STUDY OF LIFT AND DRAG FORCES ON VEHICLE BY VARYING SPOILER ANGLES

A PROJECT REPORT SUBMITED IN PARTIAL FULFILLMENT OF THE REQUIRMENTS FOR THE AWARD OF THE DEGREE OF 18

BACHELOR OF ENGINEERING

IN

MECHANICAL ENGINEERING

SUBMITTED BY

P.RAM CHARAN TEJ	:	314126520113
M.BHARAT KUMAR	:	314126520097
M.PAVAN TEJA	:	314126520091
M.RIDVIK	:	314126520092

Under the esteemed guidance of

BASANAGARAJU

ASSISTANT PROFESSOR

Dept. of Mechanical Engineering

ANITS

DEPARTMENT OF MECHANICAL ENGINEERING
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
(AFFILIATED TO ANDHRA UNIVERSITY)
SANGIVALSA, VISAKHAPATNAM (DISTRICT)

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES

(AFFILIATED TO ANDHRA UNIVERSITY)

SANGIVALSA, BHEEMUNIPATNAM (MANDAL), VISAKHAPATNAM

PRAGNANAM BRAHMA

ANITS

CERTIFICATE

This is to certify that the Project Report Work entitled **"STUDY OF LIFT AND DRAG FORCES ON VEHICLE BY VARYING SPOILER ANGLES"** has been carried out by M. Pavan Teja(314126520091), M. Bharat Kumar (314126520097), P.Ram Charan Tej(314126520113), M.Ridvik (314126520092) of Mechanical Engineering, **ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES,** Visakhapatnam during the Year 2017-2018 in partial fulfilment of the requirements for the award of the Degree of Bachelor Of Mechanical Engineering by ANDHRA UNIVERSITY, VISAKHAPATNAM.

APPROVED BY

18.4.18

(Prof.B.Naga raju)
Head of Department
Dept. of Mechanical Engineering
ANITS, Sangivalasa
Visakhapatnam

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

PROJECT GUIDE

B.N. Raju

(Mr.Basa Nagaraju)
Asst.Professor
Dept. of Mechanical Engineering
ANITS, Sangivalasa
Visakhapatnam

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

1. **INTERNAL EXAMINER:**

Dr. B. Naga Raju
M.Tech,M.E.,Ph.d
Professor & HOD
Dept.of Mechanical Engineering
ANITS, Sangivalasa,
Visakhapatnam-531 162

2. **EXTERNAL EXAMINER:**

ACKNOWLEDGMENT

We are extremely thankful to our honourable principal Prof.**T.V.Hanumatha Rao**, Anil Neerukonda Institute of Technology & Sciences for their Valuable Suggestions.

We express immensely our deep sense of gratitude towards Prof. **B.Naga Raju**, Head of Department of Mechanical Engineering and our Internal guide **Mr. Basa Nagaraju** for interest he took and constant encouragement in throughout the completion of the project.

We would like to thank staff of our Mechanical department and all our friends who helped and given Encouragement during this project.

We express our Sincere thanks to the members of non- teaching staff of Mechanical Engineering for their Kind Co-operation and Support to Carry on Work.

M.PAVAN TEJA (314126520091)

M.BHARAT KUMAR (314126520097)

M.RIDVIK (314126520092)

P.RAM CHARAN TEJ (314126520113)

ABSTRACT

Drag force is the resisting force acting in the opposite direction to the motion of a vehicle. This is one of the main constraints to reduce the fuel efficiency. The drag force increases with the increase in speed of the vehicle. Along with drag force considerable amount of lift force will also act on the vehicle at higher speeds. This will lift off the vehicle from the ground which results in decreased stability in handling the vehicles. Ongoing research of automobiles is focussed on reducing the drag and lift forces which will increase the fuel economy and the stability of the vehicle. This can be achieved by proper streamlining of vehicle body or by attaching a rear end spoiler. The process and costs involved in proper streamlining the vehicle bodies are very high compared to attaching a rear end spoiler.

Spoiler is an automobile aerodynamic device whose function is to spoil unfavourable air movements across a body of a vehicle in motion. Rear spoilers are provided on the vehicles, create negative lift which will help in non- lifting of vehicles moving at high speeds. Rear spoilers will also help in reducing the drag forces by vortex braking. In modern days, light weight vehicles are more preferable because they can attain high speeds and have good fuel economy. At high speeds, the rear end of the vehicles lifts upwards because of aerodynamic lift. To counter this upward lift, the spoiler is placed at the rear end, which will oppose it and can maintain traction.

In the present study, the drag and lift forces of a sedan type car without spoiler and spoiler with different inclinations are studied by using fluent flow solver in ANSYS workbench software

It is observed that, the lift and drag forces on car body are reduced by attaching a spoiler. It is further observed that the lift and drag forces are decreasing with increased angle of spoiler in the downward direction whereas increases with increased angle in the upward direction.

LIST OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

1. INTRODUCTION

According to international energy agency in the world, the emissions of gas with greenhouse effect may increases to the 57% in 2030 with the strong effects on the environment and the climate in the world energy outlook 2007.The human activities became major cause in increasing the greenhouse effect of gases and also the average global temperature. The activity that includes transportation sector where number of automobile is rapidly increasing and also increases the fuel consumption. It tends to create harmful effects on environment by increasing air pollution. Because of these problems, the automobile industries try to minimize the greenhouse effect by reducing the fuel consumption.

The automobile fuel consumption is related to its aerodynamic drag which influenced highly by the flow separation over its shape. Meanwhile the flow over automobile in motion is complex and presents non-linear interactions between the different parts of automobile.

External aerodynamics of road vehicles determines many aspects of an automobile like fuel consumption, stability and comfort at high cruising speeds. The flow over the vehicles is characterized by high turbulence in three dimensional flow separations and there is a growing need for more insight in the physical features of the vehicles.

The automotive industry uses the CFD package which is a commercial tool. The CFD is not only used for the improvement of the vehicles aerodynamics, but also for domain optimization like brake cooling, lighting, engine cooling, fuel system and airbags. While the product development of new road vehicles, leads to understanding the phenomenon of flow behaviour and also the aerodynamic forces that are influenced by the changes in aesthetic shape of the vehicles body. The CFD is best way for the designers to obtain the results in a short interval of time.

1.1. History and Developments of Automobiles

In the 17th century, engineers of European countries began tinkering with the vehicles of motor power. The steam, electrical and combustion motors that have been attempted in 1800's. By 1900's, it is the uncertain of which the engine type would power automobiles. At first, the most popular car is the electric car, but at that time battery doesn't exist which allowed the car to move with maximum speed. The earlier records of speed are set by the electric cars and these did not stay in the production of 20th century. The automobiles of steam driven lasted till 1920's.However,price on the steam powered engines either to build or to maintain was incomparable to gas powered engines. The price was not only the problem, but there is a risk of the boiler explosion. The combustion engine that continually beat out competition, and early American automobile pioneers such as the Ransom E. olds and the Henry Ford built reliable combustion engines, rejecting ideas of the steam or the electrical power.

The automotive production on the commercial scale was started in the France in 1890. The commercial production in the United States began at the beginning of 1900's and was equal to that of the Europe. The European industry was consisted of small independent firms which would turn out the few cars by the means of precise engineering and the handicraft methods. The American automobile plants that were assembly lined operations, which meant for using parts made by the independent suppliers and then putting them together at plant .In early 1900's, the United States had firms of about producing 2000 cars. By 1920, number of firms that had decreased to about 100 and by the 1929 to 44. In the 1976, Motor Vehicles Manufacturers Association had only 11 members. The same situation occurred in the Europe and the Japan.

The first automobile produced in the US for masses was the three horse-powered and curved-dash Oldsmobile. 425 of them were sold in the1901and 5000 in the 1904. This model is still prized by the collectors. The firm prospered and it was noted by the others, and from 1904 to 1908,24 automobile manufacturing firms went into the business in United States. One of these was Ford Motor Company and was organised in the June 1903, and sold its first car on July 23. The company produced1700 cars during its first full year of the business. The Henry ford produced the model T to be an economical car for an average American. By 1920, Ford sold over a million numbers of cars.

At the beginning of the century the automobiles entered into transportation market as a rich toy and it became increasingly popular among the general population because it gives freedom to the travellers to travel when they want and also to where they want. As a result of this, in Europe and North America, automobiles became the cheaper and also more accessible to middle class and this was facilitated by Henry Ford who performed two most important things. First, he had priced his car which is as affordable as possible and the second; he had paid his workers that purchased the cars which they are manufacturing. This made the push wages and the auto sales upward.

The history had said that Henry Ford freed the common people from geography limitations. Automobiles created the mobility on the scale never known before, and total effects on the living habits and the social customs is endless. In the transportation days of horse-drawn, the wagon's travel of practical limit was 10 to 15 miles. This serves the transportation from a railroad or city to the individual farm and vice versa. The motor vehicles on paved roads had narrowed gap between urban and rural life. The farmers shipped easily and economically by the trucks and it can drive to the town when it is in the state of convenience.

1.2. Highlights of Motor Vehicles History

The early history of the automobile can be divided into number of eras, based on the prevalent means of propulsion. Later periods were defined by trends in exterior styling, size, and utility preferences. In 1769 the first steam-powered automobile capable of human transportation was built by Nicolas Joseph Cugnot. In 1808, Francois Isaac De Rivaz designed the first car powered by an internal combustion engine fuelled by hydrogen.

In 1870 Siegfried Marcus built the first gasoline powered combustion engine, which he placed on a pushcart, building four progressively sophisticated combustion-engine cars over a 10 to 15 year span that influenced later cars. Marcus created the two-cycle combustion engine. The car's second incarnation in 1880 introduced a four-cycle, gasoline-powered engine, an ingenious carburetor design and magneto ignition. He created an additional two models further refining his design with steering, a clutch and brakes.

The four stroke petrol (gasoline) internal combustion engine that still constitutes the most prevalent form of modern automotive propulsion was patented by Nikolaus Otto. The similar four-stroke diesel engine was invented by Rudolf Diesel. The hydrogen fuel cell, one of the technologies hailed as a replacement for gasoline as an energy source for cars, was discovered in principle by Christian Friedrich in 1838. The battery electric car owes its beginnings to Anyos jedlick, one of the inventors of the electric motor and Gaston Plante who invented the lead acid battery in 1859.In 1885, Karl Benz developed a petrol or gasoline powered automobile. This is also considered to be the first "production" vehicle as Benz made several other identical copies. The automobile was powered by a single cylinder four-stroke engine.

In 1913, the Ford model T, created by the Ford Motor company five years prior, became the first automobile to be mass produced on a moving assembly line. By 1927, Ford had produced over 15,000,000 Model T automobiles.

At the turn of the 20th century electrically powered automobiles were a popular method of automobile propulsion, but their common use did not last long, and they diminished to a niche market until the turn of the 21st century.

1.3 Styling forms

A Sedan(American, Canadian, Australia and English) or Saloon(British, Irish and Indian English) is a passenger car in a three-box configuration with A, B & C-pillars and principal volumes articulated in separate compartments for the engine, passenger and cargo. The passenger compartment features two rows of seats and adequate passenger space in the rear compartment for adult passengers. In sedan class vehicles there is no D pillar

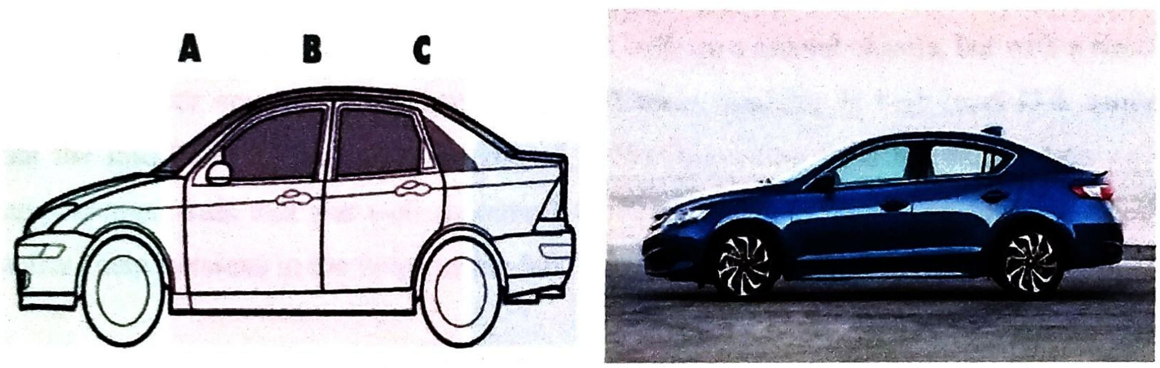


Figure 1.1: Sedan car

Hatchback is a car body configuration with a rear door that swings upward to provide access to a cargo area. Hatchbacks may feature fold-down second-row seating, where the interior can be flexibly reconfigured to prioritize passenger vs. cargo volume. Hatchbacks may feature two- or three-box design. Hatchback has three pillars namely A, B, &C pillars excluding D pillar.

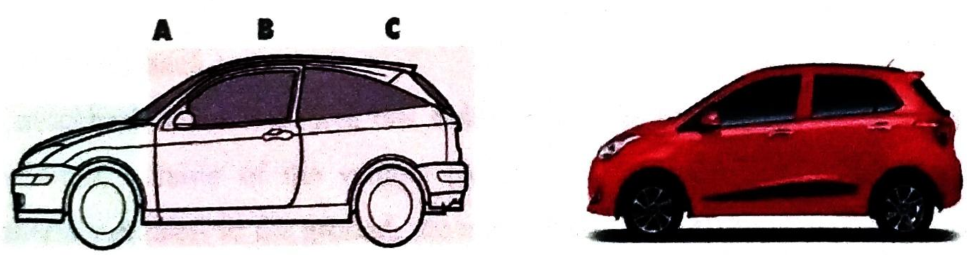


Figure 1.2: Hatchback car

SUV in Short for Sports Utility Vehicle or Suburban Utility Vehicles and is a large car, which were designed to go over sharp surfaces or off-road but is essentially driven in the city and highways. These can accommodate about 5 to 7 people. SUVs also features big ground clearance, high seating, huge body, high centre of gravity, and weigh considerably more than other car types resulting higher fuel consumption. SUV has all the four pillars namely A,B,C&D.

