EFFECT OF VORTEX GENERATORS & REAR SPOILER ON A LOW-END SEDAN PASSENGER CAR

Capstone Project Report submitted in

Partial fulfillment for the award of the

degree of

BACHELOR OF TECHNOLOGY

Submitted by

Sartaj Anwer Khan

(18021050147)

Nikhil Pasricha

(1714105008)

IN

MECHANICAL ENGINEERING WITH SPECILIZATION OF AUTOMOBILE ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

Under the Supervision of

Mr. Faisal Shameem

Assistant Professor

GALGOTIAS UNIVERSITY

(Established under Galgotias University Uttar Pradesh Act No. 14 of 2011)

JUNE 2021

GALGOTIAS UNIVERSITY

DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

It is certified that this project report **"EFFECT OF VORTEX GENERATORS & REAR SPOILER ON A LOW-END SEDAN PASSENGER CAR"** is the bonafide work of "**Sartaj Anwer Khan (18021050147) and Nikhil Pasricha (1714105008) "** who carried out the project work under my supervision.

Signature of Supervisor

DECLARATION

We, here by, declare that this written submission represents our ideas in our own words and where others' ideas or words have been included, we have adequately cited and referenced the original sources. We also declare that we have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in our submission. We understand that any violation of the above will be cause for disciplinary action by the Institute and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

1. Sartaj Anwer Khan
   (18021050147)
2. Nikhil Pasricha
   (1714105008)

Date: 11/06/2021

(Signature)

APPROVAL SHEET

EFFECT OF VORTEX GENERATORS & REAR SPOILER ON A LOW-END SEDAN PASSENGER CAR by **Nikhil Pasricha and Sartaj Anwer Khan** is approved for the degree of Bachelor of Technology in Automobile Engineering.

Supervisor

Project Coordinator

Date: ____________

Place: ____________

ACKNOWLEDGEMENT

Presentation inspiration and motivation have always played a key role in the success of any venture.

I express my sincere thanks to **Prof. P.K Nain, Dean, Department of Mechanical Engineering, Galgotias University.**

I feel to acknowledge my indebtedness and deep sense of gratitude to my guide **Assistant Prof. Mr. Faisal Shameem** whose valuable guidance and kind supervision given to me throughout the course which shaped the present work as its show.

I pay my deep sense of gratitude to especially **Assistant Prof. Mr. B.N Agarwal** and all **Staff member** of **Department of Mechanical Engineering,** to encourage me to the highest peak and to provide me the opportunity to prepare the project. I am immensely obliged to **My Friends** for their elevating inspiration, encouraging guidance and kind supervision in completion of my project.

Last, but not least, **My Parents & Family members** are also an important inspiration for me. So with due regards, I express my gratitude to them.

(Sartaj Anwer Khan)

(Nikhil Pasricha)

(Department of Mechanical engineering)

ABSTRACT

Any vehicle moving at higher speeds will experience some resisting and lifting force called drag and lift. These forces mostly makes the budget passenger cars unstable to drive at higher speeds and can also lead to accidents sometimes. This study contains the designing & simulation of the passenger sedan model connected with & without the rear spoiler & vortex generators. For designing the sedan model & aerodynamic components, we used Solidworks. This study centres on streamlined airflow & boundary layer separation for reducing drag & lift on a sedan model. Aerodynamic analysis, like computational fluid dynamics (CFD), plan improvement, drag coefficient (Cd), & lift coefficient (Cl) & control strategies like, dynamic control had been investigated. Aerodynamics analysis on the base model with rear spoiler & vortex generator attached was also analysed because of their adaptability for a broad scope of use in the diverse kind of vehicles.

Table of Contents

List of abbreviations

VGs	Vortex Generators.
RS	Rear Spoiler.
CFD	Computational Fluid Dynamics.
Cl	Co-efficient of Lift.
Cd	Co-efficient of Drag.

List of Figure

List of Table

Chapter 1

Introduction

In this chapter the topic of the thesis is introduced the introduction about aerodynamics, CFD and aerodynamic components like Rear Spoiler and Vortex Generators is given in Section 1 to 3 along with the various forces like Drag and Lift that affects the vehicle at higher speeds. The literature survey and the current research in the area of reducing Drag and Lift forces using various aerodynamic components is given in section 4 .The problem statement of our work is defined in section 5. The governing equations of mathematical formulation is given in section 6. The methods used in this work and the objectives of the work is outlined in section 7. Finally, the evaluation parameters used for quantitative result analysis is detailed in Section 8.

1. Introduction

Entering in the industrialization, need for fossil fuels increased drastically. The significant contribution of fossil fuel is to the automobile industry, but the fossil fuel is finite, & thus automobiles should be designed as efficiently as possible to keep the fuel usage to a minimum & get output as much as possible. Therefore, keeping these conditions in mind aerodynamic components were introduced. As the 20^{th} century began, there has been some significant improvements in the aerodynamics of base models. The world also witnessed some of the finest racing cars like Bugatti Veyron achieving top speed up to 267 mph. Improved aerodynamics are the only reason such top speed was achieved. Aerodynamics is a broad concept that refers to the study of road vehicles that experience air resistance while in motion. It encompasses the whole range of forces and moments operating on a vehicle, all of which affect its drag and stability. Since the drag force generates a downforce part, the drag force and lift force may also contradict each other. As a result, the lift is reduced, resulting in increased downforce and traction for improved fuel economy.

1.1 AUTOMOBILE AERODYNAMICS

1.1.1 Introduction to Aerodynamics

The science of how things move through the air is called aerodynamics. How aeroplanes fly is explained by the aerodynamics rule. From rocket take-off to kite flying, all objects moving in the air are affected by aerodynamics. Aerodynamics has an impact on automobiles travelling on roads. "Aerodynamics" is a branch of fluid dynamics concerned with the study of air motion, particularly when it interacts with moving objects. Aerodynamics is a branch of gas dynamics that shares many theories with fluid dynamics. Aerodynamics and gas dynamics are often used interchangeably, with the difference being that the gas dynamics only applies to compressible flows. The calculation of forces and moments acting on an item is made possible by understanding the motion of air (commonly known as the flow field) around the item. The flow field's usual properties as a function of position and time include velocity, pressure, density, and temperature. The flow field can be controlled by defining a control volume around it, the mass, momentum, and energy conservation equations can be derived and used to solve the properties. The scientific foundation is built on the application of aerodynamics through mathematical analysis, experimental approximations, and wind tunnel tests.

External and internal aerodynamics are the two sub-categories of aerodynamics. The study of the flow around solid bodies of diverse shapes is known as external aerodynamics. External aerodynamics includes things like calculating lift and drag on an aeroplane, airflow over a wind turbine blade, and a shock wave emerging in front of a rocket arrow. Internal aerodynamics, whereas, is study of solid bodies fluid flow channels. Internal aerodynamics, for example, is concerned with the movement of air via jet engines and air conditioning pipework. This work focuses on the outer portion of the vehicle-related aerodynamics with domain geometry and displays grids, vector graphs, contour graphs, and finally the XY graph and the graph of the results**.**

1.1.2 Scope of Aerodynamics

Design experts are under immense pressure to improve present automotive designs with little shape modifications in order to combat global warming and fast growing fuel prices. To meet the above requirements, the design engineers used aerodynamic concepts to improve the efficiency of the automobile. Although many factors affects aerodynamics, this work focuses on external devices that control the flow around the vehicle body in order to lower the vehicle's air resistance when going on roadways.

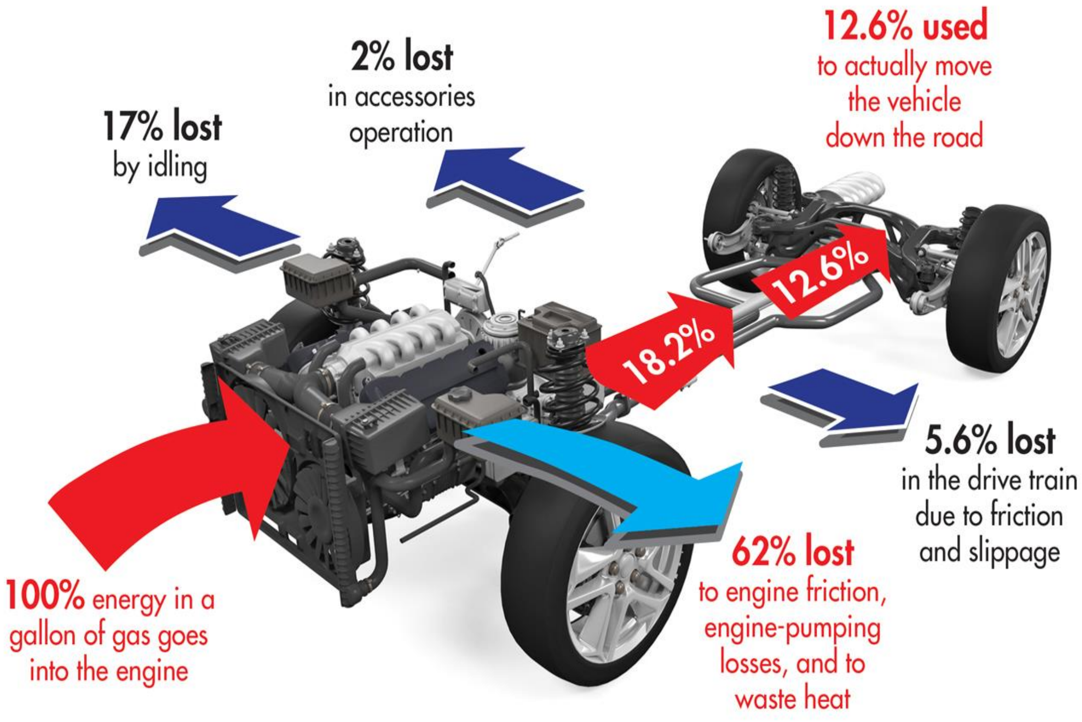


Figure 0.1 Energy usage in City driving

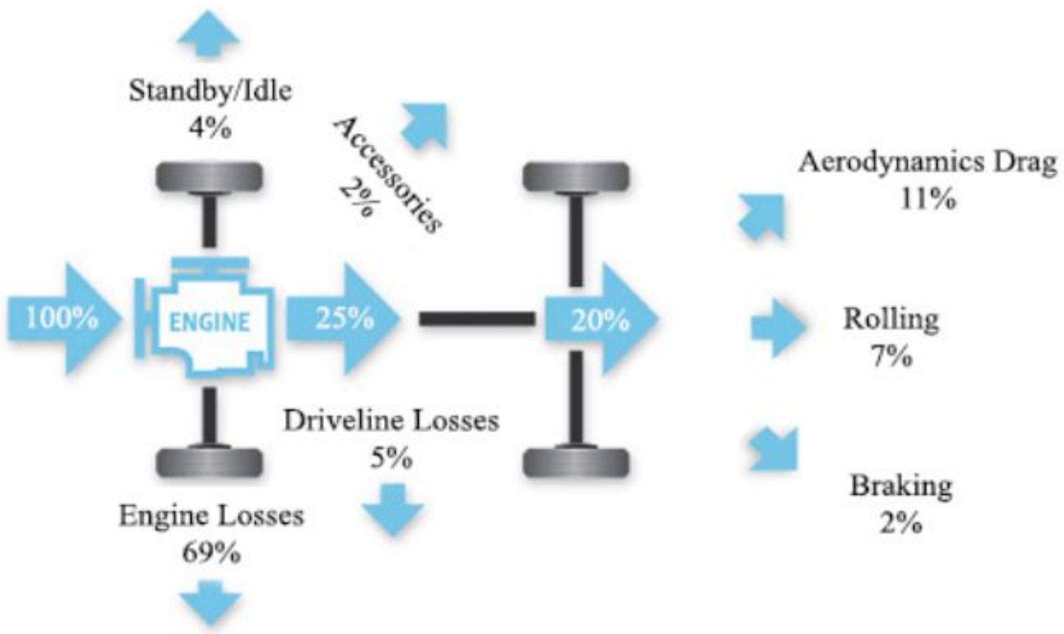


Figure 0.2 Energy usage in highway driving

Figures 1.1 and 1.2 depicts the fuel energy utilised in a modern car when driving in cities and on highways, respectively. In urban driving, the vehicle shape consumes around 3% fuel to overcome resistance, but highway travel consumes 11% of gasoline. This unusually high highway fuel consumption drew the attention of various design experts who sought to improve vehicle aerodynamics with minimum design adjustments. This suggests the use of external gadgets that can be added to an existing vehicle without requiring a body change. This work focuses on design, development, and numerical analysis of the effects of aerodynamic components, which are a rear spoiler located at the back of the vehicle and VGs mounted on the car's roof, on the existing vehicles' aerodynamics..

1.1.3 External Flow Phenomena of Automobile

The streamlined flow surrounding a vehicle in rest position is seen in Figure 1.3. The fluid's viscosity effects are limited to a small layer termed the boundary layer while a vehicle is driving at a given speed. Non-viscous flow exists outside the boundary layer. The boundary layer is compressed as a result of the fluid flow. The liquid is released

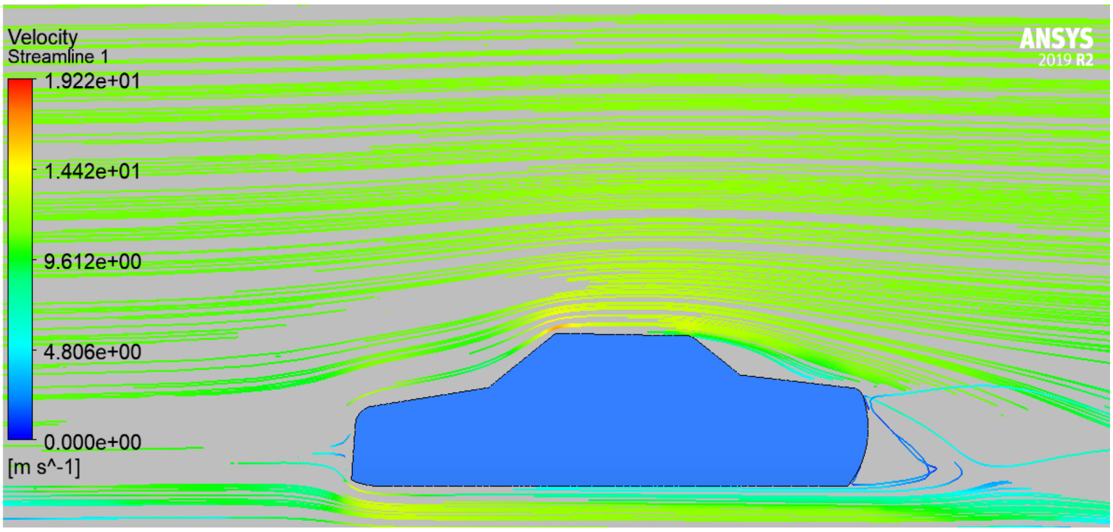


Figure 0.3 Streamline flow around a stationary car

when the air hits the back of the car. The viscous properties of the fluid totally dominate the fluid motion in the boundary layer.

The Reynolds number is calculated using the vehicle's characteristic length, kinematic viscosity, and speed. The fluid circulating around the car is determined by the automobile's geometry and Reynolds number. There is another significant phenomenon that has an impact on vehicle flow and performance. Vehicle "wake up" is a term used to describe this phenomena. When the air flowing above the car splits at the back, forming a huge low-pressure turbulent area at the rear end of the vehicle, the wake zone is generated. This wake adds to the pressure drag produced by the vehicle, lowering its performance.

1.1.4 Factors Influencing the Flow Field Around the Vehicle

Boundary layer, Flow field separation, Drag caused by friction, and finally the Drag caused by pressure are the key elements that affect the flow field around a moving vehicle.

