CFD ANALYSIS ON A PASSENGER CAR WITH & WITHOUT SPOILER TO IMPROVE ITS AERODYNAMIC PERFORMANCE

A project report submitted in partial fulfilment of the requirement for

The award of the degree of

BACHELOR OF ENGINEERING

IN

MECHANICAL ENGINEERING

Submitted by

K. JOHNY	(314126520199)
V. ANIL KUMAR	(314126520163)
K. SHANMUKHA RAO	(314126520192)
P. SANTHOSHAMMA	(314126520174)
S. GULSHANUDDIN	(314126520145)

Under the guidance of

Mr. B. ROOPSANDEEP
Assistant professor

DEPARTMENT OF MECHANICAL ENGINEERING

PRAGNANAM BRAHMA
ANITS

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES

(Permanently Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with 'A' grade)

Sangivalasa - 531162, Bheemunipatnam (Mandal), Visakhapatnam (Dist.), AndhraPradesh, India. 2018

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
(Permanently Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with 'A' grade)
Sangivalasa, Bheemunipatnam, Visakhapatnam, A.P.

PRAGNANAM BRAHMA
ANITS

CERTIFICATE

This is to certify that this project report entitled "CFD ANALYSIS ON PASSENGER CAR WITH &WITHOUT SPOILER TO IMPROVE ITS AERODYNAMIC PERFORMANCE" has been carried out by K.Johny(314126520199), V. Anil Kumar(314126520163), K. ShanmukhaRao(314126520192), P. Santhoshamma(314126520174), S. Gulshanuddin(314126520145) under the esteemed guidance of Mr. B. Roopsandeep, in partial fulfilment of the requirements for the award of "Bachelor of Engineering" in Mechanical Engineering of Andhra University, Visakhapatnam.

APPROVED BY:	PROJECT GUIDE:
Prof.B.NAGARAJU	Mr.B. ROOPSANDEEP
Head of the Department	Assistant Professor
Department of Mechanical Engineering	Department of Mechanical Engineering
ANITS	ANITS
SangivalasaSangivalasa	
Visakhapatnam.Visakhapatnam.	

Signature of HOD **Signature of Internal Guide**

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

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER:

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

EXTERNAL EXAMINER:

ACKNOWLEDGEMENT

On the submission of our project report entitled "CFD analysis on Passenger car with & without spoiler to improve its aerodynamic parameters" we would like to give our heartiest thanks and gratitude to Mr.B.Roopsandeep, Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, for his continuous motivation and constant support and guidance throughout the past year.

We are very thankful to Prof.T.V.HanumanthaRao, Principal and Prof.B.Nagaraju, Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for their valuable support and facilities.

We would like to thank the technical staff of fluid machinery lab for their continuous cooperation and their guidance in helping us to understand the technical details of hydraulic machines in the lab.

Finally, we would like to convey our thanks to everyone, who have contributed directly or indirectly for the completion of this project work.

K. JOHNY	(314126520199)
V. ANIL KUMAR	(314126520163)
K. SHANMUKHA RAO	(314126520192)
P. SANTHOSHAMMA	(314126520174)
S. GULSHANUDDIN	(314126520145)

ABSTRACT

In the present scenario, the demand of a high speed car is increasing in which vehicle aesthetics is of major concern. One of on-going automotive technological development is in field of aerodynamic drag reduction, because it has direct impact on fuel consumption. We are familiar to know that aerodynamic plays an important role while designing any automotive. Due to this, the entire performance of the automotive will be changed. In this work, attention is given only external car design which is in direct contact with the atmospheric entities.

The present work aims to reduce the drag force which improves fuel utilization and protects the environment as well.

So we are working on a passenger car with spoiler and without spoiler. The spoiler is used to reduce the aerodynamic drag force. The design of passenger car with and without spoiler has been done on SOLIDWORKS 2017 and the model is imported to ANSYS 14.5 in that we used ANSYS FLUENT for the CFD analysis.

The analysis is done for finding out drag and lift forces at different highway speeds such as 80kmph, 110kmph and 140kmph. The study of this work proposes an effective passenger car based on CFD approach to obtain the drag and lift forces of the car with and without spoiler that is concerned.

The results obtained in the form of forces and coefficients are compared. It has been observed that the passenger car model with spoiler had lower drag and lower lift while for without spoiler has more lift and more drag.

TABLE OF CONTENTS

TABLE OF FIGURES

1 INTRODUCTION

The performance, handling and comfort of an automobile are significantly affected by its aerodynamics properties. A low drag is a decisive prerequisite for good fuel economy. Increasing fuel prices and stringent legal regulations ensure that this long-established relationship becomes more widely acknowledged. But the other aspects of vehicle aerodynamics are no less important for the quality of an automobile such as side wind stability, wind noise, soiling of the body, the lights and the windows, cooling of the engine, the gear box and the brakes, and finally heating and ventilating of the passenger compartment all depend on the flow field around and through the vehicle.

In the study of road vehicle aerodynamics design, the key concepts are the lift and drag components. A road vehicle must produce down force in order to stay on the road. There are two ways of improving the vehicles down force. The first method is by increasing the car weight and second is by adding aerodynamics aids.

Aerodynamics aids are the components added to the vehicle in order to modify its aerodynamic properties. Aerodynamics aids such as spoilers and vortex generator are basically used to improve the vehicles performance without modifying the basic design of the vehicle itself. Today, almost all road vehicles are installed with aerodynamic aids. Among others, spoilers are one of the famous accessories for road vehicles. Even the new vehicles nowadays are already equipped with aerodynamics aids during manufacturing. However, some spoilers were merely installed for decoration purpose despite of its aerodynamics benefits.

Nowadays, with the high level of CAD prediction and pre-production evaluation coupled with a greater human understanding of aerodynamics, wind tunnel testing often comes into the design process later. The wind tunnel is the proving ground for the vehicle's form and allows engineers to obtain considerable advanced information within a controlled environment.

The automotive industry uses the CFD package which is a commercial tool. The CFD is not only used for the improvement of vehicle aerodynamics, but also for optimization of domains 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 vehicle body. The CFD is best way for the designers to obtain the results in a short interval of time.

The CAD software SOLDWORKS is used for designing the exterior shape of the body. The mesh generation and the analysis are done by using the ANSYS CFD analysis software to find the Drag & Lift Coefficients.

SPOILER:

A spoiler is an aerodynamics device attached to an automobile which its intended design function is to 'spoil' unfavourable air movement across a body of a vehicle to minimize the effect of lift force; spoiler is mounted at the rear of the car. Spoilers work by blocking the flow of air as it comes off the rear wind screen. This creates a high pressure area in front of the spoiler and therefore more down force.

Different angles of attack of spoilers can give different effects to the car. The objective is CFD analysis of a passenger car with and without spoiler. Analysis has been made with rear spoilers at different speeds of a basic rectangular shape using CFD. Three different speeds were chosen for the spoiler which is 80kmph, 120kmph and 140 kmph.

1.1 HISTORY AND EVOLUTION OF AERODYNAMICS

Aerodynamics is one of the branches of dynamics that concerned with study of the flow motion of air over the bodies. It is the sub-field of gas dynamics and fluids and the word "aerodynamics" is used often when referring to the gas dynamics.

The road vehicle aerodynamics is the study of aerodynamics that focuses the concepts of wind noise and drag reduction. It also minimizes noise emission and prevents the undesired lift forces causing instability of aerodynamics at higher speeds. In some cases of the racing vehicles, it is so important to produce the downforceto improve the traction and thus the cornering abilities The studies on the aerodynamics had originated from the applications of aeronautics and marine by the Hucho in 1998 and with respect to Barnard, substantial progress on the aerodynamics of aircraft is obtained and the worth of research and analysis on it was being done. The study of road vehicle aerodynamics was first began to the surface during earlier part of 20^{th} century and had continued till today and the study of road vehicle aerodynamics is associated with the performance of the vehicles. Aerodynamicists carried out research on the road vehicle aerodynamics with a target to produce vehicles which can achieve a high speed limit to the power ratio. To achieve the performance of high road vehicles, much of this attention focused on lowering road vehicle coefficient of drag.

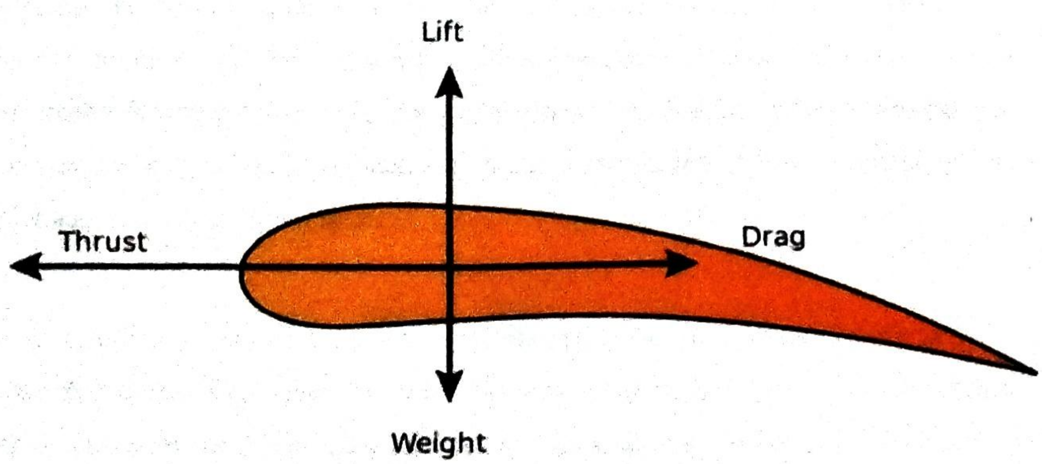


Fig-1.1 DIFFERENT TYPES OF FORCES ACTING ON A BODY

The trend that shifted again in early 1990's in the North America especially where the low fuel price coupled with increase in the popularity of sport-utility vehicles and light trucks and have a coefficient of drag which is around 0.45 and had reduced the importance of need on the research to reduce the coefficient of drag. George et al. 1997. And then aerodynamicists shifted their focus towards the designing of road vehicles which provides the maximum comfort. The vehicle comfort consists of fine-tuning areas such as heating, ventilation, air conditioning and minimizing the wind noise inside vehicle, Hucho, 1998.

The first passenger car was manufactured in the early 20th century and the attempt had been made to travel at the faster speeds. Before1950, the designers tried to make the cars as streamlined to make it easier for engine and restricted the interior layout of the car. After 1950, the aerodynamic drag raised up because of the cars that are becoming more eco and family friendly and so the consequence of that shapes which are available in making the possibility to keep the drag levels of aerodynamics at very low level. The rectangular shape that made the cars most purposeful for family and also it is so fair to speak that after the 1950 and the designing of cars is to aid lifestyle of the larger families.

The friction force of the aerodynamic drag that increases the vehicle speed significantly and in the early 1920s, the engineers had begun in considering the shape of automobiles in reducing the aerodynamic drag at limits of higher speeds. And by 1950s, the automotive engineers of German and British are analysing effects of automotive drag systematically for the vehicles of higher performances. And in late 1960s, scientists had become the aware of significant increase in the sound levels that are emitted by the limits of automobiles at higher speeds.

If the car had poor aerodynamics which make the engine to perform more work to travel the same distance as a passenger car with the best aerodynamics, so if engine performs harder that requires the more fuel which allows the engine to perform the work and so the passenger car with best aerodynamics which uses less fuel than one another.

In present days, most of the cars that are manufactured aerodynamically, by designing the car aerodynamically and then reduces the friction which it encounters and the power required to overcome will be less and fuel will be saved. In modern era of road vehicle aerodynamics where the resources of the fuel are quick and fast depleting all the efforts that are used to find the alternate energy sources or to save current resources or to minimize the use of current fuel resources. So, in present days the aerodynamics those are given much importance and as everyone like to have fuel-efficient car, stylish and good looking.

Some of the passenger cars that had a good drag coefficient from past decades to till today.

From most of the 50s, 60s and in early 70s, the automotive aerodynamicists are mostly non-existent. The enthusiasm and original promise of aerodynamics is discarded as another style fad, and also gave way to the less functional styling gimmicks that are tacked unto ever larger and even the squarer bricks

1.2 BACKGROUND INFORMATION

The road vehicle pushes the air away which surrounds it during its motion by giving a result that the body is subjected to drag which is also called fluid resistance or air resistance refers to the forces that opposes the vehicles motion. Around 55% to 60% of power is required to cruise at the speeds of highway is considered to overcome the aerodynamic drag which increases quickly at greater speeds. The vehicle experiences more aerodynamically and also aesthetically, if the exterior design of the vehicle reduces the drag which makes the engine to work less and leads to the usage of less fuel becoming better fuel-efficient. The aerodynamic effectiveness of a vehicle is measured computationally by the engineers who use a commercial tool known to be computational fluid dynamics where we can study the flow of fluid behavior over the vehicle.

Fig -1.2AERODYNAMIC SIMULATION ON A CAR

1.3 Problem Statement

The decrease in fuel consumption of the road vehicles, due to the environmental and the arguments reasons of selling, concerns the manufacturers of cars. The aerodynamic improvement of the car shapes consequently and more precisely the drag coefficient reduction that became one of the major topics in the research sectors of automotive industries. By designing a new vehicle with a minimum Drag resistance that provides the economical and the performance advantages. The decreased resistance to the forward motion allows the limits of higher speeds for same power output or the lower power output for the limits of same speeds.

The main target for drag resistance reduction is:

- Better fuel consumption
- Increase in the performance.

In focusing the target to be succeeded, to study the aerodynamic properties of a sedan with & without spoiler. The study is to bring a new model, which supports in reducing the drag that leads to fuel-efficient car. This project is about the study, modelling flow analysis of aerodynamic profile of a passenger car with & without spoiler. This project is concerned with brief study and calculations of lift forces, drag forces and coefficients respectively.

