# The Implementation of The Effect of Aerodynamics on Car Overtaking Truck Using CFD Analysis Method

**Bodhisagar J. Tayade[1], P. S. Bajaj[2]**

[1]PG Student (M.Tech. Design Eng.) Mechanical Engineering Department, SSGB College of Engineering and Technology, Bhusawal, Maharashtra

[2]Professor, Dept. of Mechanical Engineering, SSGB College of Engineering and Technology, Bhusawal, Maharashtra

---------------------------------------------------------------------***---------------------------------------------------------------------

**Abstract -** The importance of overtaking investigation is to ensure that vehicle can remain stable under turbulent conditions. The flow distribution occurring on a light vehicle when it is passing by a heavy vehicle investigated using three dimensional (3D) Computational Fluid Dynamics. Previous tests have been researched and compared, it was found that the overtaking vehicle causes the other vehicle it experiences sideward forces, leading to roll followed by undergoing a yawing moment as well as reductions in speed, caused by changing pressure distributions throughout the vehicles. An aerodynamic characteristic of a vehicle is of significant interest in reducing accidents due to wind loading and in reducing the fuel consumption. To avoid accidents, we should always reduce the wake region of the whole body of every vehicle. For that modulation in shape the physical phenomenon condition is changed and therefore the flow separation also is reduced to small.

This report presents a control and stability of a vehicle overtaking and air flow pattern around the vehicle body using Computational Fluid Dynamics (CFD) software. The car is modeled without aero kit and CFD analysis is carried out to study the static pressure, velocity and wake region. The CAD model is created by using Creo parametric software and CFD analysis is done by using ANSYS software.

***Key Words*****:** CFD, Vehicle Overtaking using CFD

## 1.INTRODUCTION

In the past, the external shape of cars has evolved particularly for safety reasons, comfort improvement and also aesthetic considerations. Consequences of these guidelines on car aerodynamics were not of major concern for many years. However, this situation changed in the 70's with the emergence of the oil crisis. The first studies on the overtaking effects, we investigated in response of the weight reduction of cars involved by the first oil crisis. After this oil crisis, car manufacturers have made substantial efforts to reduce the fuel consumption. This was achieved by improving the design of the cars, by developing efficient engine or by decreasing the vehicle weight. With the third option, vehicles became more sensitive to unsteady aerodynamic effects, such as those induced by an overtaking man oeuvre. The overtaking manoeuvre between two vehicles yields additional aerodynamic forces acting on both vehicles. These additional forces lead to sudden lateral displacements and rotations around the yaw axis of each vehicle. Such sudden change of the side force and of the yawing moment, complicates the steering corrections performed by the driver and can yield critical safety situations, in particular in adverse weather conditions, such as crosswinds or rain.

This chapter provides an introduction to this investigation of transient aerodynamics of closely spaced vehicles. As the population continues to grow, the problem of mass transportation in urban areas has become an increasing concern for travelers; Truckers, rail systems, and car are mass transportation methods whose use has been encouraged to reduce the daily congestion on heavily used public routes. However, an increasingly larger number of cars continue to make transportation difficult for people in large metropolitan areas. In characterizing the aerodynamic behavior of road vehicles, the most important factor from the viewpoint of fuel economy. From a control and stability point of view, however, the side force and yawing moment are the most crucial aerodynamic characteristics of a vehicle.

### 1.1 Overtaking

Overtaking or passing is the act of one vehicle going past another slower moving vehicle, travelling in the same direction, on road. The overtaking manoeuvre is one of the critical actions that a driver performs while travelling on a highway. Errors in this decision-making process, typically caused by driver failure to accurately and timely interpret information about other vehicles in close proximity, have often resulted in catastrophic accidents. In order to eliminate such errors, or at least minimize their impact, and increase the level of safety, the vehicles of the future would have to incorporate intelligent algorithms that will allow them to accurately consider all aspects of overtaking manoeuvre. Figure 1 shows the overtaking manoeuvre.

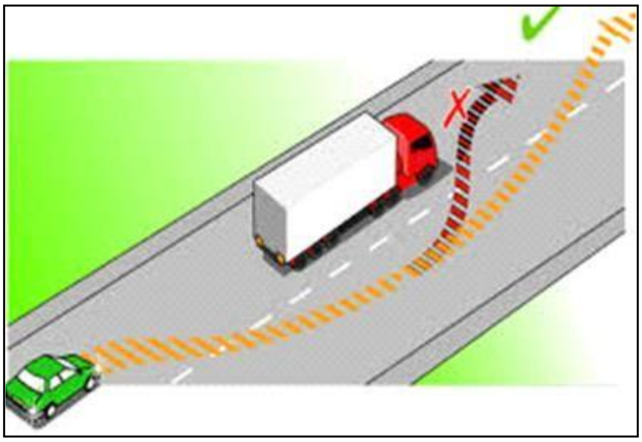


**Fig.-1.** Overtaking manoeuvre

- **Fuel Consumption**

Fuel economy is that the relationship between the space traveled and fuel consumed. The fuel economy of an automobile is that the relationship between the space traveled and herefore the amount of fuel consumed by the vehicle. Consumption is often expressed in terms of volume of fuel to travel a distance, or the space travelled per unit volume of fuel consumed. Since fuel consumption of vehicles may be a significant think about pollution, and since importation of motor fuel are often an outsized a part of a nation's foreign trade, many countries impose requirements for fuel economy. Different measurement cycles are wont to approximate the particular performance of the vehicle. The energy in fuel is required to beat various losses (wind resistance, tire drag, and others) in propelling the vehicle, and in providing power to vehicle systems like ignition or air conditioning. Various measures are often taken to scale back losses at each of the conversions between energy in fuel and K.E. of the vehicle. Driver behavior can affect fuel economy; maneuvers like sudden acceleration.

- **Drag**

Force is the summation of all forces that resist against vehicle motion. We will consider the separate sources of drag that contribute to the total drag of a vehicle. The variation of drag force as a function of airspeed seems like a graph of parabola. This indicates that the drag initially reduces with airspeed, then increases because the airspeed increases. It demonstrates that there are some parameters which will decrease drag because the velocity increases there is another parameter which will increase drag because the velocity increases. This observation shows us a nice direction for drag classification.

## 1.2 Drag Classification

In this section, a list of definitions of various types of drag is presented, and then classification of all of these drags forces is described.

### 1) Induced Drag:

The drag that results from the generation of a trailing vortex system downstream of a lifting surface with a finite ratio. In another word, this sort of drag is induced by the lift force.

### 2) Parasite Drag:

The total drags of a vehicle minus the induced drag. Thus, it's the drag indirectly related to the assembly of lift. The parasite drag consists of drag of varied aerodynamic components; the definitions of which follow.

### 3) Skin Friction Drag:

The drag on a body resulting from viscous shearing stresses (i.e., friction) over its contact surface (i.e., skin). The drag of a really streamlined shape like a skinny, flat plate is usually expressed in terms of a skin friction drag. This drag is a function of Reynolds number (Re). There are mainly two cases where the flow within the physical phenomenon is entirely laminar or entirely turbulent over the plate. The Reynolds number (Re) is based on the total length of the object in the direction of the velocity. In a usual application, the boundary layer is normally laminar near the leading edge of the object undergoing transition to a turbulent layer at some distance back along the surface.

A laminar physical phenomenon begins to develop at the vanguard and its thickness grows in downstream. At a long way from the vanguard the laminar boundary becomes unstable and is unable to suppress disturbances imposed thereon by surface roughness or fluctuations within the free stream. In a distance the physical phenomenon usually undergoes a transition to a turbulent physical phenomenon. The layer suddenly increases in thickness and is characterized by a mean velocity profile on which a random fluctuating velocity component is superimposed. The distance, from the vanguard of the thing to the transition point are often calculated from the transition Reynolds number. Skin friction factor is independent of surface roughness in streamline flow, but may be a strong function of surface roughness in flow thanks to physical phenomenon.

### 4) Form Drags (Pressure Drag):

The drag on a body resulting from the integrated effect of the static pressure acting normal to its surface resolved within the drag direction. Unlike the skin friction drag those results from viscous shearing forces tangential to a body's surface, form drag results from the distribution of pressure normal to the body's surface. In an extreme case of a flat plate normal tothe flow, the drag is totally the result of an imbalance in the pressure distribution. As with skin friction drag, form drag is usually hooked in to Reynolds number. Form drag is predicated on the projected frontal cortex. As a body begins to move through the air, the vortices in the boundary layer are shed from the upper and lower surfaces to form two vortices of opposite rotation. The coefficient of drag values during this table are supported the frontal cortex. In this table, the flow is coming from left to the proper.

## 1.3 Reduction of Drag

The issue of reducing the vehicles power consumption emerged nearly at an equivalent time when vehicle itself was invented. In the beginning fueling stations were few and a low-consumption vehicle was needed to hide the longest possible distance, between refueling. Since, the amount of cars has grown with the rise in fuel cost; it became again important to scale back the fuel consumption, both to save lots of fuel but also the environment. Vehicle power consumption reduction are often achieved by various means like improved engine efficiency and aerodynamic drag reduction. From the early days, the designers recognized the importance of drag reduction and tried to streamline the design. Since the early 20-th century, large number of studies was carried out in this field. This was also the time of formulating of basic principles of vehicle body optimization, and definition of drag lower limit. For an ideal car body configuration rock bottom possible aerodynamic coefficient of drag is 0.16.

