the CFD group has steadily increased. This long-term investment and dedicated work have already had significant impact on the aerodynamic development strategy adopted at SAUBER PETRONAS. Through extensive validation, comparing CFD results with wind tunnel measurements and race car data, new procedures for CFD analysis and design have been established.

Today, CFD is not only used for verification but actually leading the aerodynamic development in several areas. As an example, initial wing design is now almost exclusively done by the CFD department.

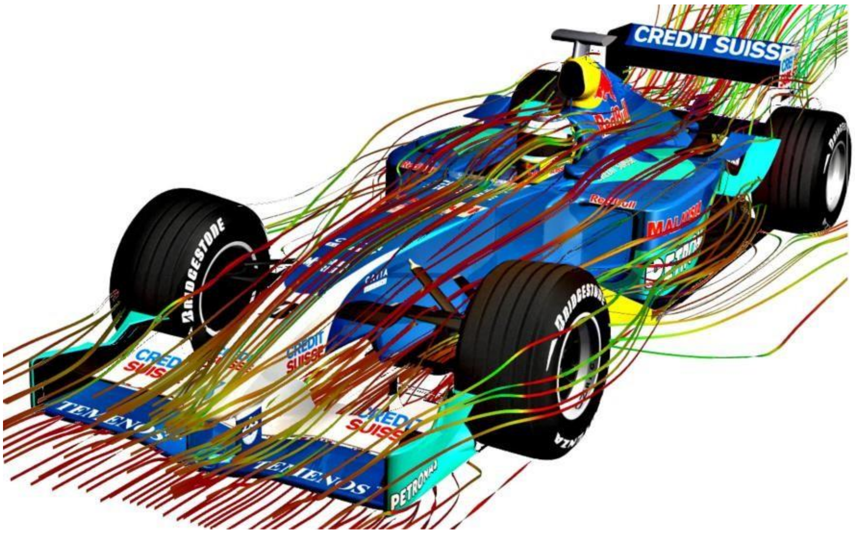

Figure 1.2 Streamlines coloured by velocity around the 2003 SAUBER PETRONAS (SOURCE: https://www.euroaviaforlibologna.eu/2020/12/04/flow-viz/)

Detailed full car simulations can provide a vast amount of information and insights not easily extracted from traditional wind tunnel measurements. Still, these complex models cannot, in a practical manner, be used for analyses of complete aero maps, which can include variations of yaw, roll and steer angles, wing settings as well as different ride height set-ups. However, even a rather small matrix of full car CFD simulations can give valuable guidance to various development routes and concepts to be further explored in the wind tunnel.

It is not likely that these advanced CFD simulations will, in the near future, reduce the number of hours spent in the wind tunnel.

But indisputably, CFD helps to improve wind tunnel testing productivity Results from CFD full car simulations can also preferably provide initial conditions and boundary conditions to smaller sub-models. This technique can significantly reduce the turnaround times and allow for more design iterations to be done within a given time frame without sacrificing too much simulation accuracy. FLUENT has proven to be a very flexible and robust tool, which has helped to create an efficient analysis and design environment.

1.3 FRONT WING DEVELOPMENT

Front wing aerodynamics is one of the most complex elements of Formula One car aerodynamics. Through the history of Formula One, the front wing has developed from a simple single element wing into a highly three-dimensional, multi-element high lift device. Endplates are visibly sophisticated with various fins and plates fitted, which are clearly not

intended only to influence the performance of the front wing. Apart from the performance of its own contribution, the influence of a front wing system to the overall aerodynamics is nowadays a major factor in its development and optimisation.

The most obvious function of a front wing is to produce downforce on the front end of the car. The wing itself generally produces approximately 25 – 30 % of the total car downforce. Besides its contribution to the overall downforce, the front wing also works as an adjustable counterbalance to the rear wing load. Since the aerodynamic balance of a car is generally tuned with the front wing loading level, it needs to have an adequate range of operating loading levels. It is typically a high lift system, thus, combined with the presence of ground in proximity, it does not pose a significant challenge in finding a wing system that produces sufficient level of downforce itself. The story suddenly becomes painfully different however, when such a front wing system is placed on the front-end of a car.

As the foremost device that disturbs the incoming (from the car point of view) airflow, it prevents the rest of the car to see a preferable ‘clean’ flow.
As a by-product of the high downforce production, it trails severe disturbances to the downstream flow field, affecting the potential aerodynamic performance of the rest of the car.

Further to its complex role in the overall aerodynamics, strict dimensional limitations by the FIA technical regulation adds even more challenges in the front wing development. The latest regulation requires any bodywork forward of the front tires, that is beyond 250mm from the car centreline, to lie within a specified box as illustrated by Figure 3. The development effort is therefore focused on the search for a front wing system that optimises the overall aerodynamic performance while producing satisfactory levels of downforce and complying with the regulations. This is where the true challenge lies in the front wing development.

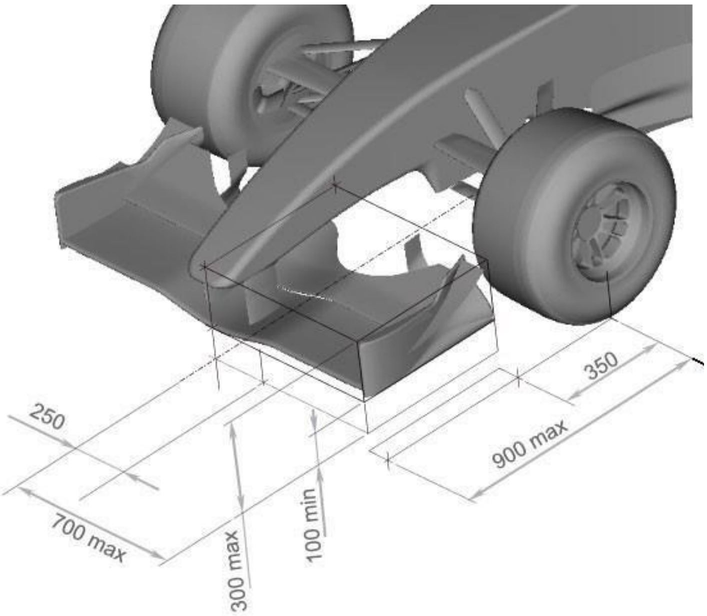

Figure 1.3 FIA technical regulation for front-end dimensions

(https://www.researchgate.net/figure/FIA-technical-regulation-for-front-end-dimensions_fig2_315717371)

The capability of CFD suits the needs for conducting analysis of a front wing, given the nature of its aerodynamic characteristics. CFD has its clear advantages of producing numerous flow field and surface data that are difficult to obtain from a physical experimental method. Flow visualization, especially of off-surface properties, can be easily performed and it provides vital information on the flow field downstream of a front wing.

With surface data, local values such as span-wise load distribution of a wing, pressure distributions over components can be obtained. Such local analysis leads to a further understanding of interactions of aerodynamic components with the overall aerodynamics.

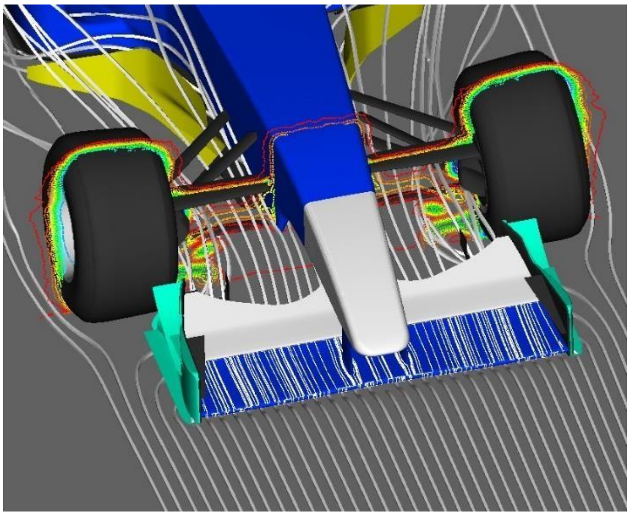

Figure 1.4 Sample image of flow visualization https://www.racecar-engineering.com/advertisement/fluid-dynamics-in-motorsport

Considerations involved in the development of a CFD model for front wing analysis are as complex as front wing aerodynamics itself. As previously discussed, understanding of its interactions with the overall aerodynamics is the key to a successful development of an optimum front wing. Clearly, stand-alone analysis is by far an inadequate approach. An ideal model candidate for this task would be of a full car with every component of a car in place. It is, however, currently an unreasonable approach that would lead to prohibitive computing times. Thus, logical compromises must be made to simplify the model in order to maintain the required computation time within reason, while maximizing the validity and viability of the solution.

Careful choice of meshing techniques, such as prism layers and refinements, is also crucial to capturing essential flow field phenomena without significant increase in the model size.

Determining the precise demand of mesh type and size is vital in achieving an efficient and effective model. Further to the modelling and meshing considerations, boundary set-up also plays a very important role in obtaining a realistic solution. A front wing in particular, operates very near the objects in motion from its point of view, namely the ground and the front tires. Minute differences in their boundary set-up and surface properties heavily impacts the solution validity. These are thus matters of investigation and an optimum set-up needs to be explored.

At SAUBER PETRONAS, CFD front wing analysis utilizes a model that incorporates all the considerations discussed. It allows the computation to be carried out in an efficient manner, thus various front wing configurations can be studied and compared in a reasonable time frame. Validity of solutions is confirmed by comparing with track and experimental data. Analysis is focused primarily on visual observations of specific properties of interest, such as vortices, streamlines, and pressure contours (Figure 4) that are difficult to obtain from experimental methods. Combined with the extensive experimental data, these insights provide broader approaches to be taken in the front wing development and are keys to understanding the flow mechanisms involved.

1.4 REAR WING DEVELOPMENT

The prime function of the rear wing is to generate aerodynamic downforce, allowing for increased cornering speeds. To achieve the correct aerodynamic balance and stability of the car, the rear wing down force is balanced with front wing adjustments.

Before the 2001 season, the FIA regulations governing the rear wing design became more restrictive. The intent was to lower attainable downforce levels, and thereby reduce cornering speeds. The rear wing assembly now has to comply with the dimensions of the two regulated boxes as drawn in Figure 5. The upper box can have at most a three-element wing configuration, whereas only one element is allowed in the lower box. This puts strict limitations on obtainable wing surface area and camber distributions.

Given the FIA regulations, including structural stiffness constraints, the task for the aerodynamicists and the CFD engineers is to design the most *efficient* rear wing assemblies over a range of down force levels to suit the race tracks throughout the season. High aerodynamic efficiency (*Downforce to Drag Ratio*) means achieving the necessary down force with the lowest possible associated drag penalty.

Figure 1.5: FIA rear wing regulation boxes. (SOURCE: https://fetchcfd.com/view-project/1603)

Through CFD optimisation, two-dimensional profiles and multi-element configurations with specific favourable aerodynamic characteristics are initially developed. A two-dimensional analysis of high- lift multi-element air foils is already an elaborate task. Strong interactions between boundary layers and wakes, laminar to turbulent transition, laminar and turbulent separation, all represent important physical phenomena that must be properly addressed on a high-lift air foil. Here, areas such as grid resolution and turbulence modelling continue to play key roles.

1.5 AERODYNAMICS FORCES

Drag Force As discussed earlier, the body shape when it exposed to air experience the different pressure on the surface. By breaking this into five constituent elements will gives a better understanding of drag force acting on the vehicle body.