2 LITERATURE SURVEY

Aerodynamics is the study of forces and the resulting motion of objects such as airplanes, cars and motorcycles through the air. The forces of aerodynamics will be generated when there is a flow of air over the body and these forces, which include pressure distributions and shear stress distributions over the body. Studying the motion of air around a body allows us to measure the forces of lift, which acts against gravity, and drag, which is the resistance the body "feels" as it moves through the air. This is due to which the theory Bernoulli says, the flow of high speeds on the area makes low pressure on that area and flow of low speeds on the area makes the area have high pressure.

Simon Watkins et al[1], has conducted tests at RMIT Industrial Wind Tunnel, a closed -jet and fixed - ground type, having a 3m wide, 9m and 2m high long test section to study the effects of vehicle spacing on the aerodynamic shape of a car. For this test, which includes two co-linear, Ahmed bodies are used in wind tunnel. Based on vehicle length, inter spacing is varied from 0.1 to 4.0. A 2:1 contraction is located before the test section. The free stream turbulence intensity was 1.8% and the blockage ratio is less than 2% at a speed of 35 m/s.

R. B. Sharma [2], developed the model of generic passenger car in the solid works-10, then generated the wind tunnel, and finally, applied the boundary conditions in ANSYS workbench 14.0 platform. Then, tests and simulations had been performed for the evaluation of drag coefficient for the passenger car model. In the other case, a more suitable design of spoiler is introduced to enhance its aerodynamics and then analysed for the evaluation of the drag coefficient for the modified passenger car model.

Dan BARBUT and**EugenMihai NEGRUS [3]**, studiedtheinfluence the lower part of the road vehicles on the global characteristics of the drag. The overall drag was reduced by redesigning the lower parts of the road vehicles that had a potential of about 20% in the breakdown of entire drag, majorly due to the effects of viscous forces and the interactions of the fluids under the car body with the flow patterns of the vehicle typically the blunt body or the bluff body. CFD analysis is most probably the only commercial efficient tool in a way to assess the design of the specific parameterization of the generic car shape, for complex interaction. The research work targeted on the sedan car configurations, and presented the examples of successful design strategies. The CFD results of this work gave possible strategies that are used in a way to reduce the global drag and the viscous drag characteristics that are proposed. The main objectives of this analysis were demonstrated and the importance of this CFD analysis in the car optimization considered the flows under the car. This is the only commercial and available tool for such kind of analysis with the potential that introduced the important changes majorly to the strategy of optimization in industry of the car.

C.J. Baker [4], presented the investigation results for the aerodynamic forces and moments on Lorries and railway containers in high cross winds that were obtained in a variety of different wind tunnel simulations. The data by which means extreme side and lift force coefficients are mainly focused. In his experiment different wind tunnel simulations can be determined, which in particular the viability of the "moving model rig" type of apparatus is considered and 1/25th scale model was used in a ground plane in the low turbulence wind tunnel. Also three different types of containers were considered in this experiment.

Based on the results of the experimental study, conclusions that can be drawn:

The lift force coefficients are largely dependent upon the nature of the wind tunnel test. It would observe that high Reynolds number tests are required to consider a conservative, which if not necessarily realistic, the upper bound. Therefore, there is a possibility that requires Reynolds number would be reduced by the addition of turbulence to the flow.

Mustafa Cakir[5], studies the aerodynamic effects of a rear wing/spoiler on a passenger vehicle using CFD.One of the design goals of a spoiler is to reduce drag and increase fuel efficiency. Adding a spoiler at the very rear of the vehicle makes the air slice longer, gentler slope from the roof to the spoiler, which helps to reduce the flow separation. Reducing flow separation decreases drag, which increases fuel economy, it also helps keep the rear window clear because the air flows smoothly through the rear window. The limitations of conventional wind tunnel experiment and rapid developments in computer hardware, considerable efforts have been invested in the last decade to study vehicle aerodynamics computationally. This thesis will present a numerical simulation of flow around racing car with spoiler positioned at the rear end using commercial fluid dynamic software ANSYS FLUENT. The thesis will focus on CFD-based lift and drag prediction on the car body after the spoiler is mounted at the rear edge of the vehicle. A 3D computer model of 4-door sedan car (which will be designed with commercial software SolidWorks will be used as the base model. Different spoilers, in different locations will be positioned at the rear end of vehicle and the simulation will be run in order to determine the aerodynamic effects of spoiler.

Mahmoud Khaled et al [6], the work that he focused on a parametric analysis of the trends in the aerodynamic forces. In his experiment work aerodynamic force measurements were carried out on a simplified vehicle model. Tests that are performed in wind tunnel S4 type. The simplified model consists flat and flexible air inlets and several types of air outlet, and includes in its body a cooling system and a simplified engine block moves in the longitudinal and lateral directions. The aerodynamic force measurements that are carried out at a wind speed of 30m/s.

The results of his research showed that configurations in which the overall drag coefficient decreased by 2% and the aerodynamic cooling drag coefficient by more than 50% and the lift coefficient by 5%. The aerodynamic drag is independent of its position of the air outlet in the wheel arch and placing this outlet at the rear of the wheel arch that decreases the lift force.

The drag increased were found for the rear Ahmed body for spacing of 0.1-1.0, when compared to the drag of the body isolation and for greater spacing, the drag of the rear body falls down the value of the isolated case, up to the maximum spacing that is considered. The lift coefficient of the rear body was also found to be very sensitive to spacing which the effect of the strong vortex system arising from the rear slant was the cause of the drag and lift that changes of the rear vehicle.

JOHAN ZAYA [7], in this project the author has been carried out as a Master Thesis together with the Volvo Cars and the Chalmers University of Technology, with a close relationship and Fluent'sAdjoint Solver that has been tested on its computation robustness, computer requirements, abilities and functionality. These tests are done on four different vehicle models that are provided by the Volvo Cars and the simulations are computed in the wide variations of different case studies and setups. Based on these results from the simulations which conclude that the Fluent Adjoint Solver which is at the moment not at the stage where it is ready to be incorporated as a part of the development processes. It is proved that one can achieve the valuable engineering insight which surely can be improved the development process, and moreover, for the external aerodynamics which the Adjoint Solver is not yet ready.

Chien-HsiungTsai[8] proposes an effective numerical model based on the Computational Fluid Dynamics (CFD) approach to obtain the flow structure around a passenger car with wing type rear spoiler. The topology of the test vehicle and grid system is constructed by a commercial package, ICEM/CFD. FLUENT is the CFD solver employed in this study. After numerical iterations are completed, the aerodynamic data and detailed complicated flow structure are visualized using commercial packages, Field View and Tecplot. The wind effect on the aerodynamic behavior of a passenger car with and without a rear spoiler and endplate is numerically investigated in the present study. It is found that the installation of a spoiler with an appropriate angle of attack can reduce the aerodynamic lift coefficient. It is clear that the vertical stability of a passenger car and its noise elimination can be improved. Finally, the aerodynamics and aero-acoustics of the most suitable design of spoiler is introduced and analysed.

Xingjun HU et al [9], Researcher's focused on the effects of under body rear diffuser aerodynamic add on device on the sedan car. Diffuser is one of the main important aerodynamic devices often found in F1. Often, it is used to reduce lift for race cars. It is widely used in case of ordinary cars. In this Researchers works on the influence of diffuser angle that was investigated without separator and end plates. The method of CFD was adopted to study the aerodynamic characteristics of sedan with a different diffuser angles. He has found that when diffuser angle increases, the under body flow and wake formation change greatly also pressure change correspondingly and which the total aerodynamic drag coefficients of car first decreases and then increases, while the total aerodynamic lift.

J.P. Howell [10],has reported basing on pressure distributions over the external surfaces of a full scale car that has been measured and the results were used to derive the distributions of side force and yawing moment components. The tested vehicle that has an early model Rover 825, an executive class saloon car with the frontal area 2.03 m^2.The car which had an initial style bumpers with short front and rear overhangs. Car was tested at the MIRA Full scale Wind Tunnel, considering the normal operating wind speed is 28 m/s. A saloon car shape, which has the largest contribution to the overall side force comes from the A pillar region and with the rear location of the moment centre at all the surfaces form the A- pillar forward that contributed to the overall yawing moment, although it is the nose region that dominates. As the moment centre moves forward to the influence of all regions rather than the extreme nose is diminished. At most transverse sections the side loading is concentrated on the upper side surfaces.

Bhagirathzala [11],has reported this thesis on aerodynamics of sedan and hatchback car models by comparing two types of models experimentally and computational fluid dynamics (CFD). The experimental investigations that were performed on an open circuit suction type wind tunnel which has a test section of 30 cm x 30 cm x 100 cm and a maximum speed of 33 m/s on a geometrically reduced scale of (1:20) Al car models. The three dimensional computational analysis that were carried out using with the help of software tools like ANSYS-CFX to simulate the flow of air around the automobiles such as ANSYS-CFX, CFD code that were used to run the simulations. Computer desktop with 3.2 GHz Intel core 2 duo with ram of 2.00GB is used to create and to simulate the flows

The investigations that were showed as sedan car produced better aerodynamic when compared to hatchback and got drag coefficient for sedan is lower than hatchback about 19%. Sedan is more stable than hatchback by the results as sedan indicates the lower value of CL about 33 % than hatchback. Sedan is more streamlined when compared to hatchback as the flow of air for hatchback detached from the car surfaces earlier than sedan. Drag force of hatchback car got computationally higher value than experimentally about 12.5% whereas drag force of sedan car obtained computationally higher value than experimentally about 11%.

R.B.Sharma and **Ram Bansal [12]**, proposed an effective numerical model that based on the Computational Fluid Dynamics approach, which is to obtain the flow structures over the passenger car with the Tail Plates. The experimental work of the vehicle that is to be tested and also the grid system is constructed by the ANSYS-14.0. FLUENT. the CFD solver is employed in the present work which the numerical iterations are completed and then after the aerodynamic data and the detailed complicated flow structures are visualized. In the present research work, generic model passenger car is developed in solid works-10 and is generated in the wind tunnel and then applied the boundary conditions in the ANSYS workbench 14.0 platform after testing and simulations are performed for the evaluation of coefficient of drag for the passenger car. In other case, the aerodynamics of the suitable design of tail plate that is introduced and analyzed for the evaluation of coefficient of drag for the passenger car.

3 INTRODUCTION TO SOLIDWORKS

The SOLIDWORKS CAD software is a mechanical design automation application that letsdesigners quickly sketch out ideas, experiment with features and dimensions, and producemodels and detailed drawings.

SolidWorks is a solid modelling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. It helps to design various products and services, testing them in very cost effective way like Model and prototype testing as you learned earlier in engineering degree class.

It provides better design visualization, design better products, faster design iterations, improved communications, and design with fewer errors; create more aesthetic design &products.

3.1 Concepts

Parts are the basic building blocks in the SOLIDWORKS software. Assemblies contain parts or other assemblies, called subassemblies.

A SOLIDWORKS model consists of 3D geometry that defines its edges, faces, and surfaces.

The SOLIDWORKS software lets you design models quickly and precisely. SOLIDWORKSmodels are:

- Defined by 3D design
- Based on components

3.1.1 3D Design

SOLIDWORKS uses a 3D design approach. As you design a part, from the initial sketch to the final result, you create a 3D model. From this model, you can create 2D drawings or mate components consisting of parts or subassemblies to create 3D assemblies. You can also create 2D drawings of 3D assemblies.

When designing a model using SOLIDWORKS, you can visualize it in three dimensions,

the way the model exists once it is manufactured.

3.1.2 Component Based

One of the most powerful features in the SOLIDWORKS application is that any change

you make to a part is reflected in all associated drawings or assemblies.

3.1.3 File format

SolidWorks files (previous to version 2015) use the Microsoft Structured Storage file format. This means that there are various files embedded within each SLDDRW (drawing files), SLDPRT (part files), SLDASM (assembly files) file, including preview bitmaps and metadata sub-files. Various third-party tools can be used to extract these sub-files, although the sub files in many cases use proprietary binary file formats.

3.1.4 Applications:

The Solid works have wide range of applications in industries such as

1. Aerospace
2. Defence
3. Automotive
4. Transportation
5. Machinery
6. Heavy Equipment
7. Consumer products
8. Mould & Tools design
9. Electronics
10. Sheet metal work
11. Process Plant

12. Energy conservation

13. Construction

14. Medical tools

15. Product design and other engineering services.

3.2 Efficient 3D design

SOLIDWORKS is an easy to use parametric design modular, meaning you can easily edit the design at any stage in the design process. Real View graphics allow you to visualise your design in real time whilst Photo View 360 can create sophisticated photo realistic renderings and animations. Both tools will give you a fantastic insight into the way your design will look without it actually being made and can be a powerful asset when presenting your work to customers. You can look at each individual part of the design, see accurate mass properties and check for interference, meaning that you won't have to build/manufacture the product before you see any errors, saving time and money and reducing the number of prototypes needed. All of this will speed up the whole process of design as you know it and increase productivity.

3.3 Built-In Applications

SOLIDWORKS is a very productive 3D CAD software tool, with its integrated analytical tools and design automation to help stimulate physical behaviour such as kinematics, dynamics, stress, deflection, vibration, temperatures or fluid flow to suit all types of design.

Organisations with more than just a few designers can utilise product data management (PDM) software, which fully integrates with SOLIDWORKS. PDM systems can do much more than just store and organize files. They can help designers find existing parts to re-use instead of reinventing them, saving many man hours. PDM systems also generate material lists for cost estimating and feed data to manufacturing resource planning (MRP) systems. More advanced PDM software can automate change-control processes to assure that out-of-date or unreleased information is not sent to factories or suppliers.