1.1.4.1 Boundary Layer

Ludwig Prandtl initially defined the aerodynamic boundary layer in a paper delivered at the third International Congress of Mathematicians in Heidelberg, Germany, on August 12, 1904. This allows aerodynamicists to simplify flow of fluids equations by separating the flow field into two zones: one inside a boundary layer and one outside a boundary layer. Inside the vehicle's boundary layer, viscosity is dominant and has a considerable influence on vehicle drag; however, viscosity is ignored outside the boundary layer because it has no substantial effect on the solution. The boundary layer is heavily emphasised in the body design to reduce drag. Designers regard the boundary layer to be a crucial contributor in aerodynamic drag for two reasons. The first is that by increasing the effective thickness of the body through displacement thickness, the boundary layer increases pressure drag. The second reason is that shear stresses at the car's surface induce skin friction drag, which is created by the fluid rubbing against the skin of the entity that is passing through it.

1.1.4.2 Separation of Flow

There are various instances where velocity shifting stops and liquid begins to flow in the opposite direction as the flow passes over the vehicle's surface. The "separation" of the fluid flow is the name for this occurrence. This commonly happens in the vehicle's back end. The pressure distribution imposed by the flow's outer layer is mostly responsible for this separation. The flow behind the vehicle changes its behaviour as a result of this separation, altering the flow field around it. This phenomenon is the most significant factor to examine while studying a vehicle's wake. Flow separation is undesirable because it results in increased wake and lower pressure on the back surface, which reduces pressure recovery. The transition of air flow from the roof to the rear window should be smoothed to avoid improper flow separation. A lack of separation might also add to the drag. If the flows operate in clean air, the aerodynamics will be more efficient (stratified flow). It is feasible to minimise the thickness of the boundary layer of by enhancing the vehicle's aerodynamics, so avoiding the worst flow separation.

1.1.4.3 Drag due to friction

Every material or wall has some friction, which prevents fluids from flowing freely. A tension is applied to every surface of the vehicle due to molecular friction. Friction drag is caused by the integration of the relevant force component in the free stream direction. If the separation does not occur, one of the main causes of total drag is friction drag**.**

1.1.4.4 Drag due to Pressure

A significant pressure slope exists behind the cars, causing the viscous flow to separate. The flow field has a high pressure value in the front, but the flow splits in the back, resulting in a large suction force in the region. The outcome of combining the force component caused by such a large shift in pressure is known as 'Pressure Drag'. The height of the vehicle, as well as the separation of the flow field, have an impact on this component.

1.1.5 Vehicle Forces and Moment

When a vehicle travels at a high rate, it is subjected to a variety of forces acting in various directions. The numerous forces operating on the vehicle body are depicted in detail in Figure 1.4. The six forces operating on the vehicle, as depicted in the FBD below are:

- Rolling Resistance
- Drag
- Lift
- Gravity
- Normal

➢ Motor

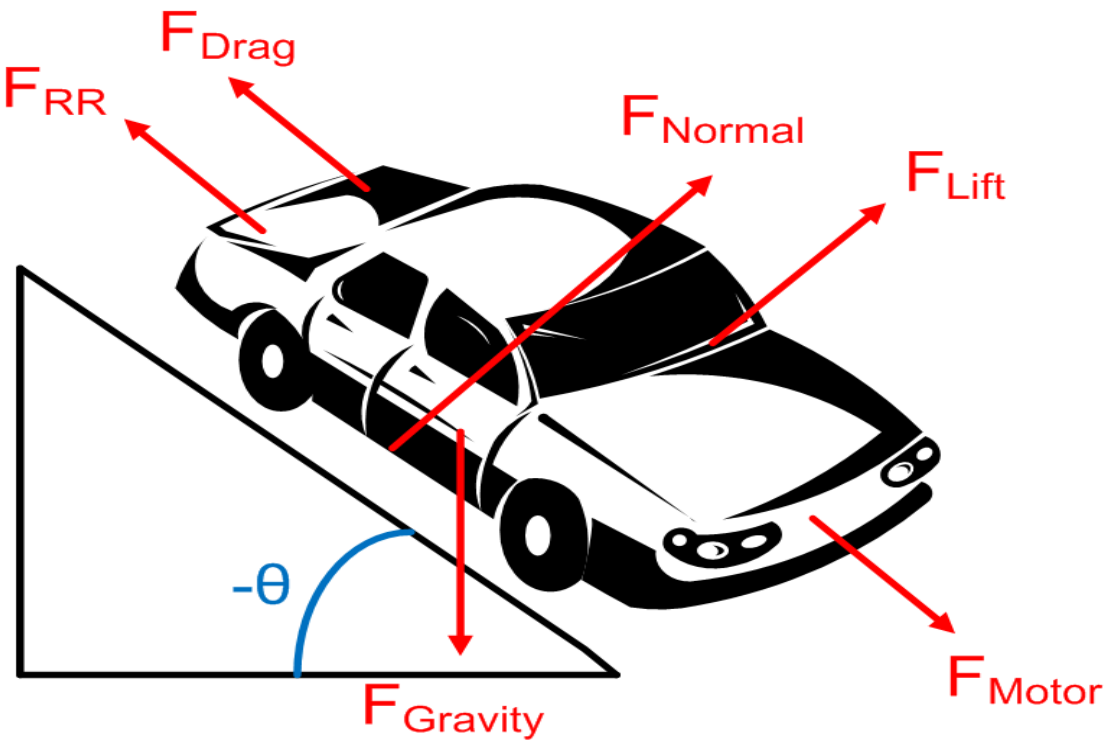


Figure 0.4 Forces on Vehicle Body

` The deformation of the tyres when they come into touch with the road surface causes rolling resistance, which varies depending on the surface the tyres are travelling on. The force exerted by the road is the normal force on the tyre. The magnitude of the normal force is equal to the force magnitude because of gravity. As the vehicle does not drive up or down, the direction relative to the road surface is important (relative to the road surface).

When applied in the positive direction, lifting force acts on the vehicle body vertically, lifting the vehicle into the air, if applied in the negative direction, it might create excessive downforce on the wheel. Engineers aim to keep this amount as low as possible to prevent creating too much downward force or lift. The following is a popular formula for calculating this force:

$$C_L = \frac{L}{\frac{1}{2} P V^2 A} \tag{1.1}$$

Where;

L: Lift force

C_L: Co-efficient of Lift

A: Frontal area of the vehicle

ρ: Air density

V: Vehicle velocity

The force acting on a vehicle's body as it moves back and forth is known as aerodynamic drag. When creating the outer body of the, this force is significant because it accounts for around 65 percent of the overall force working on the overall body of the car. The following formula can be used to determine aerodynamic drag:

$$C_D = \frac{D}{\frac{1}{2}PV^2A} \tag{1.2}$$

Where;

D: Drag force

C_D: Drag coefficient

A: Vehicle’s Frontal Area

ρ: Air density

V: Vehicle velocity

Chapter 2

CFD (COMPUTATIONAL FLUID DYNAMICS)

2.1 Introduction to CFD

"CFD (Computational Fluid Dynamics) is a collection of mathematical approaches for obtaining approximate solutions to fluid dynamics and heat transport problems."

As indicated by the definition, CFD isn't a science in and of itself, but rather a method of applying methodologies developed for one control (mathematical inquiry) to another (heat and mass transfer). It includes not just the principles of fluid mechanics with mathematics, but also software engineering, as seen in Figure 2.1. Fundamental mathematical equations that regulate a process of interest, usually in partial differential form, and are referred to as governing equations in CFD, may usually be used to characterise the physical features of fluid motion. “Computer scientists convert mathematical equations into computer programmes in order to solve them or software packages utilising high-level computer programming languages. The computational part simply refers to the numerical simulations used to analyse fluid flow, to arrive at numerical solutions, computer programmes or software packages running on high-speed digital processors are used. "Do we really need the knowledge of three specialised professionals from each area – fluids engineering, mathematics, and computer science – to join forces for the construction of CFD programmes or just to run CFD simulations?" another issue arises. No, and it's more likely that this field will demand someone who can master some portions of each discipline's knowledge.”

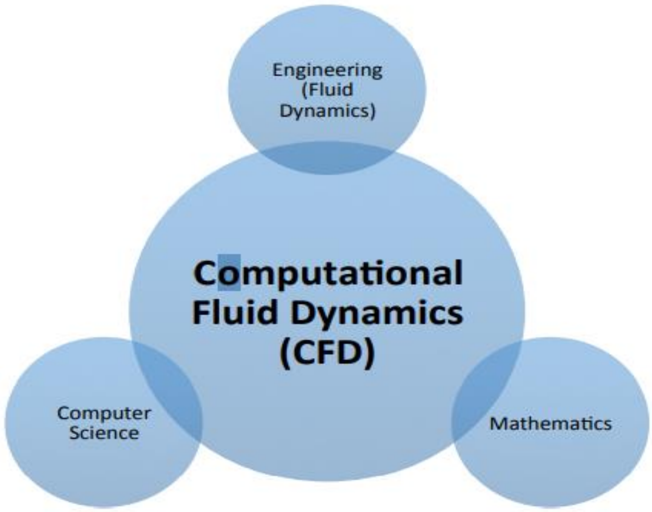


Figure 0.1 Variety of topics encompassed in CFD.

CFD is one of the three primary methods or methodologies for solving fluid dynamics and heat transfer problems. Each technique is closely interconnected and does not exist in isolation, as shown in Figure 2.2.

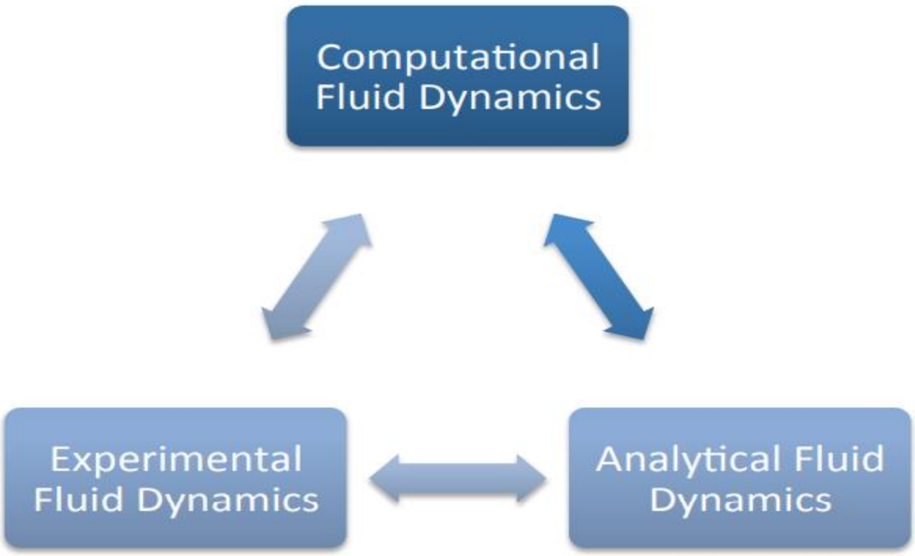


Figure 0.2 Three primary approaches in solving problems in fluid dynamics and heat transfer.

2.2 Computational Fluid Dynamics Benefits

Due to the increasing proliferation of digital computers, CFD is positioned to continue at the forefront of cutting-edge research in the disciplines of fluid dynamics and heat transport. The advent of CFD as a relevant tool in modern engineering practise has sparked a lot of interest and curiosity.

There are numerous benefits to considering CFD. The methodology and solution of the model parameters, as well as the investigation of alternative approximations to these equations, are fundamental to the theoretical development of computational sciences. CFD is a cost-effective alternative to experimental and analytical methodologies for modelling real-world fluid flows. In particular, when compared to an experimental-based approach, CFD significantly reduces lead times and costs in design and production, as well as providing the capacity to address a variety of complex flow problems when an analytical method is unavailable. CFD can be used to simulate flow conditions that are difficult to replicate in experimental research, such as nuclear disaster scenarios or situations that are too large or far distant to be replicated experimentally in geophysical and biological fluid dynamics (e.g., Indonesian Tsunami of 2004). CFD can provide significantly more detailed, observable, and comprehensive information than analytical and experimental fluid dynamics. Despite its benefits, CFD cannot simply replace experimental testing as a means of gathering data for design reasons. Despite its many benefits, the researcher must recognise the CFD's inherent limits. During computations, numerical mistakes arise; as a result, there will be disparities between the computed results and reality.

2.3 Numerical Technique

Numerical algorithms are used in CFD codes to solve fluid flow problems. Every piece of commercial CFD software on the market has three components that divide the full analysis of the numerical experiment to be done on a specific domain or geometry into three components.

The following are the three key elements:

i. Pre-processor

ii. Solver

iii. Post-Process

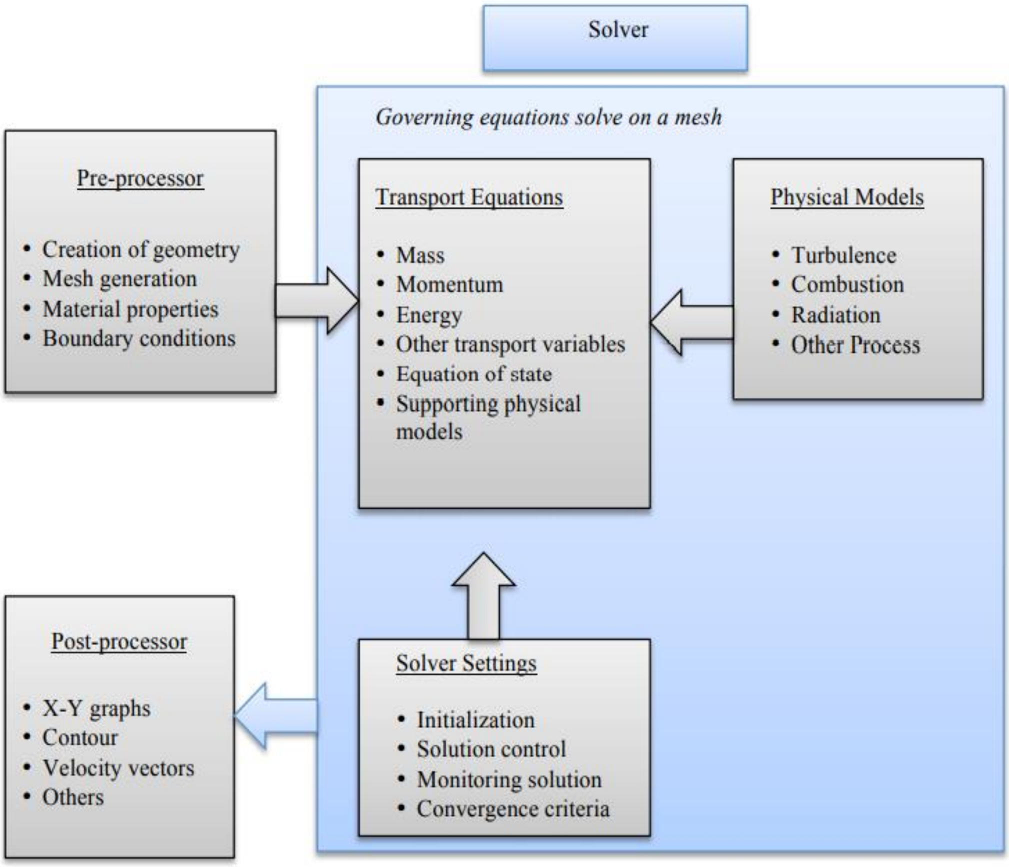


Figure 0.3 Three primary aspects of a CFD analysis framework's interconnectivity functions.

2.3.1 Pre-Processor

The pre-processor involves entering a flow problem through a user-friendly interface and then transforming that input into a format that the solver can understand. The user and the solver are linked through the pre-processor.