## 1.4 Aerodynamics

Aerodynamics may be a branch of dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics may be a subfield of fluid dynamics and gas dynamics, with much theory shared between them. Understanding the motion of air (often called a flow field) around an object enables the calculation of forces and moments working on the thing. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. In automotive field, the study of the aerodynamics of road vehicles is extremely important. The main concerns of automotive aerodynamics are reducing drag (though drag by wide wheels is dominating most cars), reducing wind noise, minimizing noise emission and preventing undesired lift forces at high speeds. For some classes of racing vehicles, it's going to even be important to supply desirable downwards aerodynamic forces to enhance traction and thus cornering abilities.

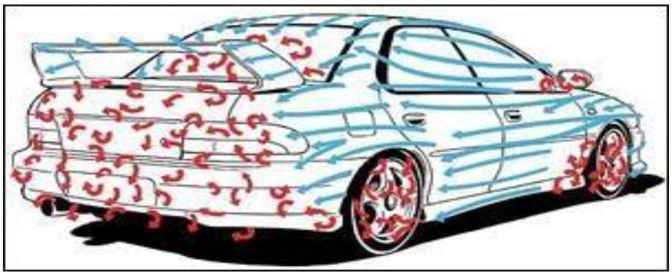


**Fig.-2.** Review of Aerodynamics

### 1) History of Aerodynamics:

Modern aerodynamics only dates back to the seventeenth century, but aerodynamic forces are harnessed by humans for thousands of years in sailboats and windmills, and pictures and stories of flight appear throughout recorded history, like the legend of ICARCUS and Daedalus Fundamental concepts of continuum drag, and pressure gradients appear within the work of Aristotle and Archimedes. In 1726, Sir Newton became the primary person to develop a theory of air resistance, making him one among the primary aerodynamicists. Dutch - Swiss HYPERLINK "http://en.wikipedia.org/wiki/Mathematician" mathematician HYPERLINK" http://en.wikipedia.org/wiki/Daniel_Bernoull i" Daniel Bernoulli followed in 1738 with Hydrodynamic, in which he described a fundamental relationship between pressure, density, and velocity for incompressible flow known today as Bernoulli's principle, which provides one method for calculating lift. In 1757, Euler published the more general Euler equations, which might be applied to both compressible and incompressible flows. The Euler equations were extended to include the consequences of viscosity within the half of the 1800s, leading to the Navier-Stokes equations. The Navier-Stokes equations are the most general governing equations of fluid flow and are difficult to solve the Wright brothers flew the first powered aircraft on December 17, 1903.As aircraft speed increased, designers began to encounter challenges related to air compressibility at speeds near or greater than the speed of sound. The differences in air flows under these conditions led to problems in aircraft control, increased drag thanks to shock waves, and structural dangers thanks to aeroelastic flutter. The ratio of the flow speed to the speed of sound was named the Mach number after Ernst Mach, who was one of the first to investigate the properties of supersonic flow. William John Macquorn Rankine and Pierre Henri Hugoniot independently developed the theory for flow properties before and after a shock wave, while Jakob Ackeret led the initial work on calculating the lift and drag of supersonic airfoils. Theodore von Kármán and Hugh Latimer Dryden introduced the term transonic to explain flow speeds around Mach 1 where drag increases rapidly. This rapid increase in drag led aerodynamicists and aviators to disagree on whether supersonic flight was achievable until the sonic barrier was broken for the primary time in 1947 using the Bell X-1aircraft.

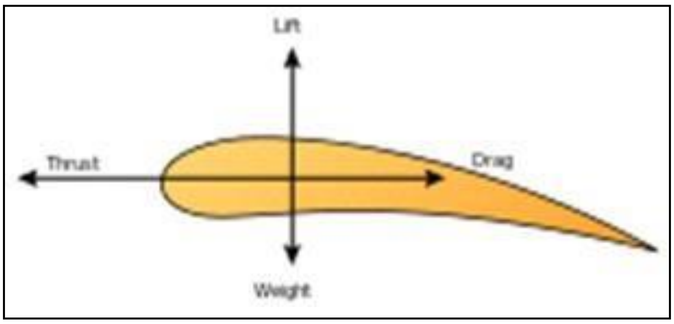



**Fig.-3.** Forces Of Flight on an Airfoil Fundamental Concepts Understanding the Motion

Air around an object (often called a flow field) enables the calculation of forces and moments working on the thing. In many aerodynamics problems, the forces of interest are the elemental forces of flight: lift, drag, thrust, and weight. Of these, lift and drag are aerodynamic forces, i.e., forces thanks to air flow over a solid body. Calculation of those quantities is usually founded upon the idea that the flow field behaves as a continuum. Continuum flow fields are characterized by properties such as velocity, pressure, density and temperature, which may be functions of spatial position and time. These properties could also be directly or indirectly measured in aerodynamics experiments, or calculated from equations for the conservation of mass, momentum, and energy in air flows. Density, velocity, and a further property, viscosity, are wont to classify flow fields. Flow velocity is employed to classify flows consistent with speed regime. Subsonic flows are flow fields in which air velocity throughout the entire flow is below the local speed of sound. Transonic flows include both regions of subsonic flow and regions during which the flow speed is bigger than the speed of sound. Supersonic flows are defined to be flows during which the flow speed is bigger than the speed of sound everywhere. A fourth classification, hypersonic flow, refers to flows where the flow speed is far greater than the speed of sound. Aerodynamicists disagree on the precise definition of hypersonic flow.
Compressibility refers as to if or not the flow during a problem can have a varying density. Subsonic flows are often assumed to be incompressible, i.e., the density is assumed to be constant. Transonic and supersonic flows are compressible, and neglecting to account for the changes in density in these flow fields when performing calculations will yield inaccurate results.
Viscosity is related to the frictional forces during a flow. In some flow fields, viscous effects are very small, and solutions may neglect to account for viscous effects. These approximations are called inviscid flows. Flows for which viscosity is not neglected are called viscous flows. Finally, aerodynamic problems can also be classified by the flow environment. External aerodynamics is that the study of flow around solid objects of varied shapes (e.g., around an airplane wing), while internal aerodynamics is that the study of flow through passages in solid objects (e.g., through a jet engine).

**2) Continuum assumption:**

Unlike liquids and solids, gases are composed of discrete molecules which occupy only a small fraction of the volume filled by the gas. On a molecular level, flow fields are made from many individual collisions between gas molecules and between gas molecules and solid surfaces. In most aerodynamics applications, however, this discrete molecular nature of gases is ignored, and therefore the flow field is assumed to behave as a continuum. This assumption allows fluid properties like density and velocity to be defined anywhere within the flow. Validity of the continuum assumption is dependent on the density of the gas and the application in question. For the continuum assumption to be valid, the mean free path length must be much smaller than the length scale of the appliance in question. For example, many aerodynamics applications affect aircraft flying in atmospheric conditions, where the mean free path length is on the order of micrometers. In these cases, the length scale of the aircraft ranges from a couple of meters to a couple of tens of meters, which is far larger than the mean free path length. For these applications, the continuum assumption holds. The continuum assumption is a smaller amount valid for very low-density flows, like those encountered by vehicles at very high altitudes (e.g., 300,000 ft/90 km) [4] or satellites in Low Earth orbit. In these cases, physics may be a more valid method of solving the matter than continuous aerodynamics. The Knudsen number are often wont to guide the selection between physics and therefore the continuous formulation of aerodynamics. Aerodynamic problems are typically solved using fluid dynamics conservation laws as applied to a fluid continuum.

### 1.5 Objectives

The main objective of this project is to decrease the wake region of the vehicle which is cross over one another. Due to the decrement of wake region, we get the control and stability of vehicle, increment in fuel efficiency. For this report, the design and analyzing of overtaking principle by using heavy vehicle (tourist truck) and light vehicle (car).

### 1.6 Outline of the Project

The report contains 5 chapters. Chapter 1 provides an overview of overtaking, fuel consumption concepts using Aerodynamics effects. Chapter 2 provides the previous work report. Chapter 3 provides the detail Design procedure. Chapter 4 Discuss and illustrate the performance of vehicle overtaking using the computational fluid dynamics tool. Chapter 5 indicates the report with their conclusion and their further recommendations.

## 2. LITERATURE REVIEW

The process of reading, evaluating, classifying and comparing the prior research studies, summarizing them and critical analysis of scholarly materials about a specific topic is known as a literature review. It is a critical and an evaluative summary of the themes, issues and crusher of a specific concept clearly defined research topic obtained from the published literature.

According to Mohd Nizam Sudin1, Mohd Azman Abdullah, Shamsul Anuar Shamsuddin, FaizRedza Ramli1, Musthafah Mohd Tahir Center for Advanced Research on Energy (CARE), the performance of aerodynamic drag - reduction is reducing pressure drag of ground vehicles it mainly focuses on the methods employed to prevent or delay air flow separation at the rear end of vehicle.

**J. Clarke, A. Filippone**, "Unsteady computational analysis of vehicle passing", (2007). Ahmed body angle of the rear end of the vehicle should be slanted surface of 30◦. Several different relative velocities and transversal distances between vehicles were passed then the aerodynamic influence of the passage on the overtaken vehicle.

Acc.to**. Krupakar Pasala**, **Bhargav**, **H.JeevanRao** , **K.Sreenath Reddy Ḃ** and **T.V. Seshaiah Naidu** ,A Department of Aeronautical Engineering, Vardhaman College of Engineering, Hyderabad, "A CFD Investigation into the Flow Distribution on a Car passing by a Truck "As the light vehicle passes the heavy vehicle during an overtaking manoeuvre the flow around the two vehicles interact generating pressure variation over the car, this leads to adverse effect on the car handling and stability. In that study case, to reduce the wake region we get the safety and stability control of a vehicle is increased, and also the fuel consumption is decreased.