4 INTODUCTION TO FLUENT

4.1 ANSYS Fluent

ANSYS Fluent software is the most-powerful computational fluid dynamics (CFD) tool available, empowering you to go further and faster as you optimize your product's performance. Fluent includes well-validated physical modelling capabilities to deliver fast, accurate results across the widest range of CFD and multi physics applications. ANSYS Fluent is a powerful and flexible general-purpose computational fluid dynamics software package used to model flow, turbulence, heat transfer, and reactions for industrial applications. The physical models allow accurate CFD analysis for a wide range of fluids problems - from airflow over an aircraft wing to combustion in a furnace. ANSYS Fluent is integrated into ANSYS Workbench.

ANSYS Fluent contains the broad physical modelling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications

4.2 Prerequisites

- A technical education and background in fluid mechanics and heat transfer is recommended but not mandatory.

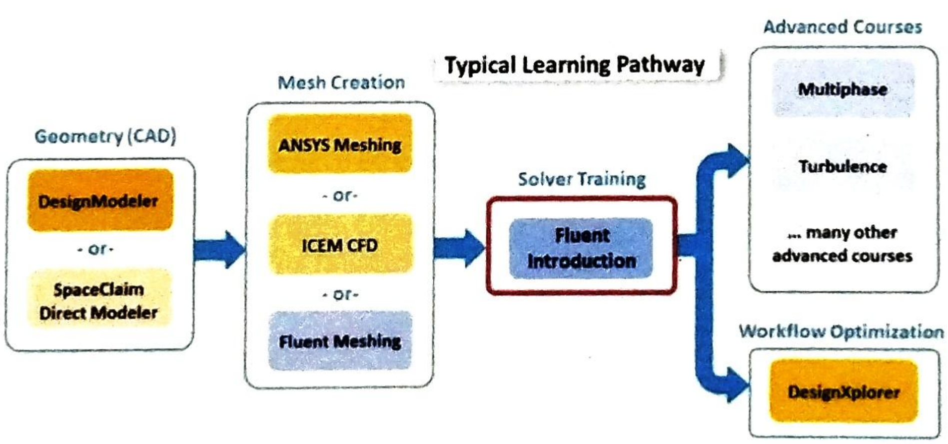


Fig-4.1 BLOCK DIAGRAM OF A FLUENT

4.3 Features

- Efficient and Flexible Workflow
- Built for Multiphysics
- Solve Complex Models with Confidence
- Go Faster with High Performance Computing (HPC)
- Turbulence Modelling
- Heat Transfer & Radiation
- Multiphase Flow
- Reacting Flow
- Acoustics
- Fluid-Structure Interaction
- Optimize Your Design - Automatically

5 GENERIC DESIGN OF A SEDAN CAR

A sedan is a passenger car with a three-box configuration A, B & Cpillars for accommodating engine, passenger & cargo respectively. The passenger compartment features two rows of seats and adequate passenger space in the rear compartment.

A generic design approach of a TATA INDICA car is taken to improve its parametrical efficiency &design aesthetics. Also, we aim to understand & differentiate the variation of aerodynamic performance of the car with the use of a spoiler.

Here, in this project work considered two different Passenger cars. They are

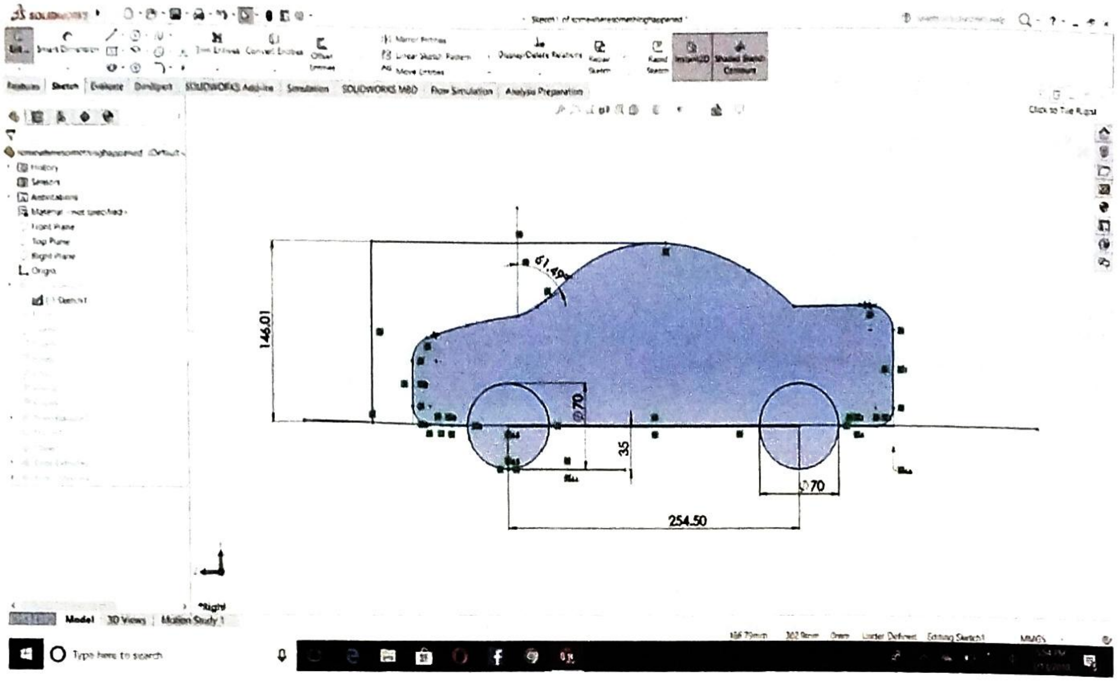
- Sedan car without spoiler
- Sedan Car with spoiler

5.1 SEDAN CAR WITHOUT SPOILER

The geometrical dimensions of the model that used to develop part modelling in Solid works 2017 are shown below.

Geometry	**Dimension**
Length	4194.9mm
Width	1500 mm
Height	1460.1 mm
Wheel Base	2545.0 mm
Ground Clearance	350 mm
Wind Shield Angle	61.1 degrees

Fig-5.1 THE ROUGH SKETCH OF A CAR DESIGN



- Development of 2-D view of the Car Body.

Fig-5.2THE ROUGH SKETCH OF A CAR DESIGN 2

- Development of 3-D view of the Car Body.

5.2 SEDAN CAR PARAMETERS FOR WITHOUT SPOILER:

For performing analysis on the body, the boundary conditions that given as input parameter is shown below.

Type of car	Without spoiler
Mesh Nodes	472445
Mesh Elements	1493668
Domain Type	Fluid
Model	Spalart: Allmaras(vorticity based)
Maximum skewness	0.89840700
Maximum face size	0.30m
Maximum size	0.30m
Use automatic inflation	Program controlled
Boundary Type	Inlet
Face sizing element size	0.02 m
Sizing	
Use advanced size function	On proximity and curvature
Relevance Centre	Fine
Transition	Slow
Number of cells across gap	10
Relative Pressure	0.0000e+00 [Pa]
Standard deviation	0.1203325

The Analysis is carried out at 80kmph in ANSYS-FLUENT and the results such as contours, vector plots, turbulent kinetic energy and streamline plots. The surface pressure contour is also observed in the analysis.

The analysis includes process of five steps such as

- Geometry
- Mesh

- Setup
- Solution
- Results.

Geometry: The part modelling of solid works file is saved in IGS format and then imported to ANSYSFLUENT

Mesh and Setup: The imported file geometry undergoes meshing then; the file is aligned to boundary conditions.

Solution and Results: After applying boundary conditions, the solution and results are as follows.

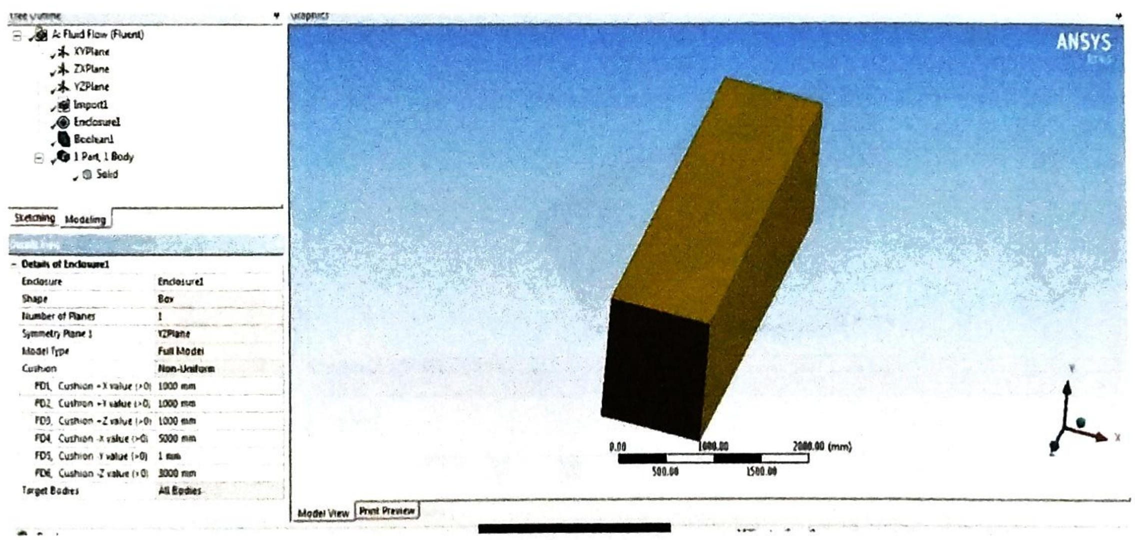


Fig-5.2.1 GEOMETRY AND ENCLOSURE

- This is the domain that we created to do the simulation on the car by flowing the fluid through it.

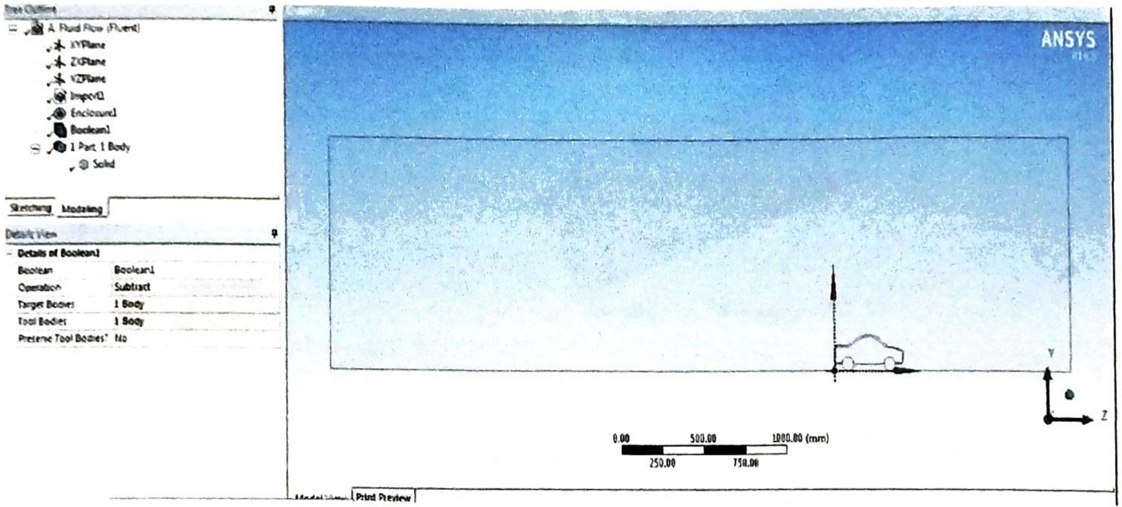


Fig 5.2.2 BOOLEAN

- In this, the Boolean operation is to separate the car body from the fluid flow.

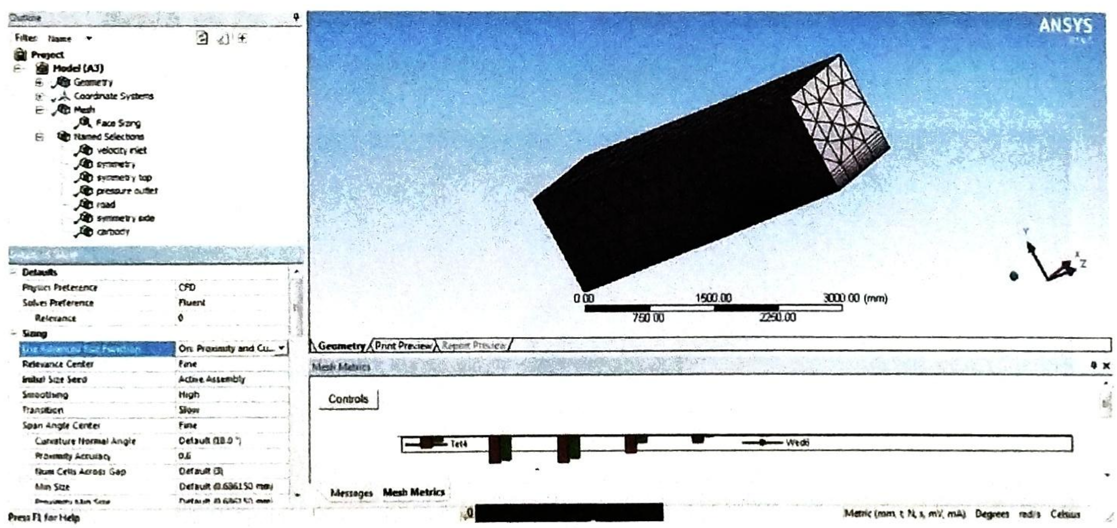


Fig 5.2.3 MESHING

- Meshing is the discretization process in which the whole part is divided into number of small parts to do the analysis in easy manner (i.e) converting the complex problem in to the simple one.