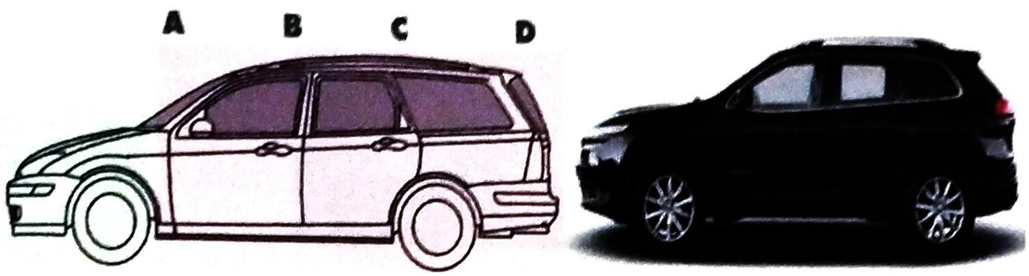


Figure 1.3: SUV car

1.4 Types of sedan vehicles

1.4.1 Club sedans

A club sedan is a two or four door design built on a normal chassis, but with a shorter roof and interior space. Club sedans were most often available in high-level U.S. models from the middle of 1920s to the middle of 1950s.Originating from the club car on a pull man passenger train that was well appointed, the "club" term imparted a sense of class to the smaller cabin versions in the range of models.

Figure 1.4: club sedans

1.4.2 Notchback sedans

A notchback sedan is a three box sedan, where the passenger volume is clearly distinct from the trunk volume of the vehicle (when seen from the side). The roof is on one plane, generally parallel to the ground, the rear window is at a sharp angle to the roof, and the trunk lid is also parallel to the ground.

Figure 1.5: notchback sedans

1.4.3 Fastback sedans

A fastback sedan is a two-box sedan, with continuous slope from the roof to the base of the deck lid (trunk lid), but excludes the hatchback feature. Typically, this design is chosen for its aerodynamic advantages.

Figure 1.6: Fastback sedan

1.4.4 Hardtop sedans

In historic terminology, a sedan door will have a frame around the door windows, while the hardtop doors end at the waist line. There is no centre or "B" pillar for roof support behind the front doors. This pillar less body style offers greater visibility. However, it requires extra under body strengthening for structural rigidity. The hardtop design can be considered separately (i.e., a vehicle can be simply called a four-door hardtop), or it can be called a hardtop sedan.

Figure 1.7: Hardtop sedan

1.4.5 Hatchback sedans

Hatchback sedans typically have the fastback profile, but instead of a trunk lid, the entire back of the vehicle lifts up (using a lift gate or hatch).

Figure 1.8: Hatchback sedan

1.4.6 Chauffeured sedans

Strictly speaking limousine sedans have a separate compartment for the driver and the passenger compartment is long enough to contain at least two comfortable, forward-facing bench seats. The term limousine can refer to a large sedan, especially if hired from a service. Chauffeured limousines are primarily used by individuals for weddings, businesses for meetings, as well as for airport and sightseeing transportation.

Figure 1.9: Chauffeured sedans

1.5 Forces on stationary bodies due to external flows:

Fluid flows over solid bodies frequently occurs in practice, and it is responsible for numerous physical phenomenon such as drag force acting on automobile, power lines, trees and underwater pipelines. Therefore, developing a good understanding of external flow is important in the design many engineering systems such as aircraft, automobile, buildings, ships, submarines and all kinds of turbines.

Sometimes fluid moves over stationary bodies, and other times body moves through a quiescent fluid. These two seemingly different processes are equivalent to each other; what matters is the relative motion between the fluid and the body. Such motions are conveniently analysed by fixing the co-ordinate system on the body and are referred to as flow over bodies or external flow. Wind blowing over buildings and trees is the case where fluid passed over a stationary body. A vehicle moving with some velocity in the air is the case where body is moving through fluid. Both these cases are similar and can be studied by assuming fluid flows over a stationary body because it is practically difficult to estimate the forces on bodies which are moving in fluid. But the same can be easily done by keeping the body fixed and by passing the fluid over it. Experimentally wind tunnels are used for these kinds of situations and the same can be studied by using software also.

The resultant force exerted on body can be resolved into two components one in the direction of fluid motion and the other in the direction perpendicular to fluid motion. The force acting in the direction of fluid motion is called as Drag force and the force which is acting on the body in the direction perpendicular to direction of fluid motion is called as Lift force.

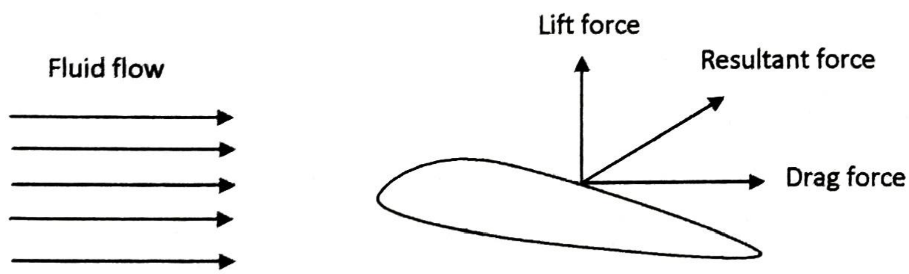


Figure 1.10: Forces on stationary body due to external fluid flow

1.6 Drag force

It is common experience that a body experiences some resistance when it is forced to move through a fluid. Higher the viscosity of fluid, higher the resistance offered. As you may notice, it is very difficult to walk in water because of the much greater resistance it offers to motion compared to air. Also, you may have seen high winds knocks down trees, power lines, even trailers felt the strong push of the wind exerts on your body. You experience the same feeling when your arm out of the window of a moving car. A fluid exerts forces and moments on a body in and above various directions. The force a flowing fluid exerts on the body in the flow direction is called drag. The drag force can be measured directly by simply attaching the body which is subjected to fluid flow to a calibrated spring and measuring the displacement in the flow direction. More sophisticated drag-measuring devices called drag balances use flexible beams fitted with strain gauges to measure the drag electronically.

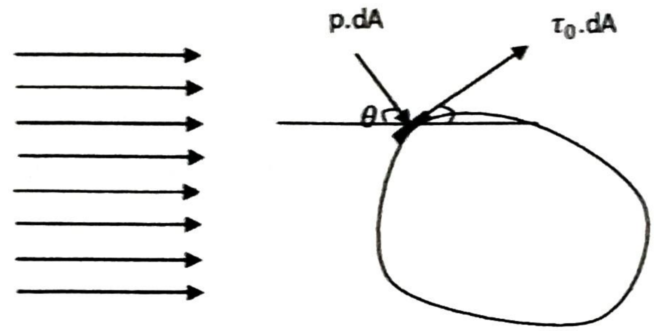


Figure 1.11: Drag and Lift

Let us consider a stationary obituary body is placed in a fluid moving with some velocity as shown in fig...... Now on a small elemental area the forces acting are pressure force and shear force. Let θ be the angle made by the pressure force with horizontal.

The pressure force acting on the element is p.dA and

Shear force acting on the element is τ_0.dA.

Therefore, drag force acting on the element = force due to pressure in the direction of motion

+ Force due to shear in the direction of motion

$= p.dA\cos\theta + \tau_0.dA.\sin\theta$........(1)

Therefore, Total drag force on the body $= \int p.dA\ \cos\theta + \int \tau_0.dA.\sin\theta$. ....(2)

Where, p = pressure

dA= elemental area

θ = Angle made by pressure force with horizontal direction

Mathematically drag force can be calculated by using, $F_d = C_d * 0.5 * \rho * A * v^2$..........(3)

Where, F_d = drag force

C_d = coefficient of drag,

ρ = density of fluid,

v = velocity of fluid over body and

A = largest projected area of the immersed body.

Drag is usually an undesirable effect, like friction, and we do our best to minimize it. Reduction of drag closely associated with the reduction of fuel consumption in automobiles, submarines, air craft and improved safety and durability of structures subjected to high winds and reduction of noise and vibration. But in some cases drag produces a very beneficial effect and we try to minimize it. Friction, for example, is a life saver in the brakes of automobiles. Likewise, it is the drag that makes it possible for the people to parachute, for pollens to fly to distant locations, and for all of us to enjoy the waves of the oceans and relaxing movements of the leaves of trees.

From the equation 1, it is clear that the drag force is a combination of two forces. Therefore, there are two types of drag force exists and they are friction drag and pressure drag.

1.6.1 Friction Drag:

The friction drag is the component of wall shear force in the direction of flow, and thus it depends on the orientation of the body as well as the magnitude of the wall shear stress. The friction drag is zero for a flat surface normal to flow and maximum for a flat surface parallel to flow since the friction drag in this case equals the total shear force on the surface. Therefore, for parallel flow over a flat surface, the drag coefficient is equal to the friction drag coefficient. Friction drag is a strong function of viscosity and increases with increase in viscosity.

The Reynolds number is inversely proportional to the viscosity of fluid. Therefore, the contribution of the friction drag to total drag for blunt bodies is less at higher Reynolds number and may be negligible at very high Reynolds number. The drag in such cases is mostly due to pressure drag. At low Reynolds number, most drag is due to friction drag. This is especially the case for higher streamlined bodies such as air foils. The friction drag is proportional to the surface area. Therefore, bodies with larger surface area experience a larger friction drag. For example, Large commercial airplanes reduce the total surface area and thus there by retracing their wing extensions when they rich cruising altitudes to save fuel. The friction drag coefficient is independent of surface roughness in laminar flow, but a strong friction of surface roughness in turbulent flow due to surface roughness elements protruding further into the boundary layer. The friction drag coefficient is analogous to the friction factor in pipe flow and its value depends on the flow regime

1.6.2 Pressure Drag:

The pressure drag is proportional to frontal area and to the difference between the pressure acting on the front and back of the immersed body. Therefore, the pressure drag is usually dominant for blunt bodies, small for stream lines bodies such as airfoils, and zero for thin flat plates parallel to the flow. The pressure drag becomes most significant when the velocity of the fluid is too high for the fluid to be able to flow the curvature of the body, and thus the fluid separates from the body at some point and creates a very low pressure region in the back, the pressure drag in this case is due to the large pressure difference between the front and back sides of the body

1.6.3. Flow Separation

When driving on country roads, it is common safety measure to slow down at sharp turns in order to avoid being thrown off the road. Many drivers have learned the hard way that a car will refuse to complain when forced to turn curves at excessive speeds. We can view this phenomenon as separation of cars from roads. This phenomenon is also observed when a fast moving vehicle jumps off hills. At low velocities the wheels of the vehicle always remain in contact with the road surface. But at high velocities, the vehicle is too fast to follow the curvature of the road and takes off at the hill, losing contact with the road.

Fluid acts much the same way when forced to flow over a curved surface at high velocities. A fluid climbs the uphill portion of the curved surface with no problem, but it has difficulty remaining attached to the surface on the downhill side. At sufficiently high velocities, the fluid streams detach itself from the surface of the body. This is called flow separation.

When a fluid separates from a body, it forms a separated region between the body and the fluid stream. This low pressure region behind the body, where recirculation and backflows occur is called the separated region. Pressure drag increases with increase in separated region. The region of the flow at the trailing edge of the body where low pressure region is created is called wake

The occurrence of separation is not limited to blunt bodies. Complete separation over the entire back surface may also occur on a streamlined body such as airplane wing at sufficiently large angle of attack, which is the angle the incoming fluid stream makes with the chord of the wing. Flow separation on the top surface of a wing reduces lift drastically and may cause the airplane to stall. Stalling has been blamed for many airplane accidents and loss of efficiency in turbo machinery.

1.7 Methods of Reducing Drag Force

Whenever we talk about any automobile the first thing comes into our mind is its fuel efficiency, and next is its speed. As we know that drag influences very much on these two, so to improve these two factors we have to reduce the drag force acting on the vehicles. There are several methods to reduce the drag force some of them are discussed below:

1.7.1 Streamlining

The first thought that comes to mind to reduce drag is to streamline a body in order to reduce flow separation and thus to reduce pressure drag. Even car salespeople are quick to point out the low drag coefficient of their cars, owing to streamlining. But streamlining has opposite effect on pressure and friction drags. It decreases pressure drag by delaying boundary layer separation and thus reducing the pressure difference between the front and back of the body and increases the friction drag by increasing the surface area. The end results depend on which effect dominates. Therefore, any optimization study to reduce the drag of a body must consider both effects and must attempt to minimize the sum of the two.

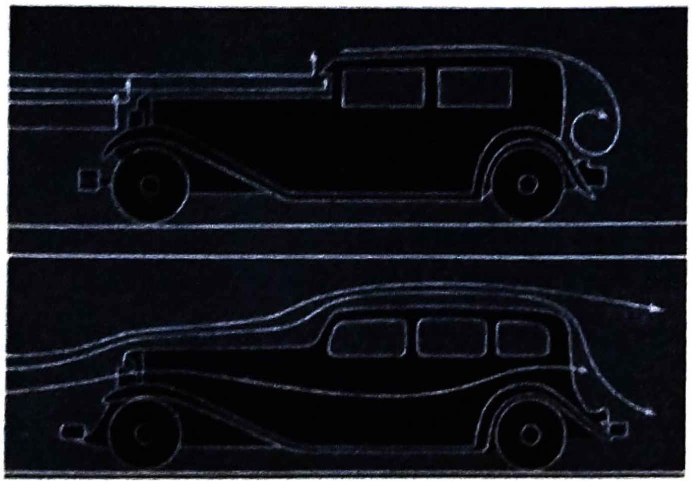


Figure 1.12 streamlined body

1.7.2 Vortex Generators

Vortex generator is an aerodynamic device, consisting of a small vane usually attached to a lifting surface or rotor blade of a wind turbine. Vortex generators may also be attached to some part of aerodynamic vehicles such as an aircraft fuselage or a car. When the airfoil or the body is in motion relative to the air, the vortex generator creates a vortex, which by removing some part of the slow moving boundary layer in contact with the airfoil surface, delays local flow separation and aerodynamic stalling, thereby improving the effectiveness of wings and control surfaces, such as flaps, elevators, ailerons, and rudders.

1.7.3 Spoiler

A spoiler is a simple plate placed somewhere on the car body so that it can interfere or spoil the flow around the vehicles, creating a controlled separation of the flow in a desired place. This is done because fast and smooth airflow leads to positive lift, so by spoiling this flow the lift is either reduced or may be completely cancelled out.

A rear spoiler is a plate in the rear of the vehicles and the plate must be integral with the body. If there is a space between the plate and the body, then it is considered as a wing. The spoiler causes separation in the rear of the car, creating turbulence just before the flow, this turbulence causes the flow to move more slowly, therefore reducing the low pressure in the area just in front of this spoiler.

Therefore, lift on the rear portion of the car is eliminated. Properly designed, a rear spoiler will create down force on the rear part of the car.