The overtaking manoeuvre between two vehicles yields additional aerodynamic forces acting on both vehicles. These additional forces lead to sudden lateral displacements and rotations around the yaw axis of each vehicle. Such sudden change of the side force and of the yawing moment, complicates the steering corrections performed by the driver and can yield critical safety situations, in particular in adverse weather conditions, such as crosswinds or rain. Intensities of these forces are extrapolated when the overtaking manoeuvre involves a light car and a heavy-truck. Moreover, such a manoeuvre implies a disturbance of the aerodynamic of vehicles.

The first studies on the overtaking effects, **Heffrey [1]** and **Howell [2],** were investigated in response of the weight reduction of cars involved by the first oil crisis. Actually, after this oil crisis, car manufacturers have made substantial efforts to reduce the fuel consumption. This was achieved by improving the design of the cars, by developing efficient engine or by decreasing the vehicle weight. With the third option, vehicles became more sensitive to unsteady aerodynamic effects, such as those induced by an overtaking manoeuvre. In addition to the works of Heffrey and Howell, several experimental studies were carried out as well. These studies were focused on different aspects.

Studies performed by **Legouis et al [3]**, **Telionis et al [4]** or **Yamamoto et al [5]** were dedicated to the car-truck overtaking. 2 More recently, several dynamic studies in order to analyze effects of the relative velocity, the transverse passing, and the crosswind during an overtaking manoeuvre were performed by **Noger et al [6, 7]**. Both studies were carried out with 7/10 scaled **Ahmed bodies, [8]**.

The two bodies of the first study were hatchback shapes (slant of 30◦), while the two bodies of the second one was squareback shapes. **Noger and Van Grevenynghe [9]** proposed a study of car-truck overtaking on one test case: one relative velocity and one lateral spacing. **Gilli´eron and Noger [10]** analyzed the transient phenomena occurring during various phenomena such as the overtaking, the crossing or tunnel exits.

The overtaking process has also been studied using numerical modelling. Some recent two-dimensional (2D) numerical studies can be found**. Clark and Filippone [11]** performed the overtaking process of two sharped edges bodies. The work aimed to provide a thorough analysis of the overtaking process. Effects of the relative velocity and the transversal spacing were studied. The authors focused on 2D overtaking as a preliminary means of investigating an appropriate simulation strategy for the complex three-dimensional (3D) flow. **Corin et al [12]** performed 2D numerical simulations of two rounded edges bodies.

The dynamic effect of the passing manoeuvre was highlighted by comparisons with quasi-steady calculations. It was shown that crosswinds yield significant dynamic 3 effects. The authors of [11, 12] agreed that their 2D approaches were first steps towards 3D calculations. In particular, the Venturi effect, occurring when vehicles move closer, is strongly overestimated. 3D numerical simulations of two Ahmed bodies overtaking were performed by **Gilli´eron [13]**. Calculations were achieved using a Reynolds Averaged Navier-Stokes numerical method with a k−ϱ turbulence model. The effects of the transversal spacing and the crosswind were studied. However, this study was limited to a quasi-steady approach. This paper presents a dynamic three-dimensional simulation of passing processes based on the ζ − f turbulence model and a deforming/sliding mesh method. The aims are to accurately predict the aerodynamic forces and moment, occurring on vehicles during a passing manoeuvre, and to give thorough analysis of the passing process. The paper starts with a description in section 2.1 of the experimental setup that is used in the present numerical study. This is followed by the numerical methodology, including the turbulence model, the numerical method and details, and the deforming/sliding mesh method, in section 2.2. Results of the simulations are presented in section 3. In section 4, a complete analysis of the effects of the passing manoeuvre is provided. Finally, the paper is summarized in section 5.

## 3. PROTO MODEL TESTING USING COMPUTATIONAL FLUID DYNAMICS

### 3.1 Introduction

As the car passes the truck during an overtaking manoeuvre the flow around the two vehicles are interact generating pressure variation over the car. This pressure variation may lead to adverse effect on the car handling and stability. This case is somewhat similar to the study of cross wind problems on vehicles. Sudden change of wind velocity in the wake of the truck causes rapid change of the Aerodynamic forces acting on a passing by car. This sudden change causes difficulty in controlling the vehicle which may result in miss- steering and an accident. To avoid such accidents, the behavior of the light vehicle to such disturbances created by the heavy/large vehicle is difficult to evaluate and are still not clear. This paper is to investigate that how a light vehicle passing by a heavy/large vehicle is affected by the flow disturbances created by the heavy vehicle. The transient aerodynamic forces acting on a vehicle model overtaking another vehicle model. However, the study considered was only two dimensional and the vehicle models overtaking each other were both similar. The impact of the ratio between the relative velocity and the overtaken vehicle velocity (k =Vr/V) is an area of some debate. A dynamic study on scale models and concluded that, for a velocity ratio of k<0.25 dynamic effects could be neglected and the problem could be modeled statically. All the experiments have their numerical quotes. These quotes are coming from the conservation principles. These are principles are classified into 3 major groups namely

- Conservation of Mass
- Conservation of Momentum
- Conservation of Energy

Three conservation principles are given below:

### 1.Conservation of Mass:

In fluid dynamics, the mathematical formulation of this principle is known as the mass continuity equation, which requires that mass is neither created nor destroyed within a flow of interest.

### 2. Conservation of Momentum:

In fluid dynamics, the mathematical formulation of this principle can be considered an application of Newton's Second Law. Momentum within a flow of interest is only created or destroyed due to the work of external forces, which may include both surface forces, such as viscous (frictional) forces, and body forces, such as weight. The momentum conservation principle may be expressed as either a single vector equation or a set of three scalar equations, derived from the components of the three-dimensional velocity vector. In its most complete form, the momentum conservation equations are known as the Navier-Stokes equations. The Navier-Stokes equations have no known analytical solution, and are solved in modern aerodynamics using computational techniques. Because of the computational cost of solving these complex equations, simplified expressions of momentum conservation may be appropriate to specific applications. The Euler equations are a set of momentum conservation equations which neglect viscous forces used widely by modern aerodynamicists in cases where the effect of viscous forces is expected to be small. Additionally, Bernoulli's equation is a solution to the momentum conservation equation of an inviscid flow, neglecting gravity.

### 3. Conservation of Energy:

The energy conservation equation states that energy is neither created nor destroyed within a flow, and that any addition or subtraction of energy is due either to the fluid flow in and out of the region of interest, heat transfer, or work.

### 3.2 Boundary Layer

The concept of boundary layer was first introduced by a German engineer, Prandtl in 1904. According to Prandtl theory, when a real fluid flows past a stationary solid boundary, the flow will be divided into two regions.
A thin layer adjoining the solid boundary where the viscous force and rotation cannot be neglected.
An outer region where the viscous force is very small and can be neglected. The flow behavior is similar to the upstream flow.
An external flow past objects encompasses an extremely wide variety of fluid mechanics phenomena. Clearly the character of the flow field is a function of the shape of the body. For a given shaped object, the characteristics of the flow depend very strongly on various parameters such as size, orientation, speed,

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 08 | August - 2022 Impact Factor: 7.185 ISSN: 2582-3930

and fluid properties. According to dimensional analysis arguments, the character of the flow should depend on the various dimensionless parameters involved. For typical external flows the most important of these parameters are the Reynolds number, Re =UL/n, Where
L – Characteristic dimension of the body.
For many high-Reynolds-number flows the flow field may be divided into two regions.
A viscous boundary layer adjacent to the surface of the vehicle
The essentially inviscid flow outside the boundary layer.
We know that fluids adhere the solid walls and they take the solid wall velocity. When the wall does not move also the velocity of fluid on the wall is zero. In region near the wall the velocity of fluid particles increases from a value of zero at the wall to the value that corresponds to the external "frictionless" flow outside the boundary layer.

## 1.Boundary Layer Concepts

The concept of the boundary layer was developed by Prandtl in 1904. It provides an important link between ideal fluid flow and real-fluid flow. Fluid having relatively small viscosity effect of internal friction during a fluid is appreciable only during a narrow region surrounding the fluid boundaries. Since the fluid at the boundaries has zero velocity, there's a steep velocity gradient from the boundary into the flow. This velocity gradient during a real fluid sets up shear forces near the boundary. Fig 3 shows the boundary layer concept.
Visualization of the flow around the car is visible the thin layer along the body cause by viscosity of the fluid. The flow outside the narrow region near the solid boundary can be considered as ideal, that reduce the flow speed to that of the boundary. That fluid layer which has had its velocity suffering from the boundary shear is named the physical phenomenon. For smooth upstream boundaries the physical phenomenon starts out as a laminar physical phenomenon during which the fluid particles move in smooth layers. As the laminar physical phenomenon increases in thickness, it becomes unstable and eventually transforms into a turbulent physical phenomenon during which the fluid particles move in haphazard paths. When the boundary layer has become turbulent, there is still a very thin layer next to the boundary layer that has laminar motion. It is called the laminar sub layer. Various definitions of boundary–layer thickness d have been suggested. The vertical scale has been greatly exaggerated and horizontal scale has been shortened.