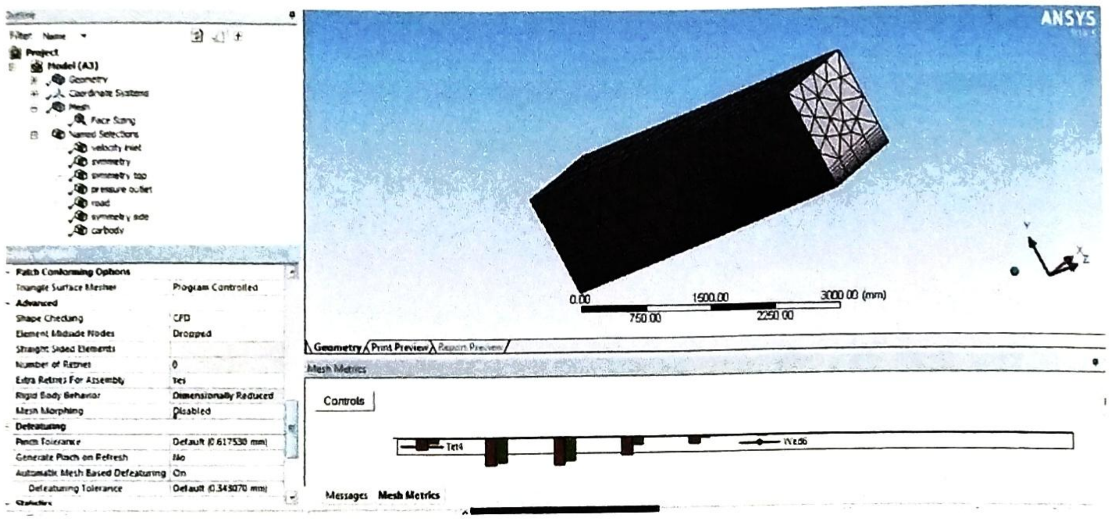


Fig 5.2.3 MESHING (2)

- In the meshing process, the skewness should be less than 93.000 to get the preferable good meshing. If the skewness is more than the given value then we should go for refining the meshing by selecting the places where the skewness quality affecting.

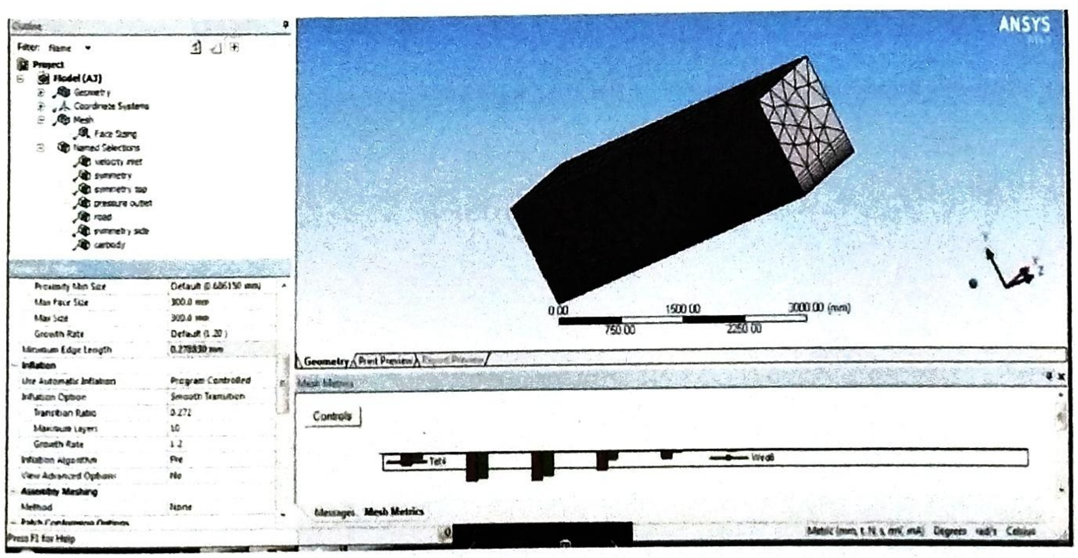


Fig 5.2.3 MESHING(3)

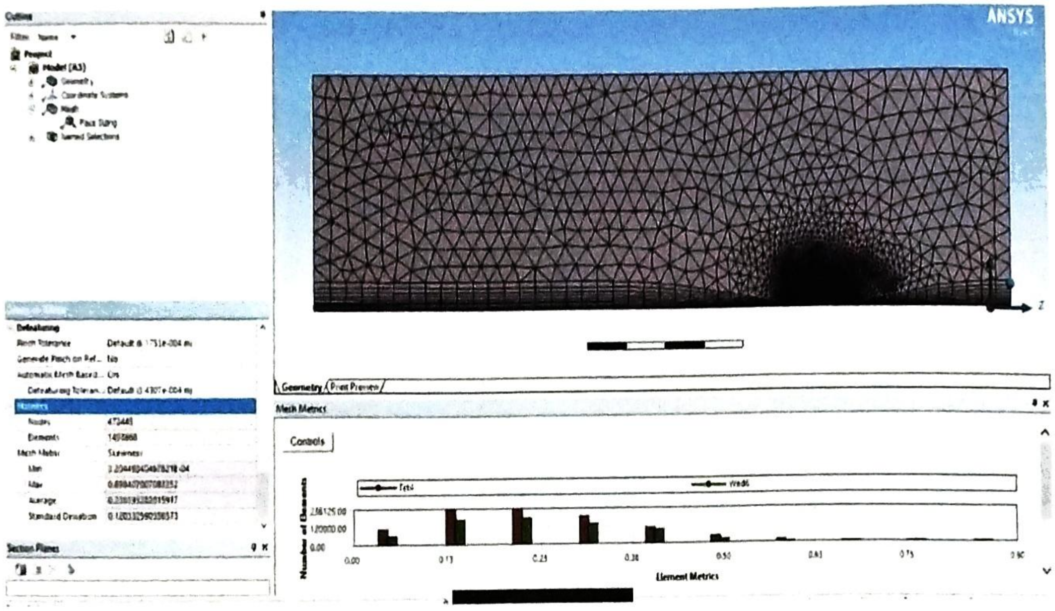


Fig 5.2.3 MESHING(4)

- Here we can see that the skewness is good enough to get the quality meshing.

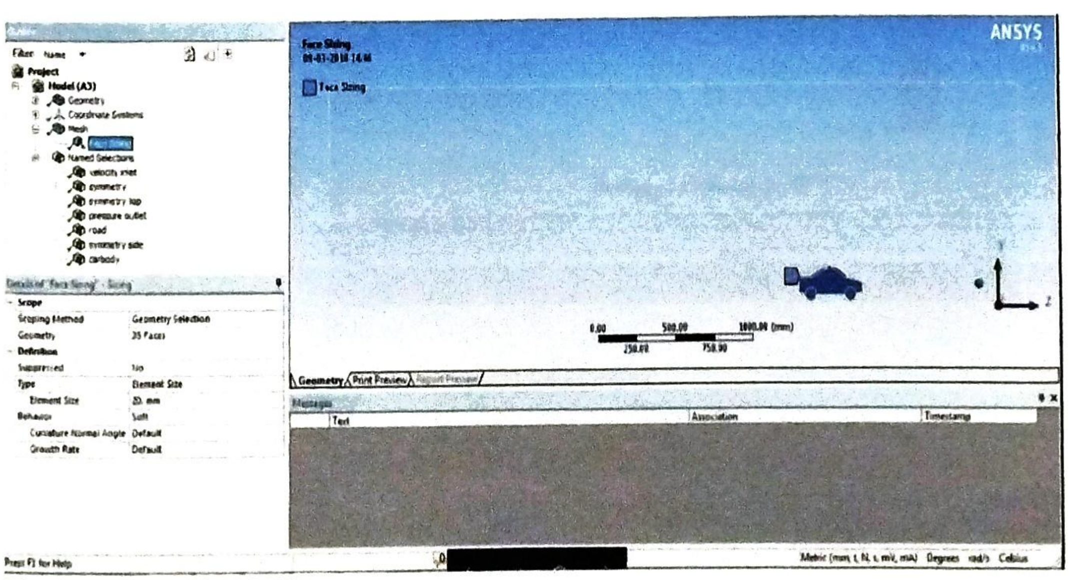


Fig 5.2.4 FACE SIZING

- Face sizing is the process in order to get the smooth curves at the corners.