1.6 PRESSURE DRAG

This is the component which is identified on the external surface of the car. As when the vehicle moves with the forward direction of the air then the surface of the car experience the pressure which is vary over the different points of the car.

To have a look it very closely the small area of the flat surface is considered then the force which is acting on the axis of the car the drag force depends on the magnitude of the pressure.

1.7 SURFACE DRAG

This type of drag is due to the stress and drag values which is from the friction between air and the body surface for a small element. This type of drag only happens due to the effect of viscosity at the surface of the car.

1.8 LIFT

In the simple words lift in the vehicle is the pressure difference between the upper and the down side of the car. Nowadays higher top speed modern cars are manufactured which high stability need while on the road. Upper surface area near the hood, wind shield and underbody such as suspension, exhaust system are the main dependents of lift force. Studies shows that it is not the common problem at the low speed but when the vehicle goes to the high speed and then the pressure difference is a lot then lift force is the problem.

CHAPTER 2

LITERATURE REVIEW

Bienz, et al… (2003), In Front of the Grid - CFD at SAUBER PETRONAS F1 Leading the Aerodynamic Development**.** At SAUBER PETRONAS Engineering AG, CFD is now playing a vital role in the aerodynamic development of the Formula One racing car. CFD has successfully been integrated into the aero design process and has become an integral part alongside wind tunnel measurements and track testing. Detailed validation work, comparing CFD simulations with experimental results and track data, has led to high confidence in the adopted CFD procedures. In many areas, CFD is nowadays leading the aero development. This presentation will highlight several examples where CFD led analysis and design, using the Fluent CFD software suite, translated into new components leading to improved car performance on the race circuit. This includes wing development, detailed full car simulations, and brake cooling analysis. Moreover, new insights regarding off-surface flows and component interactions gained through CFD will be discussed.

Mustafa Cakir (2012), did CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle**.** Aerodynamic characteristics of a racing car are of significant interest in reducing car-racing accidents due to wind loading and in reducing the fuel consumption. At the present, modified car racing becomes more popular around the world. Sports cars are most commonly seen with spoilers, such as Ford Mustang, Subaru Impreza, and Chevrolet Corvette. Even though these vehicles typically have a more rigid chassis and a stiffer suspension to aid in highspeed manoeuvrability, a spoiler can still be beneficial. One of the design goals of a spoiler is to reduce drag and increase fuel efficiency. Many vehicles have a fairly steep downward angle going from the rear edge of the roof down to the trunk or tail of the car. Air flowing across the roof tumbles over this edge at higher speeds, causing flow separation. The flow of air becomes turbulent and a low-pressure zone is created, thus increases drag. Adding a spoiler at the very rear of the vehicle makes the air slice longer, gentler slope from the roof to the spoiler, which helps to reduce the flow separation. Reducing flow separation decreases drag, which increases fuel economy; it also helps keep the rear window clear because the air flows smoothly through the rear window. The limitations of conventional wind tunnel experiment and rapid developments in computer hardware, considerable efforts have been invested in the last decade to study vehicle aerodynamics computationally. This thesis will present a numerical simulation of flow around racing car with spoiler positioned at the rear end using commercial fluid dynamic software ANSYS FLUENT®. The thesis will focus on CFD-based lift and drag prediction on the car body after the spoiler is mounted at the rear edge of the vehicle. A 3D computer model of 4-door sedan car (which will be designed with commercial software SolidWorks®) will be used as the base model. Different spoilers, in different locations will be positioned at the rear end of vehicle and the simulation will be run in order to determine the aerodynamic effects of spoiler**.**

Mohammad Mohammadi, et al. (2012), studied on various gurney flap shapes on the performance of wind turbine air foils**.** This paper gives an overview on two-dimensional numerical investigation and comparison of aerodynamic characteristics of small flaps used to increase lift on wind turbine air foils. The small flaps consist of Gurney flaps, trailing edge wedges and a devised trailing edge curved shape. The investigations were performed for a diversity of lengths and heights of these flaps on the air foil. Extensive numerical simulations have been done using RANS model using SST-Transitional turbulence model using a commercial CFD code, a CFD finite-volume based software, at the Reynolds number of 2 × 106. The results confirmed advantages of using the trailing edge curved shape over the Gurney flap, which will be more efficient as the flap height is increased.

Ali Asgar S, et al. (2014), designed and analysis of bodyworks of a formula style race car**.** The aim was to develop a body of the race car with the proper studies and analyses, taking into account several factors, to present an optimum structure as a final result. These factors include, but are not limited to, weight, cost, drag resistance, functionality and aesthetics. The expected product is to not just be appealing to the eye but also increase the performance of the vehicle.

Additional objectives include being able to accommodate the budget while maintaining a highly competitive level to perform well in on the race track. The new design will reduce the weight of the prototype and as well as the air drag, taking into consideration the ground effects desired to be implemented in the vehicle as a crucial factor. Moreover, the new body will be easier to dismantle reducing the service time.

Shobhit Senger, et al. (2014), Aerodynamic Design of F1 and Normal Cars and Effect on Performance**.** Aerodynamics is a branch of dynamics concerned with studying the motion of air, particularly when it interacts with the solid object. Aerodynamic is a subfield of fluid dynamics and gas dynamics, with much theory shared between them. In this paper we are presenting the different forces acting on a car (drag force, lift force). the measurement of forces (computational fluid dynamics CFD and wind tunnel testing WTT). On the basis of forces and measurement the comparison has been done over Hindustan ambassador,

Lamborghini Aventador LP 700-4 and the F1 car. The design of FORMULA1 is explained in this paper. After detailed observation and tests performed, we obtained that F1 car has most aerodynamic of all the vehicles. The design is made in such a way that it cuts through the air with ease and channelize the air flowing over it to the rear wings. This results in a highly reduced drag and lift force acting on the car body. It in turn, generates more amount of down force making the car stable at high speeds. It is the pinnacle of racing technology. On the other hand, the Lamborghini Aventedor LP 700-4, is a full-on super car. It was designed to give speed and performance in a coupe car. thus, the body had to be designed such that there is minimum air resistance at high speed and proper cornering stability as well as drivability.

Akshay Khot, et al. (2018), Analysed of Aerodynamic Characteristics in Front Wing of F1 Car Using CFD. In our Project we are improvising on the design of the front wing of a Formula 1 Vehicle. This helps in improving drag and down force which adds stability to the vehicle. Especially on turns/corners at high speeds. Having done CFD on the front wing we understand better designs as we learn the flow patterns and its effect our vehicle. Thus, comparing different designs helped us to achieve higher efficiency of front wing.

Singh, et al. (2019) reviewed of “performance analysis and optimization of car air spoilers. Before last decade, due to surplus amount of fuel, cars are designed for high-speed operation, comfort and safety. As scarcity of fuel increases due to high consumption of fuel in automobiles, many researchers started working on the idea of alternative fuels, redesign of car body and reducing aerodynamic losses. To overcome these losses spoilers are used and hence it is needed to optimize its shape.

This study is primarily focused to find out optimized shape of car spoiler so that mass can be minimized which help in reducing fuel consumption without affecting its aerodynamic properties and strength. Computational Fluid Dynamics (CFD) analysis of two-dimensional model of spoiler is done whose results are validated by earlier research work in this field for understanding variation of aerodynamic property of cross-section. Three dimensional CFD analysis of spoiler gives aerodynamic property and pressure data which is needed to compare result of optimized model formed by optimized cross-sectional shape. Shape optimization is done by Shape Optimization tool of ANSYS 14.0 which is further tested for design failure in ABAQUS 6.11. The result of shape optimization has saved 18.74% of material keeping all its strength and aerodynamic property intact. This study open provides a numerical tool for improvement of future model of spoiler in terms of reduction of mass.

Killi Surya Sivaji, et al. (2019), In automobile the aero design grabs the priority after getting to know the importance. Where we can counter the drawbacks by optimizing the design. The design contribution may indirectly improve the automobile efficiency. In this paper we brief the air drag due to design and the better design for automobile with five tire systemin comparison with four tire system. This system reduces the air friction by limiting the front face of the vehicle, by maintaining its stability.

Janusz Piechna, et al. (2019), Influenced of Side Spoilers on the Aerodynamic Properties of a Sports Car. This paper discusses the capabilities of side spoilers to improve the aerodynamic properties of a sports car exposed to a non-zero yaw angle flow.

In such conditions, the aerodynamic drag and lift both increase with the introduction of a side force and a yawing moment, which contribute to the decrease of the car's handling properties and force the car to change its driving path.

Elements mounted on the side of the car make it possible to obtain an asymmetric aerodynamic load distribution and generate additional forces that can be used to counter these effects. The performance of the side spoilers was analysed at yaw angles ranging from 0◦ to 15◦ using the results of numerical calculations. It was established that the side spoilers made it possible to generate at low yaw angles aerodynamic forces that exceeded those caused by a crosswind.

Ahmed T. Raheem, et al. (2019), Aerodynamics of a Formula One Car Front Cascade Wing during Cornering**.** The design for the aerodynamics of front wing based on cornering angle of a Formula One car plays an important role on the car's performance. The cascade elements above the main front wing is enhanced with new and the deep winglets that manage the airflow for the rest of the car. The front wing cascades are attached with end plates at the extremities of the front wing to reduce turbulence. However, when the car turns, the airflow behaviour on these cascades wings changes significantly. This paper presents the aerodynamic characteristics resulting from the cornering forces subjected on Formula One styled cascade wings. The study attempts to predict the down-force and drag-force acting on the front wing under the effect of airflow change in this area. In order to investigate the airflow behaviour, computational fluid dynamics (specifically the ANSYS software) was used to simulate selected cases with specific surface definitions and boundary conditions defined in a 3D domain.

Piyush Chavda, et al. (2020) Analysed of Aerodynamics Effects of a Rear Wing/Spoiler of Formula –1Car.Formula vehicle becomes very known to almost anyone due to peoples having so much love and passion on Racing and Automobiles vehicles. Aerodynamics plays a important role in efficiency of the vehicle and engine performance. To maximize the performance of the vehicle, the aerodynamics forces acting on the automobile vehicle and how to utilize those forces for increasing the performances and stability.

A wing or car spoiler is accessory that generally attached to the rear end of the automobile's vehicles like car, and normally mounted on top of a car's trunk or positioned under the front bumper. The low-pressure zone at back end creates drag force on vehicle is overcome by using rear Spoiler. The different designs of rear spoiler used are based on the different type of the automobile vehicles used, therefore aerodynamic shape of the automobile bodies and the point of the rear spoiler is important in this analysis. In this study, we have selected formula-1 car spoiler for our analysis purpose.

Pranav Salhan (2020), studied of the front-end downforce enhancing aerodynamic elements in sports cars. While designing sports cars, getting the aerodynamic balance of the vehicle is very important. Aerodynamic elements to create rear downforce are studied extensively while, this research tries to recognize the various ways of developing effective front end down force producing devices. The most widely used add-on devices include the front splitter, canards and underbody vortex generators. CFD simulations are employed to study these devices on the widely acknowledged MIRA fastback model. ANSYS Fluent CFD software was used with the efficient k- ε model to get accurate results while avoiding expensive experimentation costs. Reduction in lift force with the help of these elements generally leads to high drag force. This study aims to find out the most suitable and versatile devices. The average lift decrement on the addition of front aerodynamic elements was $\Delta Cl = 0.0778$ while the increase in drag was $\Delta Cd = 0.0393$.