2.3.1.1 Geometry Creation

This procedure necessitates the use of CAD software such as CATIA®, Solidworks®, Pro-E®, and others. The fluid flow zone of interest's topology is defined with the help of CAD software. In research analysis, this software is an important aspect of the design and optimization process.

2.3.1.2 Generation of Mesh

When the domain geometry is determined, one of the most significant activities in the pre-process stage is mesh production. CFD requires the division of the domain into a number of smaller, non-overlapping subdomains in order to solve the flow physics inside the domain geometry that has been generated, resulting in the formation of a mesh (or grid) of cells (elements or control volumes) overlaid on the overall domain geometry. The discrete values of flow attributes such as velocity, pressure, temperature, and other transport parameters of relevance are normally calculated by numerically solving the key fluid flows represented in each of these cells. This gives you the CFD solution to the flow problem you're working on. The CFD solution's accuracy is determined by the quantity of cells in the mesh inside the computational region. In general, using a greater number of cells results in a more accurate result. On the other hand, the correctness of a solution, is heavily influenced by the imposed constraints, which are dominated by processing costs and calculation turnaround times.

2.3.1.2.1 Fluid Properties

The properties of each surface or fluid domain are unique. The properties of the fluid utilised in the CFD domain are established at this stage of the CFD Process.

2.3.1.2.2 Boundary Conditions

The complicated character of many fluid flow behaviours has significant consequences for the flow problem's boundary conditions. A CFD user must set relevant criteria in order to convert a solved CFD issue into a physical representation of the fluid flow. The boundary conditions input provides an initialization for each alternative configuration of the CFD domain. The boundary conditions of the CFD issue are normally defined by the CFD code, with each cell at a specified boundary assigned finite values.

2.3.2 Numerical Solver

A fundamental grasp of the numerical aspects of the CFD solver is required for the correct application of either an in-house or commercial CFD code. The treatment of the solver element is the emphasis of this section. The solution process shown in Fig. 2.4 may normally be used to explain and visualise a CFD solver.

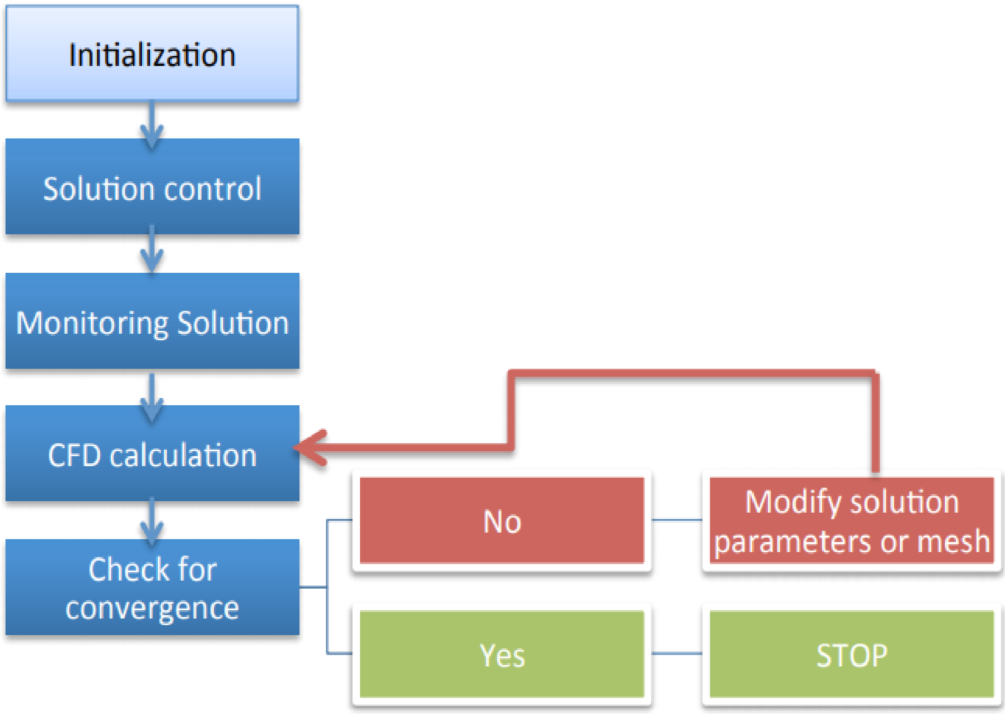


Figure 0.4 An overview of the procedure for solving the problem

In today's market, solvers often employ one of three methods to calculate solutions: the finite difference method, finite element technique, or finite volume approach. For stress and structure analysis, The methods of finite difference and finite element analysis are widely employed. In contrast, the finite volume method is the most appropriate method for the CFD procedure. The finite volume approach refers to the numerical algorithm computation approach that uses finite volume cells. The phases of this problem-solving technique are typically completed in the sequence listed below:

1. Formal integration of the fluid flow regulating equations over all of the solution domain's control volumes or finite volumes.

2. The process of converting an equation's integral form into a system of algebraic equations.

3. Using an iterative strategy, solve the algebraic equations.

2.3.3 Body of influence

The Body Sizing control also includes the ability to adjust mesh refinement using geometric bodies (These are referred to as a "body of influence" in ANSYS Meshing). A body of influence is any shape or size that intersects the main fluid domain we're trying to mesh. This zone of influence will be used by the ANSYS mesher to construct the suitable amount of local mesh refinement in the intersection region. This innovative feature also eliminates the necessity for sub-regions in the main body. This has shown to be quite useful in a variety of situations, such as when building up wake refinement regions in external aerodynamics difficulties.

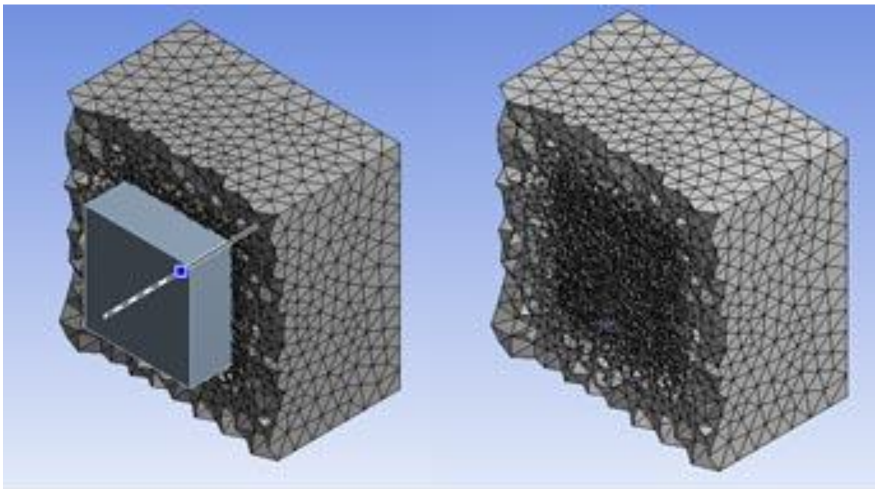


Figure 0.5 Body of Influence effect

2.3.4 Post Processor

Commercial CFD programmes, such as ANSYS Inc., CFX, ANSYS Fluent, STARCD, and others, also provide excellent visualisation features that allow users to graphically visualise the results of a CFD calculation at the end of a computational simulation through their user-friendly GUIs. The CFD solver's data visualisation features were used to analyse the following simulation results:

i. Geometry of Domain and Grid display

ii. Graphs of Vectors

iii. Line and shaded contour graphs

iv. 2D and 3D surface graph plots

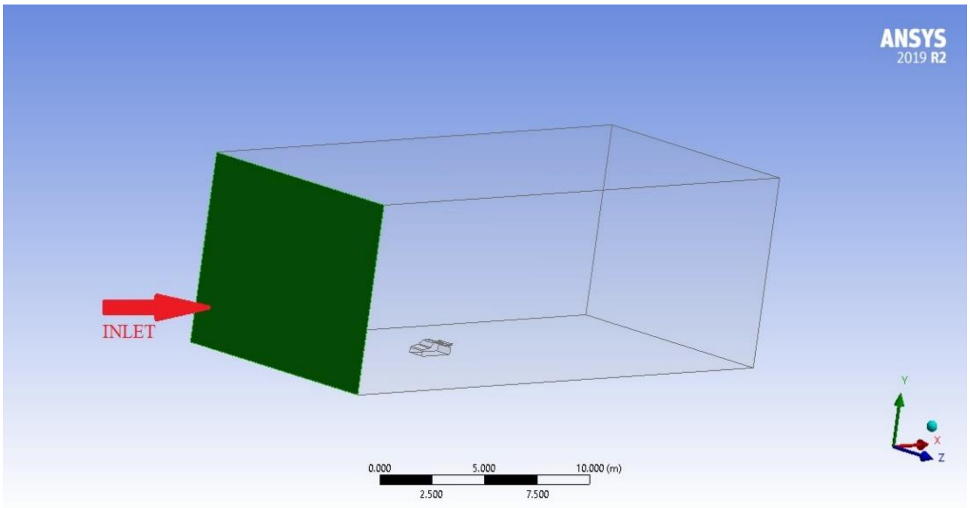
v. Particle tracking

vi. XY plots and graphs of results

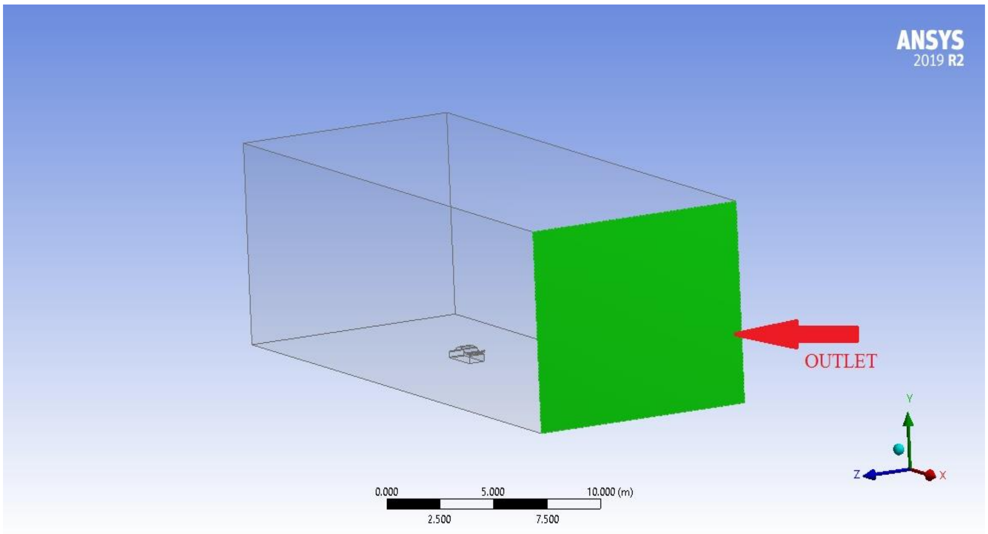
2.3.5 Wind Tunnel Testing

2.3.5.1 What is Wind Tunnel?

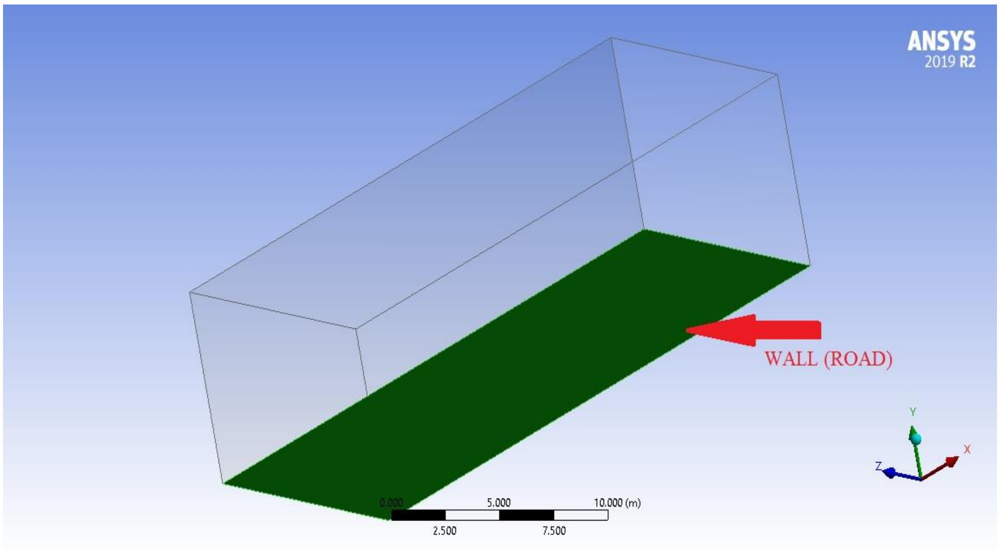
A wind tunnel is a large pipe through which air is blown to reproduce the interaction between air and flying or moving objects in the air or on the ground Wind tunnels are used by researchers to understand more about how the aircraft will fly. Wind tunnels are used by NASA to test scale models of planes and spacecraft. Some wind tunnels are large enough to fit a full-scale model of the vehicle. The air is moved around the object in the wind tunnel, giving it the appearance of flight. A large and powerful fan blasts air through the pipe most of the time. To keep the thing under test steady, it is firmly secured in the tunnel. An aerodynamic test object, such as a cylinder or airfoil, a single component, a miniature model of a vehicle, or a full-size vehicle can all be used as the object. The movement of air around a stationary item demonstrates what happens when the object moves in the air. Air flow may be examined in a variety of methods, including putting smoke or dye in the air and watching it move around objects. Colored lines can also be connected to objects to demonstrate how the air moves around them. The air force exerted on the object is usually measured with special instruments. **Figure 2.6** shows the various components of a wind tunnel.