Figure 1.13 Spoilers

Figure 1.14spoiler at rear end of car

1.7.4 Diffuser

A **diffuser**, in an automotive context, is a shaped section of the car under body which improves the car's aerodynamic properties by enhancing the transition between the high velocity airflow underneath the car and the much slower free stream airflow of the ambient atmosphere. It works by providing a space for the under body airflow to decelerate and expand (in area, as density is assumed to be constant at the speeds that cars travel). So that it does not cause excessive flow separation and drag, by providing a degree of "wake infill" or more accurately, pressure recovery. The diffuser itself accelerates the flow in front of it, which helps generate down force.

Figure 1.15 Diffuser

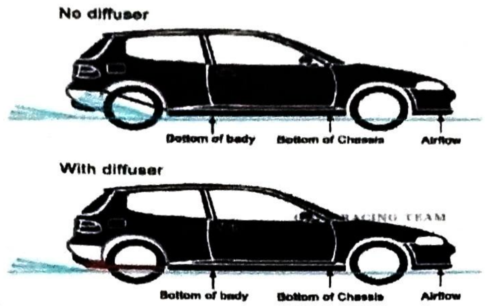


Figure 1.16Car with and without diffuser

1.7.5 Side Duct:

Side ducts make any car look real cool when put it on car. They are found behind front or rear wheel. They were first adapted for racing coupes. The concept is to make an aerodynamic design on the side that allows air to flow out from the vehicle. Aerodynamics of the vehicle requires that the wind should flow smoothly through the vehicle, speeding it up not slowing it down. With the sleek body design of most sports cars, an air duct further reduces the wind ability to affect performance by providing a vent for air to go through

Figure 1.17 side duct

1.7.6 Side skirts

Side skirts are used to reduce the amount of high pressure area on the side of the car by sending air under the car from the sides. If an air splitter is used, air under the car is at a low pressure, which causes the higher-pressure air on the outside and on the sides of the car to come rushing in diminishing ground effect and down force.

The effectiveness of the skirts depends primarily on how close the lower edge can be maintained to ground. That edge should be less than a 2 centimetres from the ground otherwise the skirts' effectiveness diminishes rapidly as the gap increases.

Figure 1.18 side skirts

1.7.7 Splitter and Air Dam

Front splitters are essential aerodynamic components that serve to balance the front vs. rear distribution of down force. Splitter is typically found on the front-end of a race car, appearing as a flat extension to the very bottom of the front bumper. This splitter extends straight out, parallel to the ground, and can be made of carbon fibre or other stiff material. It is attached to the bottom of the front bumper, and may also be supported by two or more support rods at some distance forward of the bumper mounting points. These support rods ensure that the splitter stays parallel to the ground, even when there are outside forces around the splitter. Front end splitters on a race car produce aerodynamic down force by creating difference in the air pressure on upper and lower side of the splitter when the car moves.

Figure 1.19 Splitter and air dams

1.8 Lift force:

Fluid flowing past the surface of a body exerts a force on it. Lift is the component of this force that is perpendicular to the oncoming flow direction. It contracts with the drag force, which is the component of force parallel to the flow direction. Lift conventionally acts in upward direction in order to counter the force of gravity, but it can act in any direction at right angles to the flow. If the surrounding fluid is air, the force is called an aerodynamic force. In water or any other liquid, it called a hydrodynamic force

Lift occurs when a moving fluid of gas is turned by a solid object. The flow is turned in one direction, and the lift is generated in the opposite direction, according to Newton's third law of action and reaction. Because air is a gas when the molecules are free to move about, any solid can deflect the flow.

Lift is a mechanical force. It is generated by the interaction and contact of a solid body with a fluid (liquid or gas). It is not generated by a force field, in the sense of a gravitational field or an electromagnetic field, where one object can affect another object without being in physical contact. For lift to be generated, the solid body must be in contact with the fluid. The Space Shuttle does not stay in space because of lift from its wings but because of orbital mechanics related to its speed. Space is nearly a vacuum. Without air, there is no lift generated by the wings.

Lift is generated by the difference in velocity between the solid object and the fluid. There must be motion between the object and the fluid. It makes no difference whether the object moves through a static fluid, or the fluid moves past a static solid object. Lift acts perpendicular to the motion. Drag acts in the direction opposed to the motion.

From the figure 1.11,

The lift force on elemental area = force due to pressure in the direction perpendicular to the direction of motion + force due to shear stress in the direction perpendicular to the direction of motion

$$= -pdA \sin\theta + \tau_0 dA \cos\theta \text{.........(4)}$$

Negative sign is taken with pressure force as it is acting in the downward direction while shear force is acting vertically up

$$\text{Total lift} = \int \tau_0 dA \cos\theta - \int pdA \sin\theta \text{..........(5)}$$

Where, p = pressure

dA= elemental area

θ = Angle made by pressure force with horizontal direction

Mathematically lift force can be calculated by using, $F_l = C_l * 0.5 * \rho * A * v^2$...........(6)

Where, F_l = lift force

C_l = co efficient of lift

ρ = density of fluid,

v = velocity of fluid over body and

A = largest projected area of the immersed body.

The lift force is a favourable condition in case of aeroplanes as they have to fly in air. But, in the case of automobiles, lift force should not be there as the automobile has to in touch with the road always. If lift is there in case of automobiles, the vehicles may tilt and cause accidents. Therefore, lift has to be eliminated completely in case of automobiles. We cannot make the lift force completely to zero but can be reduced as low as possible. One of the method is to reduce the lift is to attach spoiler at the rear end.

1.9 Spoiler:

A spoiler is an automotive aerodynamic device whose intended design function is to 'spoil' unfavourable air movement across a body of a vehicle in motion, usually described as turbulence or drag.

One main factor in car design is its drag coefficient or simply the drag force it experiences at various speeds. A designer should ensure the car sees least possible drag at all times. This means the flow should be smooth over the car and there should not be any separation.

One of the most conventional methods to avoid or delay separation is creating disturbances in flow in the laminar zone, thus making it turbulent. Turbulent flow by its nature has more internal energy to remain attached to the body.

The spoilers are basically located in front and rear of the vehicle. Each has a different purpose. Front spoilers work on the above principle. They convert the flow over the hood to turbulent and it flows over the rest of the car body smoothly. Rear Spoilers are slightly different in operation. In simple terms you can think of them as inverted wing (wing that generates force/lift in downward direction). At high speeds the car body generates significant amount of lift which reduces its traction and thus its manoeuvrability and stability. This could be only compensated by increasing the weight of the car and thus taking a penalty of performance. This is where the rear spoilers help in design. They provide the extra downward force to compensate for the lift generated by the car. The below points helps you understand more on spoiler design.

1.9.1 Benefits of a Car Spoiler

Installing a spoiler on a car provides a variety of benefits for owners. The main benefits, perhaps, are for better traction and to add a sporty look, but also include other advantages, such as increased fuel efficiency, added visibility, reduced car weight, and braking stability.

Benefit 1: Maintain Traction

The main benefit of installing a spoiler on a car is to help it maintain traction at very high speeds. Generally, when a car goes very fast (over 70 miles per hour), the air pressure can lift the car, which makes it difficult to manoeuvre the car without the danger of having it spin out of control. Rear spoiler, in particular, push the back of the car down so the tires can grip the road better and increase stability.

Benefit 2: Increase Fuel Efficiency

Front car spoilers or air dams can actually increase mileage in some cars. Since these types of spoilers reduce the drag (instead of increasing it) by pushing the air around the car, it does lower the amount of energy (fuel) the car needs to burn to propel itself forward.

Benefit 3: Added Visibility

Another advantage of installing a rear spoiler on a car is the added visibility. This means other drivers on the road can easily see the car and prevent rear-end collisions and other types of accidents. Certain spoilers, such as trunk cap spoiler, even have brake lights at eye-level so the driver behind can easily be alerted when the car is slowing down or braking.

Benefit 4: Reduce Weight

A spoiler can reduce the weight of a vehicle. While this may seem counterintuitive, it makes sense in a way. The only thing keeping a car stable on the road is its weight. Perhaps that's why many people have this perception that SUVs are much safer, because of their heavier mass keeps them steady. However, having a spoiler means that the car manufacturer can reduce the weight of the car by using lighter materials or doing away with unnecessary weight, without worry that driving at high speeds will cause the car to become unsteady and fly off the highway.

Benefit 5: Create a Stylish Look

Most car owners install spoilers as a fashion accessory and spoilers do a pretty good job of making a car look cool. This idea first became popular in the 1970s, when Porsche introduced the 911 Turbo, which featured whale tail spoilers on the back. Today, many cars come with built-in spoilers to evoke that "sporty" look, though many aftermarket spoilers are available for a wide variety of car makes and models.

Benefit 6: Increase Braking Stability

Adding spoilers that raise the downward force on the back of the car not only increases traction, but the braking ability as well. Drivers will have an easier time braking, even at high speeds, making driving even safer.

CHAPTER 2

2. LITERATURE REVIEW

Before going to start our project, a brief study on papers related to analysis on aerodynamics of vehicles using ANSYS software is done. Many authors gave different ideas to their related works on thestudy the effect of lift and drag on vehicles. Some are listed below.

S.M. Rakibul Hassan et.al.[1] had concentrated on different aspects analysis of aerodynamic drag of racing cars and different drag reduction techniques such as rear under body modification and exhaust gas redirection towards the rear separation zones. Through a numerical process (Finite Volume Method) of solving the Favre-averaged Navier-Stokes equations backed by k–epsilon turbulence model, the drag coefficient of the car under analysis was found to be 0.3233 and it was evident that the drag can be reduced up to 22.13% by different rear under-body modifications and up to 9.5% by exhaust gas redirection towards the separated region at the rear of the car. It was concluded that if somehow the negative pressure area and its intensity at the rear of the car can be minimized, the separation pressure drag is subsequently reduced. The aerodynamic drag coefficient of car model used was 0.3233.The main design consideration to reduce the drag of any bluff should be- keep the flow attached to the body as much as possible.

Wenhu Wang and Peiqing Liu[2] carried out the studies numerically based on the influence of the downward spoiler deflection on the boundary layer flow of a high-lift two-element airfoil consisting of a droop nose, a main wing, a downward deflecting spoiler and a single slotted flap. Both of the boundary layer of the upper surface of the spoiler and the confluent boundary layer of the upper surface of the flap become thicker, as the downward spoiler deflection increases. Compared to the attached flow at the angle of attack of 10°, the flow of the upper surface of the spoiler becomes separated at the angle of attack of 16° when the spoiler deflection is large enough, which corresponds to the boundary layer flow reversal in velocity profiles.

R. B. Sharma and Ram Bansal[3] carried out the experimental work of the test vehicle and grid system was constructed byANSYS-14.0. FLUENT Solver was employed in their work. In this study, numerical iterations were completed, then after aerodynamic data and detailed complicated flow structure were visualized. The aerodynamics of the most suitable design of tail plate was introduced and analyzed for the evaluation of drag coefficient for passenger car. The addition of tail plates results in a reduction of the drag-coefficient 3.87% and lift coefficient 16.62% in head-on wind. Rounding the edges partially reduces drag in head-on wind but does not bring about the significant improvements in the aerodynamic efficiency of the passenger car with tail plates. The effects of different aerodynamic add-on devices on flow and its structure over a generic passenger car may be analyzed using CFD approach.

C.N. Patil1et.al.[4] carried out the Aerodynamic flow simulation on one of conventional BUS was performed to demonstrate the possibility of improving the performance with benefits of Aerodynamic features around the BUS by reducing C_d which in turn decreases the fuel consumption. Analysis was carried out by adding spoilers and panels at rear portion. This assessment has shown that drag can be decreased without altering the internal passenger space and by least investment. Simple features installed and modifications performed at the rear end of the vehicle have shown capable of reattaching the flow along the BUS surface and reducing aerodynamic drag. This study demonstrates the possibility of improving the aerodynamic performance by different geometrical features around a passenger BUS. These features contributed towards reduction of C_d which impacts fuel consumption.

Bhavini Bijlani et.al.[5] carried out the work mainly focussed on investigation of aerodynamic of sedan and square-back car, air flow around the car body and measuring of drag coefficient. The investigation was carried out both experimentally and by using software.1:20 aluminium scale model of popular sedan and square-back car was used for experiment. Experiment was done on subsonic wind tunnel which test section (30cm x 30cm x 100cm). Computational analysis was carried out in ANSYS CFX-13. The drag co efficient (C_d) evaluated for exterior profile of Sedan and Square-back, to be of the order of 0.38 and0.66, which are acceptable. Sedan is more aerodynamic than square-back. However, further optimization of this wind tunnel estimation is strongly recommended. Experimental investigations were further validated computationally. Combining wind tunnel experiments and CFD computation, integration on both methods can be lead to a better aerodynamic design.

Mustafa Cakir[6] carried out the numerical simulation of flow around racing car with spoiler positioned at the rear end using commercial fluid dynamic software ANSYS FLUENT. The work focused on CFD-based lift and drag prediction on the car body after the spoiler is mounted at the rear edge of the vehicle. A3D computer model of 4-door sedan car (which was designed with commercial software Solid Works) was used as the base model. Different spoilers, in different locations were positioned at the rear end of vehicle and the simulations were run in order to determine the aerodynamic effects of spoiler. The aerodynamic lift, drag and flow characteristics of a high-speed (~65 mph) generic sedan passenger vehicle with a spoiler and without a spoiler situations were numerically investigated.

Rubel Chandra Das and Mahmud Riyad[7] performed the analysis based on the design, developments and numeral calculation of the effects of external device, which will be spoiler that mounted at the rear side of the vehicle to make the present vehicles more aerodynamically attractive. The influence of rear spoiler on the generated lift, drag, and pressure distributions were investigated and reported using commercially available Autodesk Simulation CFD software tool. It was concluded that at a particular spoiler height the spoiler that possess smaller angle of wind collision gives higher drag force. This is due to the fact that with smaller angle of wind collision, the spoiler would create smaller recirculation zone behind the rear end of the running vehicle. This implies to higher pressure behind spoiler but lower pressure behind the rear end of the vehicle. Rear spoilers redirect the airflow behind the vehicle & increase the negative lift of the vehicle. In the investigation, six modifications were simulated & 12 degree spoiler inclination angle model was the most optimum though it created 1.56% extra C_d than 4 degree inclination angle. Minimum C_L was maintained in the model which is basic concern for better stability of high speedy vehicle.