Xabier Castro, et al… (2020), The aerodynamic loads generated in a wing are critical in its structural design. When multi-element wings with wingtip devices are selected, it is essential to identify and to quantify their structural behaviour to avoid undesirable deformations which degrade the aerodynamic performance. This research investigates these questions using numerical methods (Computational Fluid Dynamics and Finite Elements Analysis), employing exhaustive validation methods to ensure the accuracy of the results and to assess their uncertainty.

Firstly, a thorough investigation of four baseline configurations is carried out, employing Reynolds Averaged Navier–Stokes equations and the kω SST (Shear Stress Transport) turbulence model to analyse and quantify the most important aerodynamic and structural parameters. Several structural configurations are analysed, including different materials (metal alloys and two designed fibre-reinforced composites). A 2022 front wing is designed based on a bidimensional three-element wing adapted to the 2022 FIA Formula One regulations and its structural components are selected based on a sensitivity analysis of the previous results. The outcome is a high-rigidity-weight wing which satisfies the technical regulations and lies under the maximum deformation established before the analysis. Additionally, the superposition principle is proven to be an excellent method to carry out high-performance structural designs**.**

Zhaowen Deng, et al. (2020), Reviewed of effects the rear spoiler aerodynamic analysis on ground vehicle performance**.** Recently reduce in fuel consumption and concern regarding the ground vehicle safety, automotive engineers are faced with the immediate task of introducing more efficient aerodynamic performance vehicles. Hence, the rear spoiler attached on the ground vehicle was presented. Rear spoiler is a device commonly found on road vehicle for improving their aerodynamic performance. Review on the research effects of the rear spoiler aerodynamic analysis on ground vehicle performance is reported in this paper. This review intends to enhance the stability and safety by attaching the rear spoiler on ground vehicle.

The review mainly focusses on aerodynamic analysis and control of the rear spoiler for different types ground vehicle in term of improving vehicle performance. Researches carried out by a number of researchers with regard to analyse and control the rear spoiler of ground vehicle and their effect on ground vehicle performance. Aerodynamic analysis i.e., computational fluid dynamics (CFD), design optimization, drag coefficient ($CCdd$) and lift coefficient ($CCll$) and control methods i.e., passive control and active control had been reviewed. Meanwhile, progressive research on the rear spoiler aerodynamic analysis was observed due to its flexibility for wide range of application in the different type vehicles.

Alex Guerrero, et al. (2020), Aerodynamic Study of the Wake Effects on a Formula 1 Car**.** The high complexity of current Formula One aerodynamics has raised the question of whether an urgent modification in the existing aerodynamic package is required. The present study is based on the evaluation and quantification of the aerodynamic performance on a 2017 spec. adapted Formula 1 car (the latest major aerodynamic update) by means of Computational Fluid Dynamics (CFD) analysis in order to argue whether the 2022 changes in the regulations are justified in terms of aerodynamic necessities. Both free stream and flow disturbance (wake effects) conditions are evaluated in order to study and quantify the effects that the wake may cause on the latter case. The studied behaviour suggests that modern F1 cars are designed and well optimised to run under free stream flows, but they experience drastic aerodynamic losses (ranging from −23% to 62% in downforce coefficients) when running under wake flows. Although the overall aerodynamic loads are reduced, there is a fuel efficiency improvement as the power that is required to overcome the drag is smaller. The modern performance of Ground Effect by means of vortices management represent a very unique and complex way of modelling modern aerodynamics, but at the same time notably compromises the performance of the cars when an overtaking manoeuvre is intended.

Darshan Ajuida, et al. (2021), Analysed of Aerodynamics Effects of a Rear Wing/Spoiler of Formula –1Car.Formula vehicle becomes very known to almost anyone due to peoples having so much love and passion on Racing and Automobiles vehicles. Aerodynamics plays a important role in efficiency of the vehicle and engine performance. To maximize the performance of the vehicle, the aerodynamics forces acting on the automobile vehicle and how to utilize those forces for increasing the performances and stability. A wing or car spoiler is accessory that generally attached to the rear end of the automobile’s vehicles like car, and normally mounted on top of a car's trunk or positioned under the front bumper. The different designs of rear spoiler used are based on the different type of the automobile vehicles used, therefore aerodynamic shape of the automobile bodies and the point of the rear spoiler is important in this analysis.

In this study, we have selected formula-1 car spoiler for our analysis purpose. To perform analysis, we use commercial software CREO for solid modelling of F1 car body. After that for analysis we use CFD tools. By this analysis we can find out lift and drag forces, pressure and velocity distributions. Possibly we may improve aerodynamics of F1 car body.

2.1 RESEARCH GAP

- To design and model the existing f1 car and modifying the f1 car front, rear wing and diffuser.

- To improve the aerodynamic efficiency of the F1 car by comparing the existing f1 car lift and co efficient of drag with modified f1 car.

- Design, Analysis and fabrication of the front wing, rear wing, diffuser & side pods.
- To wind tunnel test the fabricated f1 car model.

2.2 SCOPE & OBJECTIVES

- The main objective of this project is to design a F1 car for better co-efficient of drag, overall performance of a F1.
- Designing the f1 car through its geometric measures, maintain the geometric measures for particular components, change the design of some main components which are most required for drag, lift and flow purpose.
- The components are diffuser, front wing & rear wing. These components are going to re-design with help of its geometric measures, into better shapes & curves to improve the flow of air to move quicker and reduce lift, to avoid it from flying.
- After all this to do flow analysis in solidworks. fabricate the designed f1 car and test in the wind tunnel. The main objective of our project is this.
- Due to all these improvements, we can increase its speed, reduce its drag, lift.

CHAPTER 3

METHODOLOGY

DESIGN

1. F1 body
2. Front wing
3. Rear wing
4. Wheels

FABRICATION OF PROTOTYPE

1. 3D Printing of f1 model

FLOW SIMULATION

1. Computational domain
2. Meshing
3. Boundary conditions
4. Flow trajectory

WIND TUNNEL TESTING

1. 10m/s at 320rpm
2. 20m/s at 635rpm
3. 30m/s at 950

COMPARATIVE ANALYSIS

1. Lift and drag coefficient of both existing and modified model has been compared.

3.1 DESIGN

3.1.1 EXISTING F1 CAR MODEL

the modelling is done by using solidworks software. Both the models are modelled using surface modelling method. the front and rear wing of both existing and modified f1 car model has been designed. The wheels also have been designed in such a way that the front wheels will be slightly less in width compared to the rear wheel of the f1 car. and assembled to the body of the f1. In the modified f1 car model the front and rear wing have been modified for betterment of aerodynamic efficiency. All dimensions are in mm.

3.1.2 VERTICAL AIRFOILS

In this case, the aim of this small airfoil would be to dilute the super pressure located on the wheels and maybe to decrease drag but it would not be significant. This would in fact transfer some drag from the tyre to the front wing. To allow more air to be deflected inboard the wheels, the vertical airfoil should be twisted at its bottom near its trailing edge to increase the gap between the tyre and this airfoil. A greater amount of air may be deflected in this gap so that less air would have to hit the tyre and so again in drag may be obtained. To avoid the wing tip vortex at the top of the vertical airfoil, a winglet may be added at the top of the vertical airfoil. It should be done inboard (at 0.3 meter above the reference plane, i.e., at the limit allowed by the regulation or by following the airfoils elements upper surface shape to drive the air) and outboard. It could be possible to control the vortex at the tips (so at the top and at the bottom) by playing with the tip curvature of the vertical airfoil.