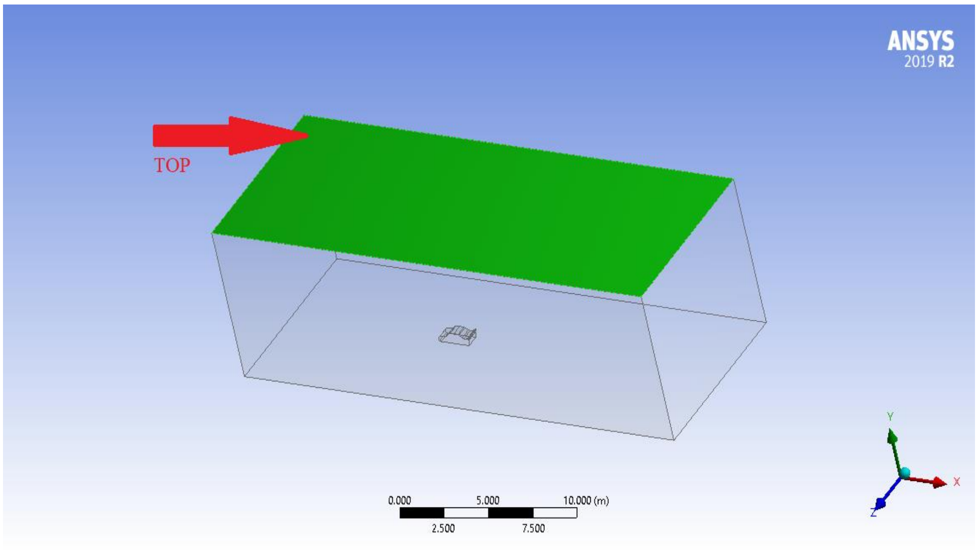
(a) Velocity Inlet



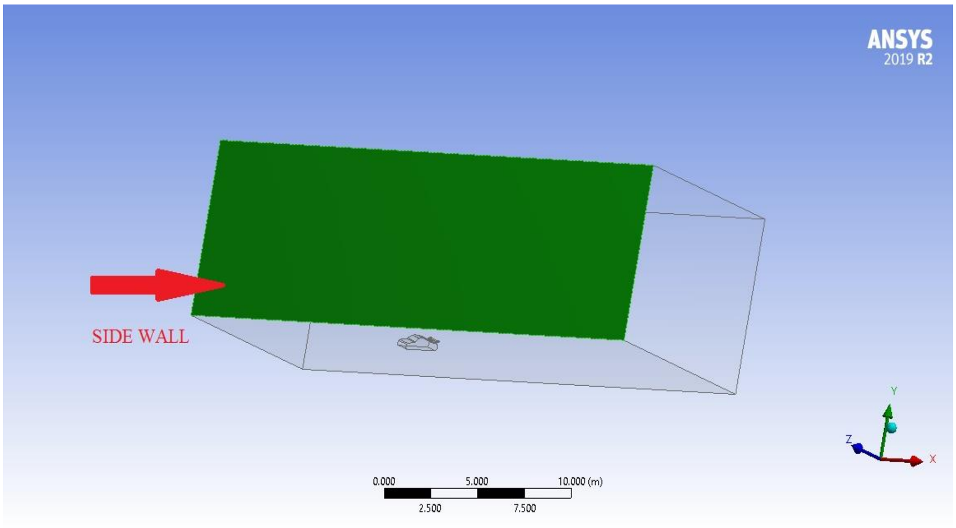
(b) Pressure Outlet



(c) Wall (Road)



(d) Symmetry Top



(e) Side walls

Figure 0.6 Virtual wind tunnel setup labelling for important boundary conditions. a) Velocity-inlet, b) Pressure Outlet, c) Wall (Road), d) symmetry-top, e) side walls

Chapter 3

INTRODUCTION TO THE SPOILER & VORTEX GENERATORS

3.1 Spoiler

A spoiler is an aftermarket accessory that becomes an integral part of the vehicle's body. The spoiler improves the streamlined flow of air around the engine, making forward propulsion more effective and straightforward. It also produces the required downforce, which reduces lift and provides stabilization.

A well-designed rear spoiler can support a vehicle's rear-axle lift. This is significant because a vehicle with a lower rear-axle lift than the front has better stability, requiring less driver intervention to maintain a straight line. Spoilers are most commonly seen on race cars and high-performance sports cars, but they've also become popular on civilian automobiles. Some spoilers are installed on cars primarily for cosmetic purposes, with little or no aerodynamic benefit. Automobile manufacturers are being pushed by environmental concerns and rising fuel prices to produce more fuel-efficient vehicles with lower emissions. Fuel quality has become increasingly important in the automotive industry in recent years. As a result, extensive research is being conducted to create aerodynamically optimized vehicle designs.

Now, Modern customers want sportier cars, so new cars manufactured according to current trends must be sportier. The current aerodynamic qualities are insufficient to deal with high power and speed, and it cannot accept downforce. Aerodynamic characteristics are important not only for the vehicle's efficiency, but also for the comfort and safety of the passengers. Other devices that can be utilised to achieve the purpose include splitters, canards, and rear diffusers, to name a few. Many researchers have noticed, however, that the spoiler only works at high speeds, resulting in a small increase in drat the expense of downforce and increased traction. According to some studies, there is also a slight reduction in drag. Nonetheless, the spoiler can be concluded to have a moderate impact on drag and a positive effect on downforce and lift. This work briefly shows how the aerodynamics components can help in reducing drag & lift in passenger sedan base models & help increase fuel economy while improving performance

Figure 0.1 Rear Spoiler on car

3.2 Vortex Generator

A vortex generator (VG) is an aerodynamic component containing small vane attached to a lifting surface (or airfoil, such as an aeroplane wing) or a wind turbine rotor blade. VGs can also be attached to components of an aerodynamic vehicle, such as an airplane's fuselage or a car's body. As the airfoil or body moves in relation to the air, the VG creates a vortex, delaying local flow separation and aerodynamic stalling by removing some of the slow-moving boundary layer in contact with the airfoil surface, improving flaps, elevators, ailerons, and rudders efficiency.

Vortex generators are most commonly used to delay flow separation. They are frequently used on the exterior surfaces of cars and wind turbine blades to achieve this. Vortex generators, regardless of design, have the same goal: to control boundary layer separation. They accomplish this by forming vortices that draw high-energy turbulent air into the low-energy boundary layer, hence the term. As a result, the boundary layer becomes more turbulent, increasing surface drag slightly. A turbulent boundary layer remains connected for a much longer period of time than a non-turbulent one. A turbulent boundary layer remains connected for a much longer period of time than a non-turbulent one. As a result, the turbulent wake diminishes and the pressure drag virtually disappears. This reduces the car's overall drag, which is particularly important for race cars. Vortex generators also transmit more air to wings and spoilers by

Figure 0.2 Vortex Generators on Cars

‘cleaning up' the airflow in this way. Thus, more downforce, is generated on the vehicle. Vortex generators aid sports vehicles on and off the track, but they must be correctly developed and fitted for the specific vehicle and its current aero features.

Chapter 4

LITERATURE REVIEW

The main focus for most of the literature that were studied was Minimising Aerodynamic Drag, mostly done on higher-end base models & race base models. Some studies modified the Aerodynamics of the Maruti 800 with double spoiler design & concluded that model with both upper & lower spoiler showed minimum Cd value. Some literature showed work on varying the taper angle or rear underbody angle. Few studies analysed Ahmed model fitted with a rear roof spoiler at different spoiler angles [1]. None of the literature mentioned use of Vortex Generators or Spoilers on Passenger base model. T. D. Ipilakyaa et al. designed a car model & simulated it with a wing type rear spoiler & reported increased drag coefficient value but decrease in lift coefficient value which is good for stability of cars at higher speed [2]. A passanger car model was studied with different spoiler angle at a particular height & it was concluded that at angle of 12^0 minimum C_L was observed which is necessary for car stability at higher speeds [3]. Madane et al. analysed a racing car model with different styles of spoiler & concluded that divided rear spoiler gave the best results in reducing drag & lift on the base model [4]. Dickison et al.used various design components like intercooler vent & rear air outlet on a racing car model & observed. The changes in drag & lift values were also recorded with & without added components. The addition of spoiler & other components increased the drag dorce but the down force helped in decreasing the lift force & stabilizing the base model [5] . A Santro car model stimulation at different velocities showed that with increasing velocity, drag on the base model increased & lift values were higher than usual & it was concluded that using a base model with rear spoiler improves the overall performance of the base model [6]. In a study of generic car model, it was observed that drag force & lift force decreased with the use of rear spoiler, which shows the effectiveness of rear spoiler at high speeds [7]. In a study on sedan car, different Vortex generator shapes was taken & it was concluded that GOTHIC type of VG was the most efficient of all the other shapes [8]. Yakkundi et al. analysed sedan model with wing type rear spoiler and saw an increase of around 8.2% in ΔCd values at speed of 70 km/hr [9].

Chapter 5

PROBLEM DESCRIPTION

Increased Fuel consumption now-a-days is a great concern for customers as well as for carmakers. To overcome this problem carmakers often try to improve the aerodynamics of the car so that fuel consumption can be minimized. Aerodynamics can be improved by making design changes to the car or adding some aerodynamics components like spoilers, roof rails or vortex generator to the car. Components like vortex generators and spoilers offers great drag resistance and reduce the lift of the car. They also increases the mileage of a car.

To increase fuel efficiency and performance to greater extent some design changes can be made to the existing designs of Spoilers and vortex generator.

Shapes can be modified to reduce the mass and to increase aerodynamics more effectively so that fuel consumption can decrease to a good extent even at higher speeds.

Chapter 6

GOVERNING EQUATIONS

The use of a Computational Fluid Dynamics (CFD) model is critical in determining the most appropriate arrangement method. The Navier-Stokes equation is seen to be the popular equation for tackling the flow of air problem and examining the changes in the stream. The Navier-Stokes equation is the "standard" approach to turbulence modelling and is used in all turbulence models. Based on the Reynolds transport theory, the three conservation equations can be combined to form Navier-Stokes equations. The following arrangement of conservation equations is independent of Navier-Stokes conditions.

a) Continuity of mass:

$$\frac{\partial p}{\partial t} + \frac{\partial y}{\partial x_i}(pu_i) = 0 \tag{1}$$

b) Momentum equation:

$$\frac{\partial pu_i}{\partial t} + \frac{\partial}{\partial x_j}(pu_i u_j) = \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + pg \tag{2}$$

c) Energy equation:

$$\frac{\partial pc_pT}{\partial t} + \frac{\partial}{\partial x_j}\left(pc_p u_j T\right) = \frac{\partial p}{\partial x_i}\left(\lambda \frac{\partial T}{\partial x_i}\right) + \frac{\partial}{\partial x_j}(u_i \tau_{ij}) \tag{3}$$

Where, x_i stands for a coordinate direction,

u_i denotes a velocity component,

$\boldsymbol{\tau}_{ij}$ denotes components of stress tensor.

P denotes density and

g denotes gravity.

Eddy viscosity of turbulent heat flux can be modelled as below:

$$q_j^{Re} = -\bar{p}c_p u_j^{"} T^{"} = \frac{u_t c_p(T)\partial T}{Pr_t \partial x_j} \tag{4}$$

6.1 Turbulence models

In general, determining which of these two widely used turbulence modelling methodologies is clearly superior is challenging. In fact, the supremacy of these models is influenced by not only the nature of the problem and simulation, but also the CFD user's abilities and competence. Lokhande et al. [10] advocate adopting the k–e turbulence model for aerodynamics and the LES turbulence model for aeroacoustics, respectively. The current work computes the aerodynamic and aero-acoustic characteristics of an automobile and its spoiler using the RNG k–e turbulence model, a version of the conventional k–e turbulence model, and the LES turbulence model, as proposed by them. The computing procedure is divided into two parts. In the first stage, the RNG k–e turbulence model is utilised to assess the overall aerodynamics of the passenger car and its spoiler. As the starting condition for the second phase, this computational result is used. In the second phase, the LES turbulence model is used to quantify pressure variations on the car body and spoiler surfaces, which are associated to noise creation.

6.2 K–ε turbulence model

In the Navier-Stokes condition to yield the Reynolds Averaged Navier-Stokes (RANS) equation, Reynolds time averaging procedure was utilized to represent the turbulence effect on the stream field, which can be numerically expressed as:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{p}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(v \frac{\partial \bar{u}_i}{\partial \bar{u}_j} - \tau_{ij}\right); i = 1,2,4; j = 1,2,3; \tag{5}$$

Chapter 7

METHODOLOGY

A passenger sedan model with suitable dimensions was designed in CAD software Solidworks. Various edges were tuned and fined for improving the aerodynamics of the model. The model was the meshed with suitable inputs and simulated at different speeds in ANSYS Fluent software. Spoiler and VGs were designed and scaled according to the car model and were then used with model. The final model was then meshed and simulated same ways as the base model. The results were then recorded and observed for the final conclusion.

i. Modelling of Base model & mesh it.
ii. Analyse the Base model at various speed.
iii. Modelling & meshing of the base model with Vortex Generator & Rear Spoiler.
iv. Analyse the final model at the same speed as the base model.

7.1 Base model Modelling & Meshing:

The base model was designed & drafted in Solidworks & was converted in solid model. The dimensions of the base model was considered similar to a budget sedan car so that the analysis results will be accurate.

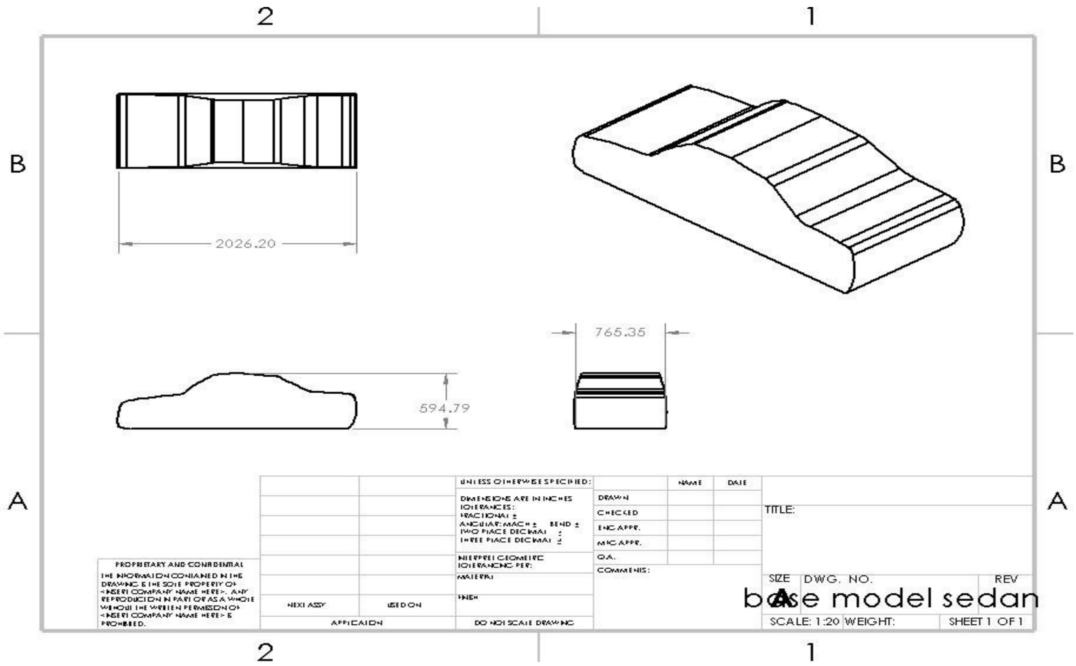


Figure 0.1 Draft model of car

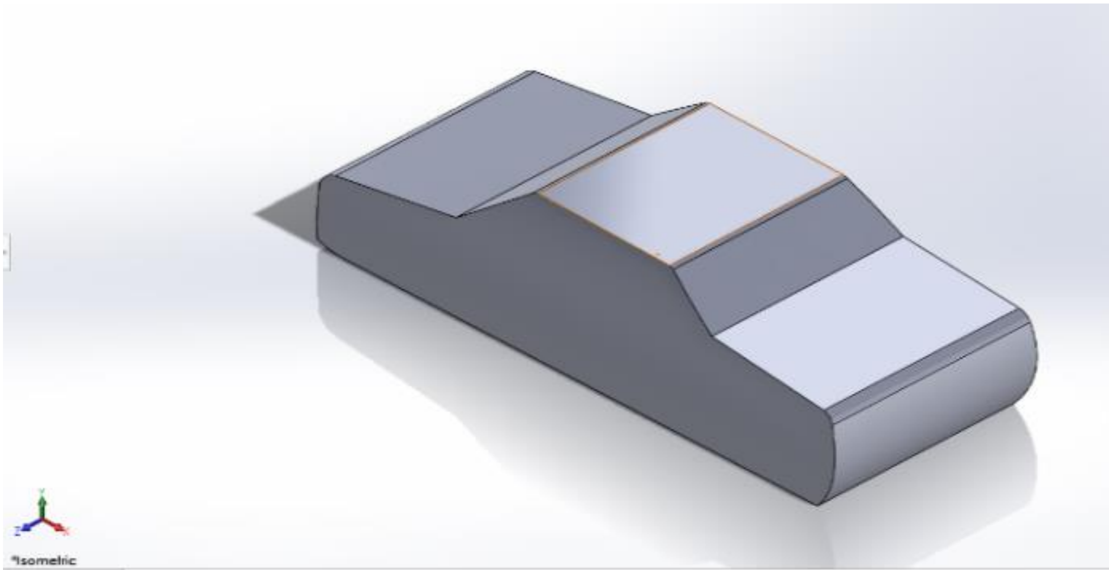


Figure 0.2 Solid model of car

The base model was then imported in IGS format in ANSYS Fluent 19 R2 & Wind Tunnel Setup enclosure was formed around the model.