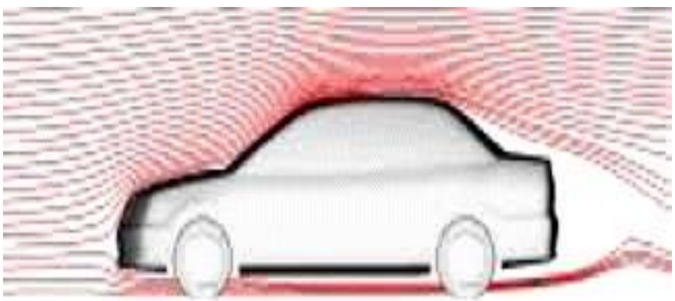


**Fig.-4.** Boundry Layer Concept

## 2. Flow Separation and Wake Region

An important observation is that fluid flow is usually irreversible. It is irreversible because fluids have viscosity, and whenever velocity gradients appear within the flow there'll be friction and energy dissipation thanks to viscous stresses. The flow is often reversible as long as there are not any velocity gradients anywhere, since that's the sole condition under which there's no friction. However, when frictional effects are small, a flow could also be approximately reversible. For instance, within the flow through an outsized duct where the boundary layers are very thin, viscous effects are confined to a rather small region, and therefore the fluid friction may sometimes be neglected. This is not the case for most practical flows. In most pipe and duct flows, for instance, the speed gradients extend over the whole cross- section and frictional stresses are all important. Even if the boundary layers are thin to start with, they will thicken rapidly under some circumstances, and therefore the flow can separate, as within the diffuser flow shown in figure. Fig 5 shows the boundary layer condition.

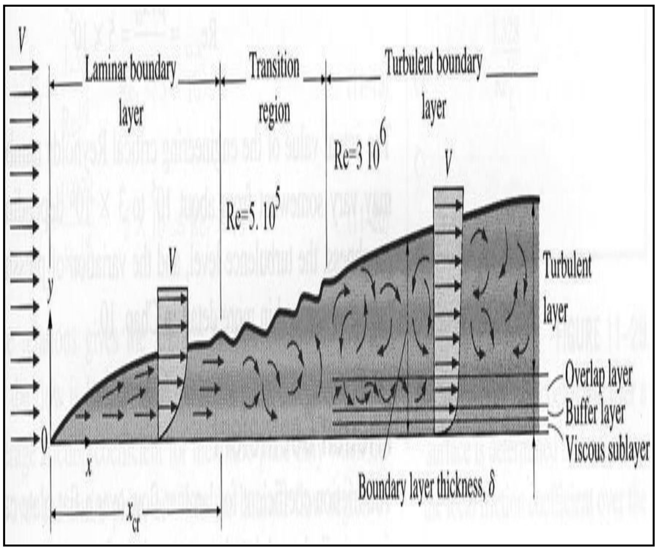



**Fig.-5.** Boundry Layer Conditions

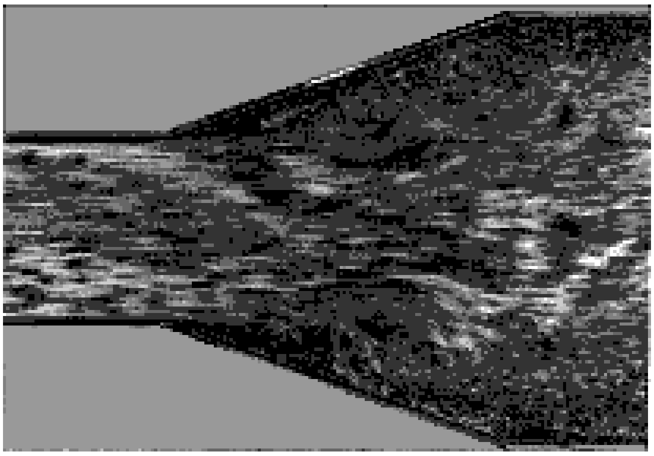


**Fig.-6.** Flow Separation and Wake Region

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 08 | August - 2022 Impact Factor: 7.185 ISSN: 2582-3930

If the flow is within the direction of accelerating area (this is named a diffuser flow), the consequences of friction and separation are usually small, if the angle of divergence is very small. For larger angles (an included angle greater than about 70 is sufficient), there is a possibility of flow separation, by which we mean that in some parts of the flow the fluid is really getting into a direction opposite to the majority flow direction. Large losses can occur, because the energy of the fluid is employed to drive large unsteady eddying motions which eventually dissipate into heat. This type of flow can occur for quite small angles. In contrast, when the flow is in the direction of decreasing area, as in the contraction flow, there is very little risk of producing large separated regions, even for very large values of the included angle (up to 450), and generally losses are quite small. So great care needs to be taken in considering frictional effects sometimes they are negligible, and sometimes they are not. When separation occurs, however, frictional effects are always important. It is worth noting that sharp corners almost always produce a separated flow. The sudden expansion and contraction flows graphically demonstrate the problem, and these flows are clearly irreversible. Fig 6 shows the flow separation and wake region.

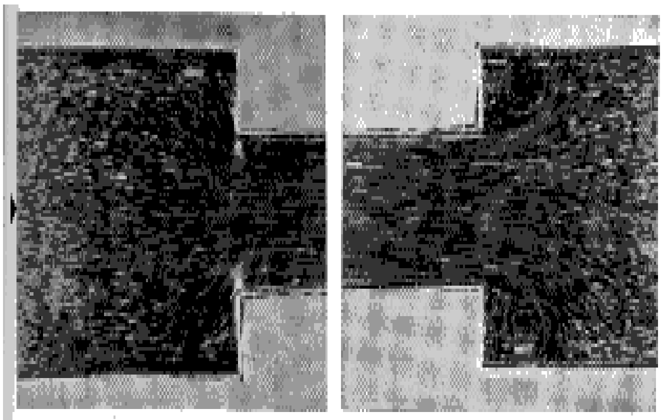


**Fig.-7.** Active Flow in Pipe

Sharp corners typically produce a separated flow, but they are not the only cause of separation. For example, the flow over a cylinder can produce a large region of separated flow downstream of the cylinder. This region is called a wake. Fig 7 shows the active flow in pipe and Fig 8 shows vortex and wake created by 2 circular particles.

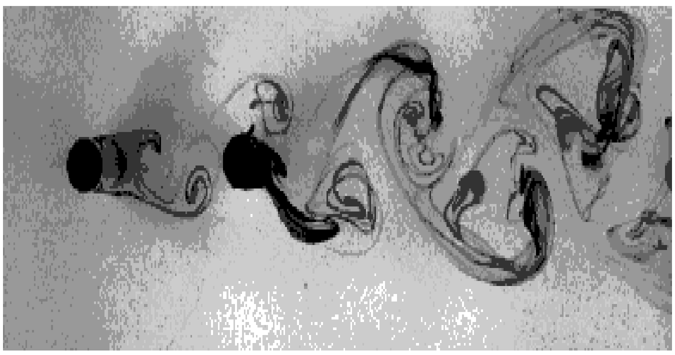


**Fig.-8.** Vortex and wake created by 2 circular particles The flow patterns over the front

Cylinder is quite different. In the front, the flow smoothly passes over the cylinder, but within the wake the flow is typically highly unsteady and enormous eddies or vortices are shed downstream. The large eddies are formed at a daily frequency and that they produce pressure disturbances within the flow which we will sometimes hear as sound waves. When we talk of the wind whistling in the trees, it is the sound of eddies being shed. A Greek instrument, the Aeolian harp, made use of this regular vortex shedding to produce music. The alternate shedding of vortices produces an alternating lift force, oscillating in direction and magnitude. If the frequency of the shedding couples to a natural frequency of the cylinder and its supports, large cross-stream oscillations can occur in the cylinder position. This kind of aerodynamic instability was responsible for the destruction of the Tacoma Narrows bridge, and it has led to a number of spectacular cooling tower failures in England. Wakes nearly always contain large eddying motions which are shed downstream, although they'll not shed at a daily frequency. For instance, the wake of a ship often contains large eddying motions, and this region is usually called a ``dead-water" region. Similar flow patterns are observed downstream of pylons supporting a bridge. In any case eddies dissipate energy. Friction is that the source of the irreversibility of fluid flow either through the formation of boundary layers over the surface, or large eddies within the wake (in thermodynamics we say that the entropy of the flow increases). In an imaginary fluid that has no friction (an inviscid fluid), there would be no drag, the entropy does not change, and the flow would be reversible. For example, a cylinder within the flow of an inviscid fluid has no vortex shedding and it's no drag, contrary to our practical experience.

### 3.3 Governing Equations of CFD

Applying the elemental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is and the conservation of momentum equation is these equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is impossible to unravel these equations analytically for many engineering problems. However, it's possible to get approximate computer-based solutions to the governing equations for a spread of engineering problems. This is the topic matter of Computational Fluid Dynamics (CFD).

### 3.4 Computational Fluid Dynamics

Computational fluid dynamics (CFD) is that the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. As a results of these factors, Computational Fluid Dynamics is now a longtime industrial design tool, helping to scale back design time scales and improve processes throughout the engineering world. CFD provides an economical and accurate alternative to scale model testing, with variations on the simulation being performed quickly, offering obvious advantages.