Sneh Hetawal et.al.[8]carried out the information about the design and CFD analysis of a Formula SAE car. A numerical study of a rear engine SAE race car was presented. The focus of the study was to investigate the aerodynamics characteristics of a SAE race car with front spoiler, without front spoiler and with firewall vents. The aerodynamics study of the SAE car was made to reduce the drag force. The study was performed using the CFD package. The main goal of the study was to enhance the stability of the vehicle and reduce the drag. With this the track performance will be increased also the resistance of air to the vehicle gets reduced. The CFD analysis was done on full scale model. The aerodynamic study was conducted in the ANSYS Fluent software to perform a turbulent stimulation (using k -epsilon model) of the air flow on the SAE car. To increase the aerodynamic performance of race car, an attempt is made to modify the design of a Formula SAE car. Comparative study was done on three car models by carrying out CFD simulations. Cutting out the section of firewall and providing wing at front end. Drag co-efficient was found to get reduced from 0.85 for the standard race car to 0.70 for the modified car with front wing, whereas negative lift was increased from 0.2 for standard race car to 0.25 for the model 3. Model 3 having wing at the front end and having cut section at firewall shows less drag and lift, shows better aerodynamics characteristics than other two models.

Rizal E. M. Nasir et.al.[9] performed the analysis based on study the effects of fluid flow and the effective drag of the vehicle over a 3D standard car (BOLERO) with attached Rear Spoiler by using Computational Fluid Dynamics (CFD) simulation. A 1:1 scale model of the actual vehicle was designed in CAD package SOLIDWORKS and CATIA V5 R20. CFD analysis was done over the scaled model keeping conditions as close as possible to the actual road conditions. For evaluation, optimization, the Reynolds-Averaged Navier-Stokes (RANS) equations with Reliable k-ε turbulence model was used over commercial package ANSYS 14, FLUENT CFD Solver. The simulation was done for vehicle speed at 80kmph and the results were compared with scaled base vehicle. Various velocity, pressure, streamline contours and velocity plots were examined and analyzed at rear part of the vehicle. It was concluded that, the Co-efficient of drag (C_d) of the vehicle with attached Rear Spoiler went down by 4.8% with 3^0 spoiler.

Based on the literature review, it is understood that the drag force causes increase in fuel consumption and is required to be reduced. Drag force is reduced by proper stream lining of car body or by attaching a spoiler. By attaching a spoiler, lift force can be reduced at high speeds along with drag force. Therefore, effect of spoiler at different angles on lift and drag forces can be studied and can suggest a suitable angle of spoiler which will give least drag and lift forces.

CHAPTER 3

3. SIMULATION PROCEDURE

A sedan type (Honda city) car is selected for the study purpose. The car geometry is drawn in solid works and fluid flow analysis is carried out using fluent software in ANSYS work bench. The analysis procedure is discussed in detail in this chapter. The steps involved in the analysis are

- Geometry
- Mesh
- Setup
- Solution
- Result

3.1 Geometry-

- IGES file of car geometry is imported to ANSYS fluent design modeller.
- Geometry is generated by right clicking on import icon.
- Virtual wind tunnel is created using enclosure option fluent design modeller.
- Dimensions of wind tunnel enclosure is set by keeping two times of car body length in the front side, four times of car body length in the rear side and 0.5 m in the side and upward of car body.
- A symmetrical plane is taken and car body is cut into two equal halves by using this.
- Another symmetrical plane is taken to touch the bottom of tyres.
- Boolean operation is performed and car body is subtracted from the enclosure.
- Boolean operation is completed by clicking generate icon.

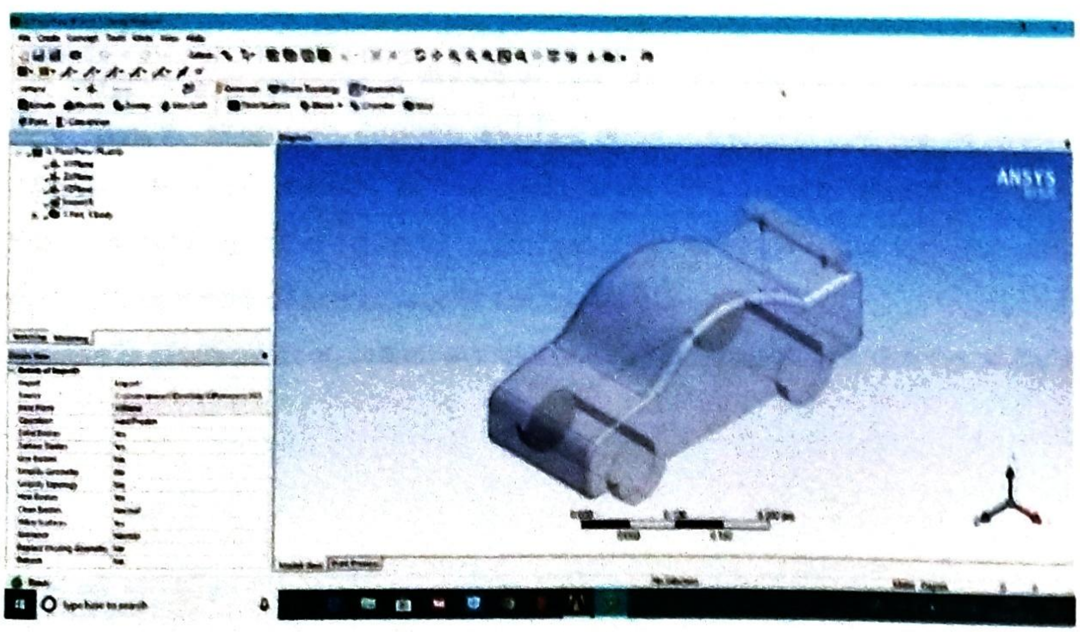


Figure 3.1 Geometry of the car

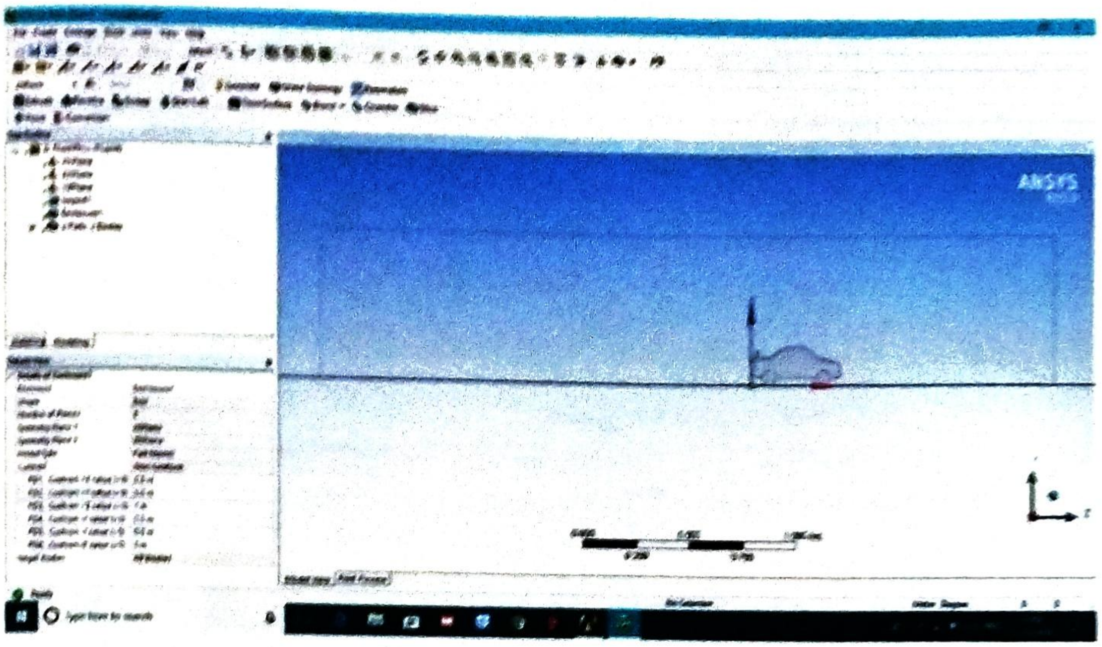


Figure 3.2 Car inside virtual wind tunnel

3.2 Meshing-

- Fine mesh is created by using the following options
  - smoothness to high,
  - Transition to slow and
  - Advanced size function as on proximity and curvature
- Maximum face size and maximum size is taken as 0.250m and number of cells across the gap is 15.
- Mesh is generated by right clicking generate option.
- The velocity and pressure distribution near the car body are of more interest than away from it. Therefore, uniform and fine mesh is required near the car body. Apart from this, the mesh can be coarser to reduce the computational time.
- Sizing of mesh near the car body is given as 0.001 m and inflation option is created by giving 20 numbers of layers around the car body.
- Naming is given as inlet, outlet, symmetry, road to the enclosure and car as car body.