The domain size was taken as-

In X direction: +X=8L, -X=5L,

In Y direction: +Y=5L, -Y=0.2L,

In Z direction: +Z=2.5L, -Z=2.5LWhere,

L is length of the car

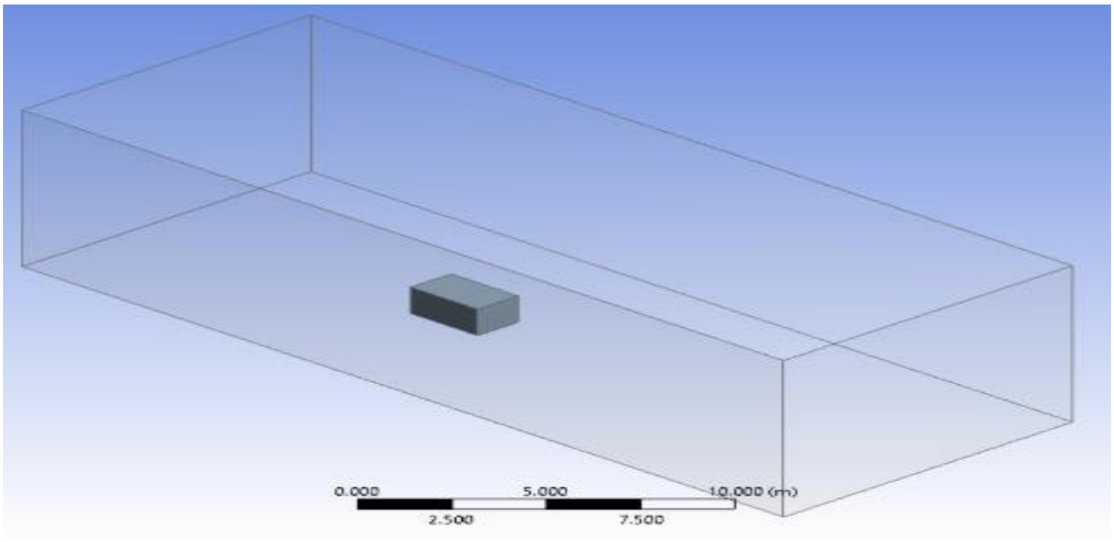


Figure 0.3 Wind Tunnel setup in ANSYS

For better results in meshing a primitive box of around the same dimension as of car model was taken. The primitive box was used as body of influence on the car so that better mesh results and dense mesh can be observed near the car.

The meshing was done linear with Element size 2.5m (global) & in inflation, first layer thickness was taken as 0.001m with 30 boundary layers & growth rate was set at 1.1. Face sizing also was done on base model with element size .008. The Body sizing was also done on a primitive box taken around the base model with type body of influence & element size is 0.05, & the total number of mesh element were 6006993, & the total number of nodes formed were 2751102.

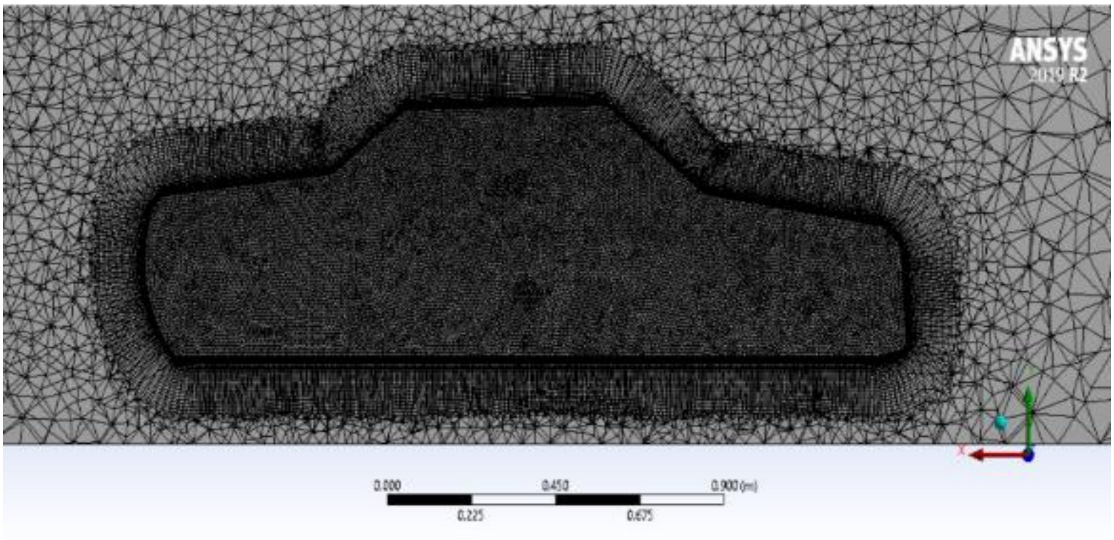


Figure 0.4 Meshing of base model

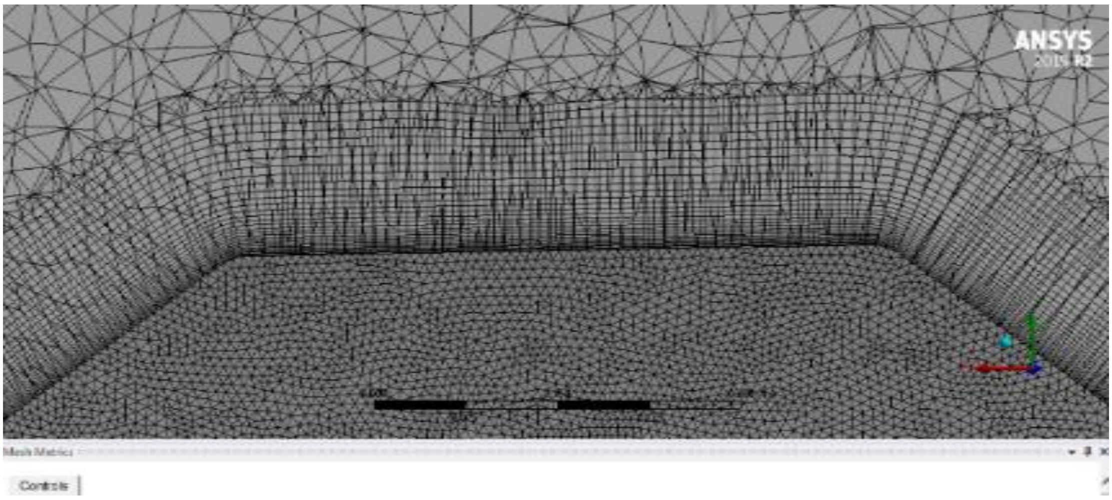


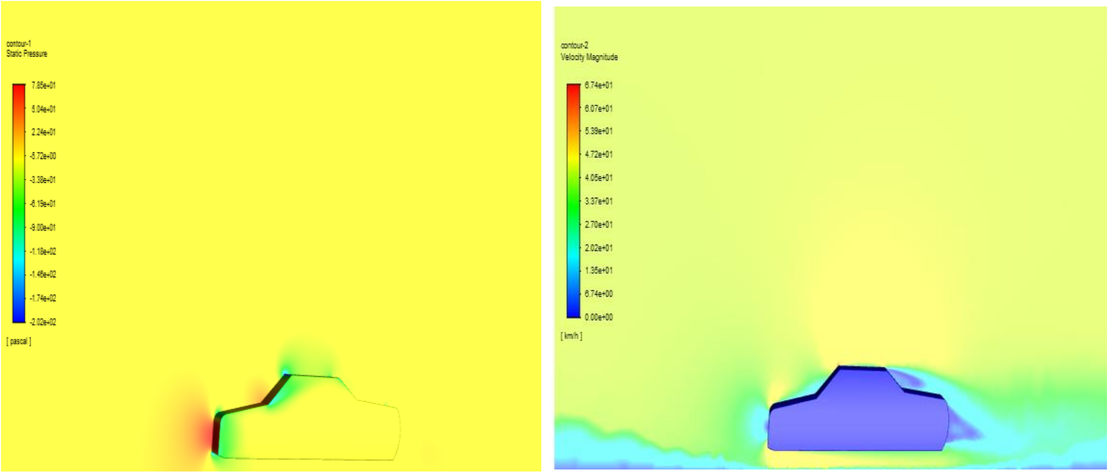
Figure 0.5 Boundary Layers

7.2 Analysis of Base Model:

The simulation was done at various velocities to get the better results of the base model. The viscous model was taken as standard k-epsilon, & steady flow was used for the simulation. The solution method scheme used was set as "SIMPLE", & the special Discretization "Gradient" was "Least Squares Cell Based" the pressure was taken as

“Second Order” whereas momentum was taken as “Second Order Upwind”. After simulating the wind tunnel test following results were observed.

i) At 40 km/hr



(a) (b)

Figure 0.6 40km/hr (a) Pressure (b) Velocity Contours

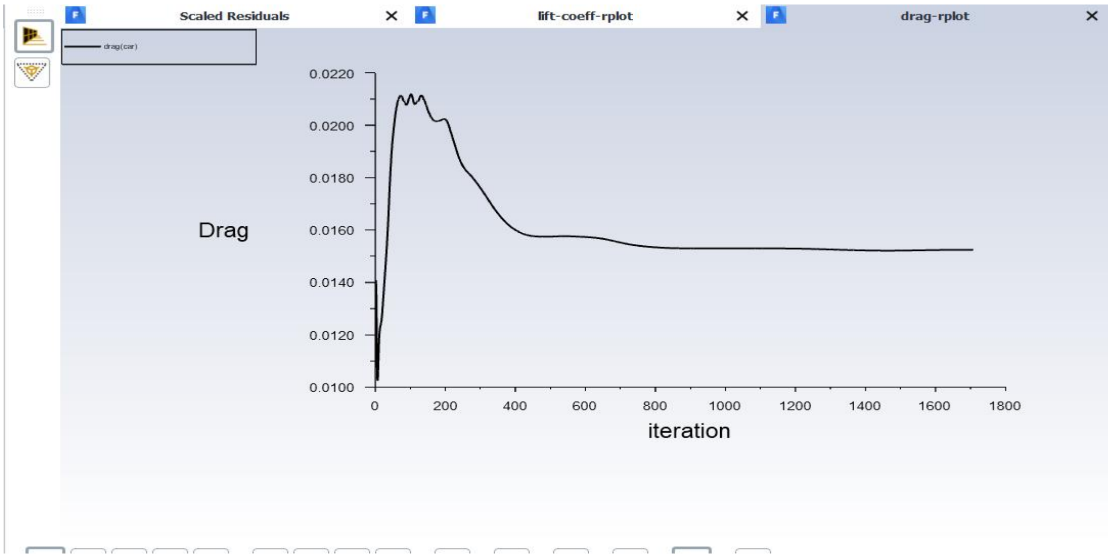


Figure 0.7 40km/hr Drag Coefficient plot

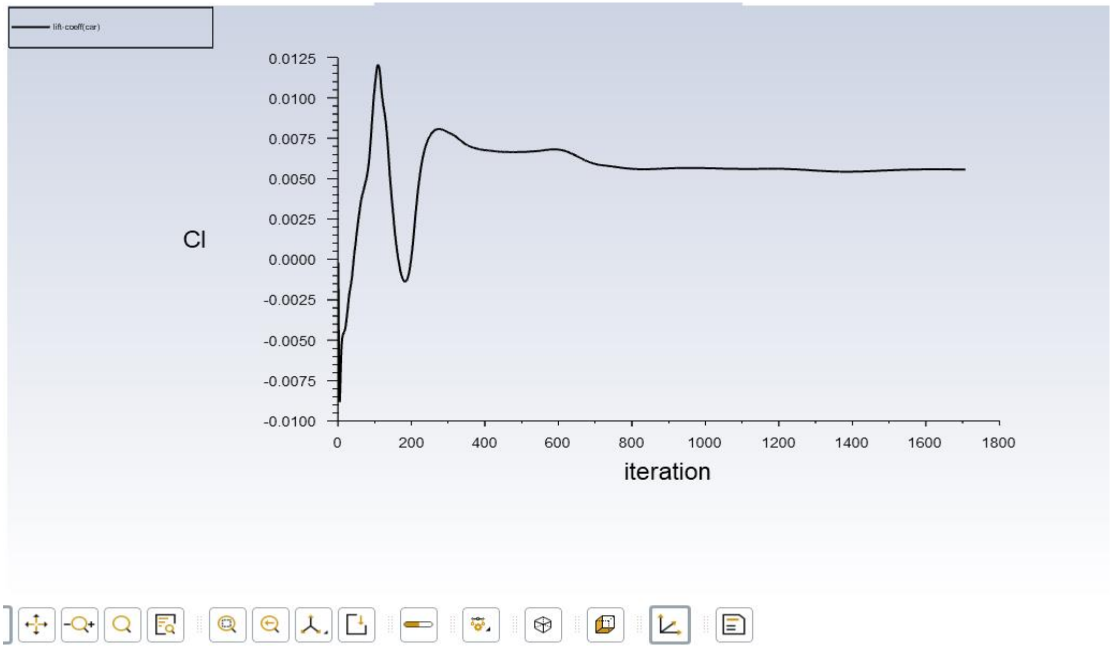
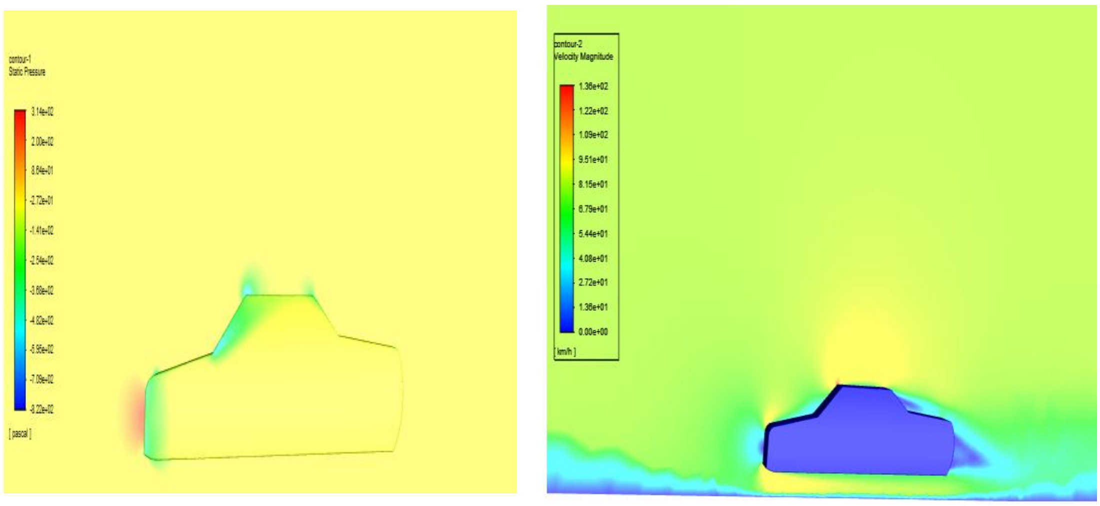