- **Steps Followed in CFD**

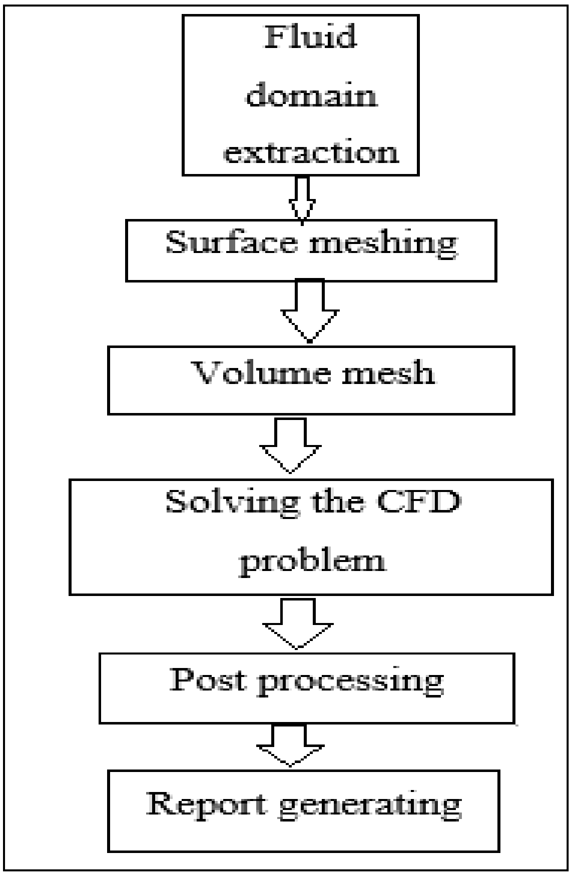



**Fig.-9.** Steps Followed in CFD Building a Mesh

One of the most cumbersome and time-consuming part of the CFD is the mesh generation. Although for very simple flows, mesh generation is easy, it becomes very complex when the problem has many cavities and passages, Mesh generation is basically the discretization of the computational domain. The mesh in finite difference methods consists of a set of points, which are called nodes. The finite volume methods consider points that form a set of volumes which are called cells. The finite element methods used sub-volumes called elements which have nodes where the variables are defined. Values of the dependent variables, such as velocity, pressure, temperature, etc., will be described for each element. Various forms of elements can be used. However, the most common type in CFD programs is a hexahedron with eight nodes, one at each corner, and this is known as a brick element or volume. For two-dimensional applications the equivalent element is a four-node quadrilateral. Some finite volume programs have now been released which have the ability to use tetrahedral in three dimensions or triangles in two dimensions. Most finite element CFD codes will allow these elements to use together with a small range of other element types.

## 1.Surface Meshing

This package provides a function template to compute a triangular mesh approximating a surface. The meshing algorithm requires to know the surface to be meshed only through an oracle able to tell whether a given segment, line or ray intersects the surface or not and to compute an intersection point if any. This feature makes the package generic enough to be applied in a wide variety of situations. For instance, it can be used to mesh implicit surfaces described as the zero-level set of some function. It may also be used in the field of medical imaging to mesh surfaces described as a gray level set in a three-dimensional image.

The meshing algorithm is based on the notion of the restricted Delaunay triangulation. Basically, the algorithm computes a set of sample points on the surface, and extract an interpolating surface mesh from the three-dimensional triangulation of these sample points. Points are iteratively added to the sample, as in a Delaunay refinement process, until some size and shape criteria on mesh elements are satisfied.

The size and shape criteria guide the behaviour of the refinement process and control its termination. They also condition the size and shape of the elements in the final mesh. Naturally, those criteria can be customized to satisfy the user needs. The Surface mesh generation package offers a set of standard criteria that can be scaled through three numerical values. Also, the user can also plug in its own set of refinement criteria. There is no restriction on the topology and number of components of the surface provided that the oracle (or the user) is able to provide one initial sample point on each connected component. If the surface is smooth enough, and if the size criteria are small enough, the algorithm guarantees that the output mesh is homeomorphic to the surface, and is within a small bounded distance (Hausdorff or even Frechet distance) from the surface. The algorithm can also be used for non-smooth surfaces but then there is no guarantee.

## 2. Volumetric Meshes

Volumetric meshes are a polygonal representation of the interior volume of an object. Unlike polygon meshes which represent only the surface as polygons, volumetric meshes also discretize the interior structure of the object. One application of volumetric meshes is in finite element analysis, which may use regular or irregular volumetric meshes to compute internal stresses and forces in an object throughout the entire volume of the object.

In this research, a procedure called the Reference Jacobian based Mesh Optimization has been developed for the optimization of 3D mesh quality by node repositioning. The procedure is designed to improve the quality or geometric shape of mesh regions and boundary mesh faces while keeping the improved mesh as close as possible to the original mesh. The quality measure optimized in the procedure is the Condition Number shape measure which quantifies the distortion of a trivalent element corner from an "ideal" corner of a canonical element. This "ideal" corner is usually considered to be formed by 3-unit vectors coincident at the origin and lying along the coordinate axes but any other definition may be used in its place.

The procedure also includes a method for improving the quality of mesh faces on internal and external boundaries while preserving surface characteristics as described in the articles on polygonal surface mesh quality improvement. The overall procedure consists of iterations involving node repositioning on the boundary followed by node repositioning in the interior (with boundary nodes fixed).

The procedure has proved to be very effective in improving mesh quality of multi-material tetrahedral and hexahedral meshes while minimizing changes to the mesh characteristics and to the discrete boundary surfaces.

### 3. Solving the CFD Problem

- Reading the file. The reading the file should clear as case file or data file or case and data file. In this we have to read case and data file.
- Scaling the grid.
- Checking the grid.
- Defining the models. Model should define whether it is steady or unsteady and whether it is viscous. The model is defined here is steady and viscous.
- Defining the materials.
- Defining the boundary condition
- Controls
- Initialize
- Monitor
- Iterate

The component that solves the CFD problem is called the Solver. It produces the required results in a non-interactive/batch process. A CFD problem is solved as follows: The partial differential equations are integrated over all the control volumes within the region of interest. This is like applying a basic conservation law (for example, for mass or momentum) to every control volume these integral equations are converted to a system of algebraic equations by generating a set of approximations for the terms in the integral equations. The algebraic equations are solved iteratively. An iterative approach is required because of the non-linear nature of the equations, and as the solution approaches the exact solution, it is said to converge. For each iteration a mistake, or residual, is reported as a measure of the general conservation of the flow properties.
How close the final solution is to the exact solution depends on a number of factors, including the size and shape of the control volumes and the size of the final residuals. Complex physical processes, such as contraction and turbulence, are often modeled using empirical relationships. The approximations inherent in these models also contribute to differences between the CFD solution and therefore the real flow.
The solution process requires no user interaction and is, therefore, usually carried out as a batch process. The solver produces a results file which is then passed to the post-processor.

### 4. Post Processing

The post-processor is that the component wants to analyze, visualize and present the results interactively. Post- processing includes anything from obtaining point values to complex animated sequences.
Examples of some important features of post- processors are:

- Visualization of the geometry and control volumes
- Vector plots showing the direction and magnitude of the flow
- Visualization of the variation of scalar variables (variables which have only magnitude, not direction, such as temperature, pressure and speed) through the domain
- Quantitative numerical calculations
- Animation
- Charts showing graphical plots of variables
- Hardcopy and online output.

### 5. Report Generating

All charts, tables, figures, and comments automatically become report content. The report component order can be adjusted and figures can be 3D Viewer files or bitmaps. Different output formats are available, including HTML.

### 3.5. Limitations

#### 1) Boundary Conditions

1. As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.
CFD software is employed during this work to calculate the pressure drop and overall heat transfer coefficient for five models. The methodology and the results are discussed in the next chapters.
The governing equation of fluid motion may result in a solution when the boundary conditions and the initial conditions of specified. The form of boundary conditions that is required by any partial differential equation depends on the equation itself and the way that it has been discredited.
Common boundary conditions are classified either in terms of the numerical value that have to be set or in terms of the physical type of boundary condition.

#### 2) Solid Walls

Many boundaries within the fluid flow domain are going to be solid walls, and these are often either stationary or moving walls. If the flow is laminar then the velocity components can be set to be the velocity of walls. When the flow is turbulent, however, the situation is more complex.
At an inlet, fluid enters the domain and thus, its fluid velocity or pressure or the mass flow could also be known. Also, the fluid may have certain characteristics, like turbulence characterizes which require to specified.

#### 3) Symmetry Boundaries

When the flow is symmetrical about some plane there is no flow through the boundary and the derivatives of the variables normal to the boundary are zero.

#### 4) Cyclic or Periodic Boundaries

These boundaries come in pairs and are used to specify the flow has the same values of the variables at equivalent position and both of the boundaries.

#### 5) Pressure Boundary Conditions

The ability to specify a pressure condition at one or more boundaries of a computational region is a crucial and useful computational tool. Pressure boundaries represent such things as confined reservoirs of fluid, ambient laboratory conditions and applied pressures arising from mechanical devices. Generally, pressure condition can't be used at boundary where velocities also are specified, because velocities are influenced by pressure gradients. The only exception is when pressures are necessary to specify the fluid properties. E.g., density crossing a boundary condition, mentioned as static or stagnation pressure conditions. In a static condition the pressure is more or less continuous across the boundary and the velocity at the boundary is assigned a value based on a zero normal-derivative condition across the boundary. In contrast, a stagnation pressure condition assumes stagnation conditions outside the boundary in order that the speed at the boundary is zero. since the static pressure condition says nothing about fluid outside the boundary (i.e., other than it is supposed to be the same as the velocity inside the boundary) it is less specific than the stagnation pressure condition. In this sense the stagnation pressure condition is usually more physical and is suggested for many applications.

- **Outflow Boundary Conditions**

In many simulations there is need to have fluid flow out of one or more boundaries of the computational region. In compressible flow, when the flow speed at the outflow boundary is supersonic, it makes little difference how the boundary conditions are specified since flow disturbances cannot propagate upstream. In low speed and incompressible flows, however, disturbances introduced at an outflow boundary can have an impact on the whole computational region. The simplest and most commonly used outflow condition is that of "continuative "boundary.
Continuative boundary conditions contain zero normal derivates at the boundary for all quantities. The zero- derivative condition is meant to represent a smooth continuation of the flow through the boundary.
Such as a specified pressure condition, should be used at out flow boundaries, whenever possible. When a continuative condition is used it should be placed as far from the main flow region as is practical so that any influence on the main flow will be minimal.

- **Opening Boundary Condition**

If the fluid flow crosses the boundary surface in either direction a gap boundary conditions must be utilized. All of the fluid might effuse of the domain, or into the domain, or a mixture of the 2 might happen.