3.1.1 EXISTING F1 CAR

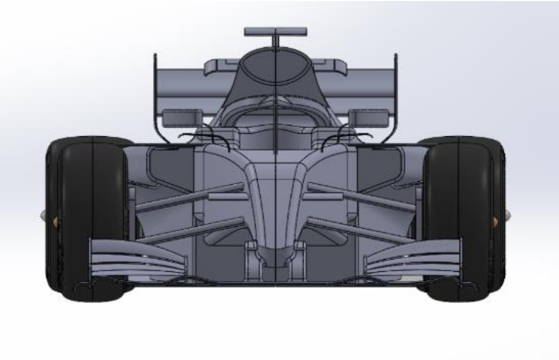

Figure 3.1 Front view of existing f1 car model

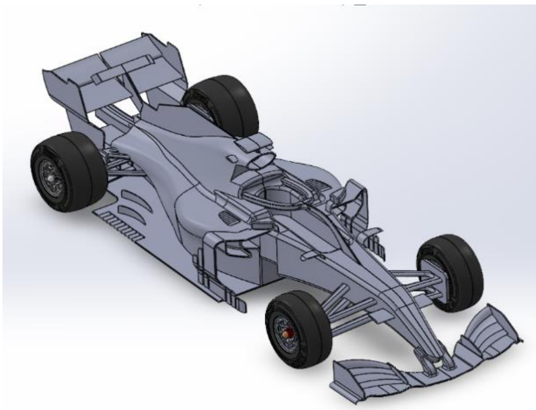

Figure 3.2 Isometric view of existing f1 car model

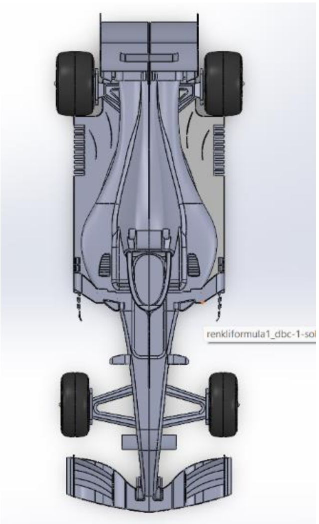

Figure 3.3 Top view of existing f1 car model

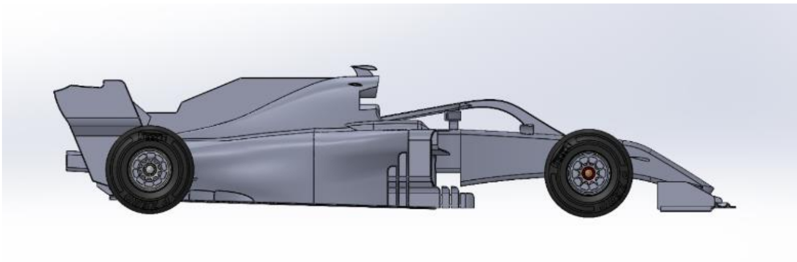

Figure 3.4 Side view of existing f1 car model

3.1.3 MODIFIED F1 CAR

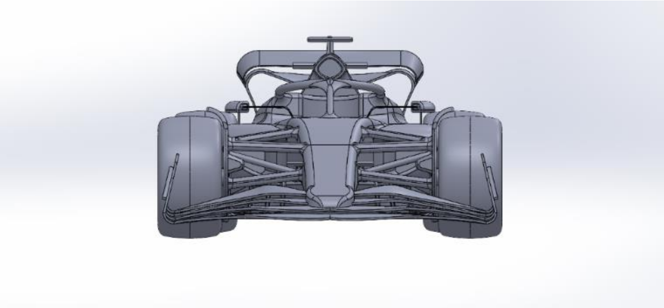

Figure 3.5 Front view of modified f1 car model

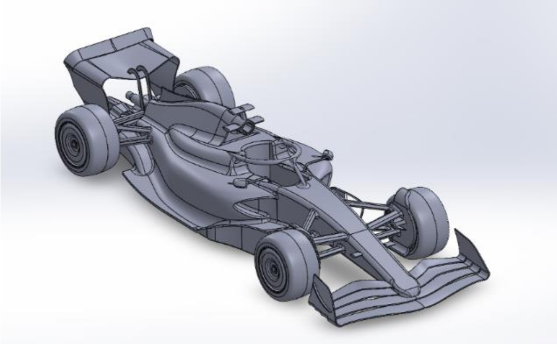

Figure 3.6 Isometric view of modified f1 car model

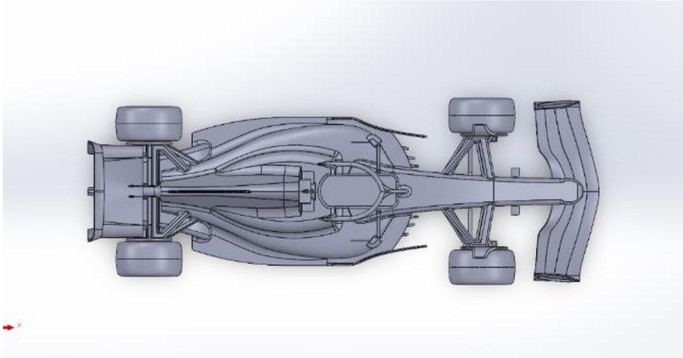

Figure 3.7 Top view of modified f1 car model

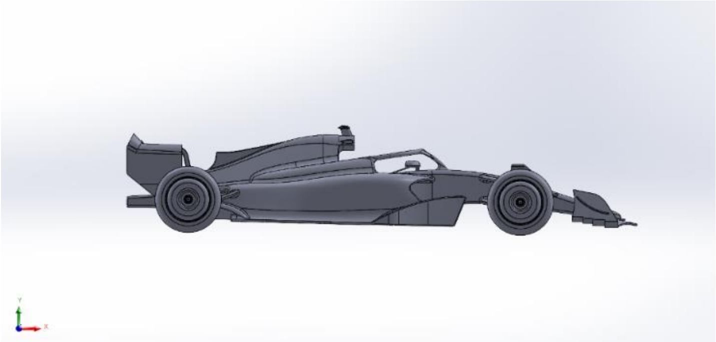

Figure 3.8 Side view of modified f1 car model

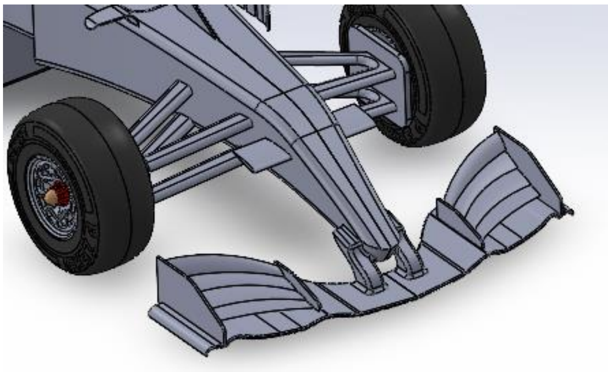

Figure 3.9 Front wing of existing f1 car model

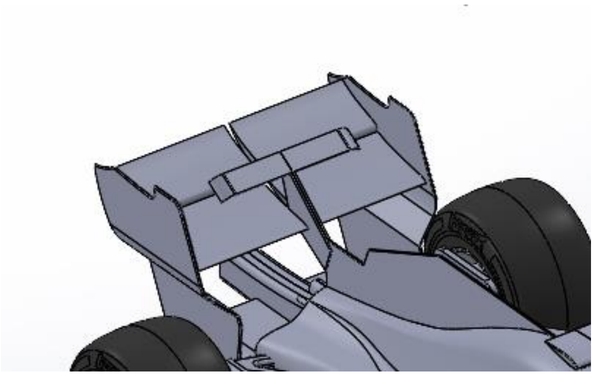

Figure 3.10 Rear wing of existing f1 car model

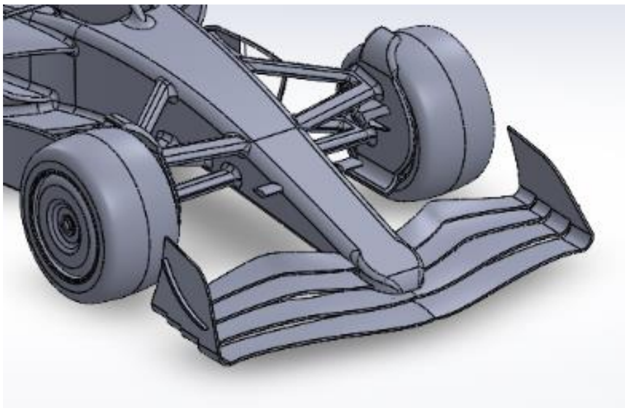

Figure 3.11 Front wing of modified f1 car model

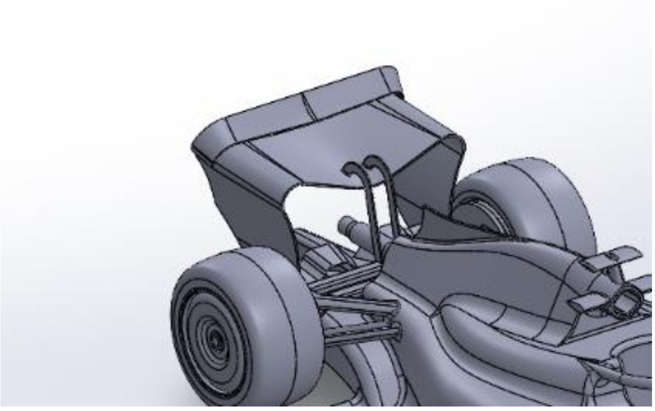

Figure 3.12 Rear wing of modified f1 car model

Actual dimensions	**Existing f1**	**Modified f1**
Wheel base	3439mm	3416mm
Total length	5345mm	5250mm
Total height	1188mm	1120mm
Total width	1763mm	1911.53mm

Table 3.1 Actual and modified dimensions of f1 model

Scaled down dimensions	**Existing f1**	**modified f1**
Wheel base	191.10mm	185.67mm
Total length	296.75mm	284.59mm
Total height	65.64mm	60.67mm
Total width	97.97mm	100.77mm

Table 3.2 Scaledown dimensions of actual and modified f1 model

3.2 FABRICATION OF PROTOTYPE

Name	Creator 3 Pro
Extruder Type	IDEX (Independent dual extruders)
Printing Precision	±0.2mm
Build Volume	300x250x200mm
Nozzle Diameter	0.4mm (0.6/0.8mm)
Print Speed	10-150mm/s
Maximum Extruder Temperature	320℃
Platform Heating Temperature	120℃
Filament	PLA (Polylactic Acid)

3.3 MATERIAL

POLYLACTIC ACID

PLA, also known as polylactic acid polylactide, is a thermoplastic made from renewable resources such as corn starch, tapioca roots or sugar cane, unlike other industrial materials made primarily from petroleum. Due to its more ecological origins this material has become popular within the 3D printing industry, we have begun to see it in medical applications and in food products. PLA filament has gained wide acceptance within additive manufacturing partly because it is made from renewable products and also because of its mechanical properties.

It is often the preferred choice for beginners in 3D printing as it is a very easy material to work with. This material, considered a semi-crystalline polymer, has a melting temperature of 180ºC, lower than ABS filament, which starts melting between 200ºC and 260ºC.