Figure 0.8 40km/hr Lift Coefficient Plot

ii) At 80km/hr



(a) **(b)**

Figure 0.9 80km/hr (a) Pressure and (b) Velocity Contours

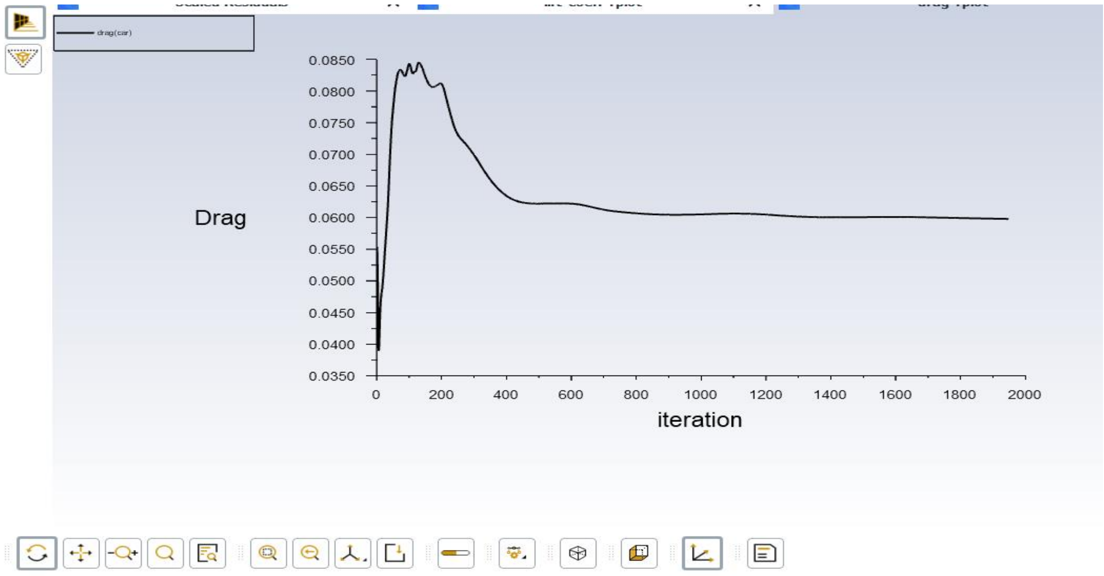


Figure 0.10 80km/hr Drag Coefficient plot

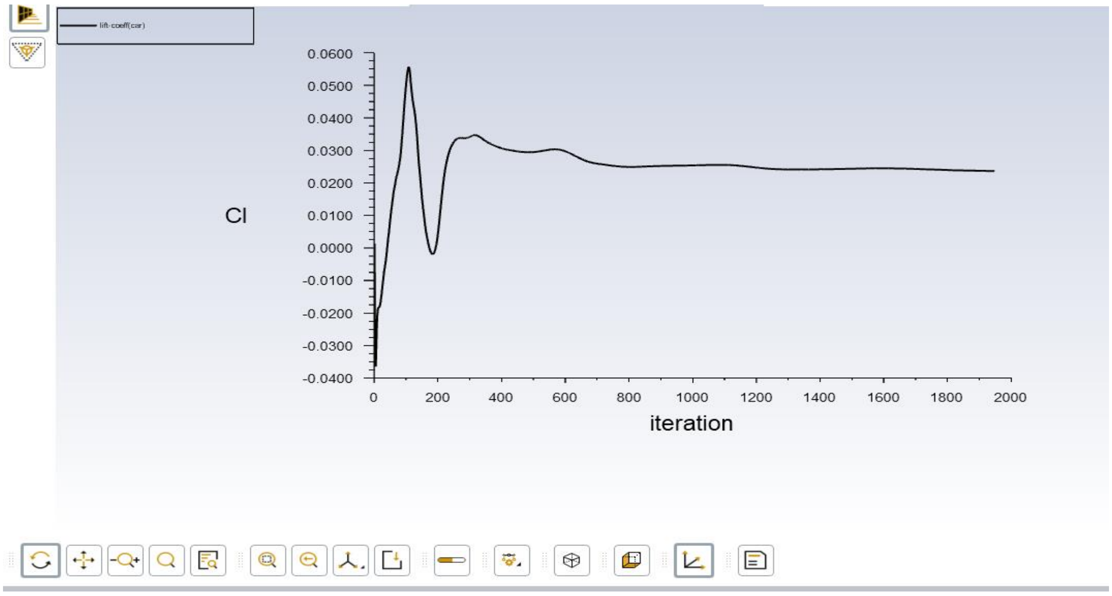
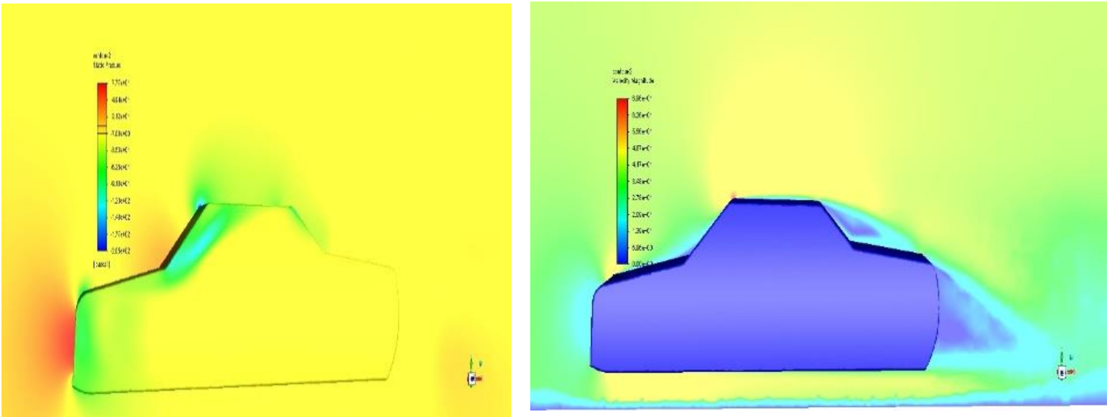


Figure 0.11 80km/hr Lift Coefficient Plot

iii) At 120km/h



(a) **(b)**

Figure 0.12 120km/hr Pressure and Velocity

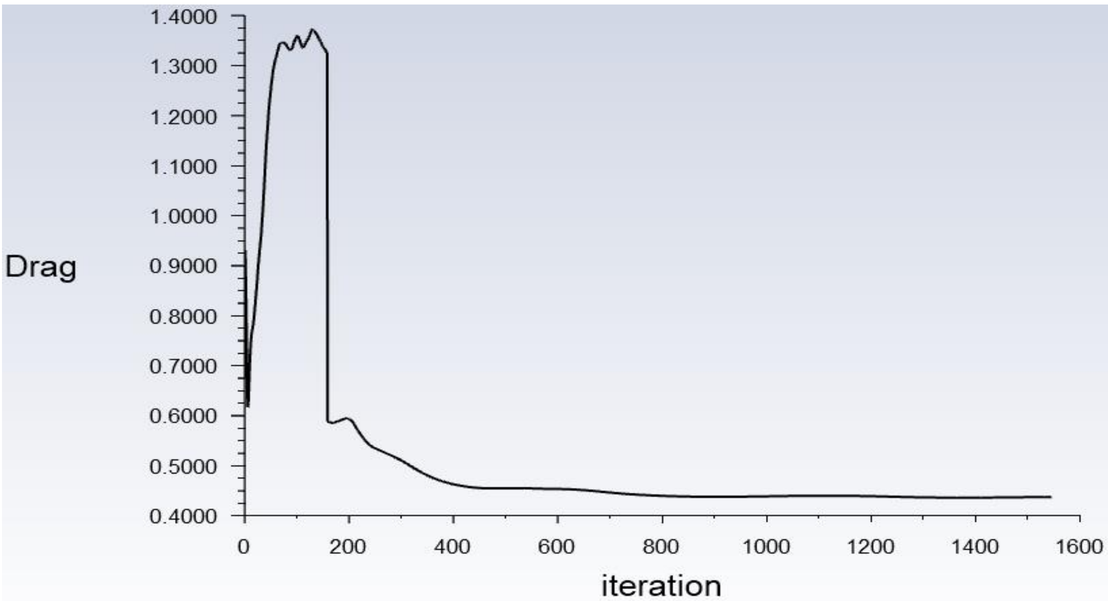


Figure 0.13 120km/hour Drag Coefficient Plot

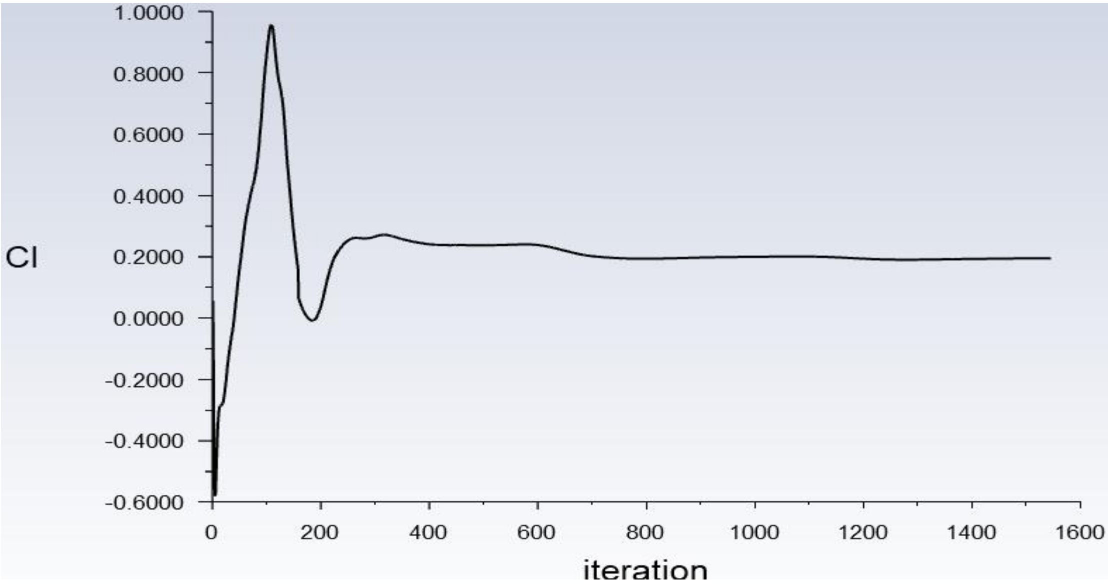


Figure 0.14 120km/hour Lift Coefficient plot

Table 1 Analysis results of Base model

Velocity (Km/h)	**C_D**	**C_L**
40	.42	.059
80	.42	.098
120	.43	.195

7.3 Model & Meshing with VGs & Rear Spoiler:

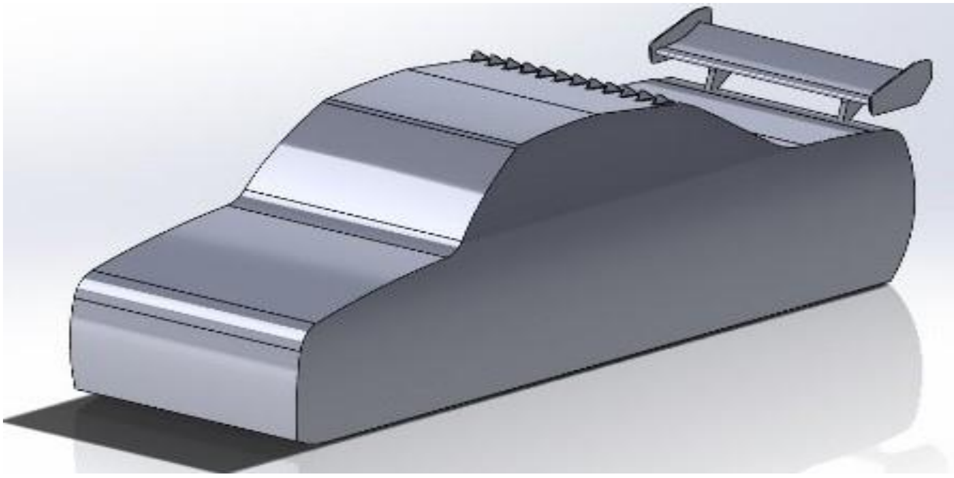


Figure 0.15 Base model with components (solid)