- **Free Surfaces and Interfaces**

If the fluid features a free surface, then the physical phenomenon forces got to be considered. This requires utilization of the Laplace's equation which specifies the surface tension- induced jump in the normal stress p s across the interface.

### 1) Base Case

Geometric model is generated in 'CREO Parametric 2.0' which is very popular modeling software. The generated model is exported to the further process within the sort of IGES because it may be a third-party format which may be taken in to the other tools. Extracting the fluid region is that the next step during which all the surfaces which are within the contact of fluid are taken alone and every one other surface are removed completely. To keep the domain air /water tight some extra surfaces are created. This cleanup is done in ANSA meshing tool which is very rotruckt clean up tool. Extracted domain for vortex generation and finder assemblies are shown below.

### 2) Model Creation

The Modeling is done using CREO Parametric 2.0, but when modeling for a CFD analysis it is to be considered developing a separate part model for the fluid, this can only do using Boolean operation.

### 3) Introduction about Creo Parametric

In Part modeling you'll create a neighborhood from a conceptual sketch through solid feature-based modeling, also as build and modify parts through direct and intuitive graphical manipulation. The Part Modeling Help introduces you to the terminology, basic design concepts, and procedures that you simply must know before you begin building a neighborhood.
Part Modeling shows you ways to draft a 2D conceptual layout, create precise geometry using basic geometric entities, and dimension and constrain your geometry. You can find out how to create a 3D parametric part from a 2D sketch by combining basic and advanced features, like extrusions, sweeps, cuts, holes, slots, and rounds.
Finally, Part Modeling Help provides procedures for modifying part features and resolving failures. From the above modeling concept, we modeled the component and the representation of the truck and car as follows and shown in Fig 10 and Fig 11.

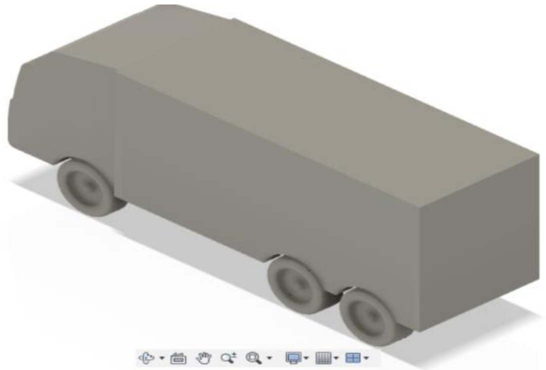


**Fig.-10.** Creo Parametric Model of Truck

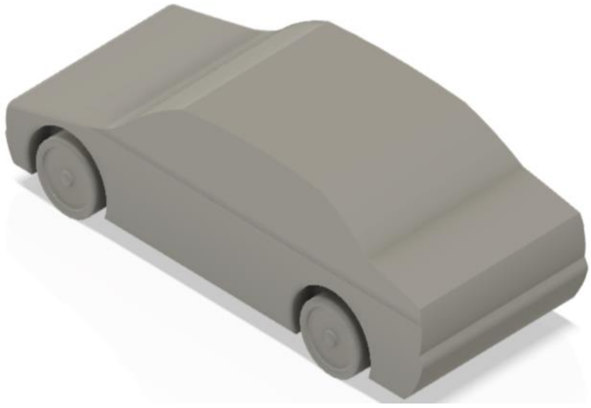


**Fig.-11.** Creo Parametric model of car

### 3.6 Transformation of Model

Then the model is converted in to the IGES and STP format which is most suitable and easy access for any other software's. Using the STP format we can import the car wheel rim model from CREO PARAMETRIC to ANSYS. Now we can make CFD model for flow analysis, for the flow analysis we required air domain. This air domain is created using the Ansys workbench design modular. Figure 12 shows the CAD model after fluid domain extraction.

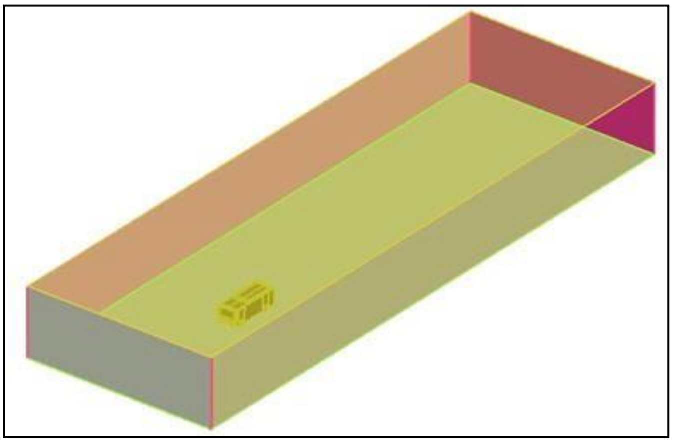


**Fig.-12.** CAD model after fluid domain extraction

### 3.7 Meshing

After cleaning up the geometry surface mesh is generated in ANSA tool itself. All the surfaces are discredited using tri surface element. As the geometry has some complicated and skewed surfaces tri surface elements are used to capture the geometry. The meshing are 2 types. They are

- **Surface Meshing**

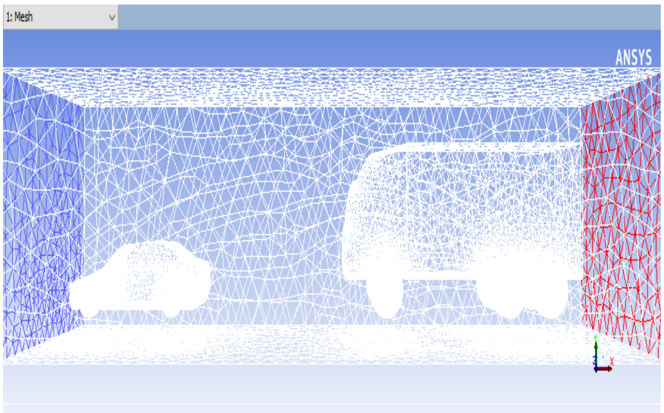
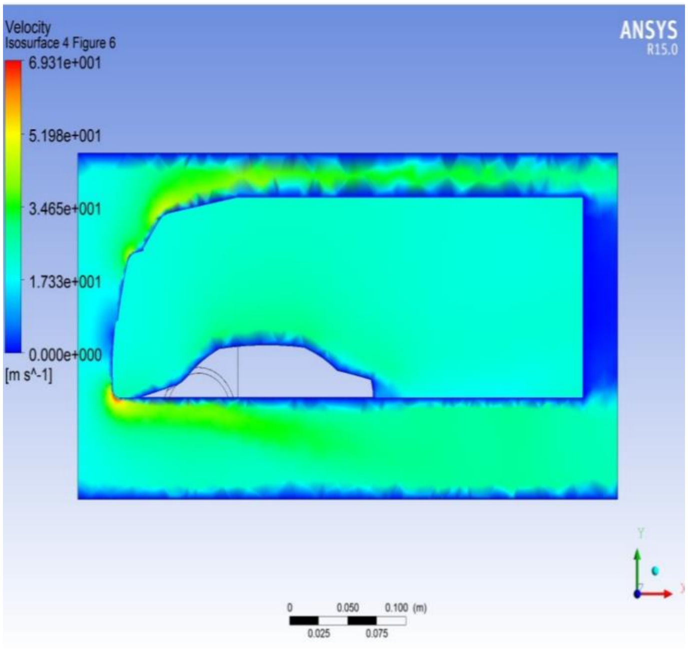


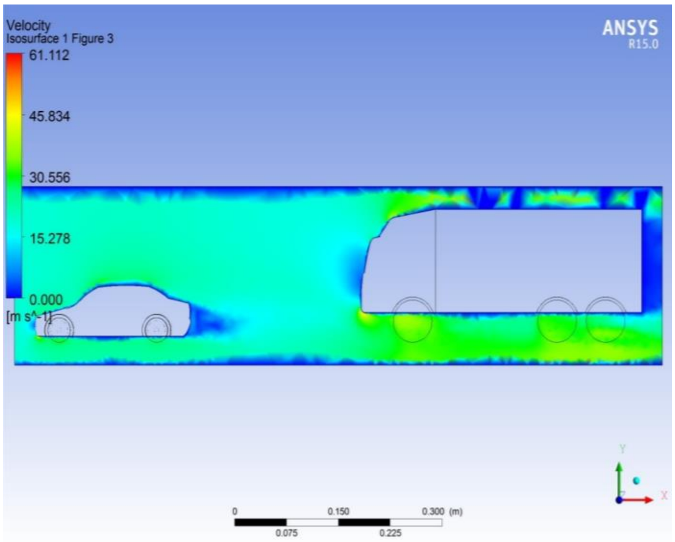

**Fig.-12.** Surface Mesh on Fluid Domain

- **Volume Meshing**

Volume mesh is generated in T-Grid which is a retract volume mesh generator. Volume is discretize using tetrahedron. Each and every cell centroid is the co-ordinate at which the navier-stokes system of equations is solved.

### 3.8 Velocity Magnitude



| Mesh | Count | Quality |
|---|---|---|
| Surface Mesh | 6931346 | 0.6 |
| Volume Mesh | 2324879 | 0.83 |

**Table-1.** Mesh Details

- **Solver Setup**

Ansys-fluent is used as the solver for this case.

- Fluid is assumed to be 3-D, turbulent, Incompressible in nature
- Turbulence is model by K-ε model
- Simple algorithm is used to solve the problem
- Segregated solver is used for pressure-velocity coupling.

- **Cell Zone Conditions**

The working fluid in the domain is air.

- **Boundary Conditions**

For flow analysis

- Inlet is assumed to be velocity inlet at 16.6 m/s
- The outlet is assumed to be pressure outlet at 0 Pascal.