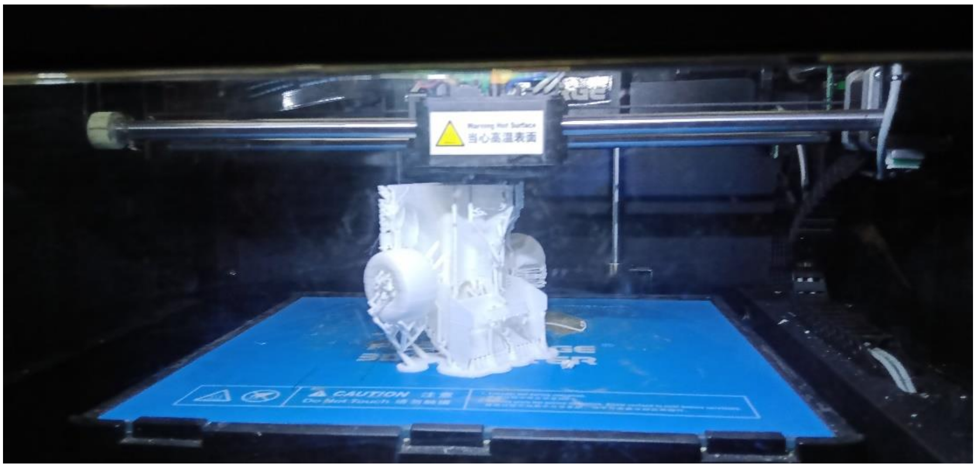

Figure 3.13 3d printing

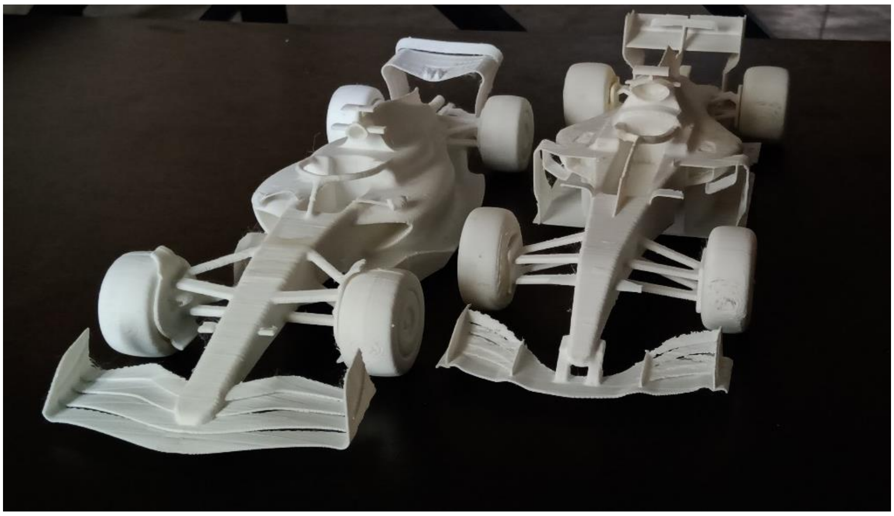

Figure 3.14 Fabricated model of both existing and modified f1 car

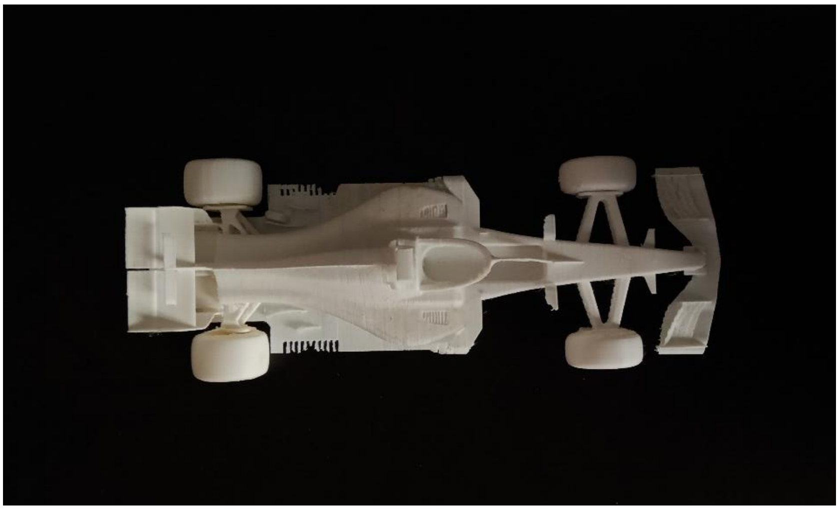

Figure 3.15 Fabricated model of existing f1 car model

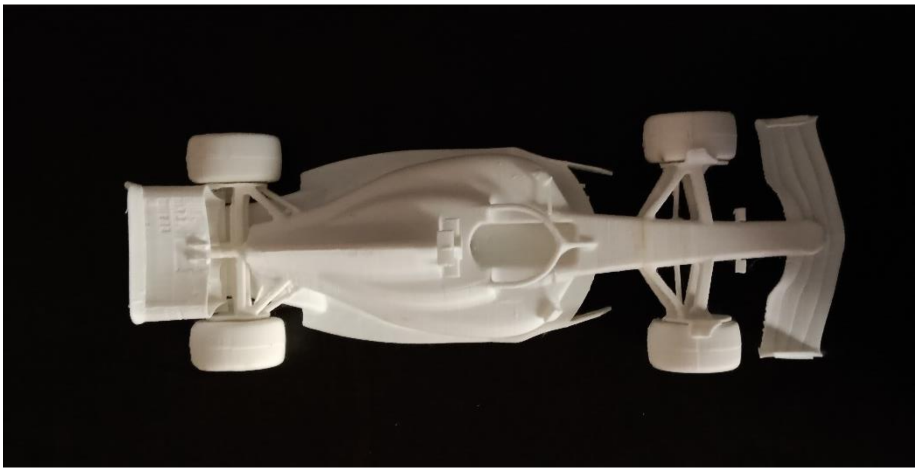

Figure 3.16 Fabricated model of modified f1 car model

3.4 FLOW ANALYSIS USING SOLIDWORKDS

The domain, meshing and boundary conditions has been created for flow simulation. Both models have been simulated at 10, 20 and 30 m/s respectively. Domain dimensions are +X(1.2m), -X(1m), +Y(0.3m), -Y(0m), +Z(0.2m), -Z(0.2m). The dimensions of the domain are setup at length where the flow of the air will be completely flow through the f1 car body.

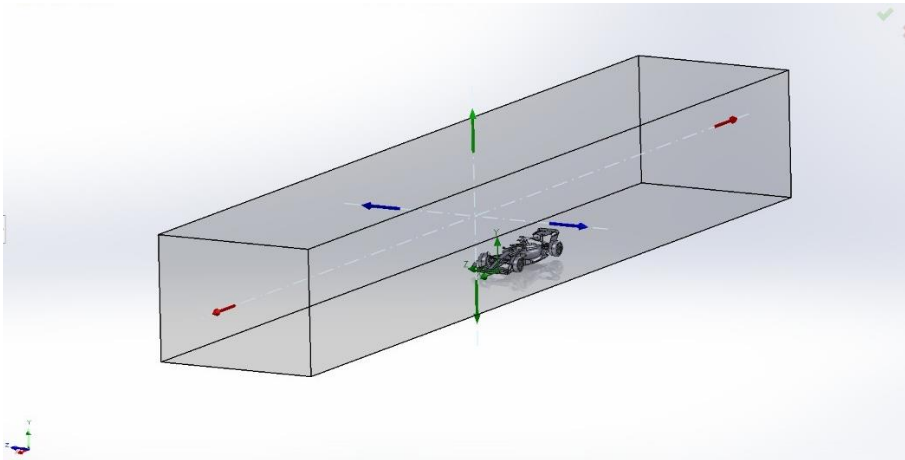

Figure 3.17 Domain

The meshing is done to improve the efficiency of the flow simulation and flow trajectory lines, Total cells 325219, Fluid cells contacting solids 116354 and Iterations 110. Global Mesh Size sets a mesh size that will be used as a default value in the process of overall meshing. Global Size is applied only as a default value. Meshing is a very crucial step in design analysis. The automatic mesher in the software generates a mesh based on a global element size, tolerance, and local mesh control specifications. Mesh control lets you specify different sizes of elements for components, faces, edges, and vertices.

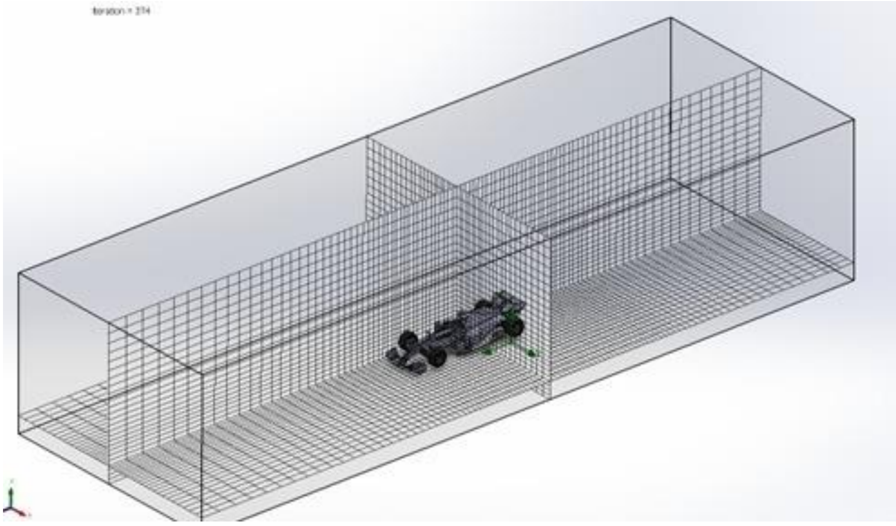

Figure 3.18 Meshing

Inlet at Z direction and outlet.

Inlet velocities are 10,20 and 30 m/s.

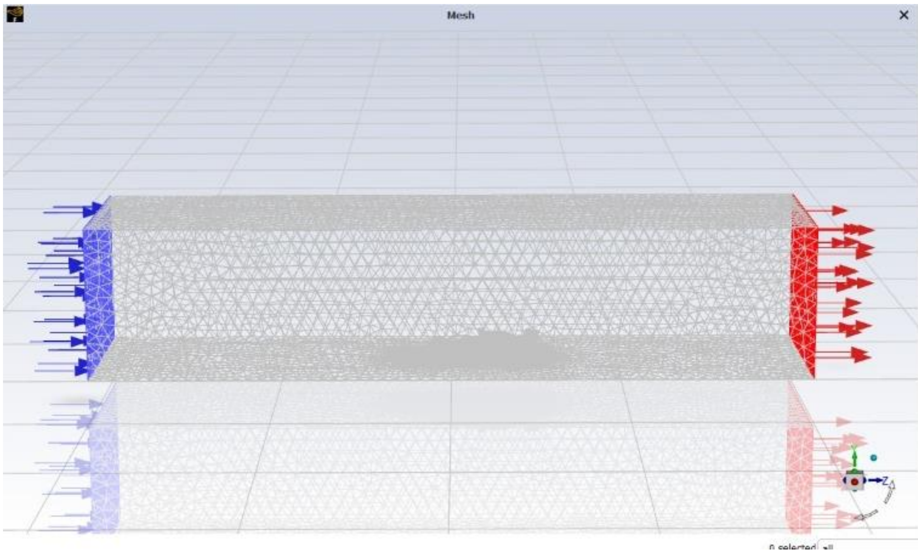

Figure 3.19 Boundary conditions

3.5 WIND TUNNEL TEST

3.5.1 EXPERIMENTAL ARRANGEMENT

The testing is done for the existing f1 car, f1 car is placed inside the wind tunnel for real time flow simulation to take readings of lift and co efficient of drag and compare the values with modified f1 testing values.

Wind tunnels reverse atmospheric circumstances out of necessity; the test subject is static while wind flows over and around the car, posing an issue with the ground surface.

A procedure using scale models loaded by fan-driven airflow to predict responses of a structure, structural components, and cladding to actual wind storm conditions.

Figure 3.20 Wind tunnel testing equipment

Air enters the tunnel through an aerodynamically designed effuse (cone) that accelerates the air linearly. It then enters the working section and passes through a grille before moving through a diffuser and then to a variable-speed axial fan. The grill protects the fan from damage by loose objects. The air leaves the fan, passes through a silencer unit and then back out to atmosphere.

A separate control and instrumentation unit controls the speed of the air velocity in the working section and supplies electrical power to other instruments.

The working area of the tunnel is a square section with a clear roof, sides and floor. The sides are removable. The floor and each side panel have a special position to support wind tunnel models. Supplied with the wind tunnel are a protractor and a model holder to support and accurately adjust the angle of any model

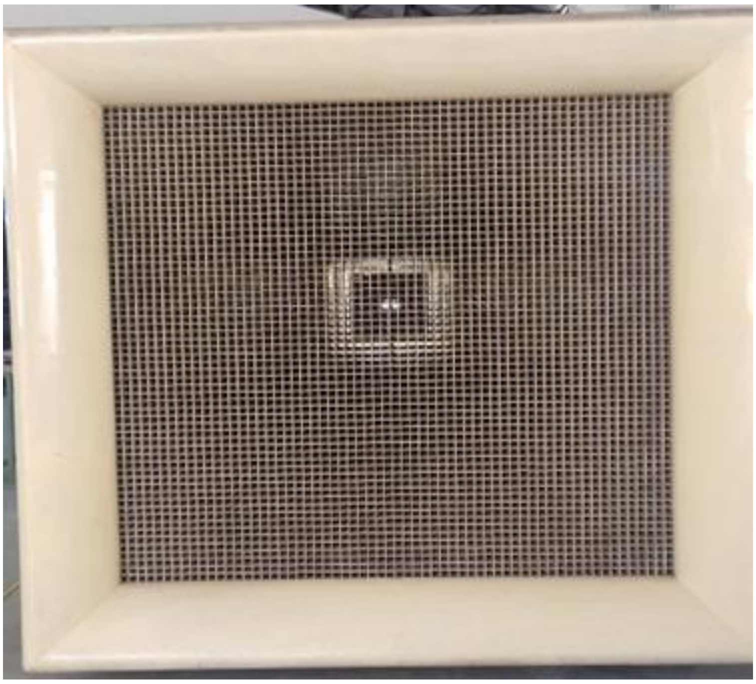

Figure 3.21 Inlet of wind tunnel

3.5.2 TEST SECTION

•Maximum operating speed – 50m/s.

•Maximum speed – 1000rpm.

•Maximum 600mm length, 300mm width & depth.

The f1 scale down car model is placed inside the wind tunnel for testing the lift and co efficient drag. It fixed with bolt and nuts to avoid the model from fly which the air comes in opposite direction.

The first test section is to air blow at 300 rpm of velocity 10 m/s.

The second test section is to air blow at 600 rpm of velocity 20 m/s.