5.3 NAMED SELECTIONS:

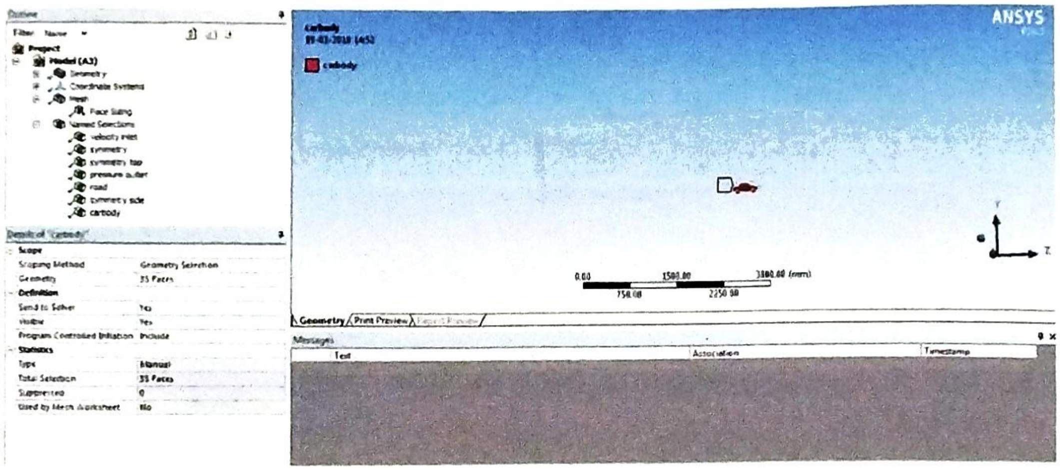


Fig 5.3.1 MAIN CAR BODY

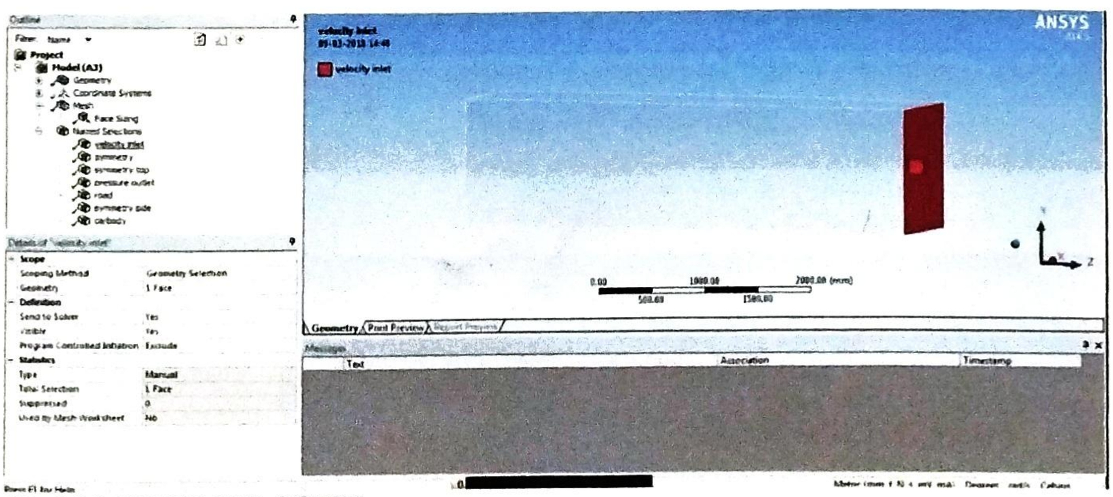


Fig 5.3.2 INLET VELOCITY

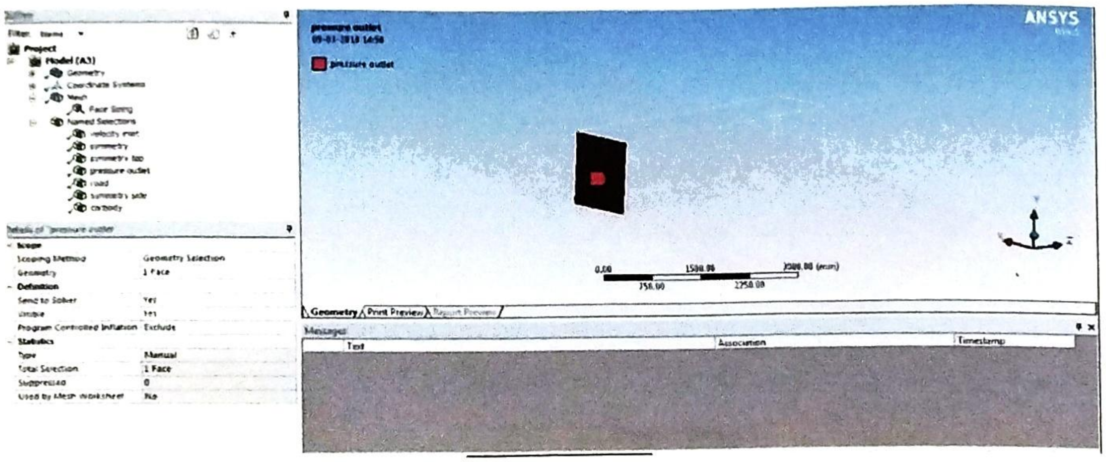


Fig 5.3.3 PRESSURE OUTLET

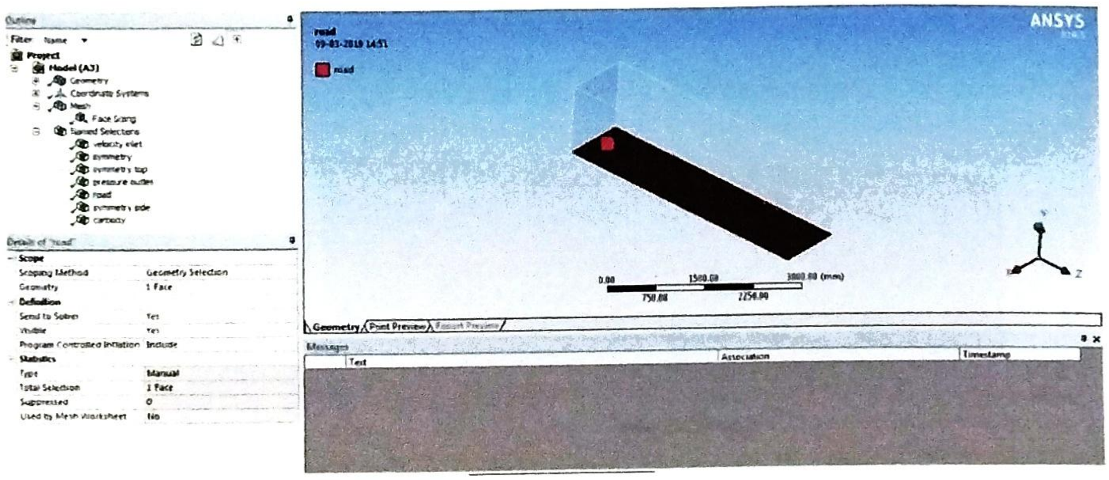


Fig 5.3.4 ROAD

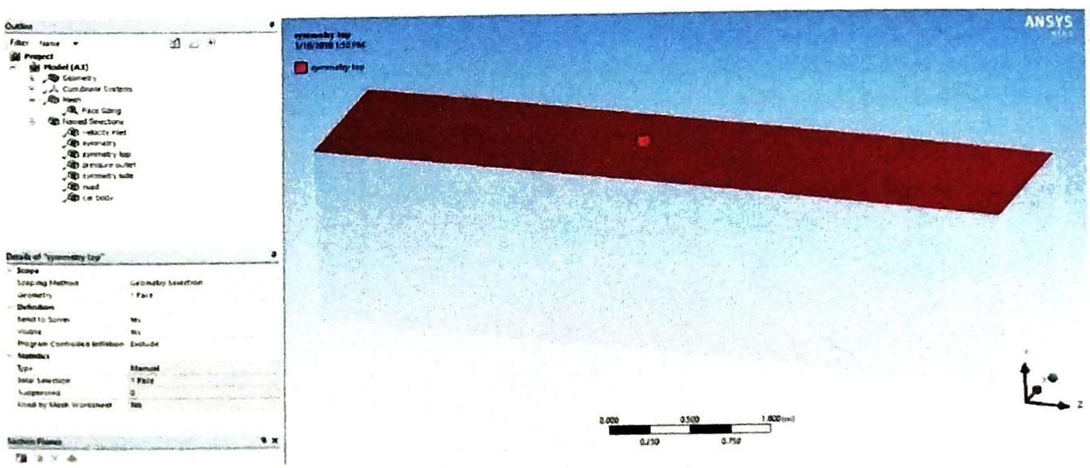


Fig 5.3.5 SYMMETRY TOP

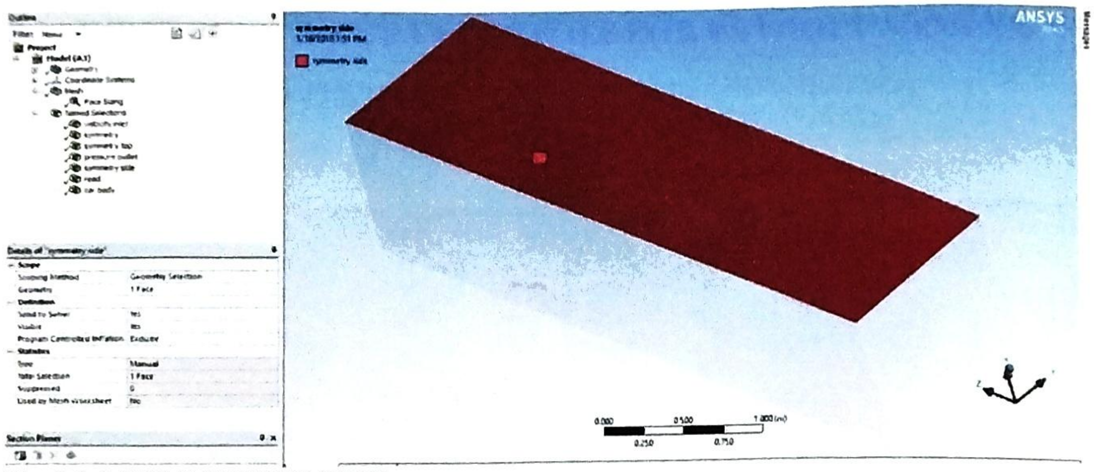


Fig 5.3.6 SYMMETTRY SIDE

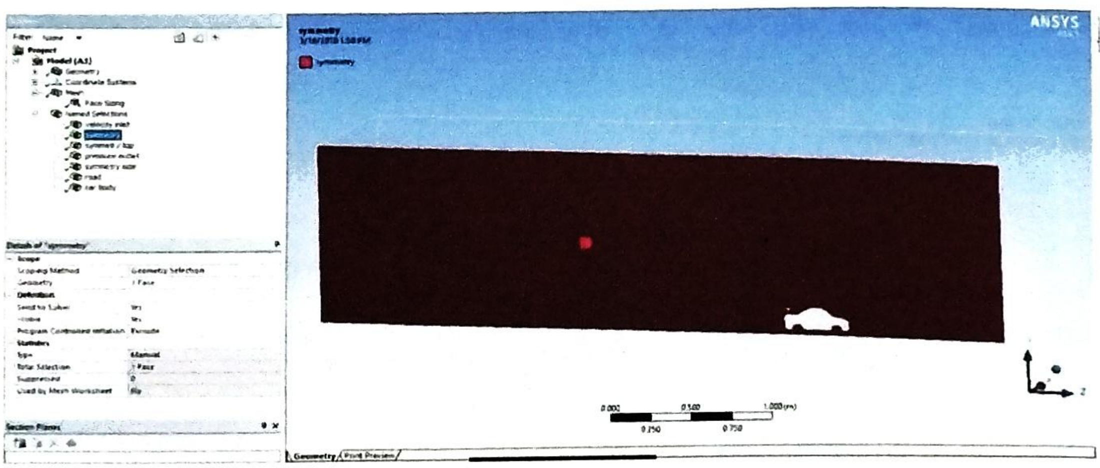


FIG 5.3.7 SYMMETRY

5.4 DIFFERENT INLET VELOCITIES FOR WITHOUT SPOILER

5.4.1 Velocity inlet :22.22 m/sec

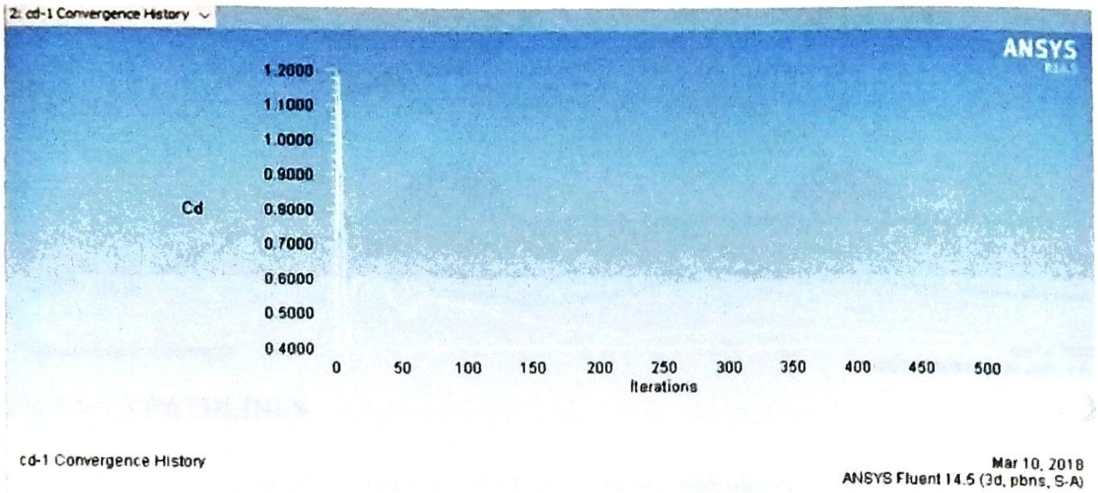


Fig 5.4.1.1 FOR CD

- This is the convergence history of the co-efficient of drag.

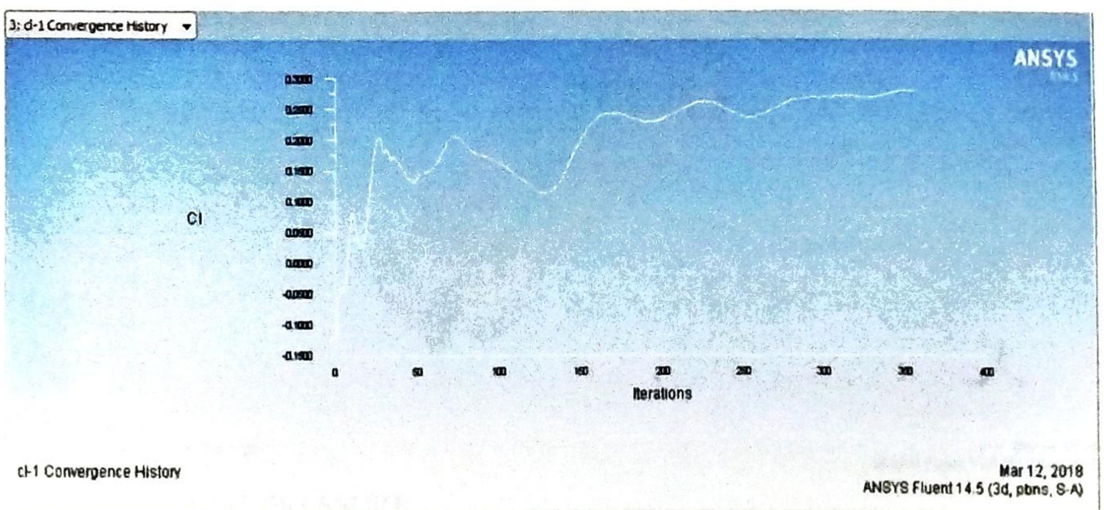


Fig 5.4.1.2 FOR CL

- This is the convergence history of the co-efficient of lift.

Fig 5.4.1.3 PATHLINES

- Here we can observe that the path lines creating turbulence behind the car.

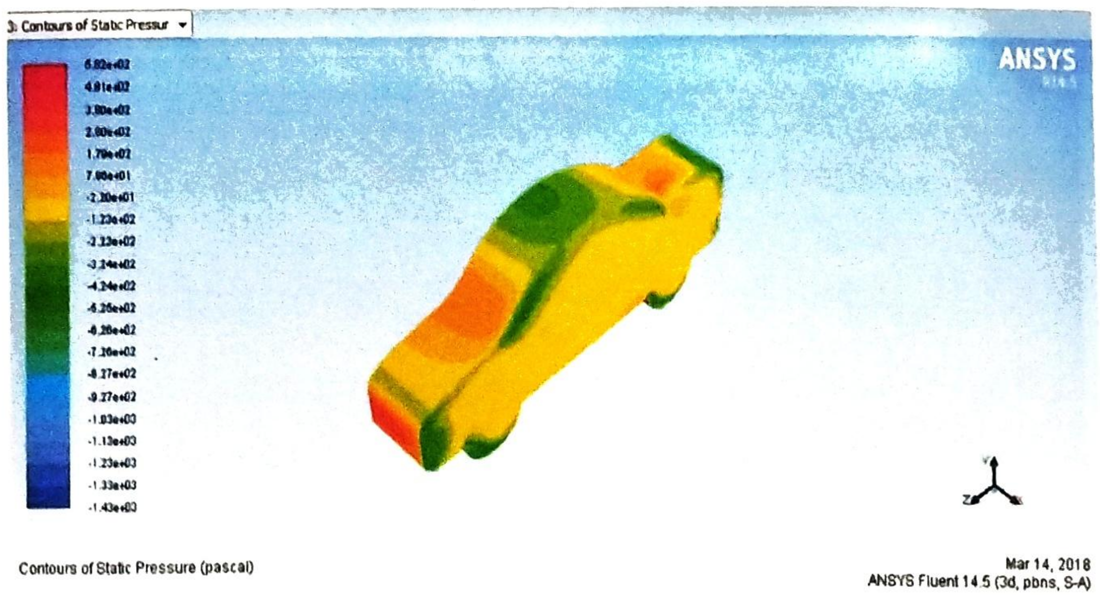


Fig 5.4.1.4 STATIC PRESSURE

- The more pressure will be acting on the frontal areas, which are normal in direction to the flow of fluid.