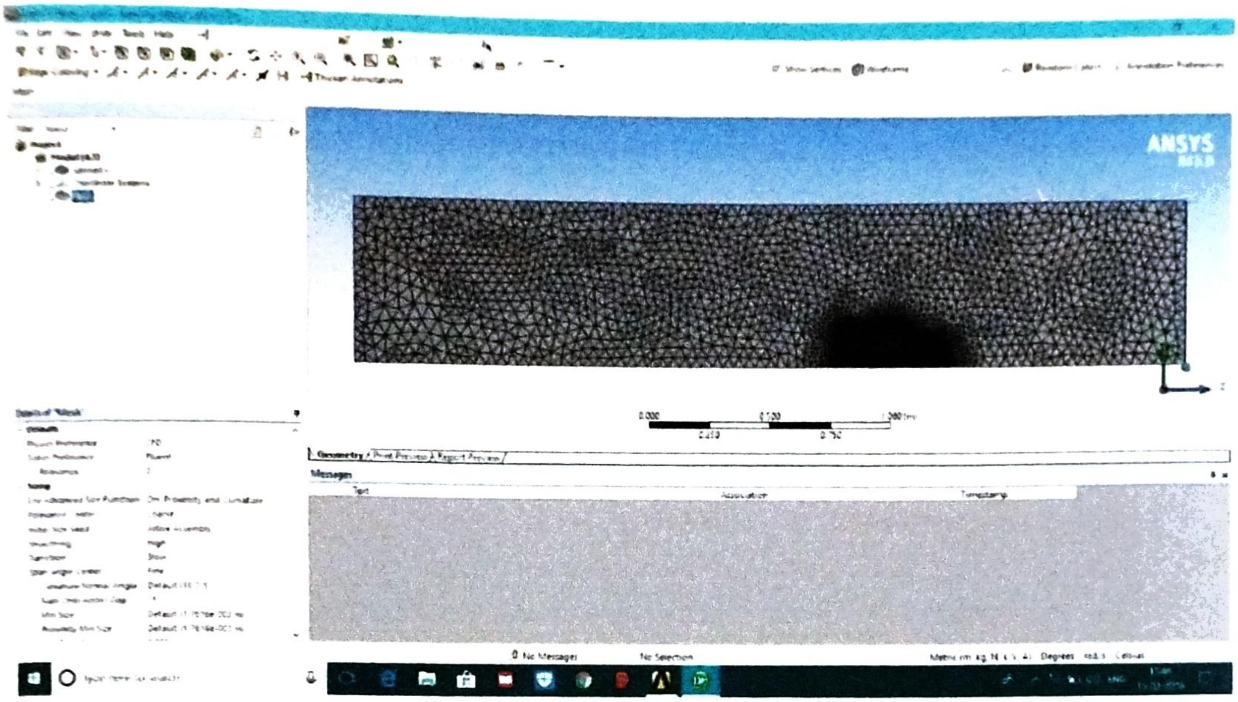


Figure 3.3 Meshing of the wind tunnel and car body

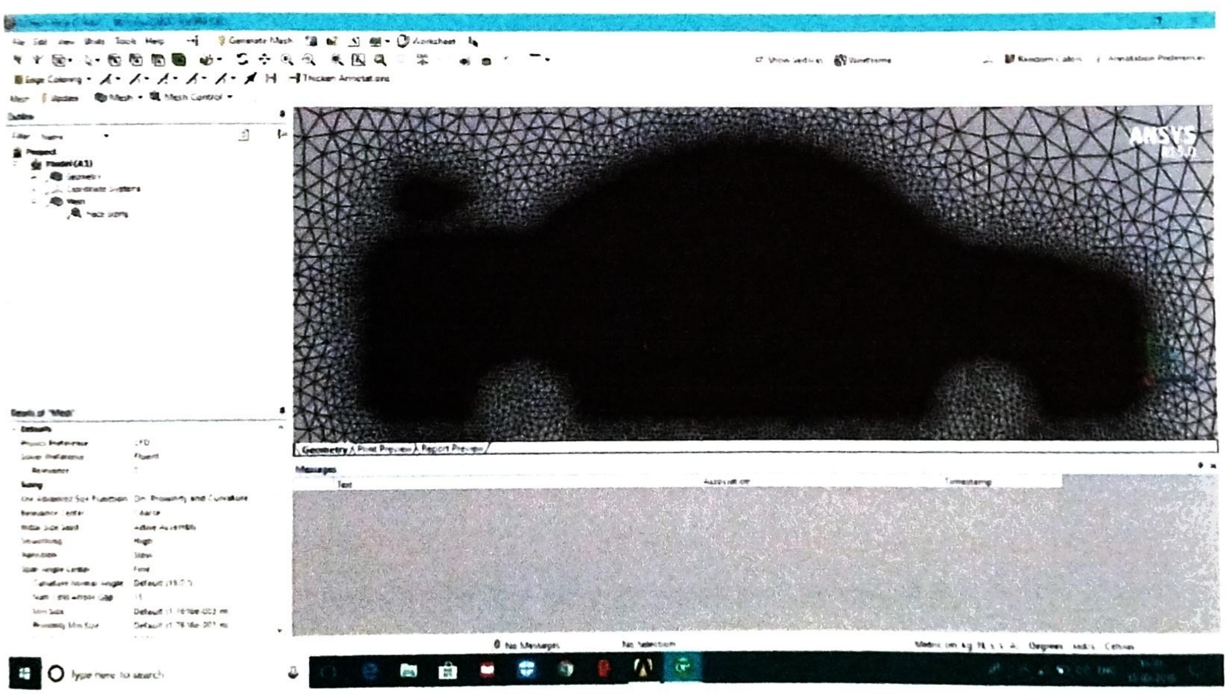


Figure 3.4 Face sizing applied to car body

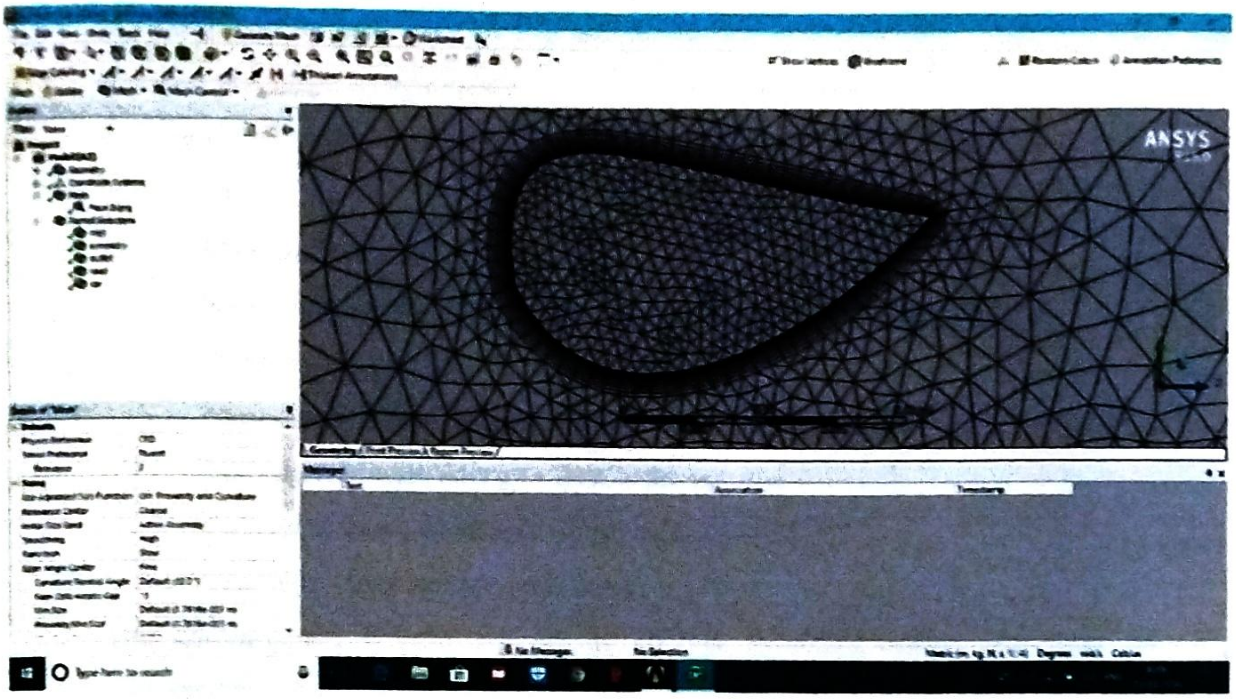


Figure 3.5 Mesh showing inflation layer around the spoiler

3.3 SETUP:

- Single precision and serial processer is selected for the setup.
- Quality of mesh is checked and report is generated.
- Pressure based Navier-Stokes solver for steady type flow with velocity formulation as absolute.
- Turbulent model is taken as K-epsilon turbulence model with 2 equations. In this, k-epsilon model is taken as realizable and near wall treatment as non-equilibrium wall function.
- The fluid inside the enclosure is selected as air and solid (car) as aluminium in the material option.
- The boundary condition for the inlet is given as velocity inlet, outlet as pressure outlet, enclosure as symmetry and road & car as wall.
- Required velocity along with 1% turbulence intensity and turbulence ratio as 10 is selected.
- The boundary condition at outlet is taken as zero gauge pressure and turbulent intensity as 1%.
- Wall boundary condition for road and car is selected as no slip and stationary wall.
- Symmetry condition is given for enclosure side and top wall.

- We take road as wall with no slip and stationary wall condition. Side two walls and top wall is a taken as symmetry.
- Reference values are taken from inlet and the default area is changed as per frontal area of car.
- The solution methods are selected as,

  Pressure velocity coupling scheme is simple spatial discretisation

  Pressure - standard

  Momentum - first order upwind

  Turbulent kinetic energy - first order upwind

  Turbulent dissipation rate - first order upwind

  Gradient - least square cell based
- The convergence criteria are set at 10^{-3} in monitors option and drag and lift convergence plot is selected.
- Solution initialization is set as standard initialization.
- Number of iterations is set as 1000 and calculation is started.

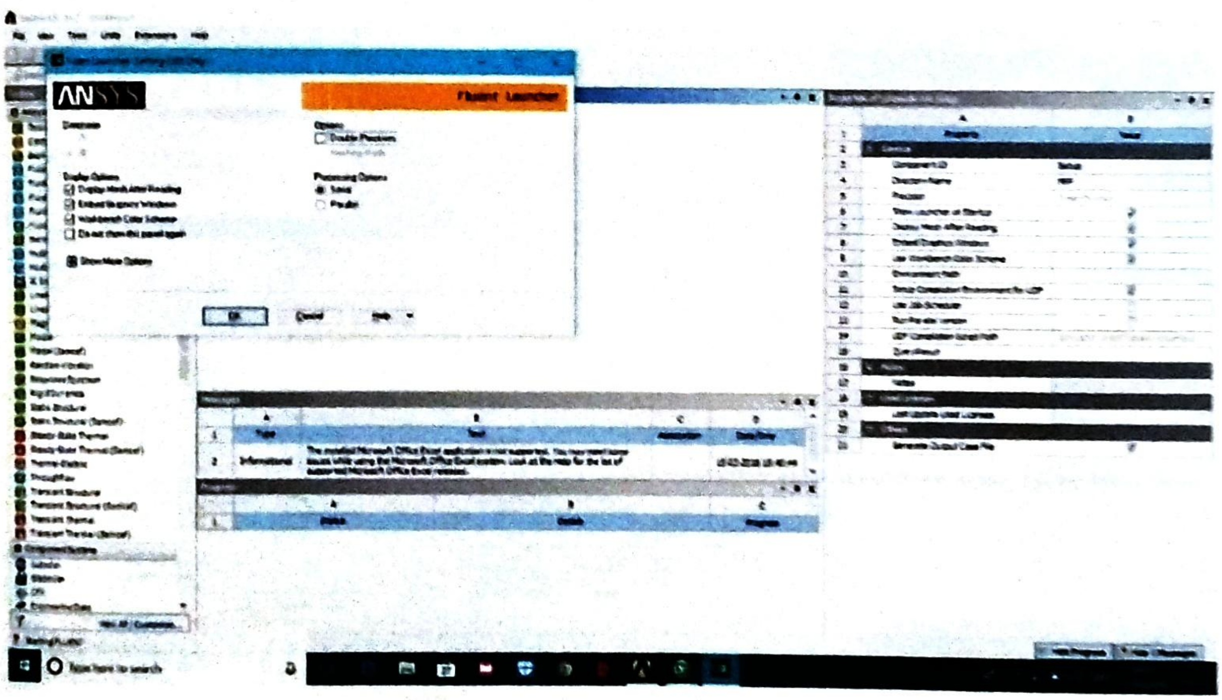


Figure 3.6 Starting of ANSYS fluent setup

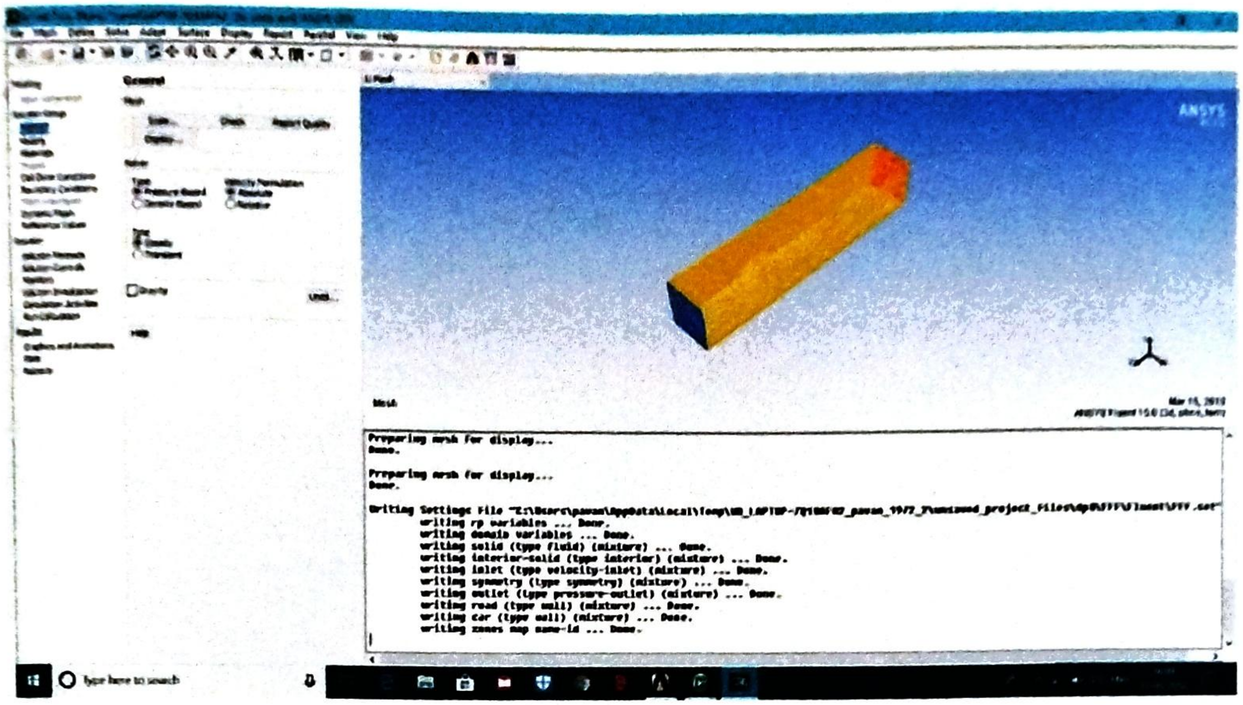


Figure 3.7 Work space of ANSYS fluent

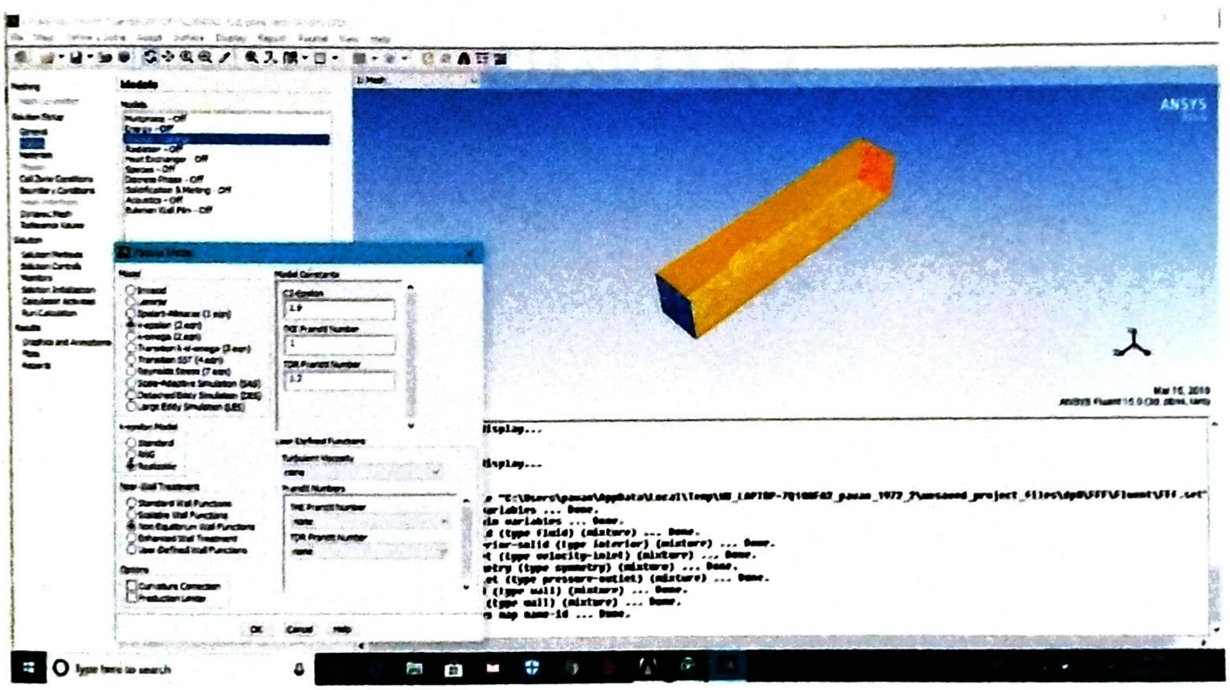


Figure 3.8 Selection of viscous model

CHAPTER 4

4. RESULTS AND DISCUSSIONS

Analysis is carried out on sedan type (Honda City) car without rear end spoiler and with rear end spoiler at different angles at different car velocities. The car velocities one at normal road speed of 60 kmph and the other at high speed of 160 kmph are selected for the study purpose. The pressure, velocity distributions on the car body along with lift and drag coefficients are found by the simulation procedure as explained in chapter 3. The obtained results are discussed in this chapter.