The last and final test is to air blow at 950 rpm of velocity 30 m/s because the maximum operating speed is 50 m/s.

wind tunnel loads where measured using weight scales, much like the ones that existed in doctor's offices, and that's why today they're called balances. Balances are made of flexures that deflect with load is applied. These flexures are designed to respond to load in a particular axis

Figure 3.22 Component balance instrumentation

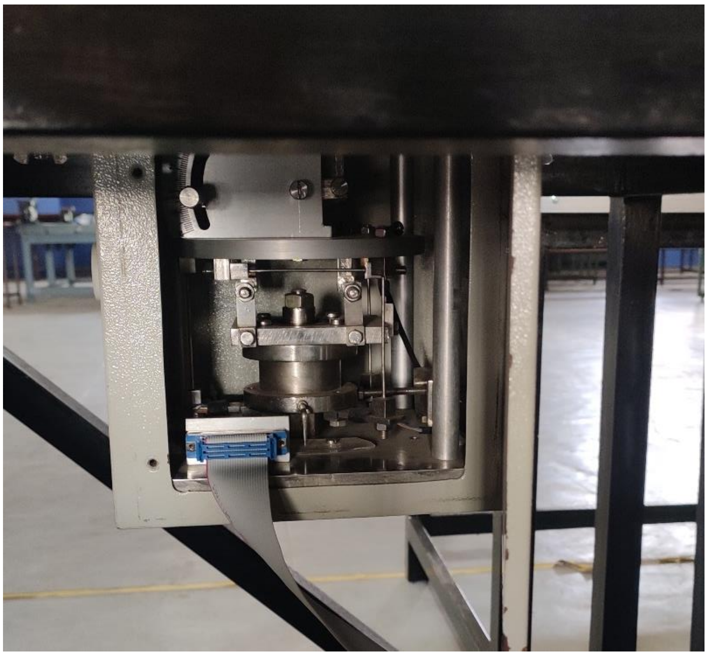

Figure 3.23 Component balance instrumentation device

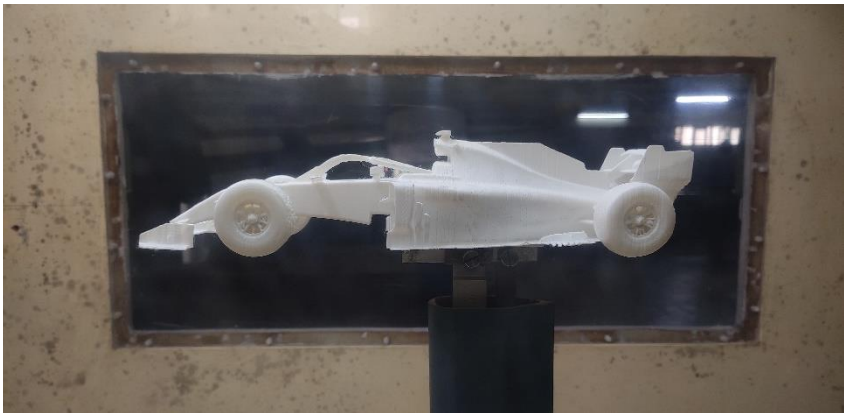

Figure 3.24 Testing of existing f1 car model

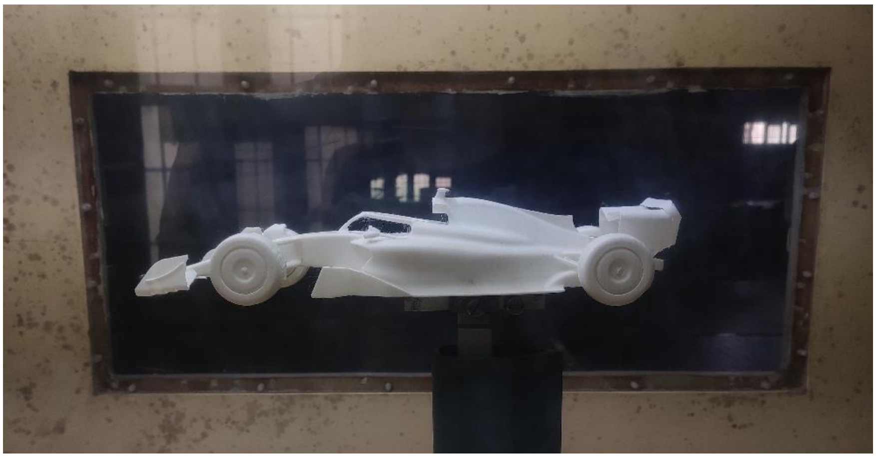

Figure 3.25 Testing of modified f1 car model

3.5.3SCALEDOWN MODEL OF EXISTING & MODIFIED F1 CAR

The models are scaled down at a scale of 1/18 for wind tunnel testing. The comparison of both existing & modified f1 car models. All dimensions are in mm.