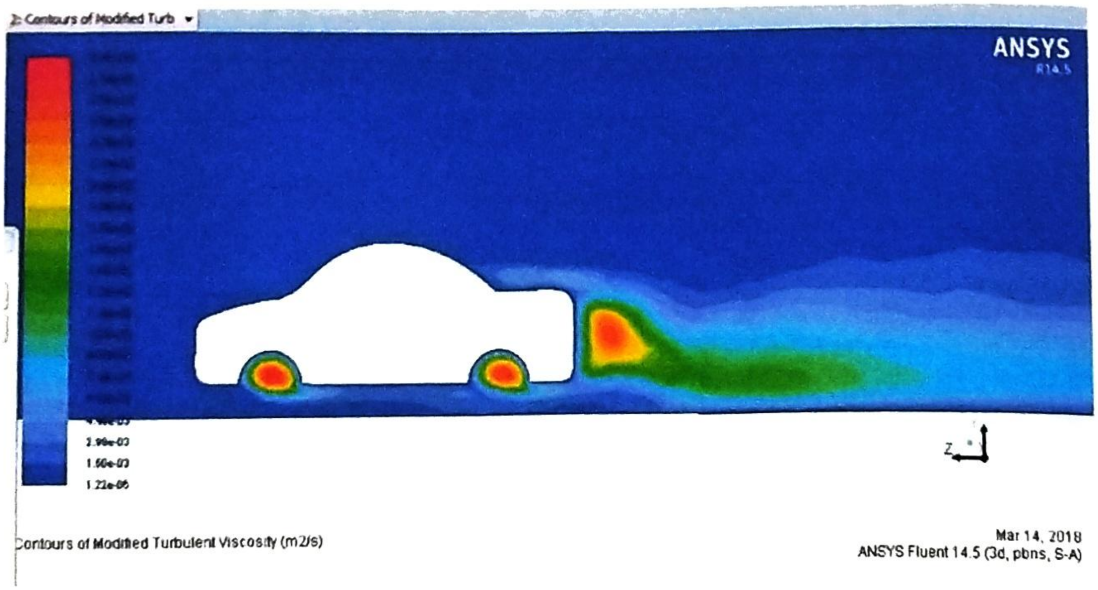


Fig 5.4.1.5 TURBULANCE

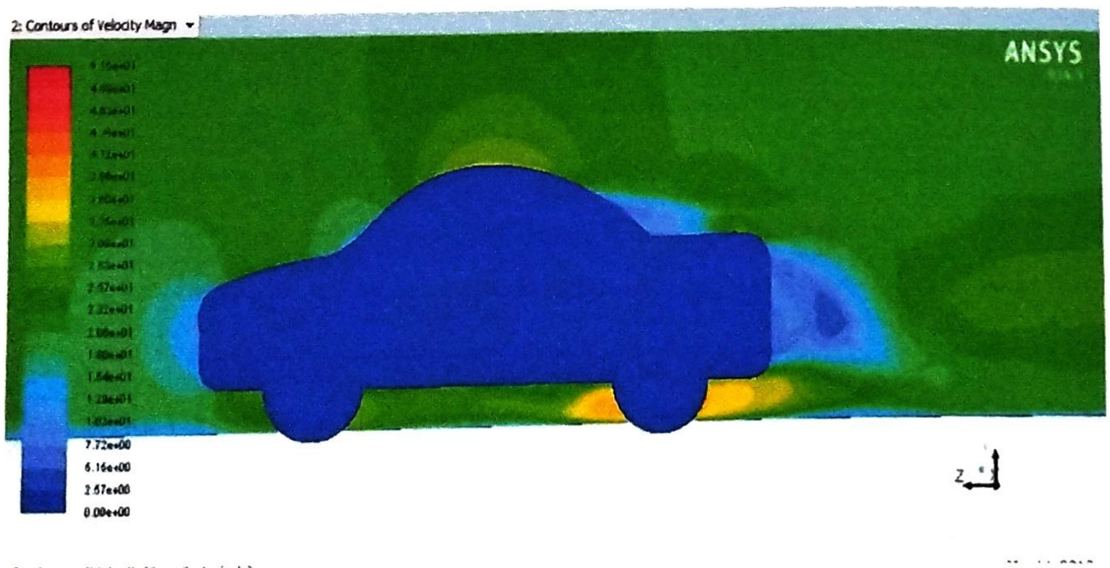


Fig 5.4.1.6 VELOCITY CONTOUR

5.4.2 Velocity inlet: 33.33 m/sec

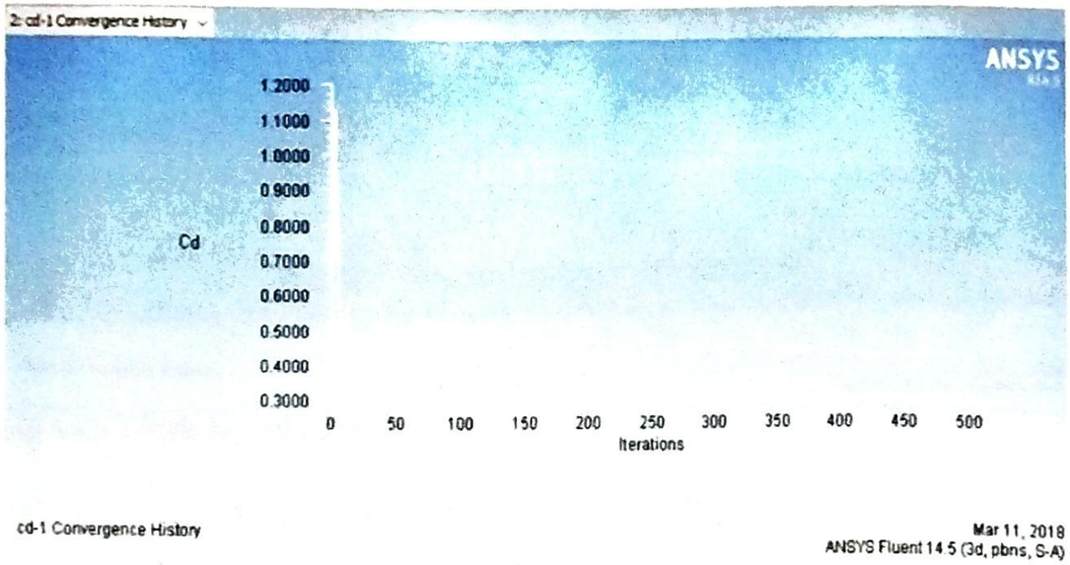


Fig 5.4.2.1 FOR CD

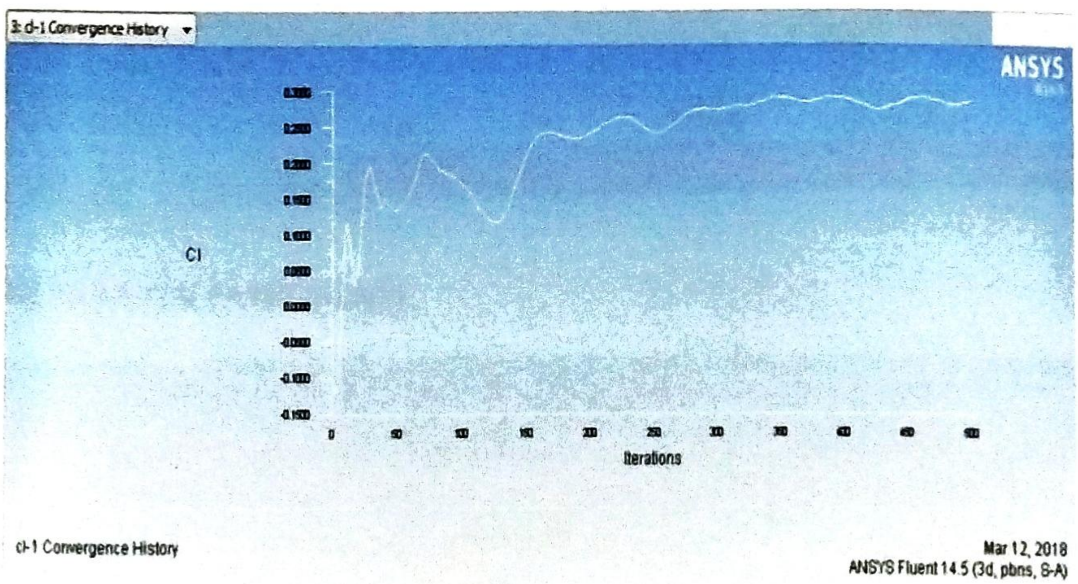


Fig 5.4.2.2 FOR CL

Fig 5.4.2.3 FOR PATHLINES

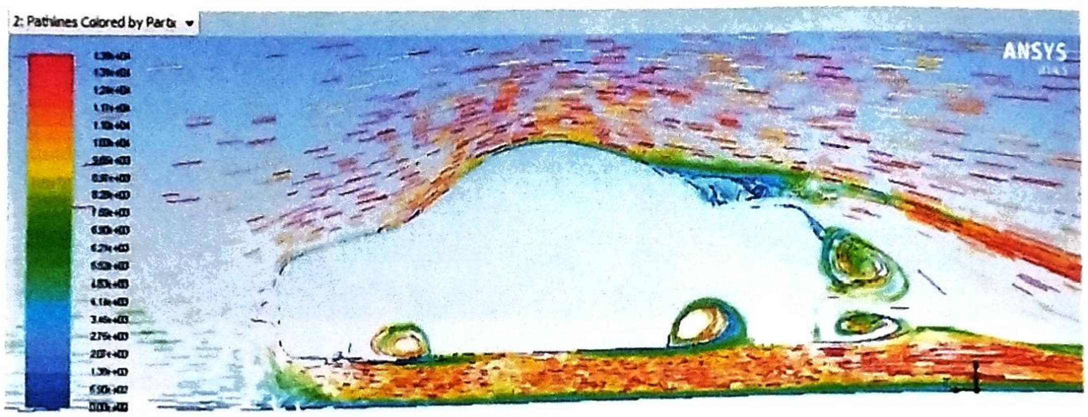


Fig 5.4.2.3 FOR PATHLINES(2)

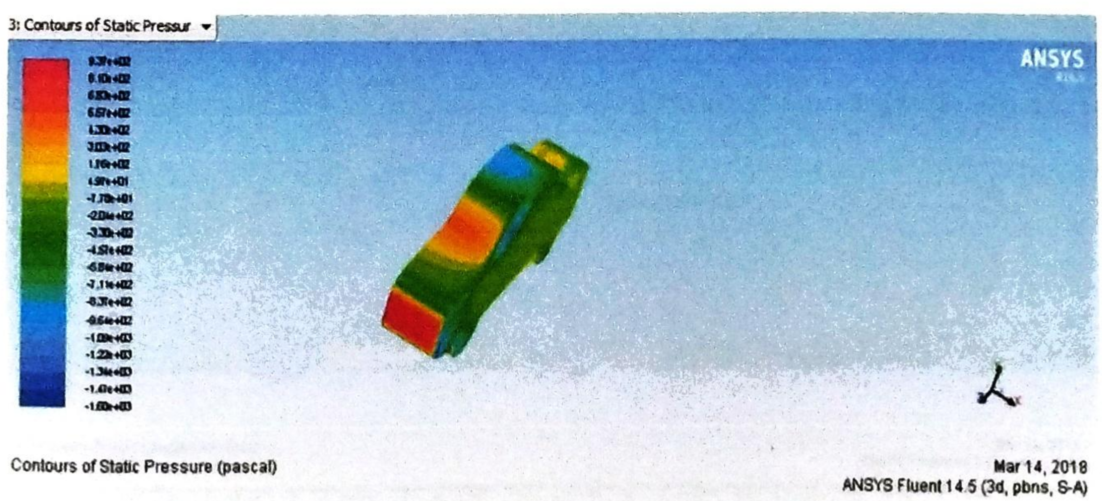


Fig 5.4.2.4 STATIC PRESSURE

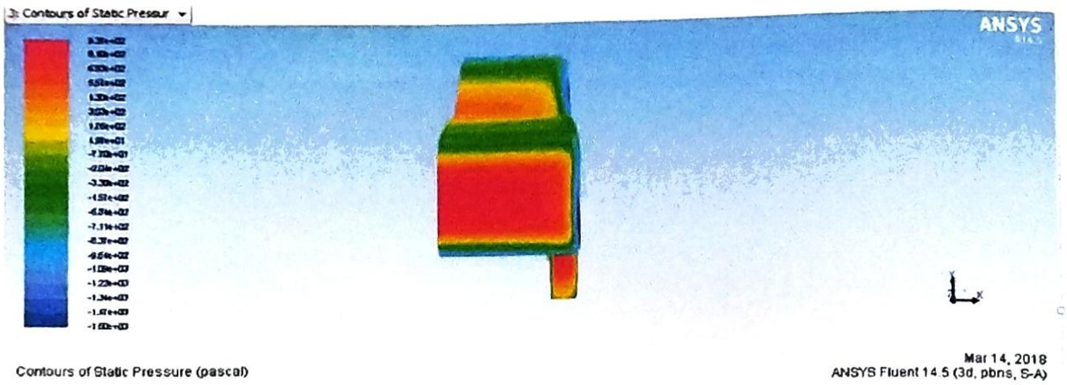


Fig 5.4.2.4 STATIC PRESSURE(2)