## 4. RESULTS AND DISCUSSION

### 4.1. Base Case

In this type of experiments are did previous stage. Their results are shown below. Every aerodynamic experiment contains main 2 results, there are:

- Static pressure contours.
- Velocity magnitude.

- **Static Pressure Contours:**

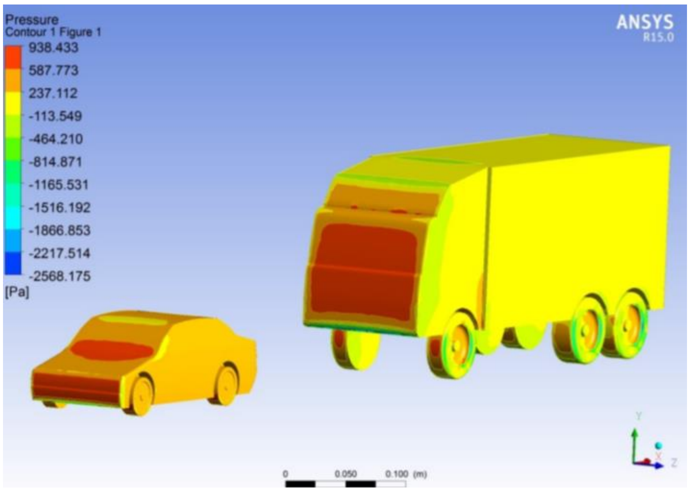



**Fig.-13.** Static Pressure Contours

Fig 13 indicates that the static pressure contours of vehicle having the medium. This image shows that the pressure values in static medium. The minimum static pressure is -2568.175 pa The maximum static pressure is 983.433 pa

- **Velocity Magnitude**

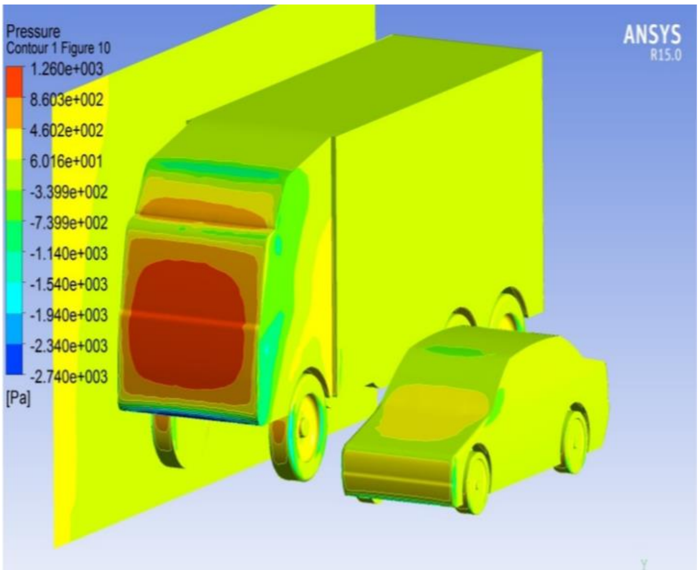



**Fig.-14.** Velocity vector diagram of reduction of wake region while over passing

Fig 14 illustrates that the velocity magnitude of wake region. The minimum velocity magnitude 0 m/s. The maximum is velocity magnitude is 61.112 m/s The velocity level of 16.6m/s is applied. A circulation is indicating the velocity speed. Fig 15 shows the velocity vector diagram of reduction of wake region while over passing. This speed creates some vortexes. This image explains that the re-circulation of velocity in both vehicles.

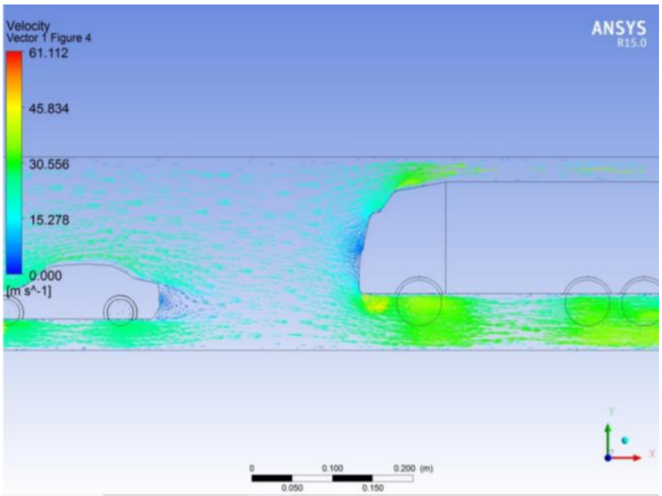



**Fig.-15.** The velocity vector diagram of reduction of wake region while overpassing.

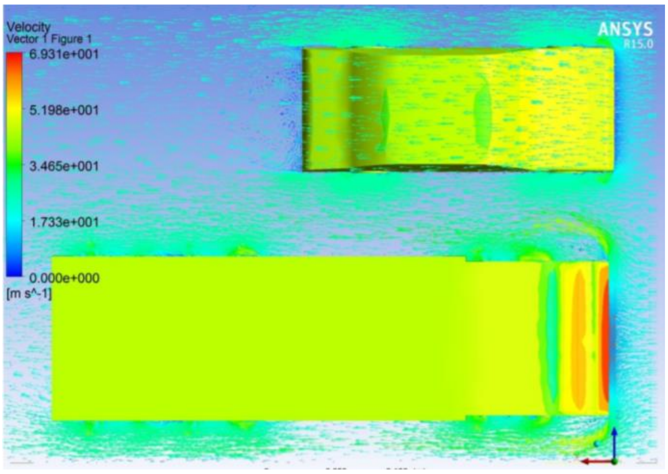
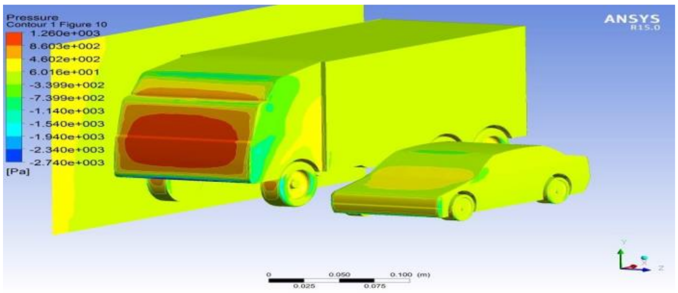
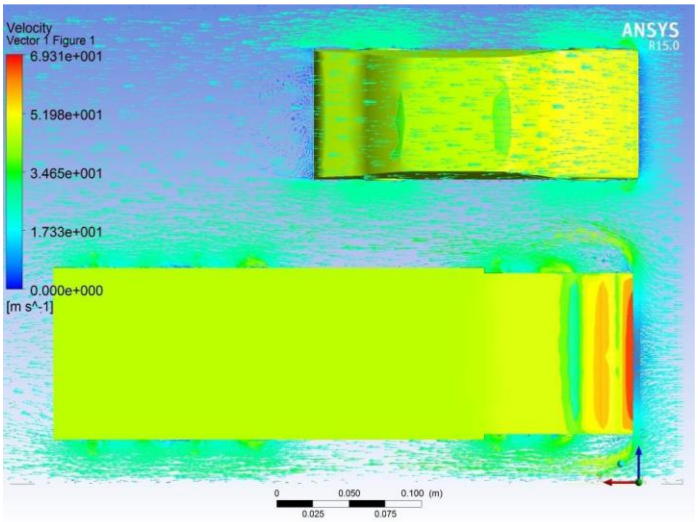


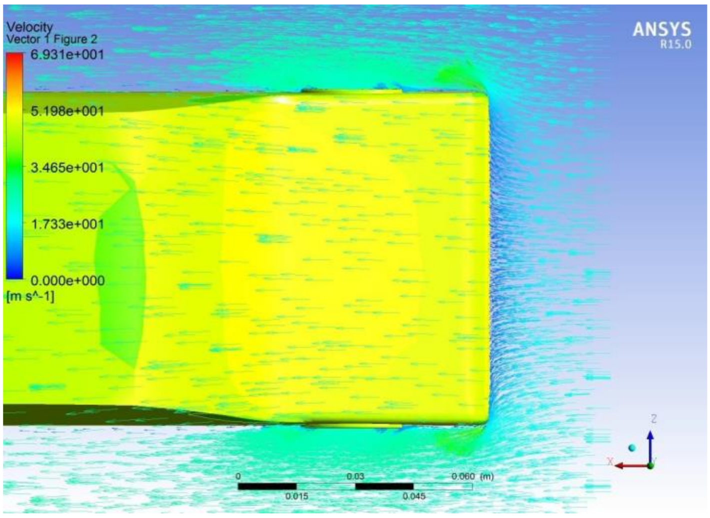

**Fig.-16.** The velocity vector recirculation zone of truck and car.

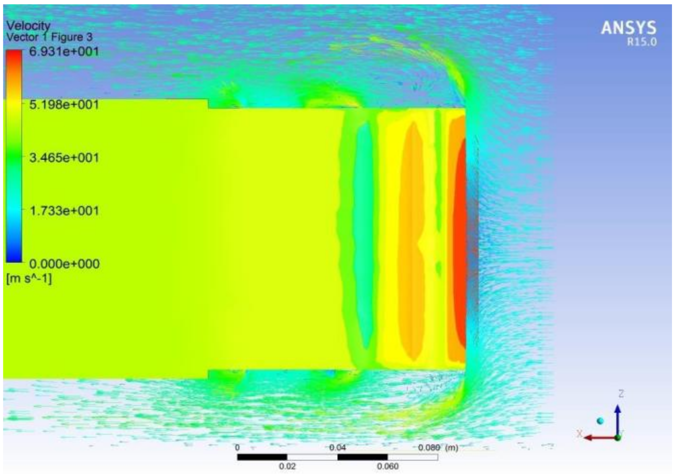
The recirculation of vortex is creating some drag. Fig 16 shows the velocity vector recirculation zone of truck and car. Using this diffuser (aero kit) we reduce the drag also, by using diffusers we reduce the recirculation velocity.