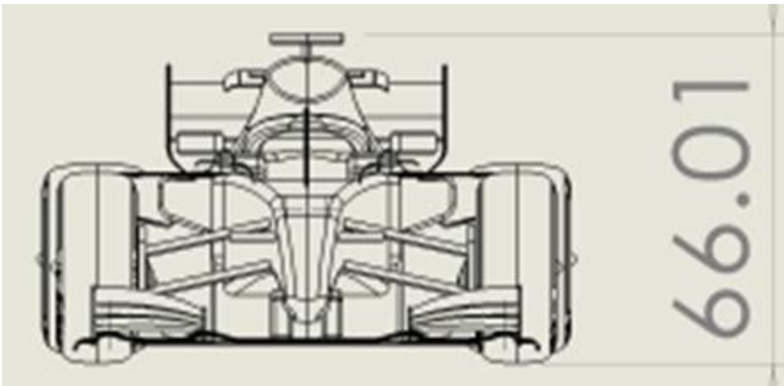

Figure 3.26 Scale down front view of existing f1 car model

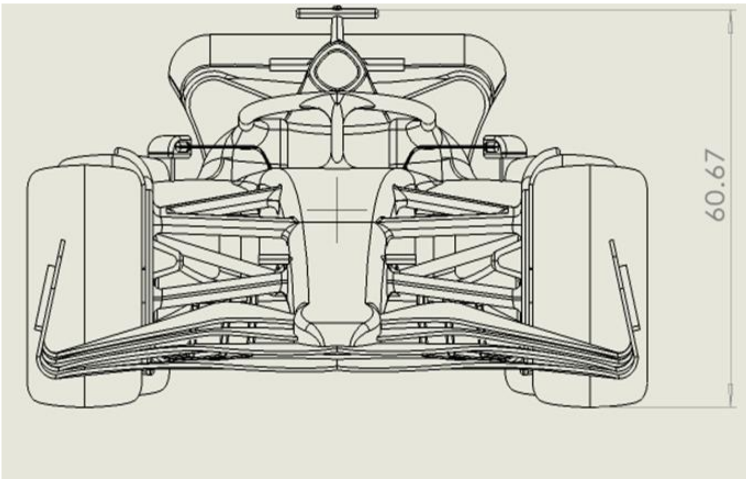

Figure 3.27 Scale down front view of modified f1 car model

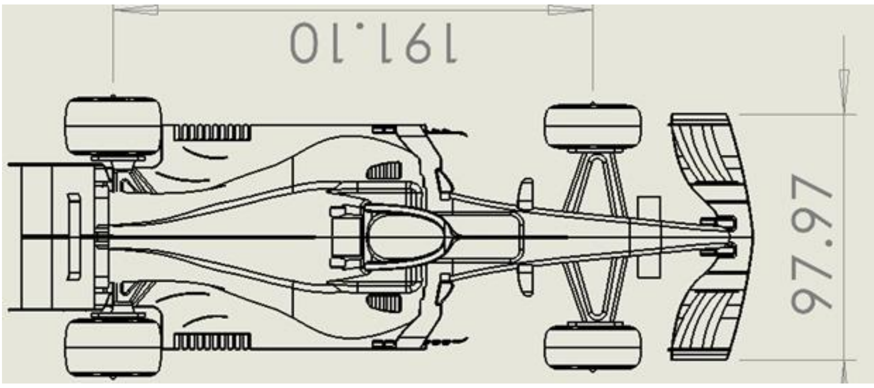

Figure 3.28 Scale down top view of existing f1 car model

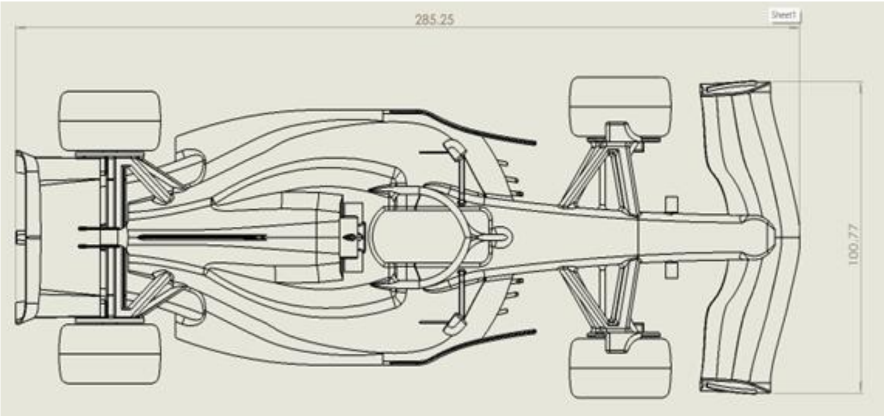

Figure 3.29 Scale down top view of modified f1 car model

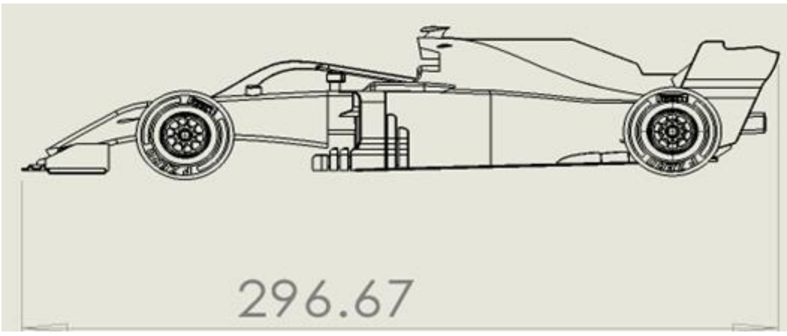

Figure 3.30 Scaledown side view of existing f1 car model

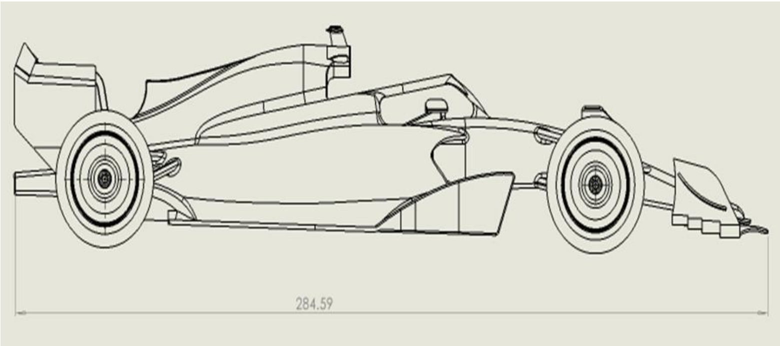

Figure 3.31 Scale down side view of modified f1 car model

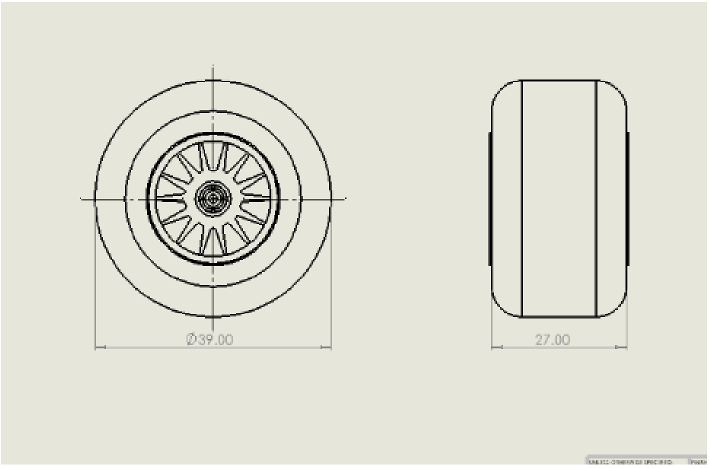

Figure 3.32 Scale down of existing f1 wheels top and side view

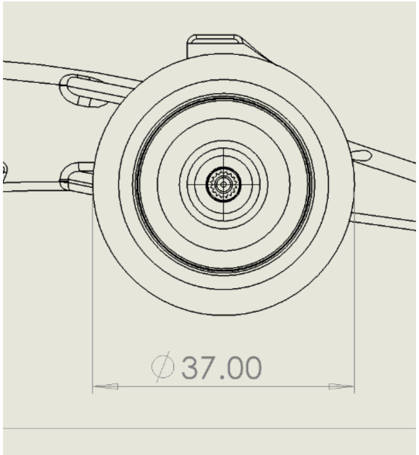

Figure 3.33 Scale down of modified f1 wheels side view

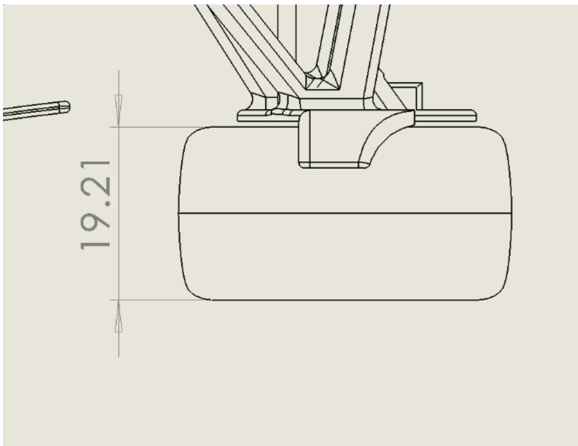

Figure 3.34 Scale down of modified f1 wheels top view

3.6 COST ANALYSIS

The cost analysis gives an insight on the budget required for the fabrication of this project.

AMOUNT SPEND		
SL.NO	ITEM	COST
1	EXISTING F1 CAR MODEL	₹1750
2	MODIFIED F1 CAR MODEL	₹2750
	TOTAL	₹4500

CHAPTER 4

RESULTS AND DISCUSSION

The f1 models are designed as per the dimensions and scaled down. The scaled down model is 3d printed for wind tunnel testing. The lift and drag values are calculated by the readings from the wind tunnel testing for the f1models respectively. The flow trajectories of the existing model and modified model are simulated using the solidworks flow simulation. The flow of the air is better and fluent in the modified f1 than the existing model. The front and rear spoilers are modified for better lift and drag co efficient. The lift and drag values are taken at three different speeds 10m/s, 20m/s, 30m/s, at 320rpm, 635rpm, 950rpm respectively. Thus, the modified f1 has the better lift and drag compared to existing f1.

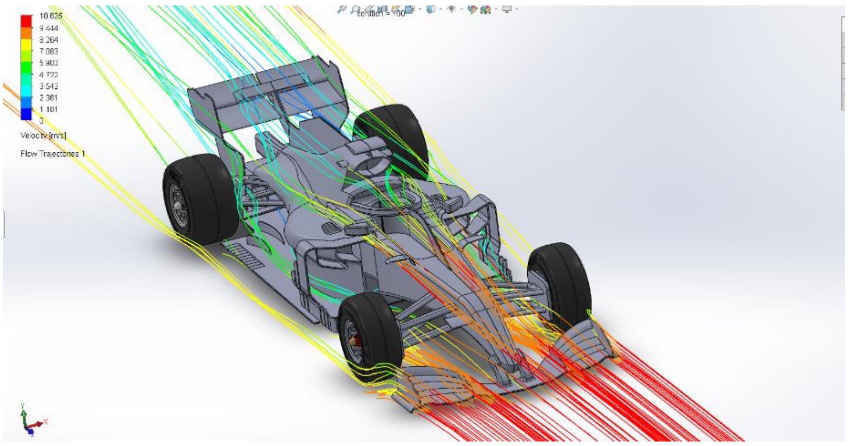

Figure 4.1 Flow trajectory of existing f1 car model

The above image shows the flow trajectory of the existing f1 car model. The air flow at the front wing gets deviated to the front tyres, causes more drag force. Which affects the speed of the car.

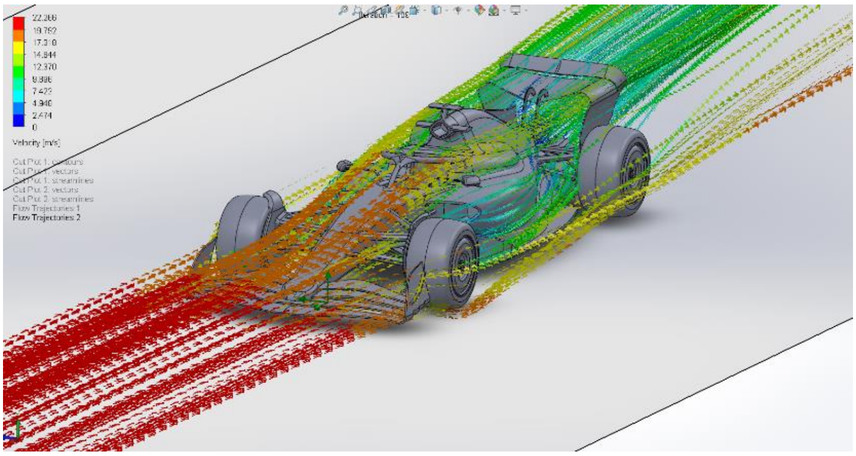

Figure 4.2 Flow trajectory of modified f1 car model

The above image shows the flow trajectory of the modified f1 car model. The front wing has been modified in such a way that the air flow is not deviated to the front wheels and the air passes away from the wheels. This results less drag and allows the vehicle to move faster than the existing f1 car model.

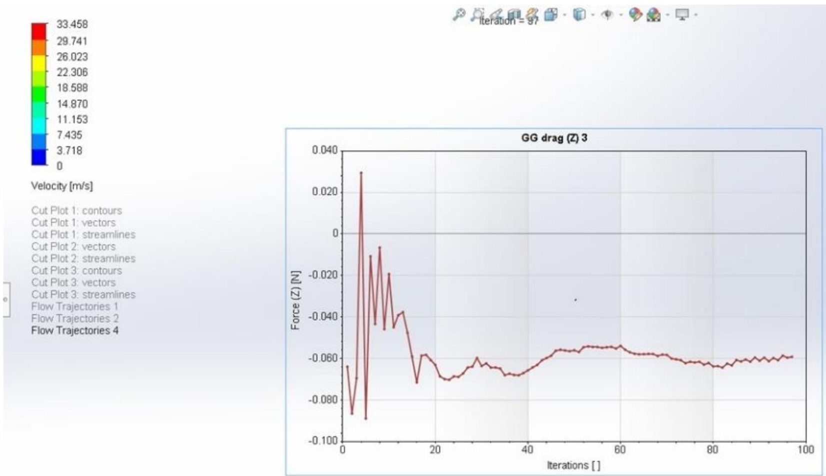

Figure 4.3 Drag of modified of f1 car model

The figure 4.3 shows the drag force of modified f1 car model. This graph has been taken from solidworks flow simulation.

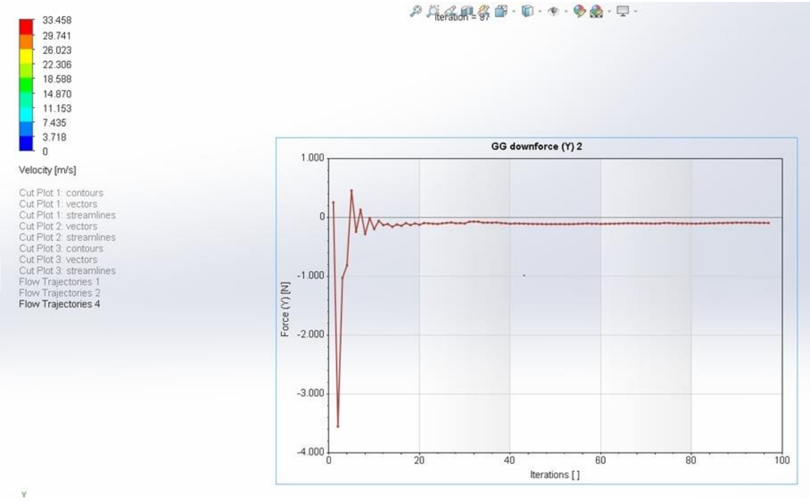

Figure 4.4 Downforce of modified f1 car model

The figure 4.4 shows the downforce of modified f1 car model. This graph has been taken from solidworks flow simulation.

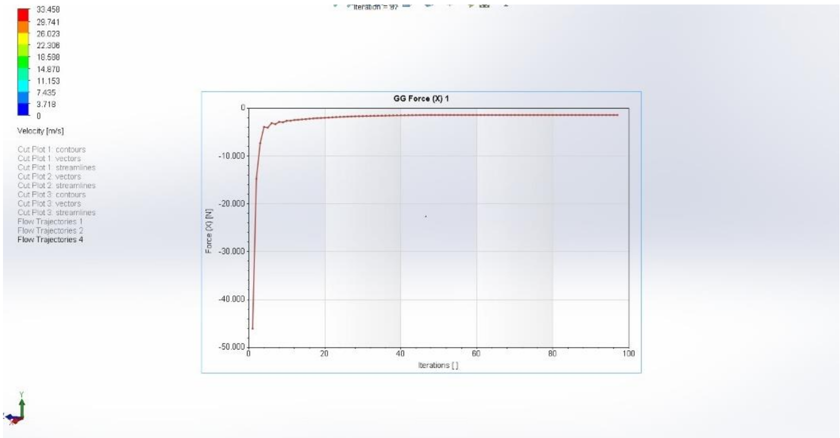

Figure 4.5 Force of modified f1 car model

The figure 4.5 shows the force of modified f1 car model. This graph has been taken from solidworks flow simulation.

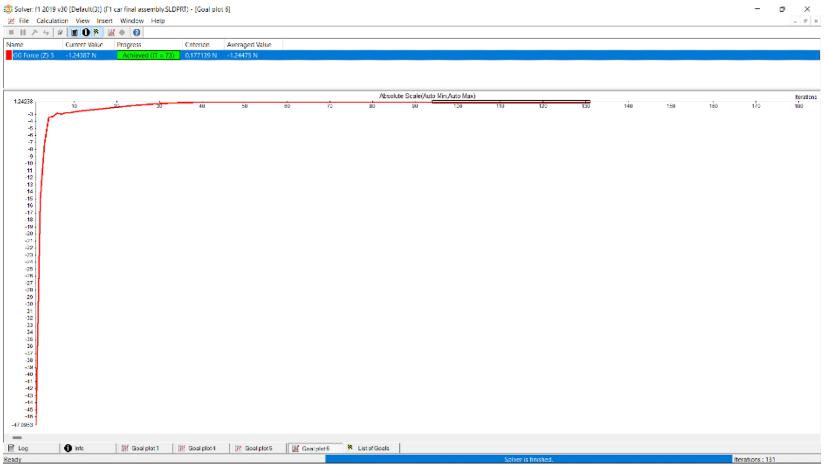

Figure 4.6 Downforce of existing f1 car model

The figure 4.6 shows the downforce of existing f1 car model. This graph has been taken from solidworks flow simulation.