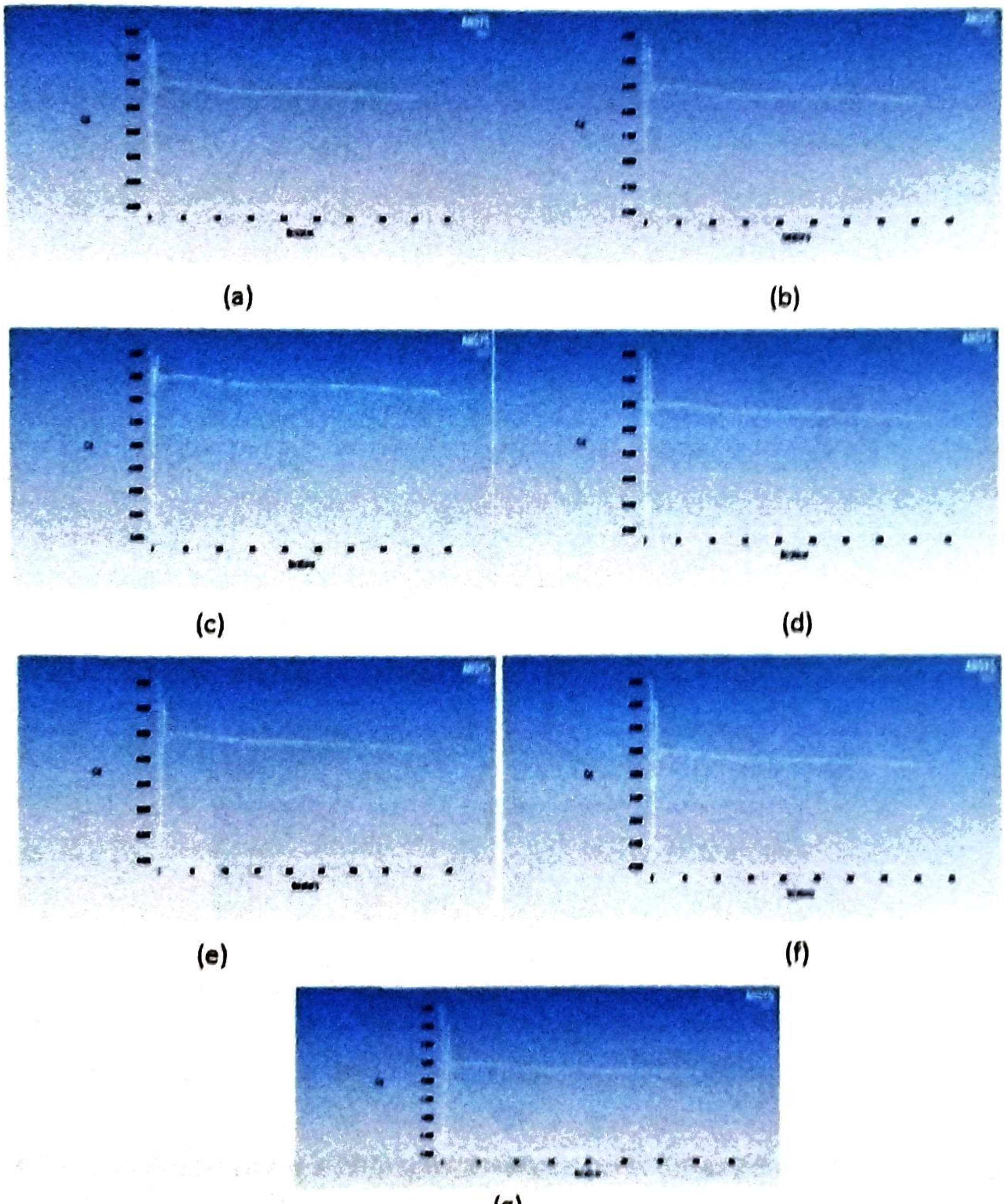
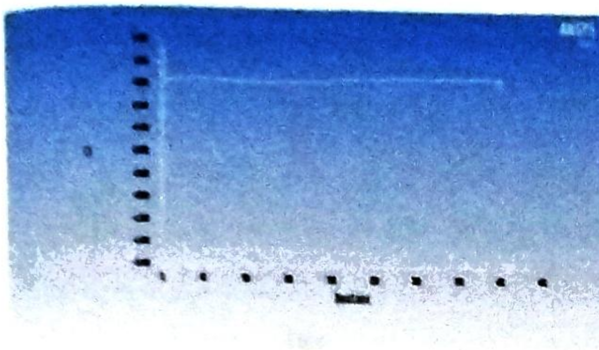
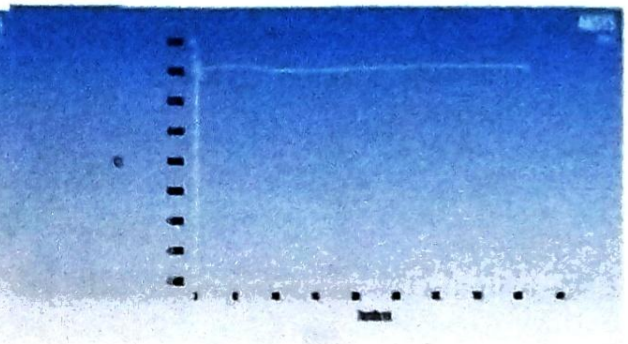


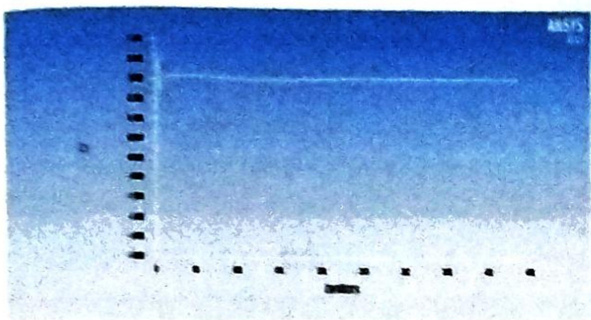
Figure 4.1. Drag coefficient on car at 60 kmph (a) without spoiler (b) spoiler at 0 degrees (c) spoiler at 10 degrees downwards (d) spoiler with 20 degrees downwards (e) spoiler with 30 degrees downwards (f) spoiler with 10 degrees upwards (g) spoiler with 20 degrees upwards.



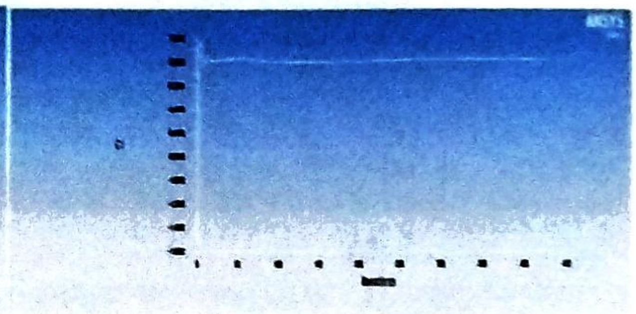
(a)



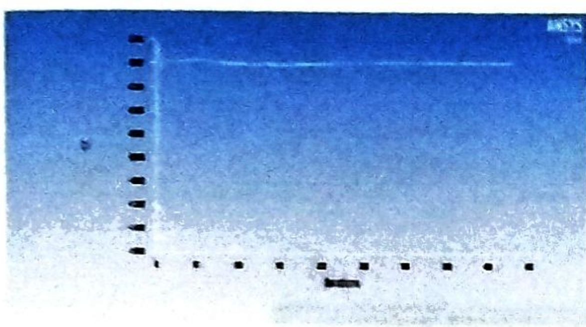
(b)



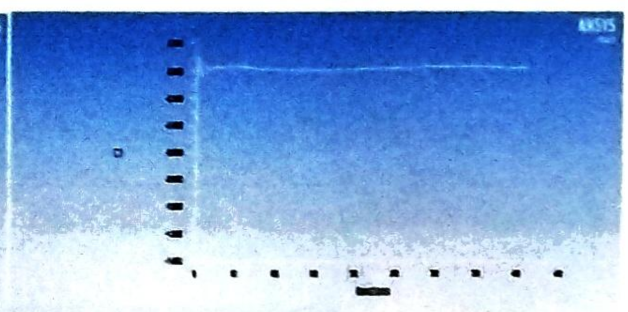
(C)



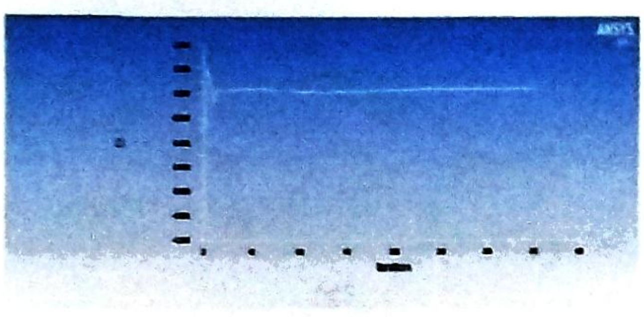
(d)



(e)



(f)



(g)

Figure 4.2. Lift coefficient on car at 60 kmph (a) without spoiler (b) spoiler at 0 degrees (c) spoiler at 10 degrees downwards (d) spoiler with 20 degrees downwards (e) spoiler with 30 degrees downwards (f) spoiler with 10 degrees upwards (g) spoiler with 20 degrees upwards.

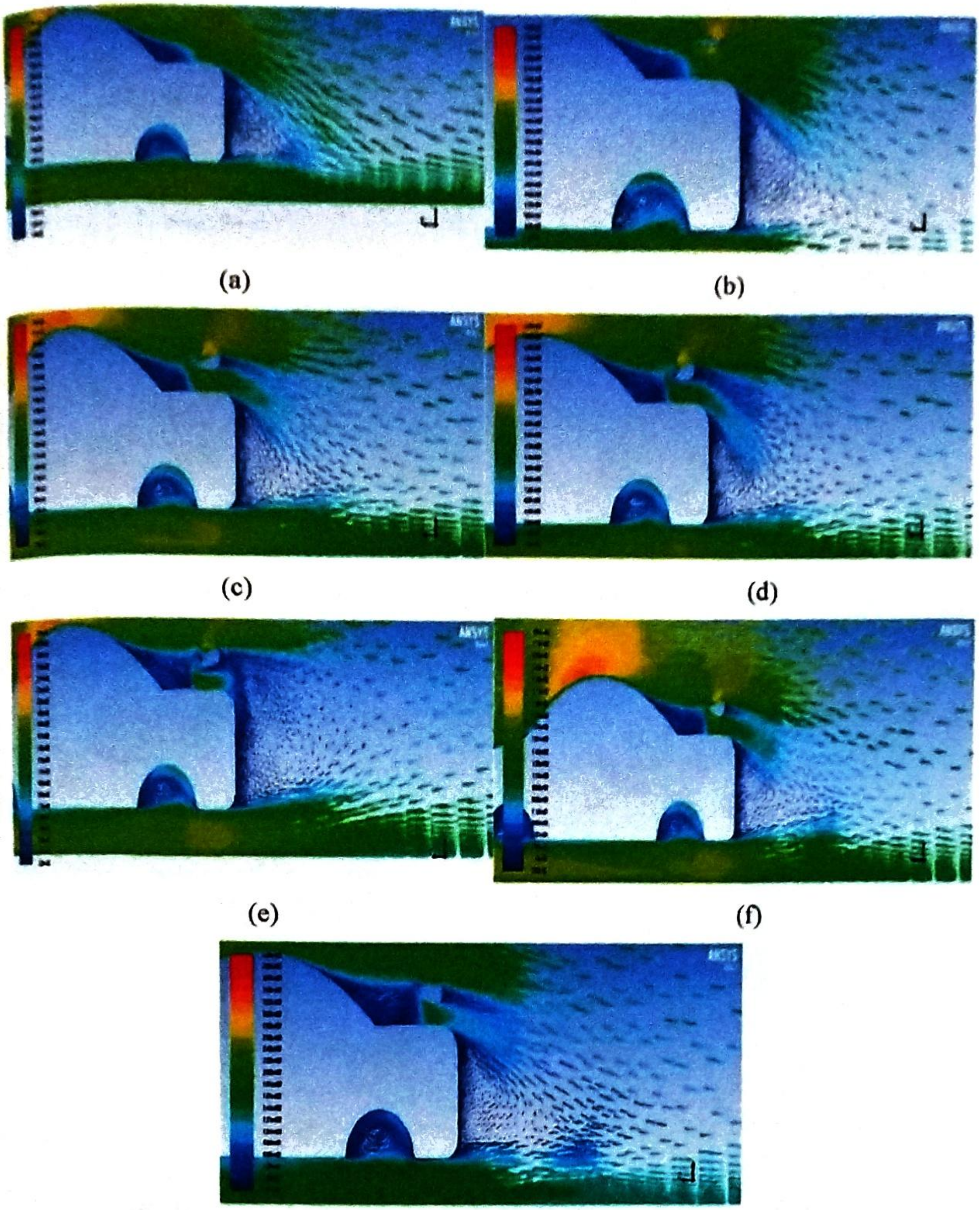


Figure 4.3. Recirculation zones at the rear end of car at 60 kmph (a) without spoiler (b) spoiler at 0 degrees (c) spoiler at 10 degrees downwards (d) spoiler with 20 degrees downwards (e) spoiler with 30 degrees downwards (f) spoiler with 10 degrees upwards (g) spoiler with 20 degrees upwards

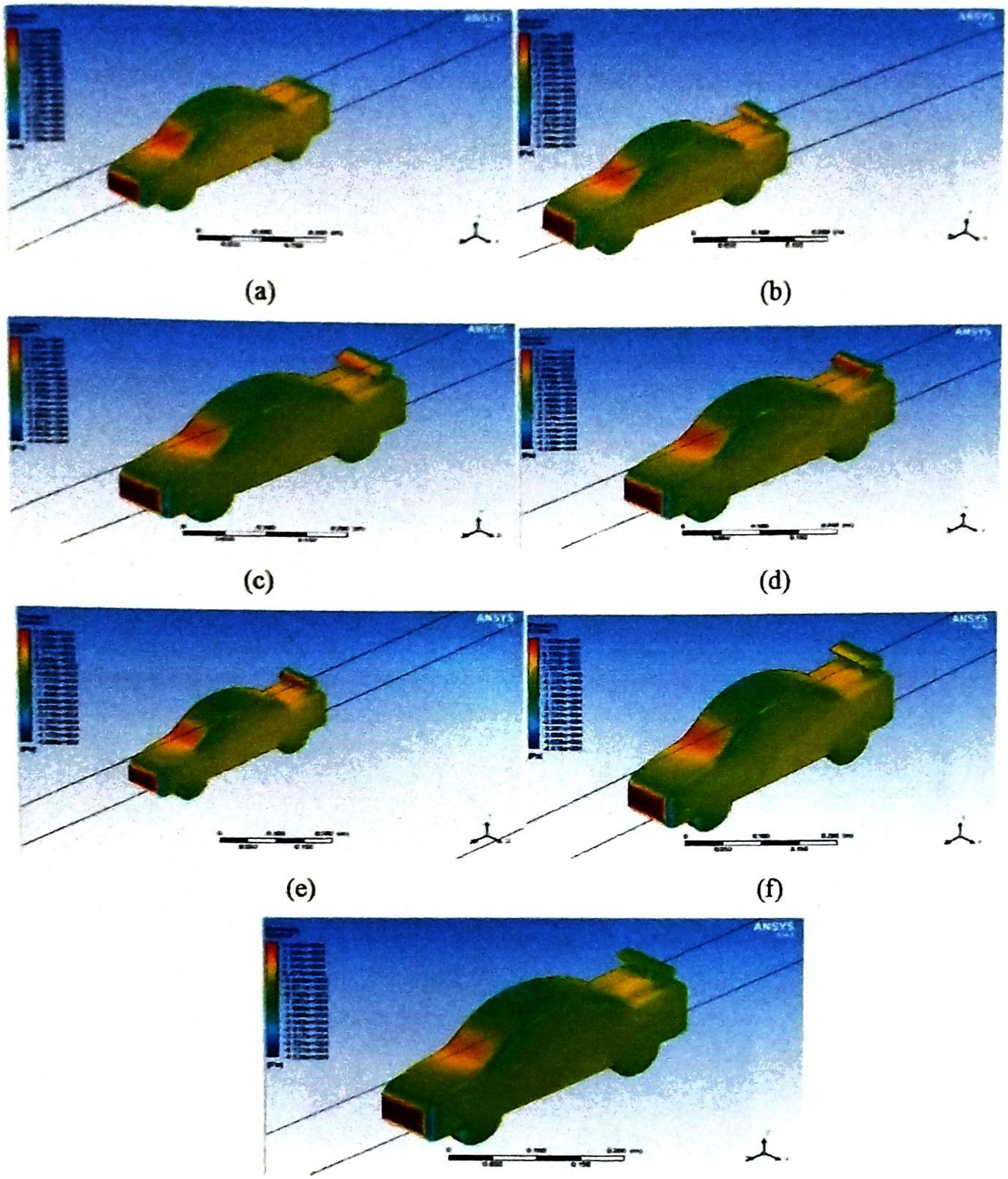
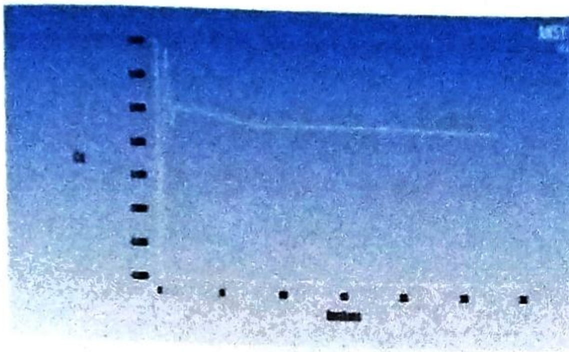
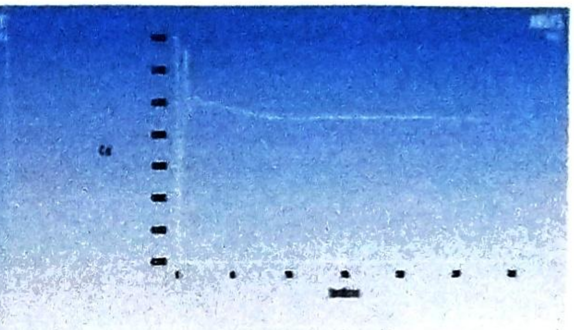
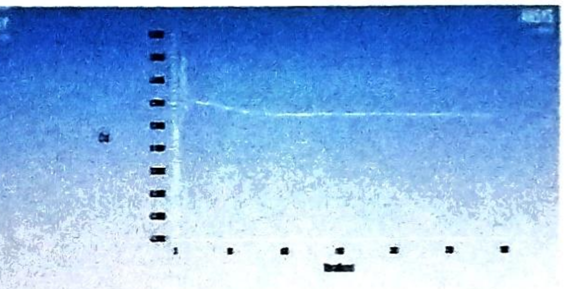