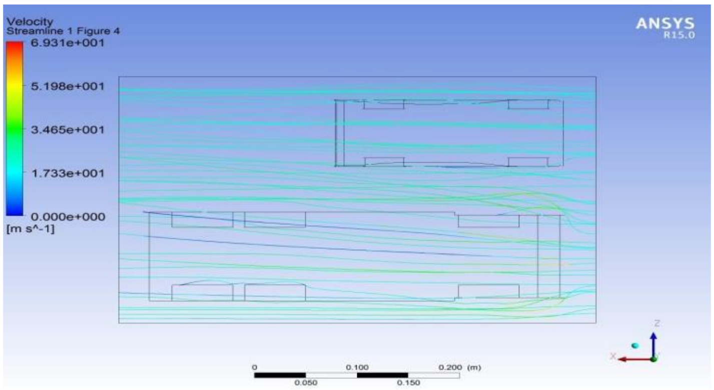
## 5. ANSYS REPORT

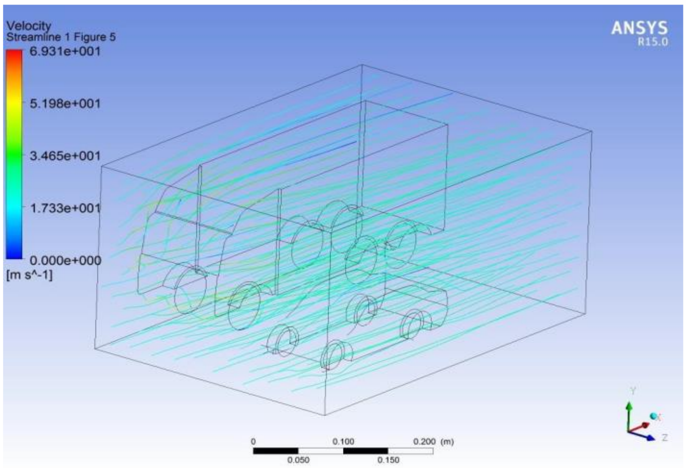


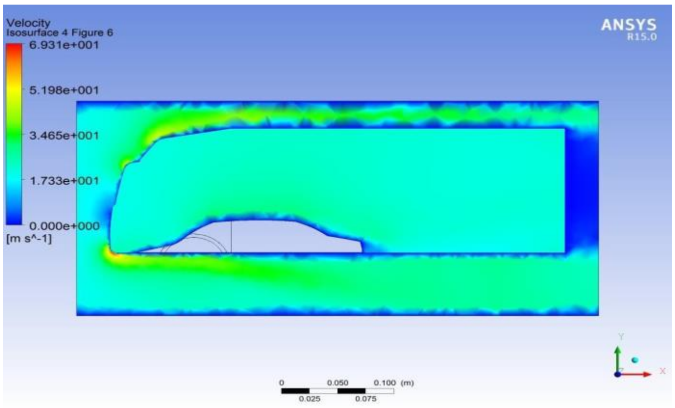



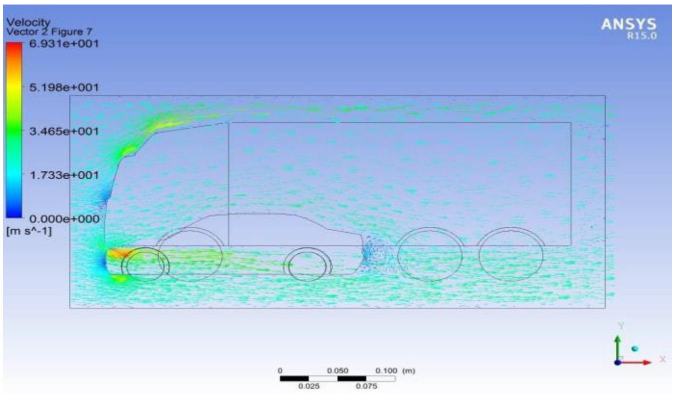



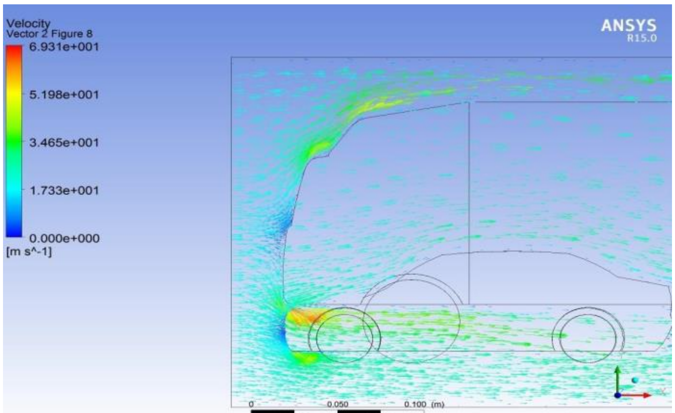



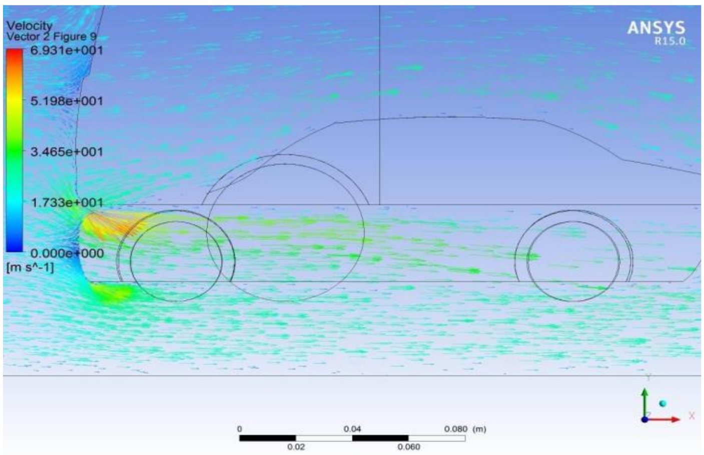






## 6. CONCLUSION

Thus, a novel approach is taken to analyze and understand the hydrodynamic behavior when the car travels around the truck. Various information regarding flow separation, localized suction, turbulence nature is understood from the analysis. By using the diffuser which is placed at the side way of heavy vehicle the flow separation is deviated automatically, and gives the effect in reduces of wake region. When the wake region is reduced means automatically the drag also reduces. This information can be very much used to avoid or delay the point of separation which can be used to increase the stability of driving. This reduces the possibility of accident at the same time reduce the fuel consumption. These are the models which are used in overtaking. This project is shown the flow separation in overtaking. According to this project, we clearly understand the point where we get the stability of driving and decrement of fuel consumption.

## ACKNOWLEDGEMENT

I take this opportunity to thanks **Prof. P.S. Bajaj** for his valuable guidance and for providing all the necessary facilities, which were indispensable in completion of this work. First of all, I am thankful to **Prof. A. V. Patil** (H.O.D., Mechanical Engineering Department.) and also thankful to **Pri.Prof. Dr. R B Barjibhe** (Dean, Academics & Administration) to give us presentation facility.
I am also thankful to all staff member of the mechanical Engineering Department. I would also like to thank the college for providing required journals, books and access to the internet for collecting information related to the project. Finally, I am also thankful to my friends and well-wishers for their valuable comments and suggestions.
.

## REFERENCES

1. **U.S. Department of Energy,** Transportation Energy Data Book, Oak Ridge, Tennessee, 2012.
2. **W.-H. Hucho,** The Aerodynamics of Road Vehicles, 1988.E.
3. **Mu Wang1, Qiang Li, Dengfeng Wang, Changhai Yang, Jinlong Zhao, Guijin Wen,** "The Air-deflector and the drag: A Case Study of Low Drag Cab Styling for a Heavy Truck," Jinan, 2009.
4. **R. Drollinger,** Heavy duty truck aerodynamics, Society of Automotive Engineers, 1987.
5. **R. Wood,** "A Discussion of a Heavy Truck Advanced Aerodynamic Trailer System," 2010.
6. **J. Eaton,** "Session 26: Computational Fluid Dynamics," Galway, 2013.
7. **F. White,** Fluid Mechanics, WCB McGraw-Hill, 2005.
8. **Ansys,** "Ansys Advantage Volume 5 issue 1," 2011.
9. **M. Lanfrit,** "Best practice Guidelines for Handling Automotive External Vehicle Aerodynamics with FLUENT," Darmstadt, 2005.
10. **F. Browand,** "Reducing Aerodynamic Drag and Fuel Consumption," 2011.

11. **F.-H. Hsu,** "Drag Reduction of Tractor-Trailers Using Optimized Add-On Devices," American Society of Mechanical Engineers, Los Angeles, 2010.
12. **Subrahmanya Veluri, Christopher Roy,** "Joint Computational/Experimental Aerodynamic Study of a Simplified Tractor/Trailer Geometry," American Society of Mechanical Engineers, Blacksburg, 2009.
13. **T. Favre,** "Aerodynamic Simulations of ground vehicles in Unsteady Crosswind," 2011.
14. **F.-H. Hsu,** "Design of Tractor-Trailer Add-On Drag Reduction Devices Using CFD," Los Angeles, 2009.
15. **Li Song, Zhang Jicheng**, Liu Yongxue, Hu Tieyu, "Aerodynamic Drag Reduction Design of van Body Truck by Numerical Simulation Method," in Second International Conference on Digital Manufacturing & Automation, Changchun, 2011.