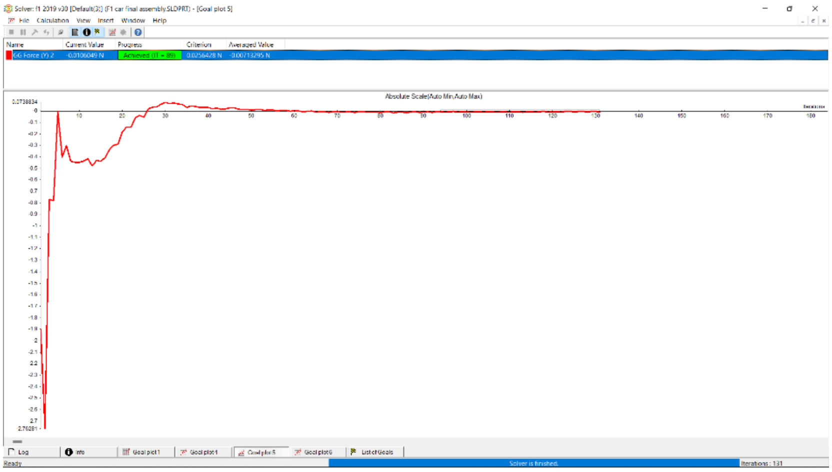

Figure 4.7 Drag of existing f1 car model

The figure 4.7 shows the drag force of existing f1 car model. This graph has been taken from solidworks flow simulation.

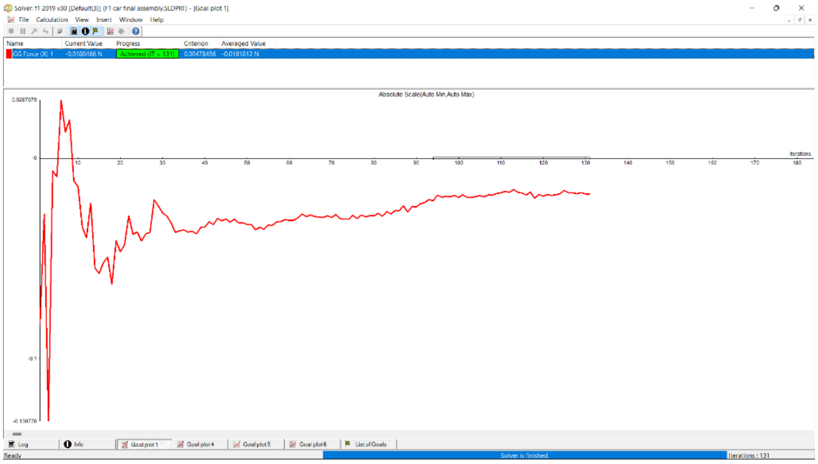

Figure 4.8 Force of existing f1 car model

The figure 4.8 shows the force of existing f1 car model. This graph has been taken from solidworks flow simulation.

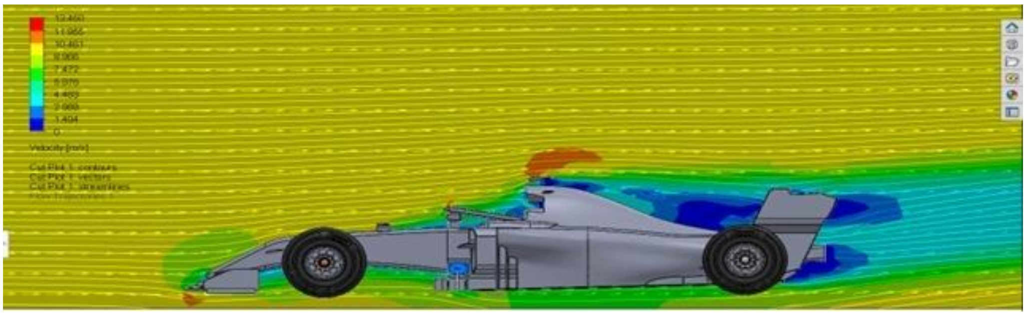

Figure 4.9 Contour and streamline existing f1 car model

The above image shows the contour and stream line flow of the existing f1 car model. At the rear wing low pressure has been created due to poor aerodynamic design.

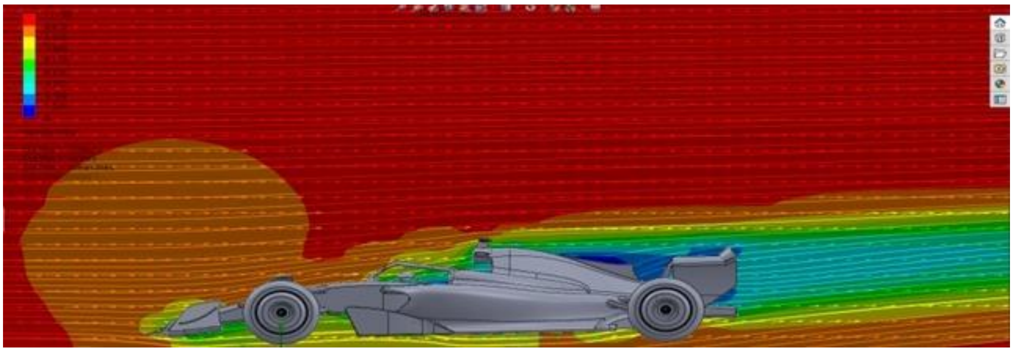

Figure 4.10 Contour and streamline of modified f1 car model

the above image shows the contour and streamline of modified f1 car model which has minimum low pressure at the rear wing. The flow is also streamline which has been improved by redesigning the rear wing.

Velocity(m/s)	Speed of the rotor (rpm)	Lift (kg)/(N)	Co-efficient of drag (kg)/(N)
10	320	0.053/0.51	0.040/0.39
20	635	0.081/0.79	0.122/1.19
30	950	0.11/0.07	0.153/1.5

Table 4.1 results of existing f1 car model

The above table show the co-efficient of drag and lift values of the existing f1 car model which have been tested in the wind tunnel testing equipment. The units are in both kg and N.

Velocity(m/s)	Speed of the rotor (rpm)	Lift (kg)/(N)	Co-efficient of drag (kg)/(N)
10	320	0.049/0.48	0.039/0.38
20	635	0.079/0.77	0.118/1.15
30	950	0.09/0.88	0.148/1.45

Table 4.2 Results of modified f1 car model

The above shows the results of co-efficient of drag and lift values of modified f1 car model which have been tested in wind tunnel testing equipment. The values are comparatively better than the existing f1 model. The overall aerodynamic efficiency has been better by 2.9%.

CHAPTER 5

CONCLUSION

This computational analysis shows that there is possibility of improving the aerodynamic performance of car by modifications in exterior design of car body. These modifications are helpful in reducing the coefficient of drag i.e., Cd which effects the fuel consumption. By these modifications the coefficient of drag is reduced by approximately 2.9%.

In aerodynamics flow separation can often result in increased drag, particularly drag which is caused by the flow differential between the front and rear wings of the object as it travels through the air flow. For this reason, much effort and research has gone into the design of aerodynamics. Thereby improving the effectiveness of wings. In this project flow simulation has been done at three different velocities are 10, 20 & 30m/s. At similar velocity has been done in wind tunnel testing also.

REFERENCES

1. Gadhvi, G. G., Chavda, P., & Pandya, P. (2020). Citation Patterns followed in Research Papers of the DESIDOC Journal of Library & Information Technology. *Library Philosophy and Practice*, 1-16.

2. Salhan, P. (2020). Computational Study of the Front-end Downforce Enhancing Aerodynamic Elements in Sports Cars.

3. Singh, K. K., Bahri, H., Singh, H., & Singh, J. (2019, November). A review of "performance analysis and optimization of car air spoilers". In *IOP Conference Series: Materials Science and Engineering* (Vol. 691, No. 1, p. 012056). IOP Publishing.

4. Sivaji, K. S., & Dikshita, P. (2019). Aero Design for Automobiles.

5. Senger, S., Bhardwaj, S. R., & Bhard, R. (2014). Aerodynamic design of F1 and normal cars and their effect on performance. *Int. Rev. Appl. Eng. Res*, 2248-9967.

6. Popova, D., Khot, A., & Thomo, A. (2018). Data Structures for Efficient Computation of Influence Maximization and Influence Estimation. In *EDBT* (pp. 505-508).

7. Etemadi, A., Abnet, C. C., Kamangar, F., Islami, F., Khademi, H., Pourshams, A., ... & Dawsey, S. M. (2014). Impact of body size and physical activity 29(2), 95-109.

8. Castro, X., & Rana, Z. A. (2020). Aerodynamic and Structural Design of a 2022 Formula One Front Wing Assembly. *Fluids*, *5*(4), 237.

9. Kurec, K., Remer, M., & Piechna, J. (2019). The influence of different aerodynamic setups on enhancing a sports car's braking. *International Journal of Mechanical Sciences*, *164*, 105140.

10. Deng, Z., Yu, S., Gao, W., Yi, Q., & Yu, W. (2020, July). Review of effects the rear spoiler aerodynamic analysis on ground vehicle performance. In *Journal of Physics: Conference Series* (Vol. 1600, No. 1, p. 012027). IOP Publishing.

11. Cakir, M. (2012). CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle.

12. Mohammadi, M., Doosttalab, A., & Doosttalab, M. (2012, November). The effect of various gurney flaps shapes on the performace of wind turbine airfoils. In *Proceedings of the ASME Early Career Technical Conference, Atlanta, GA, USA* (pp. 2-3).

13. Bienz, C., Larsson, T., Sato, T., & Ullbrand, B. (2003, June). In Front of the Grid–CFD at SAUBER PETRONAS F1 Leading the Aerodynamic Development. In *1st European Automotive CFD Conference, Bingen, Germany*.

14. Raheem, A. T., Sapit, A., & Mohammed, A. N. (2019). Aerodynamics of a Formula One Car Front Cascade Wing during Cornering. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, *53*(1), 5360.

15. Guerrero, A., & Castilla, R. (2020). Aerodynamic Study of the Wake Effects on a Formula 1 Car. *Energies*, *13*(19), 5183.

INDIVIDUAL CONTRIBUTION

1. D. Venkat ramana reddy
    - PPT content collection.
    - Fabrication.
    - Photography.
    - Wind tunnel testing.
2. JS. Karthikeyan
    - Literature review.
    - Report content collection.
    - Existing f1 car Modelling in solidworks.
    - Wind tunnel testing.
3. G. Monish kumar
    - PPT content collection.
    - Report correction.
    - Flow simulation of modified f1 car model.
    - Wind tunnel testing.
4. G. Anurag parida
    - Modified f1 car modelling in solidworks.
    - Flow simulation of existing f1 car model.
    - Wind tunnel testing.
    - Fabrication.

PO – PROJECT MAPPING

PO NO.	PO DESCRIPTION	PROJECT MAPPING
PO 1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.	3
PO 2	**Problem Analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.	3
PO 3	**Design Development of Solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.	3
PO 4	**Conduct Investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.	2
PO 5	**Modern Tool Usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.	3
PO 6	**The Engineer & Society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety,	2

	legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.	
PO 7	**Environment & Sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.	3
PO 8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.	3
PO 9	**Individual & Team Work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.	3
PO 10	**Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.	2
PO 11	**Project Management & Finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.	2
PO 12	**Life-Long Learning:** Recognize the need for, and have the preparation and ability to engage in independent and lifelong learning in the broadest context of technological change.	3

PSO – PROJECT MAPPING

PSO NO.	PSO DESCRIPTION	PROJECT MAPPING
PSO 1	Design, Analysis, Fabrication and Testing of vehicles, which enable the students to compete globally.	3
PSO 2	Carry out research in fuel economy, emission reductions, alternate fuels and solar vehicle for the benefit of the society and environment.	2