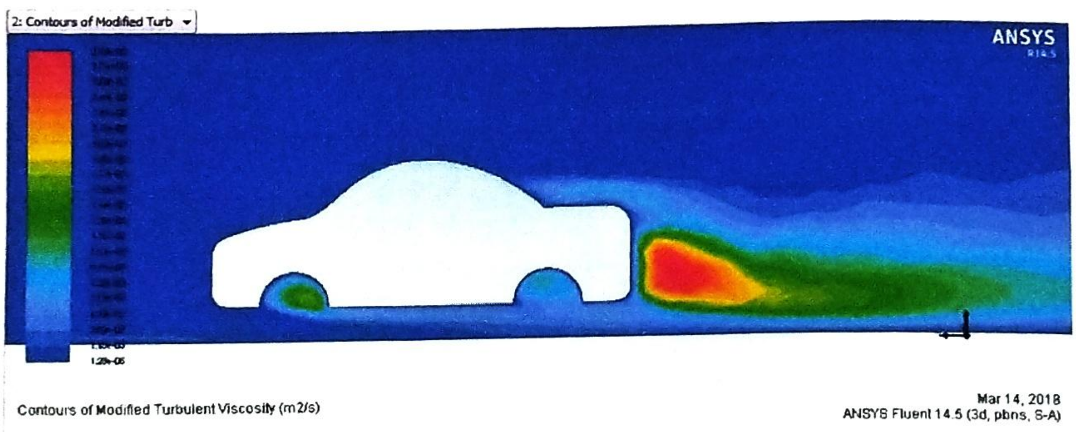


Fig 5.4.2.5 TURBULANCE

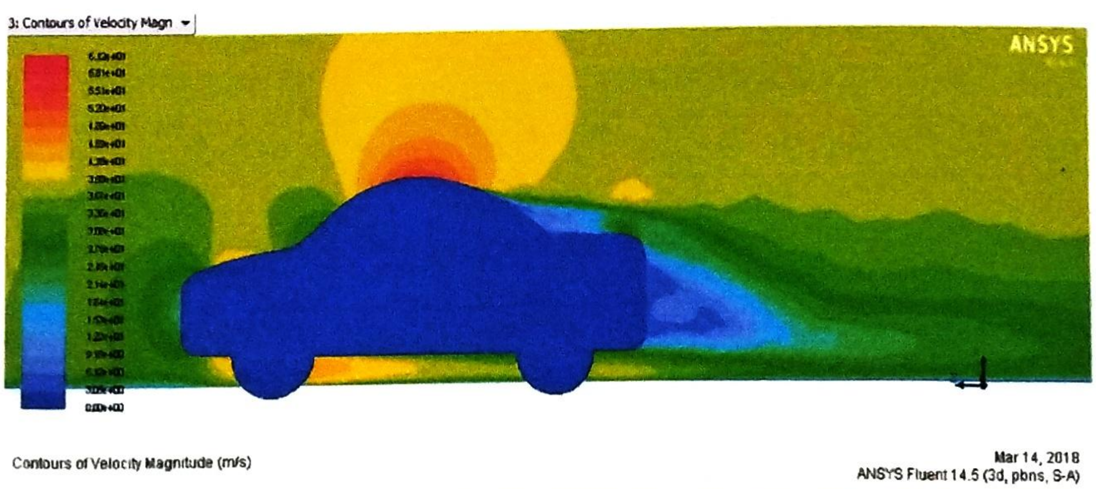


Fig 5.4.2.6 VELOCITY CONTOURS

5.4.3 Velocity at 38.3 m/sec

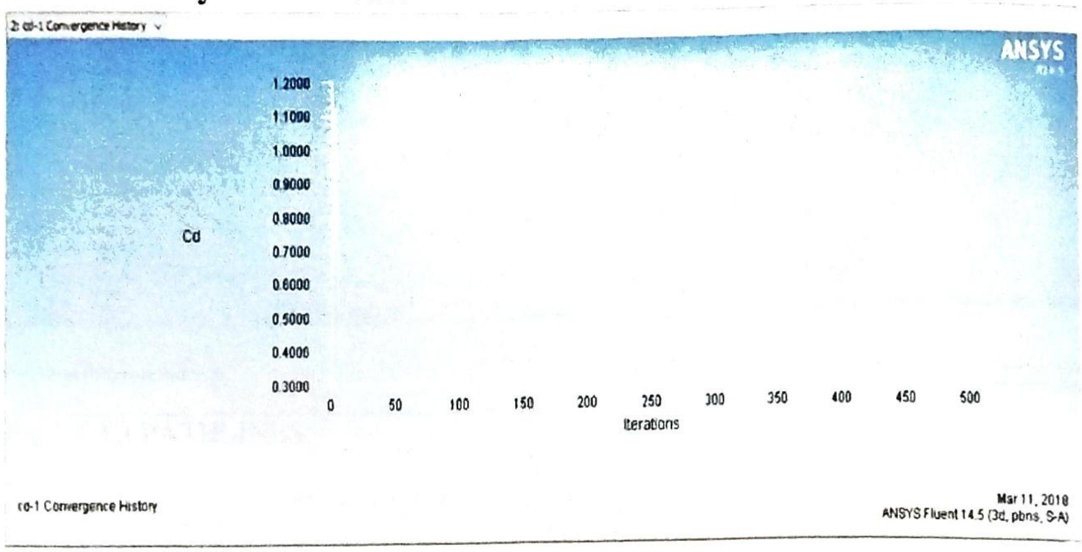


Fig 5.4.3.1FOR CD

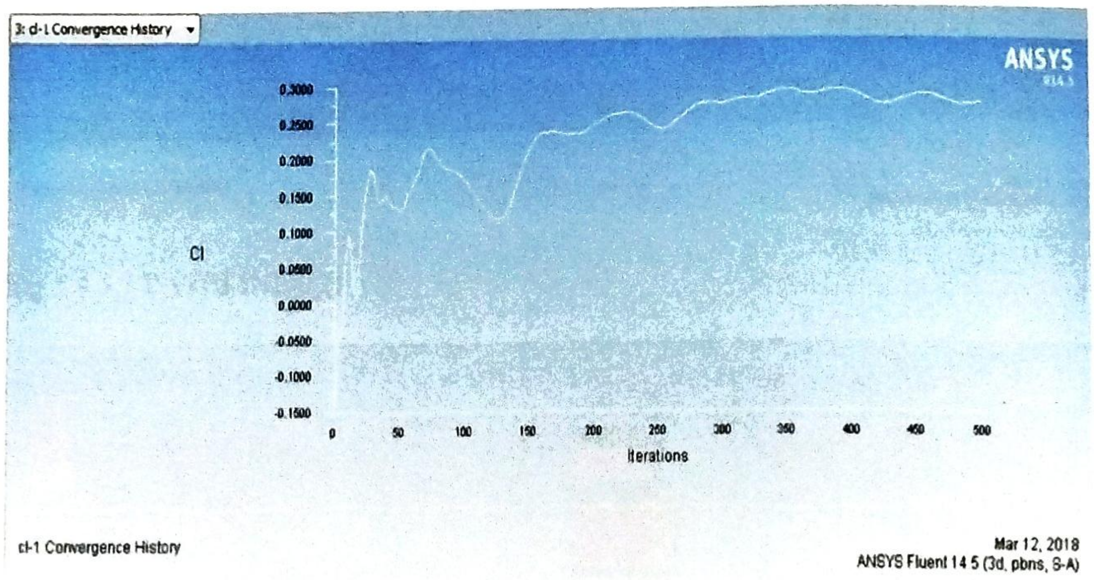


Fig5.4.3.2 FOR CL

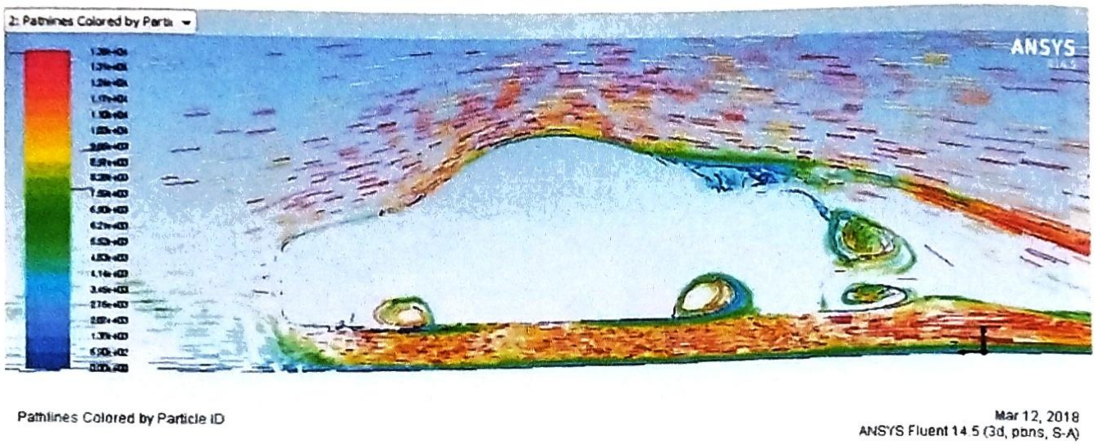


Fig 5.4.3.3 PATHLINES

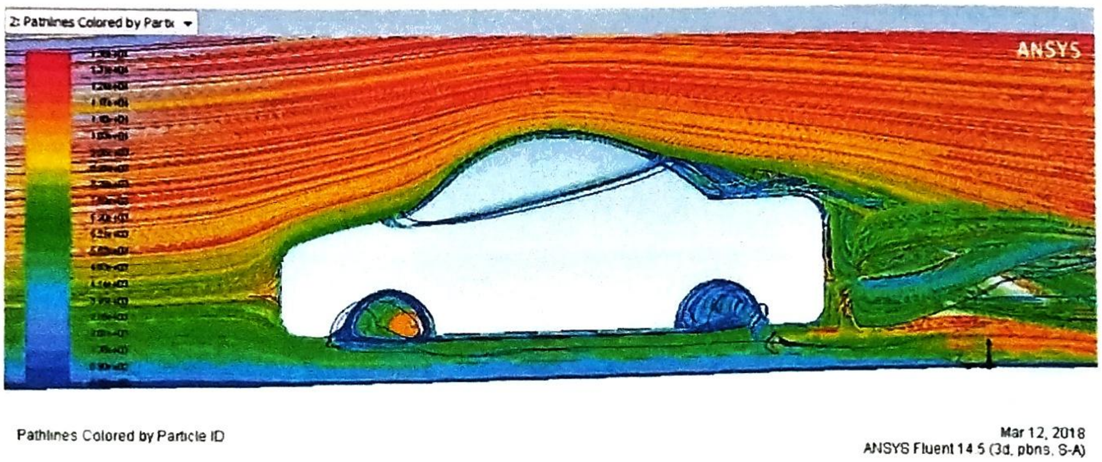


Fig 5.4.3.3 PATHLINES(2)