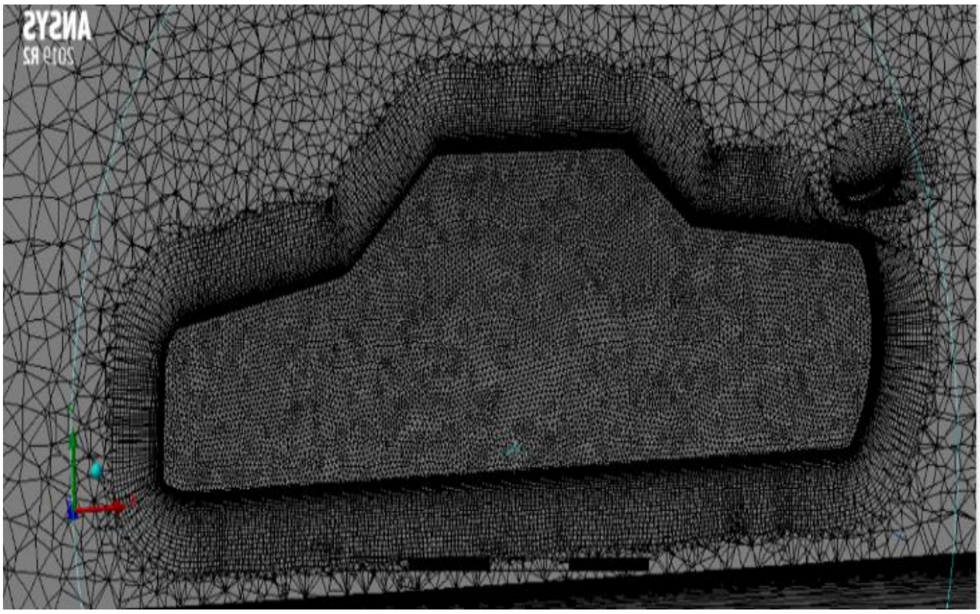


Figure 0.16 Meshing of the final model

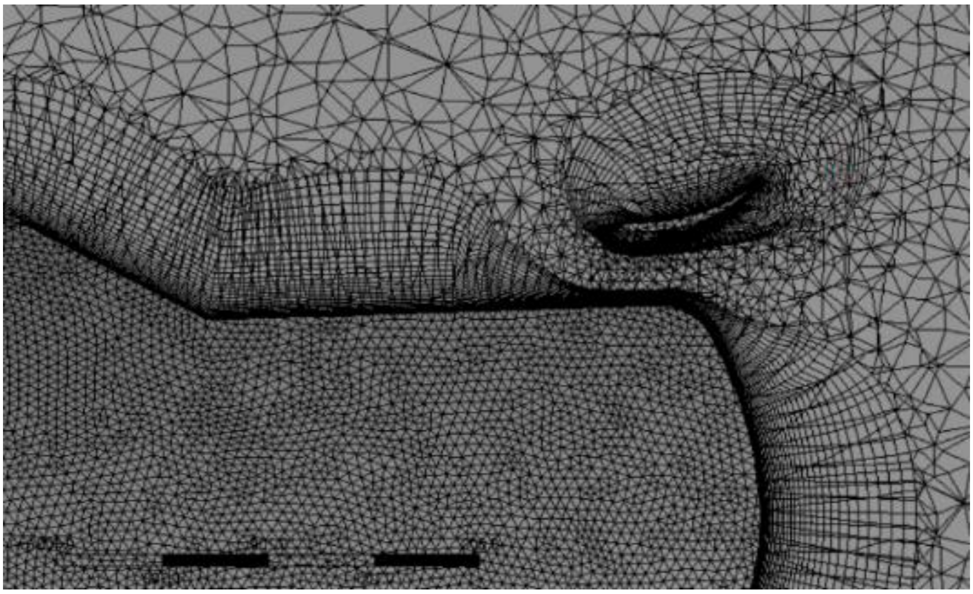


Figure 0.17 Meshing of Rear spoiler

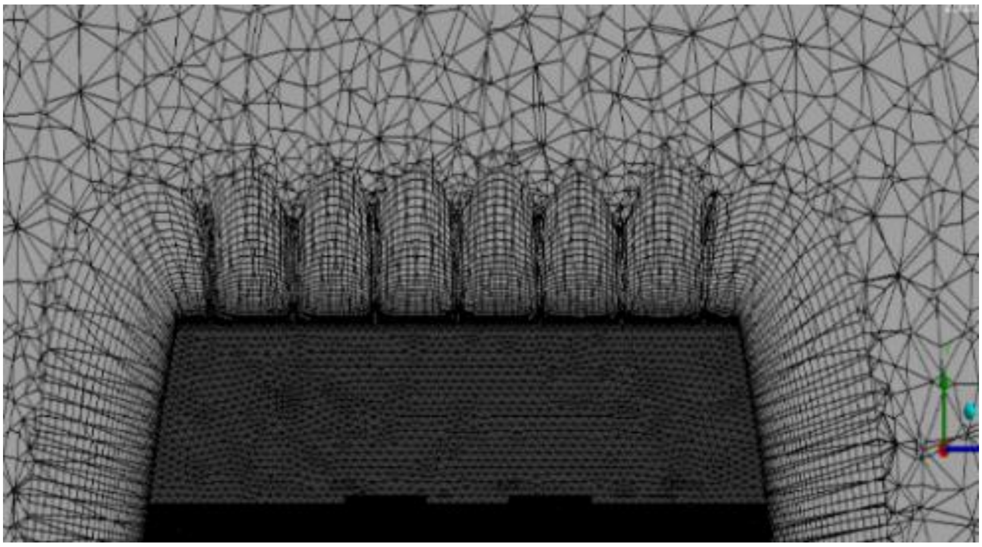


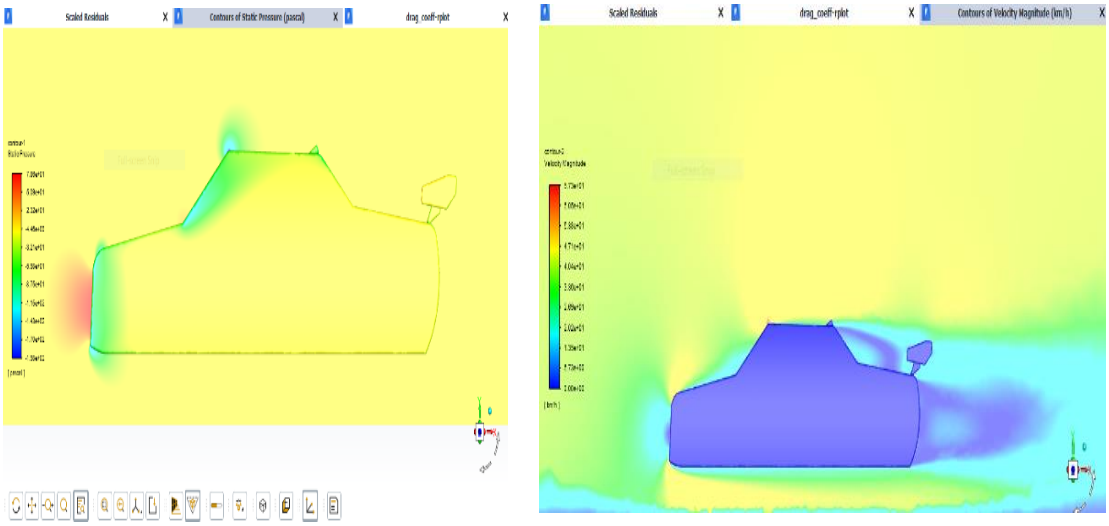
Figure 0.18 Meshing of vortex Generator

The same settings of base model were used in meshing the final model with the addition of face sizing of each VG & rear spoiler so that better mesh quality could be obtained & better results can be recorded. The values of nodes & elements recorded were 2001992 & 4489714.

7.4 Analysis of Final Model:

The simulation was done at various velocities to get the better results of our base model. The viscous model was taken as standard k-epsilon and steady flow was used for the simulation. The solution method scheme used was set as "SIMPLE" and the special Discretization "Gradient" was "Least Squares Cell Based" the pressure was taken as "Second Order" whereas momentum was taken as "Second Order Upwind". After simulating the wind tunnel test at different velocities following results were observed.

i) At 40km/hr



(a) (b)

Figure 0.19 40km/hour Pressure and Velocity Contours

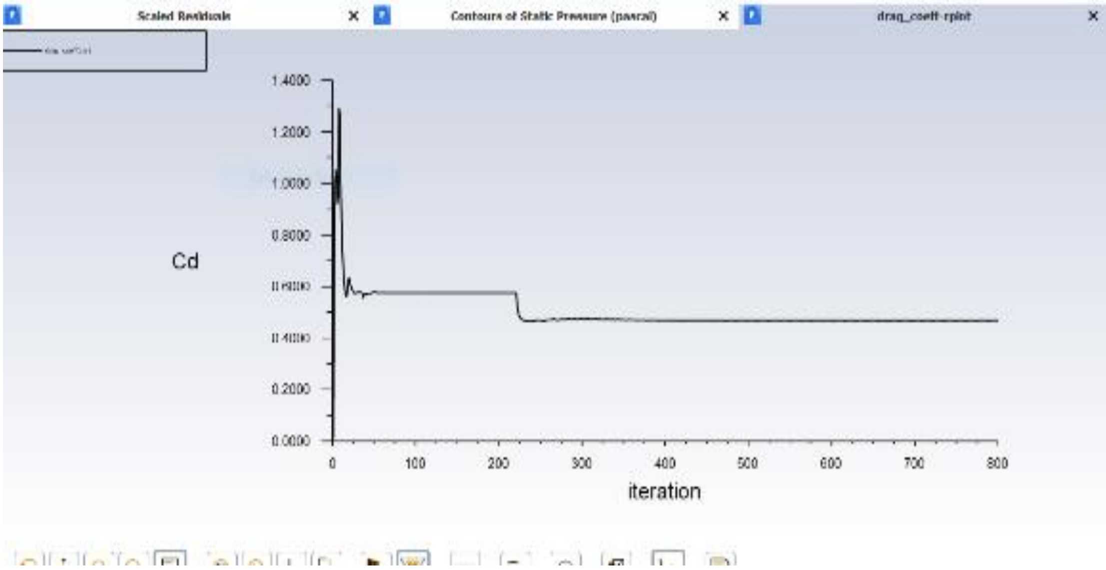


Figure 0.20 40km/hour Drag Coefficient Plot

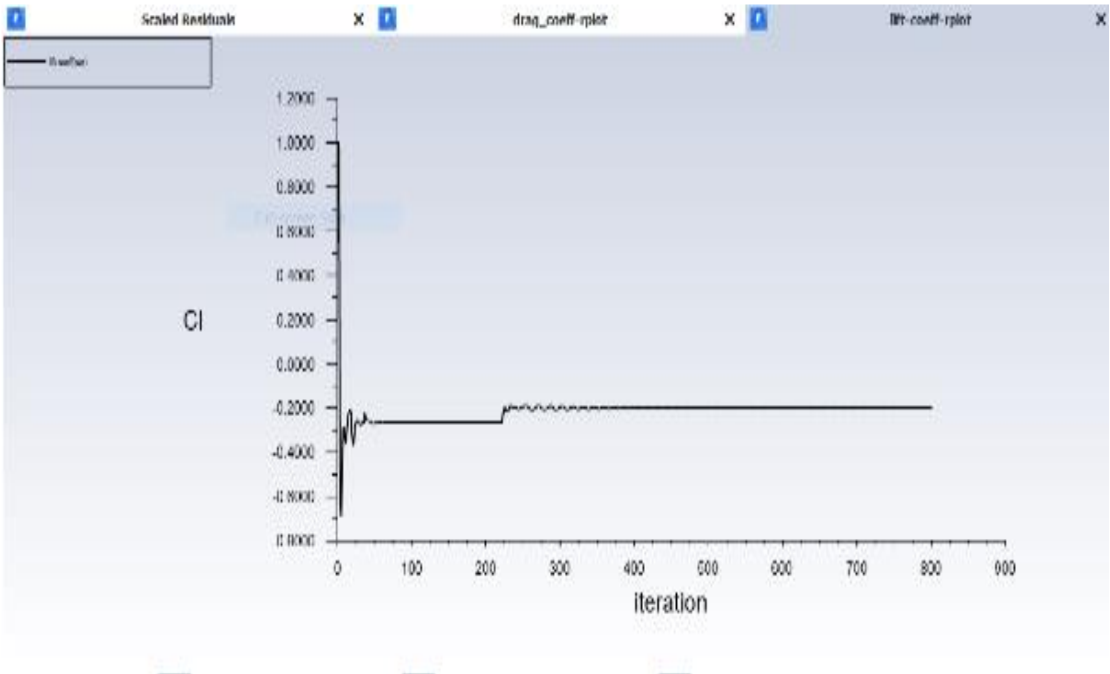


Figure 0.21 40km/hour Lift Coefficient plot

ii) At 80km/hr

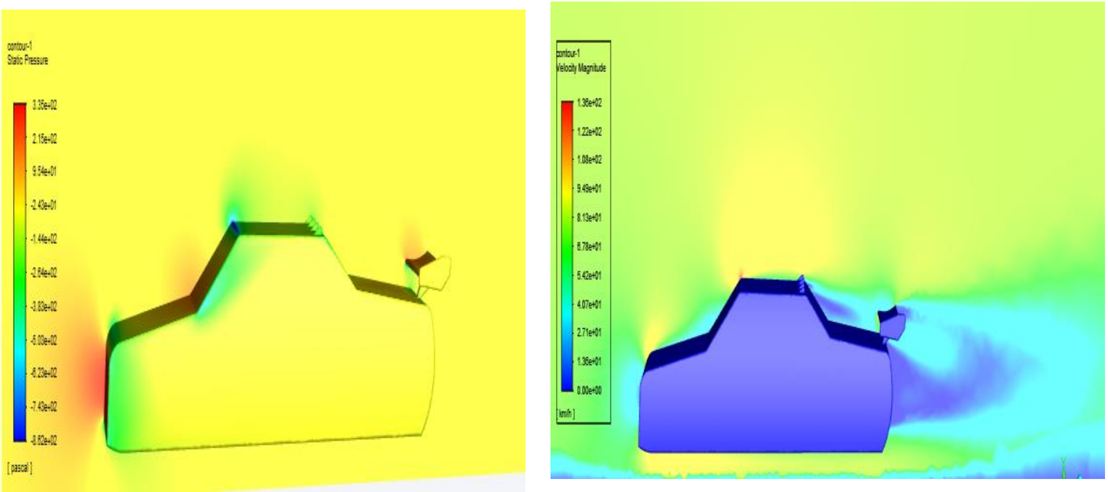


Figure 0.22 80km/hour Pressure and velocity contours

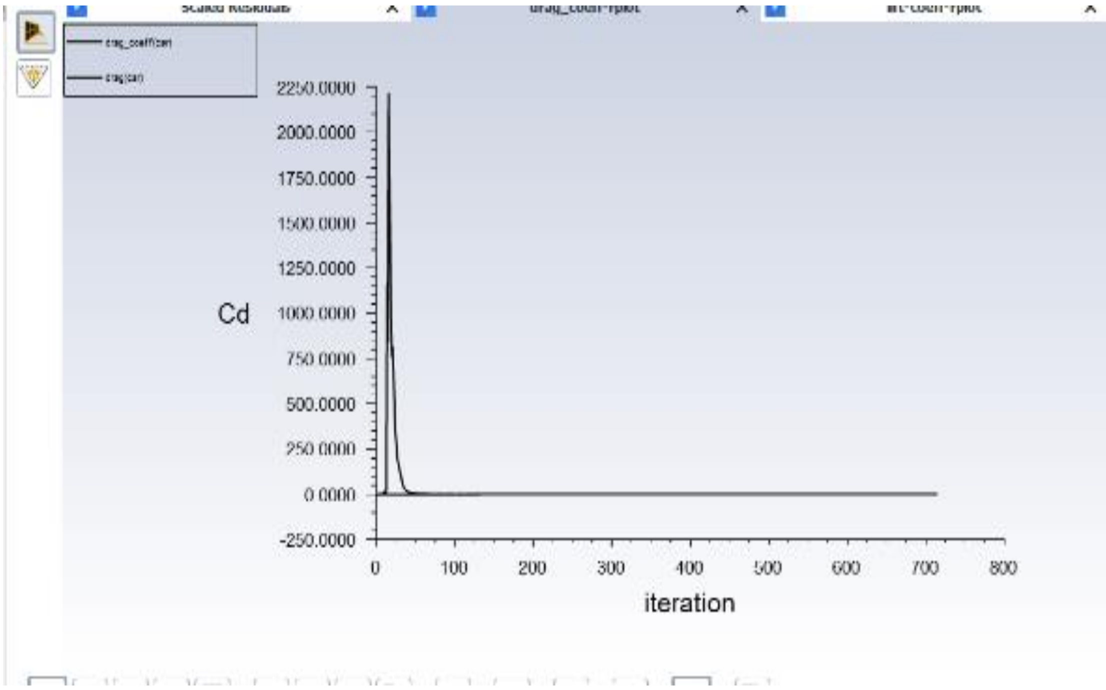


Figure 0.23 80km/hour Drag Coefficient Plot

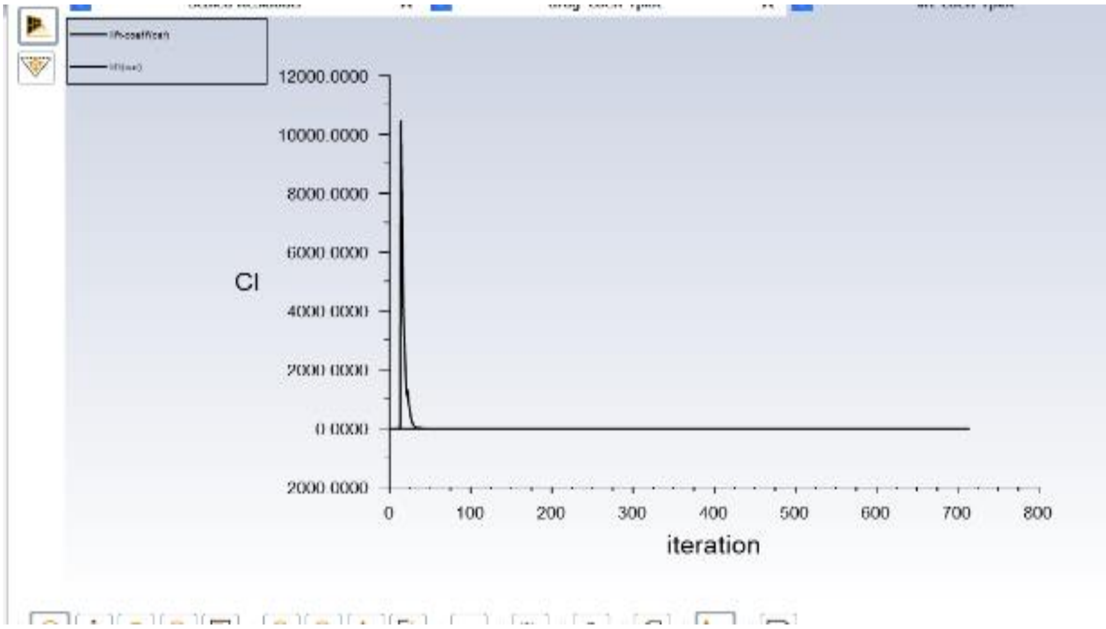
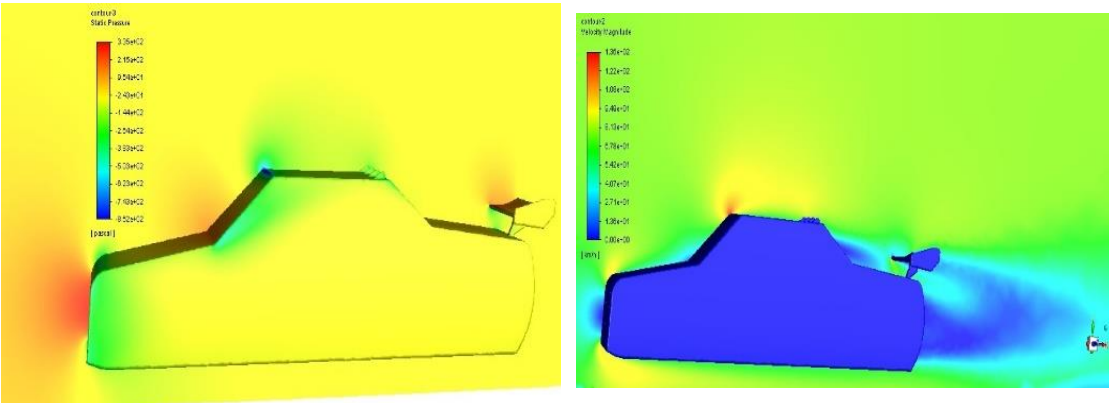