Figure 4.4. pressure contours over the car at 60 kmph with (a) withouit spoiler (b) zero degree spoiler (c) ten degree inclination spoiler downwards (d) twenty degree inclination spoiler downwards (e) thirty degree inclination spoiler downwards (f) ten degree inclination spoiler upwards (g) twenty degree inclination spoiler upwards.

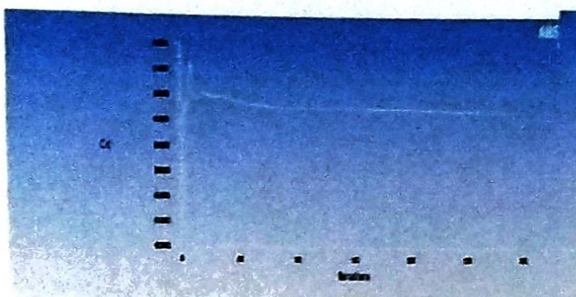


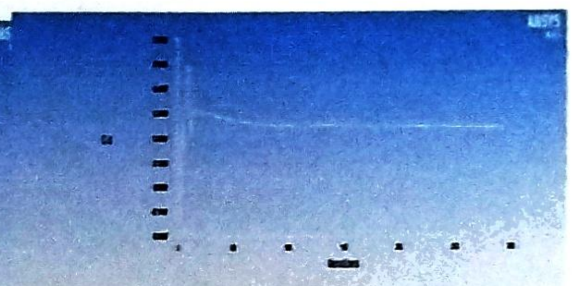


(a) (b)

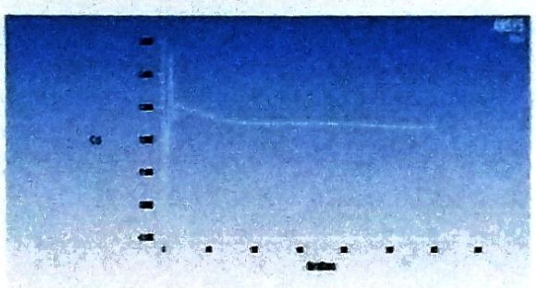


(c) (d)



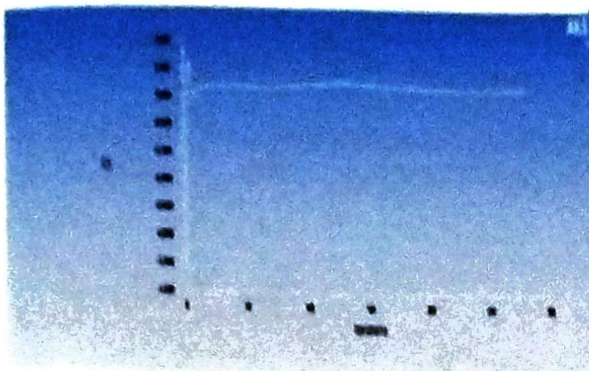


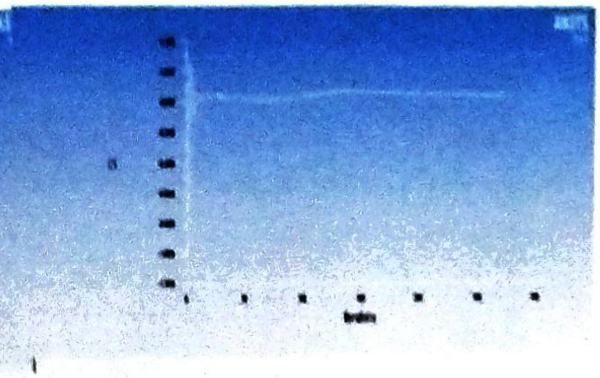
(e) (f)



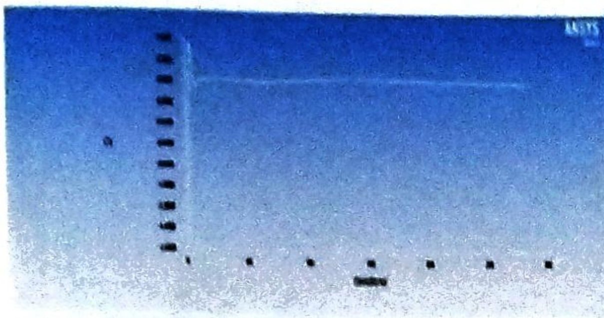
(g)

Figure 4.5. Drag coefficient on car at 160 kmph (a) without spoiler (b) spoiler at 0 degrees (c) spoiler at 10 degrees downwards (d) spoiler with 20 degrees downwards (e) spoiler with 30 degrees downwards (f) spoiler with 10 degrees upwards (g) spoiler with 20 degrees upwards

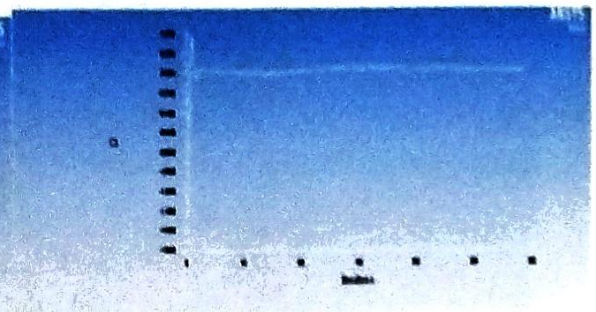




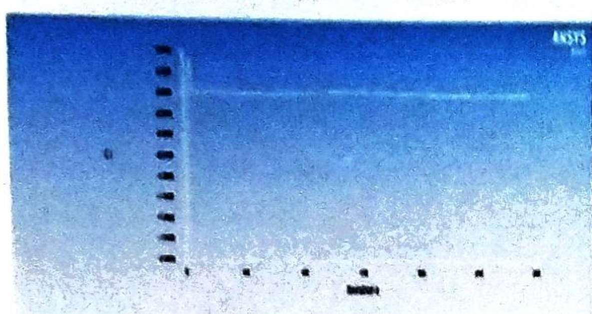
(a) (b)



(c)

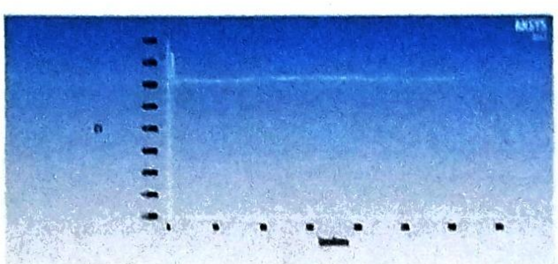


(d)



(e)

(f)



(g)

Figure 4.6. Lift coefficient on car at 160 kmph (a) without spoiler (b) spoiler at 0 degrees (c) spoiler at 10 degrees downwards (d) spoiler with 20 degrees downwards (e) spoiler with 30 degrees downwards (f) spoiler with 10 degrees upwards (g) spoiler with 20 degrees upwards

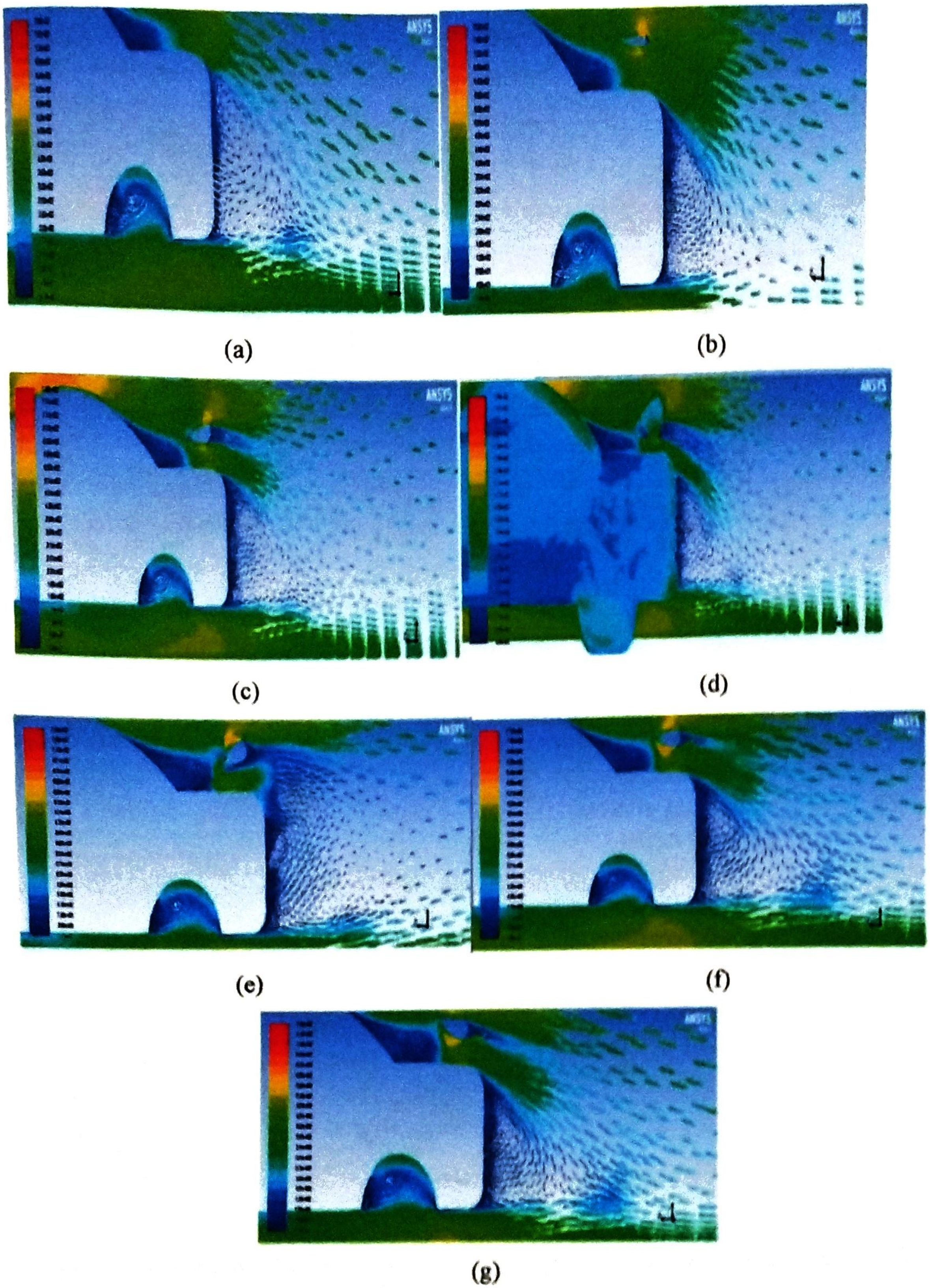


Figure 4.7. Recirculation zones at the rear end of car at 160 kmph (a) without spoiler (b) spoiler at 0 degrees (c) spoiler at 10 degrees downwards (d) spoiler with 20 degrees downwards (e) spoiler with 30 degrees downwards (f) spoiler with 10 degrees upwards (g) spoiler with 20 degrees upwards