Figure 0.24 80km/hour Lift Coefficient plots

iii) At 120km/hr



(a) **(b)**

Figure 0.25 Pressure & Velocity Contour of final Model, 120km/hour

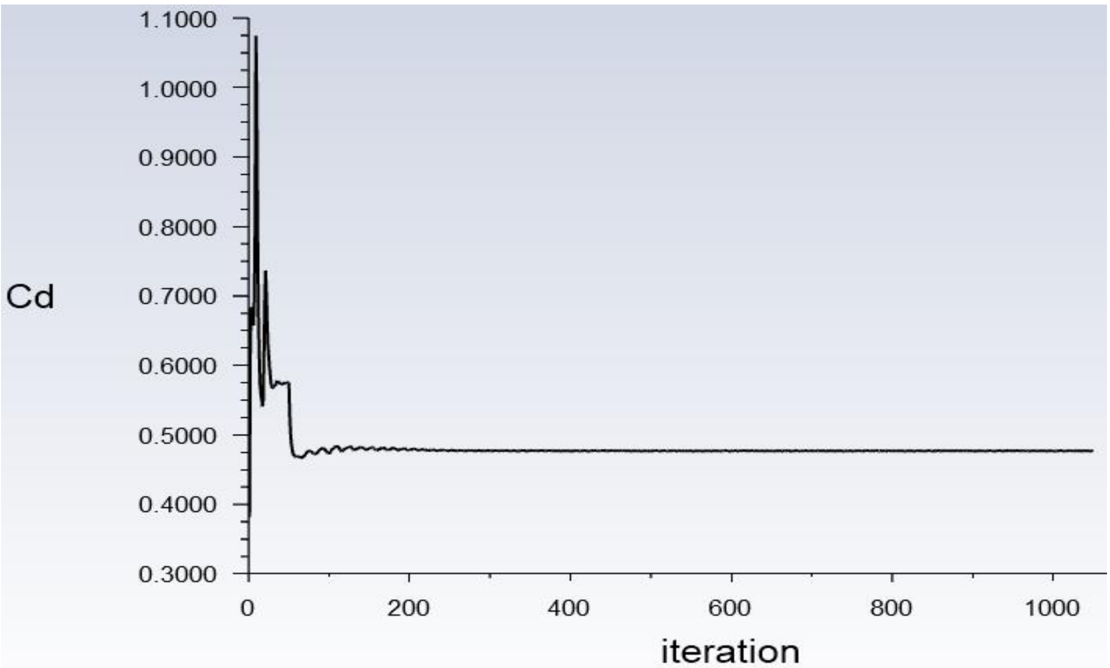


Figure 0.26 Drag Co-efficient plot of final model, 120km/hour

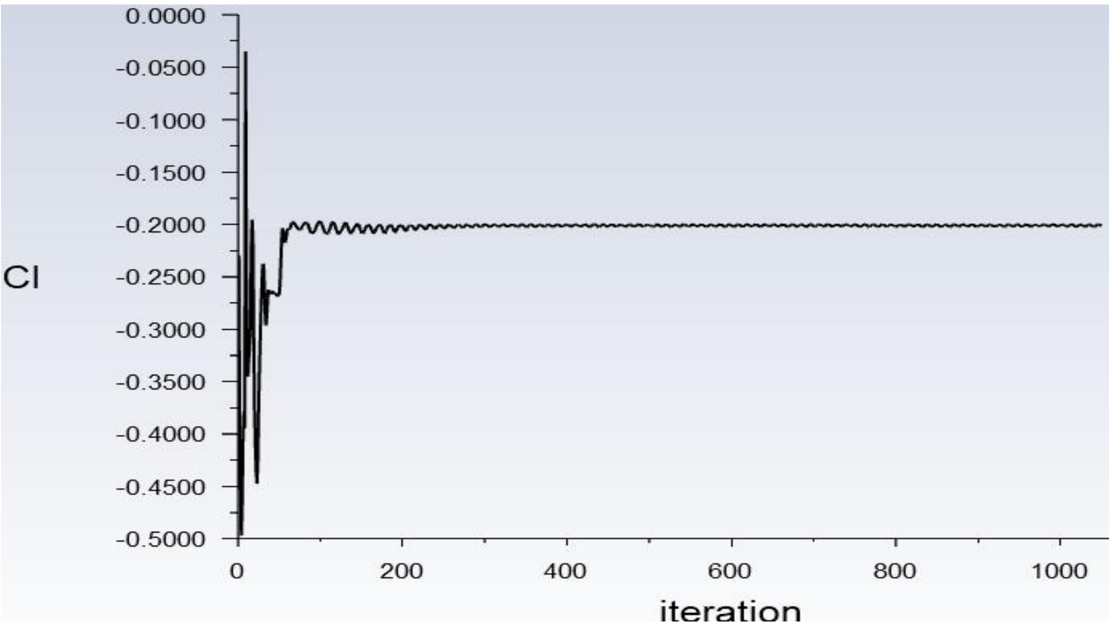


Figure 0.27 Lift Coefficient Plot of final model, 120km/hour

Table 2 Result analysis of final model

Velocity (km/h)	C_D	C_L
40	.44	-.188
60	.46	-.196
120	.47	-.201

Chapter 8

RESULTS & DISCUSSION

In comparing the base model results at changing velocities, it can be seen that the drag & lift coefficients values are increasing steadily with the increase in velocity. At 40 km/h the base model didn't show any significant drag coefficient value because of low speed, & the coefficient value recorded was 0.42, whereas at 80 km/h & 120 km/h, the values of ΔC_d were 0.42 & 0.43.The lift coefficient values also increased steadily with ΔC_l (at 40 km/h) .055 whereas ΔC_l (at 80 & 120 km/h) was .098 & .195. These results clearly show that with the increase in vehicle speed, the value of drag & lift coefficients increases steadily with maximum drag & lift co-efficient value at 120 km/h. Increased ΔC_l can makes the car unstable at higher speeds thus, aerodynamic components must be used.

Now with use of aerodynamic components with the base model. The value of ΔC_d increased marginally around 9% change with increasing speeds, but ΔC_l values decreased around 200% as expected because of aerodynamic components. Rear spoiler and VGs helped the car to achieve the negative lift value which will help the car model to generate downforce, which will further be beneficial in maintaining better stability & control at higher speeds.

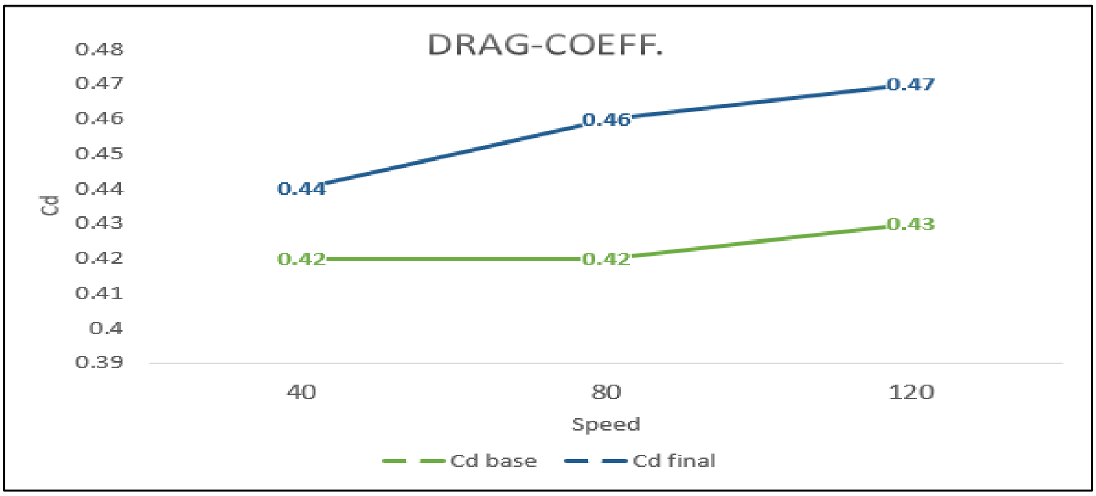


Figure 0.1 Comparison of Co-efficient of Drag of final model to the base model

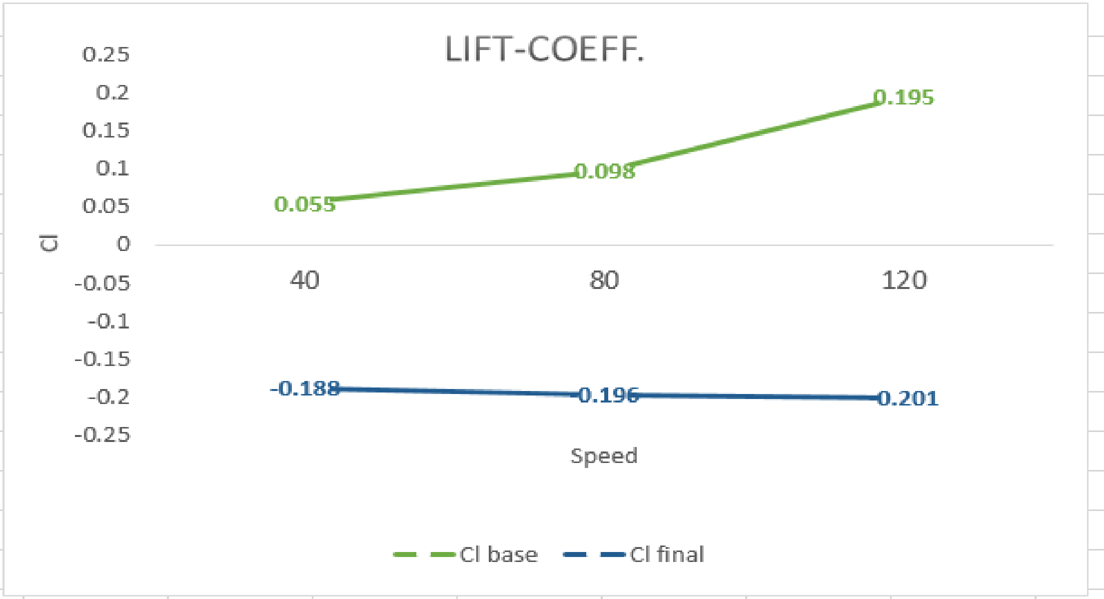


Figure 0.2 Comparison of Co-efficient of Lift of final model to the base model

Chapter 9

CONCLUSION

This study shows the Cd observed for the base model was .43 at higher speed, whereas with Vortex Generators & Rear spoiler attached, we saw an increase of about 9.3% in Cd value when compared to the base model at a speed of 120 km/h. Addition of spoiler can be considered as the reason for increased ΔCd value. The Cl value observed was much lower (around 200% change) which is one of the basic concerns of the model at higher speeds.

Thus, it can be concluded that adding aerodynamic components to the budget sedan cars can help get much better stability & control at higher speeds, which will also help in smooth driving. Increased drag can also be improved by making some design changes in the overall shape of the car, & thus, lower budget sedan cars can also be much stable to drive at higher speeds.

Chapter 10

LIST OF PUBLICATIONS

1. Nikhil Pasricha, Sartaj Anwer Khan, Faisal Shameem (2021) EFFECT OF VORTEX GENERATORS & REAR SPOILER ON A LOW-END SEDAN PASSENGER CAR.

References

1. America E .: Showstopper. 1–2 (2010)
2. Madane P., Pande K., Gote P., Dongare P.: Study and Overview of Aerodynamic Active Rear Wing of High speed Vehicles. 4127–4132 (2020)
3. Guda NT., Surisetti BV., Ram S ., Kolla C.: Enhancing Aerodynamic Performance of a HatchBack Model Passenger Car using Ansys Fluent Software Volume IX Issue V , MAY / 2020 Page No : 2593 Issn No : 2347-3150. IX:2593–2605 (2020)
4. Yuan CS., Mansor S., Abdullah MA.: Effect of spoiler angle on the aerodynamic performance of hatchback model. Int J Appl Eng Res 12:12927–12933 (2017)
5. pilakyaa TD., Tuleun LT., Kekung MO.: Computational fluid dynamics modelling of an aerodynamic rear spoiler on cars. Niger J Technol 37:975. https://doi.org/10.4314/njt.v37i4.17 (2018)
6. Das RC., Riyad M.: CFD analysis of passenger vehicleat various angle of rear end spoiler. Procedia Eng 194:160–165. https://doi.org/10.1016/j.proeng.2017.08.130 (2017)
7. Dickison M., Ghaleeh M., Milady S.: et al, Investigation into the aerodynamic performance of a concept sports car. J Appl Fluid Mech 13:583–601. https://doi.org/10.29252/jafm.13.02.30179 (2020)
8. Patel JR.,: Fluid Dynamics Simulation of a Car Spoiler. 0–11. https://doi.org/10.13140/RG.2.2.18149.91364 (2017)
9. Prabhu L., Krishnamoorthi S., Gokul P.: et al, Aerodynamics analysis of the car using solidworks flow simulation with rear spoiler using CFD. IOP Conf Ser Mater Sci Eng 993:. https://doi.org/10.1088/1757-899X/993/1/012002 (2020)
10. Sen W.: Experimental and CFD analysis on car with several types of icmere2019-pi-185 experimental and CFD analysis on car with several types of vortex generators (2020)

11. Yakkundi V., Mantha SS.: Effect of Spoilers on Aerodynamic Properties of Car Effect of Spoilers on aerodynamic properties of a car. 7:271–280 (2018)
12. Elewe AM .: Numerical simulation of surface curvature effect on aerodynamic performance of different types of airfoils. IOP Conf Ser Mater Sci Eng 928:. https://doi.org/10.1088/1757-899X/928/3/032003 (2020)
13. Tsai CH ., Fu LM ., Tai CH .: Computational aero-acoustic analysis of a passenger car with a rear spoiler. Appl Math Model 33:3661–3673. https://doi.org/10.1016/j.apm.2008.12.004 (2009)
14. ANSYS Fluent 2019 R2 Theory Guide, https://ansyshelp.ansys.com/, last accessed 2021/04/16
15. Velagapudi NK ., K. LN ., Rao LNVN ., Y. SR .: Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler using CFD. Int J Adv Sci Res 1:331. https://doi.org/10.7439/ijasr.v1i8.2510 (2015)
16. Cakir M .: Scholar Commons CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle. 1–72 (2012)