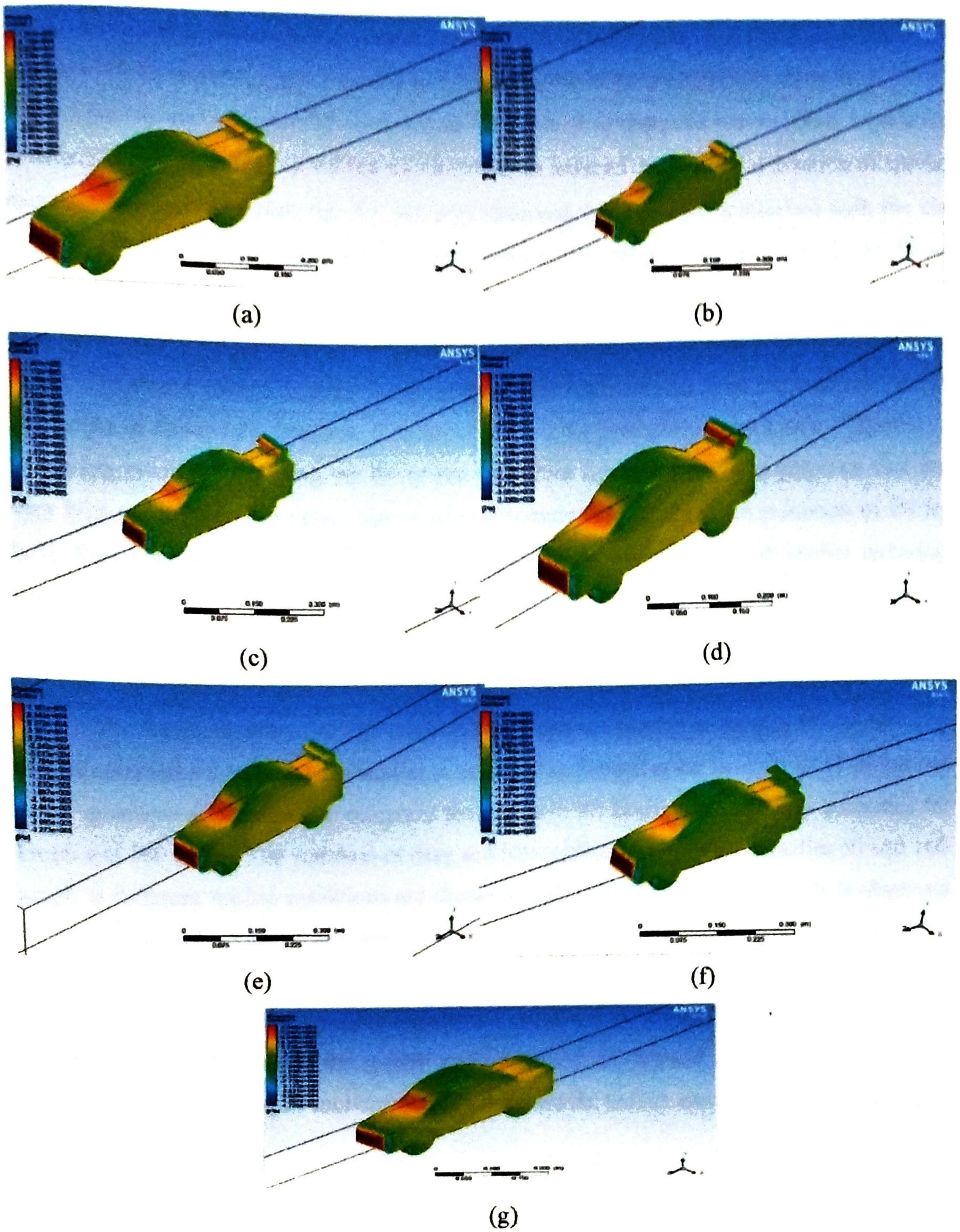


Figure 4.8 pressure contours over the car at 160 kmph with (a) zero degree spoiler (b) ten degree inclination spoiler downwards (c) twenty degree inclination spoiler downwards (d) thirty degree inclination spoiler downwards (e) ten degree inclination spoiler upwards (f) twenty degree inclination spoiler upwards (g) without spoiler

The results obtained at car velocity 60 kmph are shown in figures 4.3 to 4.4 and are discussed in detail here. Figure 4.3 (a) to 4.3 (e) shows the recirculation of air at the rear end without spoiler and with spoiler inclining downwards. it is observed that the flow separation is reduced from fig 4.3 (a) to 4.3 (e) which results in reduced drag as the inclination of spoiler increases downwards. From fig. 4.3 (e), it is observed that the flow is attached with the car body completely which results in lower drag compared to others. Figure 4.3 (f) & (g) shows the recirculation of air with spoiler inclining upwards. It is observed that, as the inclination of spoiler increases upwards flow separation is increased which results in increased drag.

Figure 4.4 (a) to (e) show the pressure distribution over car body with spoiler angles increasing in downward direction. It is observed that the pressure on upper side of spoiler is increasing results in reducing the lift of the car. From figure 4.4 (e), it is observed that the very high pressure on the upper side of spoiler compared to others gives reduction of lift by 96%. Figure 4.4 (f) & (g) represents the pressure distribution over car with spoiler inclining upwards at 10 and 20 degrees respectively. It is observed that the pressure on upper side of spoiler is lower and is decreased as inclination increases which results in increased lift of the car.

The results obtained at car velocity 160 kmph are shown in figures 4.5 to 4.8. The results obtained are qualitatively similar to the results obtained at car velocity of 60 kmph but differ quantitatively. Table 4.1 compares the drag and lift coefficients of car at velocities 60 kmph and 160 kmph. The variation of drag and lift coefficients of car at velocities 60 and 160 kmph at different spoiler conditions are shown in fig. 4.9 & 4.10 respectively. It is observed that, the values of lift and drag coefficients are less at lower velocities as expected. Fig. 4.9 & 4.10 shows the reduction of lift and drag coefficients of car with spoiler. Therefore, it is clear that the spoiler addition is always suggestible irrespective of velocity of car. Further it is observed that, by increasing the spoiler angle upwards, the lift and drag coefficients shows increasing trend whereas by increasing angle downwards, the lift and drag coefficients shows reducing trend.

Table 4.1 Comparison of drag and lift coefficients without and with spoiler at different inclinations and at car velocities of 60 kmph and 160 kmph.

Spoiler Condition / Velocity of car	60 kmph		160 kmph	
	Drag coefficient(C_D)	Lift coefficient(C_L)	Drag coefficient(C_D)	Lift coefficient(C_L)
without spoiler	0.252960	0.072735	0.251190	0.068970
Spoiler with zero inclination	0.238320	0.028473	0.247070	0.040476
Spoiler with 10° inclination upwards	0.238250	0.027836	0.240420	0.028241
Spoiler with 20° inclination upwards	0.238620	0.037613	0.241750	0.049448
Spoiler with 10° inclination downwards	0.238930	0.021108	0.242640	0.018175
Spoiler with 20° inclination downwards	0.237990	0.024149	0.241750	0.021224
Spoiler with 30 ° inclination downwards	0.237850	-0.002232	0.239910	0.002540

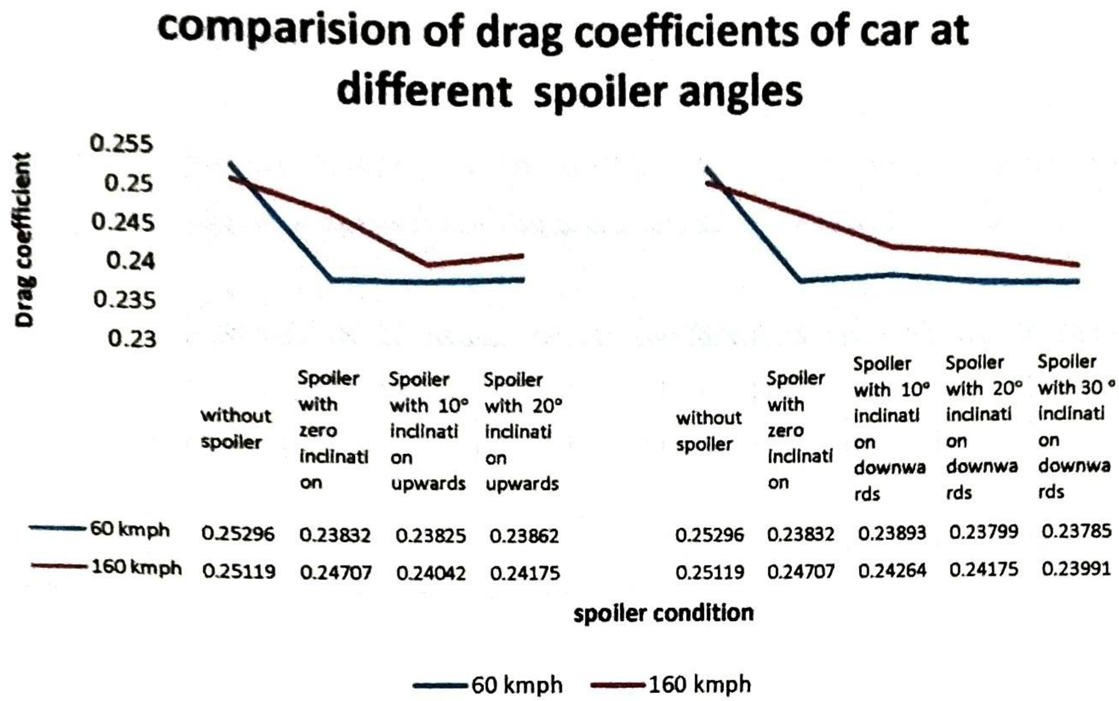


Figure. 4.9. comparison of drag coefficient of car at diffferent velocities with varying spoiler angles.

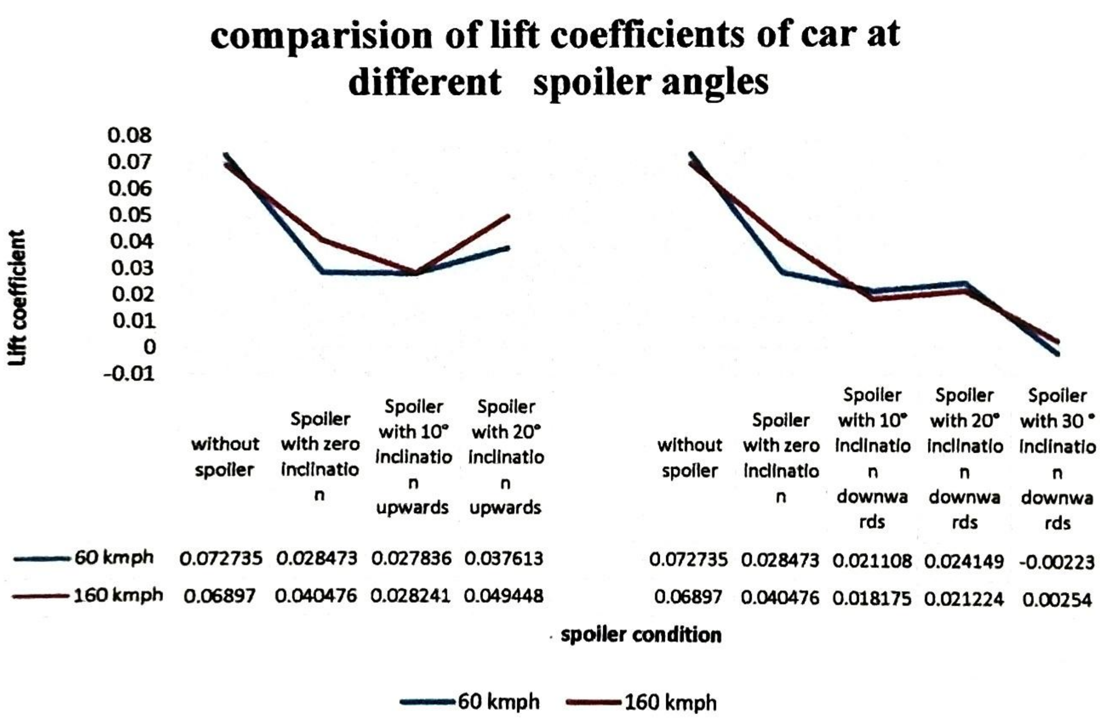


Figure. 4.10. comparison of lift coefficient of car at diffferent velocities with varying spoiler angles.

The drag coefficient reduces by 5.97% and lift coefficient by 96.3% with the attachment of spoiler at inclination of 30 degrees downwards at a velocity of 60 kmph. The drag coefficient reduces by 4.5% and lift coefficient by 103.06% with the attachment of spoiler at inclination of 30 degrees downwards at a velocity of 160 kmph.

At a car velocity of 60 kmph, the lift coefficient of car with spoiler inclining downwards at an angle 30 degrees is -0.002322. It shows that the car will not lift off from ground and it is always pushed towards the ground maintaining better stability.

CONCLUSIONS

Lift and drag forces are lesser for a vehicle fitted with spoiler. The spoiler inclining downwards reduces the lift and drag forces compared to inclining upwards. The drag coefficient reduces by 5.97% and lift coefficient by 96.3% with the attachment of spoiler at inclination of 30 degrees downwards at a velocity of 60 kmph. The drag coefficient reduces by 4.5% and lift coefficient by 103.06% with the attachment of spoiler at inclination of 30 degrees downwards at a velocity of 160 kmph. Therefore, by attaching a spoiler with downward inclination reduces drag and lift drastically and gives better fuel economy and stability.

Future scope

1. Simulations can be run at different velocities and different spoiler angles and there by optimum angle of spoiler at each velocity can be determined.
2. Once the optimum value of spoiler angle at different velocity is obtained, a sensor based spoiler model which can adjust automatically to an optimum angle corresponding to velocity of car.

REFERENCES

[1]S.M. Rakibul Hassan ,et.al. "Numerical Study on Aerodynamic Drag Reduction of Racing Cars" , Published by Elsevier Ltd on behalf of The Chinese Society of Theoretical and Applied MechanicsProcedia Engineering 90 (2014) ,pg 308 – 313

[2]Wenhu Wang, Peiqing Liu . "Numerical study of the influence of spoiler deflection on high-lift configuration".Theoretical & Applied Mechanics Letters 7 (2017) ,pg159–163

[3] R. B. Sharma, Ram Bansa."CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction ". *(IOSR-JMCE) Volume 7, Issue 5 (Jul. - Aug. 2013), PP 28-35*

[4]C.N. Patil . "Aerodynamic study and drag coefficient optimization of passenger vehicle".(IJERT), vol. 1 issue 7, sept-2012

[5]Bhavini Bijlani. "EXPERIMENTAL AND COMPUTATIONAL DRAG ANALYSIS OF SEDAN AND SQUARE-BACK CAR ".(IJAET)Vol. 4, Issue 2(April-June, 2013), pg63-65

[6]Mustafa Cakir . "CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle ". MASTER THESIS ,In the School of Engineering atSanta Clara University, December 2012

[7]Rubel Chandra Dasa,, Mahmud Riyada . "CFD Analysis of Passenger Vehicleat Various Angle of Rear End Spoiler ".*Department of Mechanical Engineering, Khulna University of Engineering &Technology,Fulbarigate,Khulna-9203,Bangladesh* ,Procedia Engineering 194 (2017),pg160 – 165

[8]Sneh Hetawala et.al. " Aerodynamic Study of Formula SAE Car ". 12th GLOBAL CONGRESS ON MANUFACTURING AND MANAGEMENT, GCMM 2014, Procedia Engineering 97 (2014), pg 1198 – 1207

[9]Rizal E. M. Nasira et.al. "Aerodynamics of ARTeC's PEC 2011 EMo-C Car, (IRIS 2012), Procedia Engineering 41 (2012) ,pg1775 – 1780

[10] CH. jayasri ."Study and Analysis of Flow Over Passenger Cars Using CFD ".

[11] A text book on Fluid Mechanics and Hydraulic Machines by R.K Bansal

[12] A text book on Fluid Mechanics Fundamentals and Applications by Yunus A Cengel and John M Cimbela