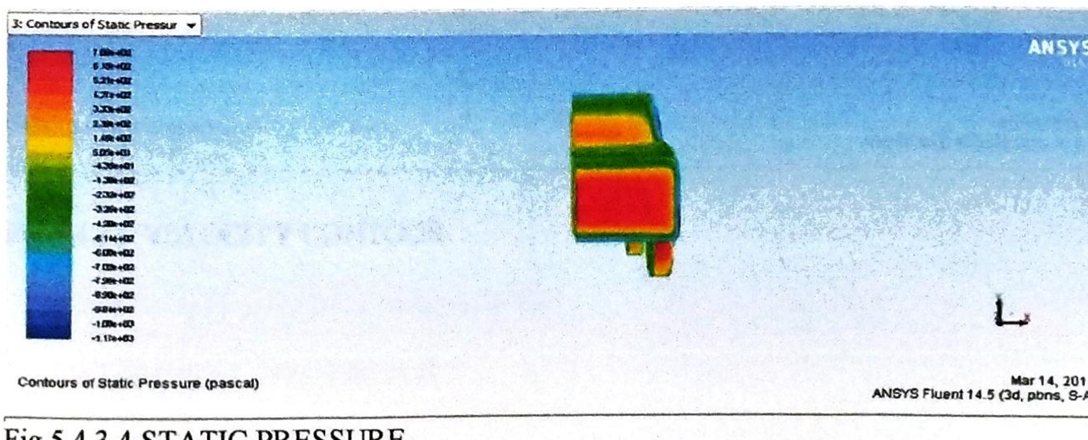


Fig 5.4.3.4 STATIC PRESSURE

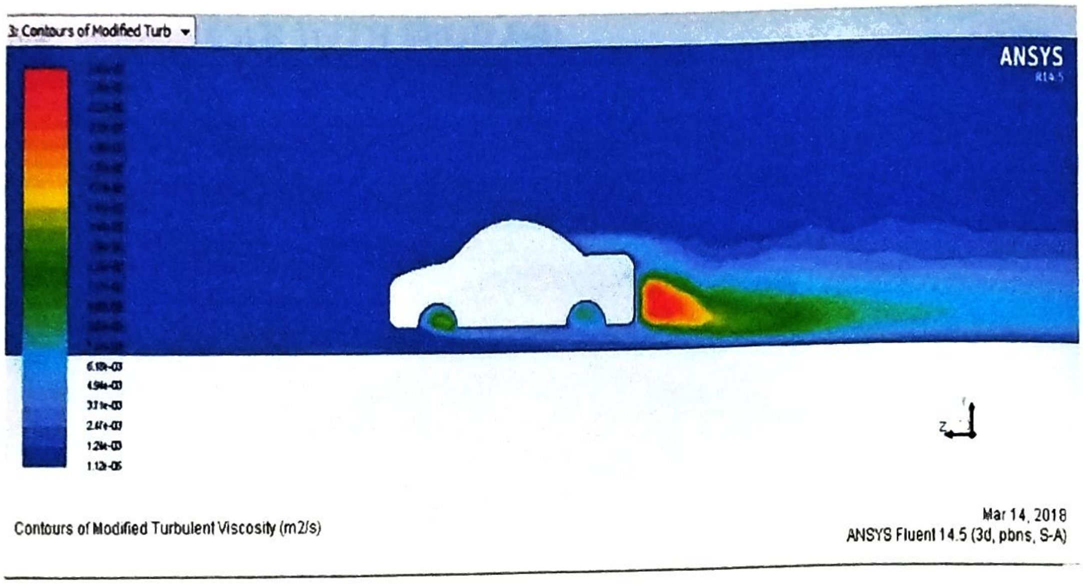


Fig 5.4.3.5 TURBULANCE

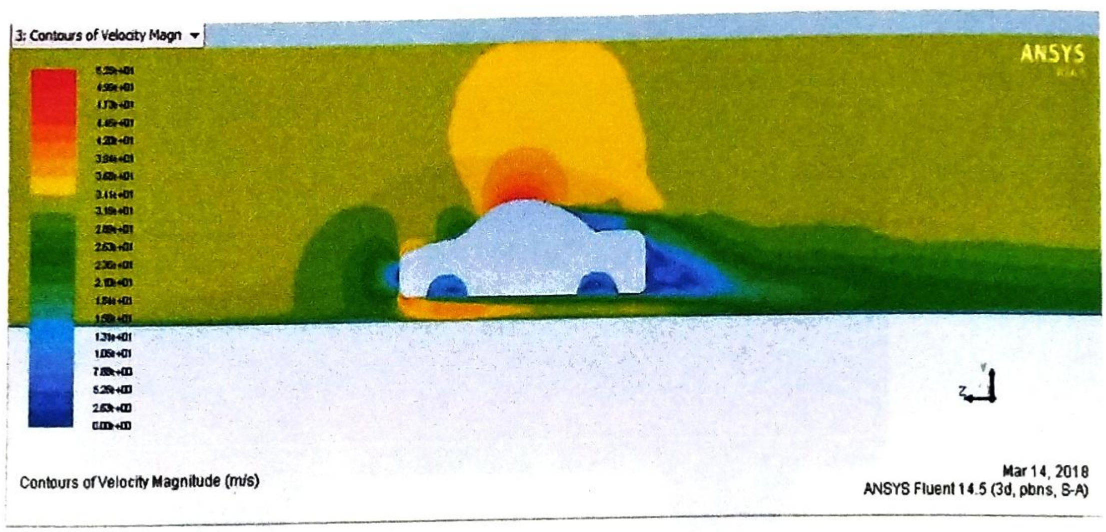


Fig 5.4.3.6 VELOCITY CONTOUR

5.5 SEDAN CAR WITH SPOILER:

Fig 5.5.1 CAR MODELLING FOR A CAR WITH SPOILER

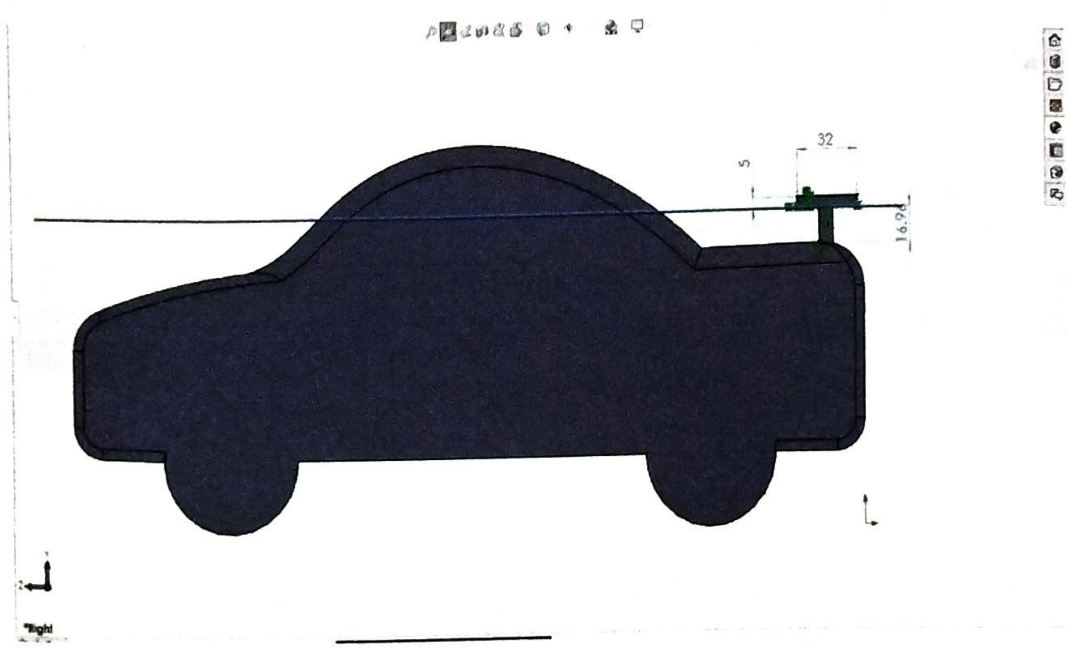


Fig 5.5.2 SPOILER DIMENSIONS

5.6 WITH SPOILER PARAMETERS

For performing analysis on the body, the boundary conditions that given as input parameter is shown below.

Type of car	With spoiler
Mesh Nodes	472445
Mesh Elements	1493668
Domain Type	Fluid
Model	Spalart: Allmaras(vorticity based)
Maximum skewness	0.89840700
Maximum face size	0.30m
Maximum size	0.30m
Use automatic inflation	Program controlled
Boundary Type	Inlet
Face sizing element size	0.02 m
Sizing	
Use advanced size function	On proximity and curvature
Relevance Centre	Fine
Transition	Slow
Number of cells across gap	10
Relative Pressure	0.0000e+00 [Pa]
Standard deviation	0.1203325

The Analysis is carried out at different speeds in Ansys- FLUENT and the results such as contours, vector plots, turbulent kinetic energy and streamline plots. The surface pressure contour is also observed in the analysis.

The analysis includes process of five steps such as

- Geometry
- Mesh
- Setup
- Solution
- Results.

Geometry: the part modeling of solid works file is saved in .igs format and then imported to Ansys fluent

Mesh and Setup: The imported file geometry undergoes meshing then; the file is aligned to boundary conditions.

Solution and Results: After applying boundary conditions, the solution and results are as follows.

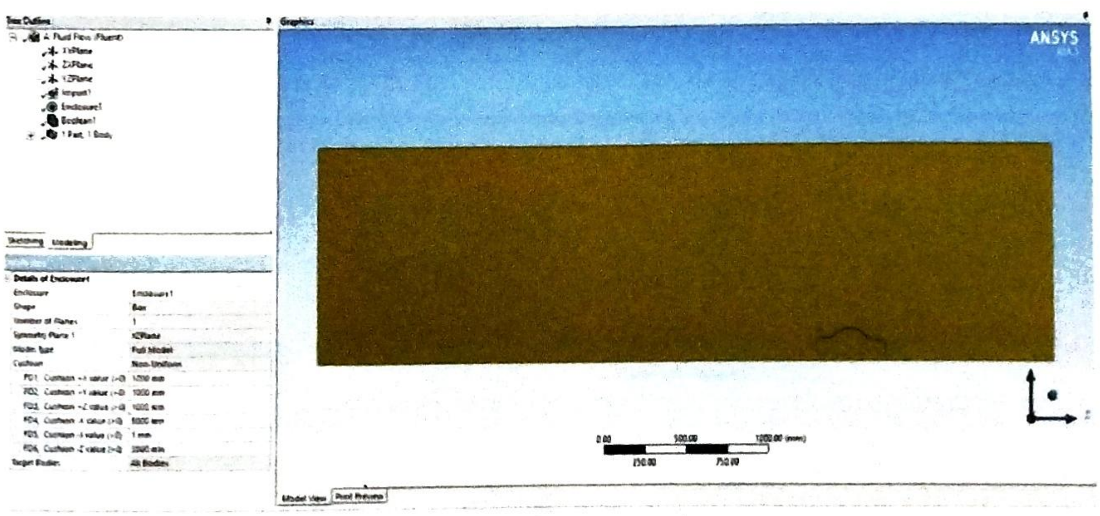


Fig 5.6.1 GEOMETRY AND ENCLOSURE

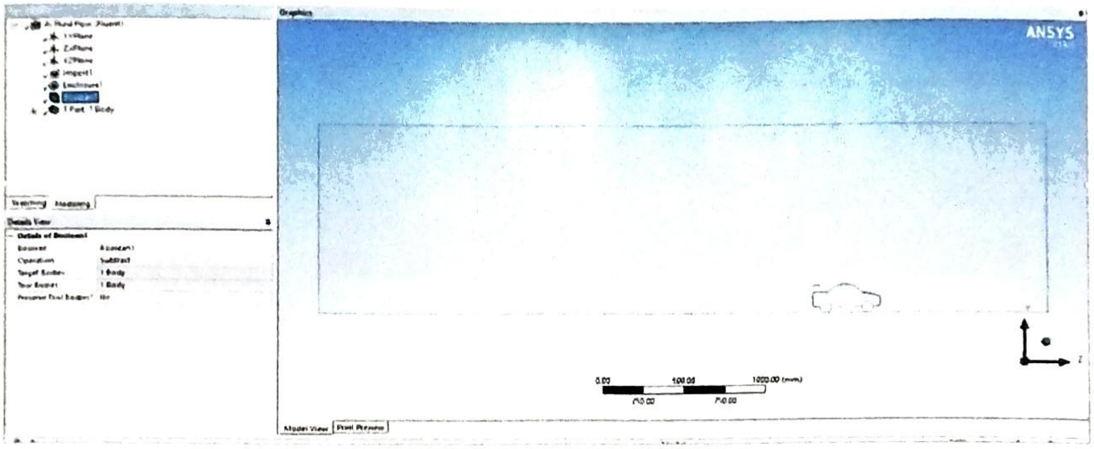


Fig 5.6.2 BOOLEAN

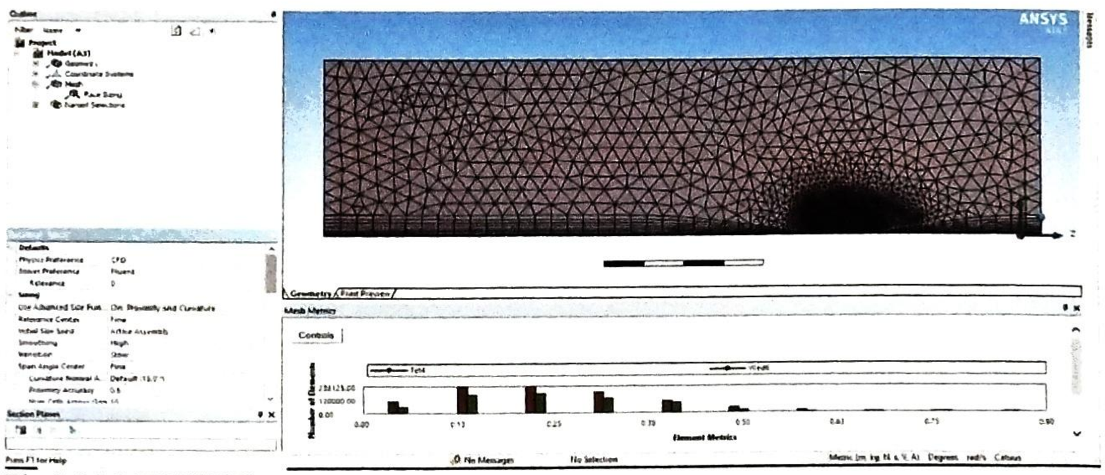


Fig 5.6.3 MESHING

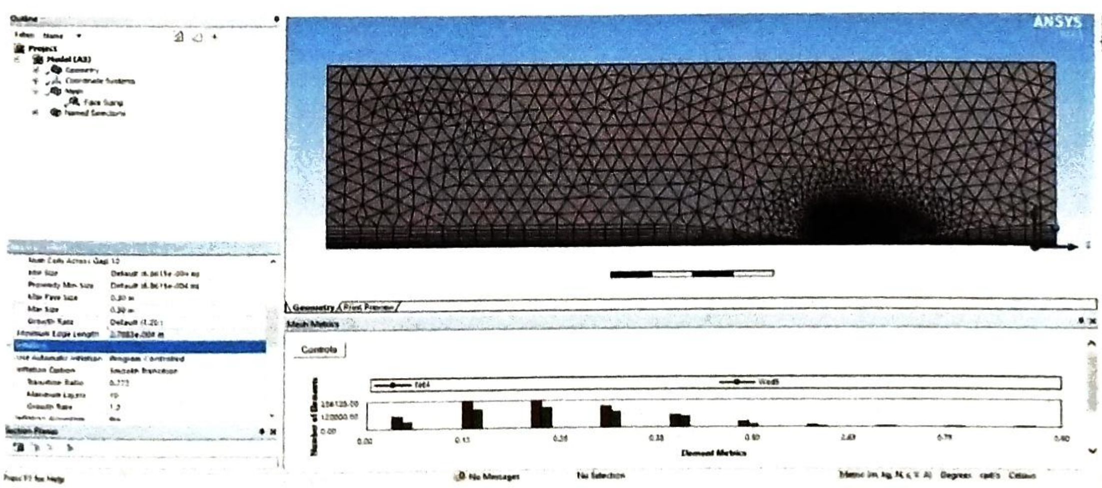


Fig 5.6.3 MESHING(2)

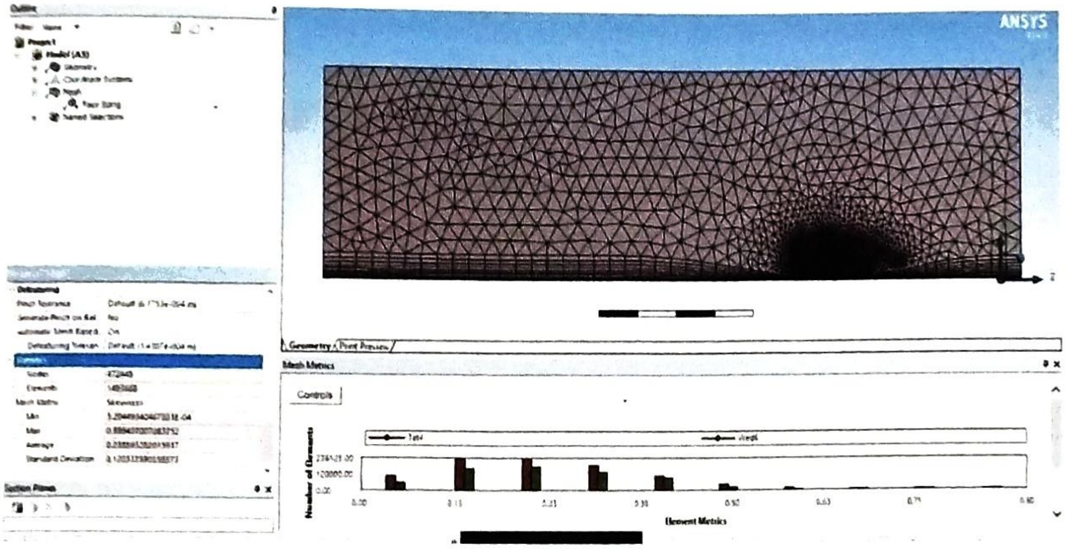


Fig 5.6.3 MESHING(3)

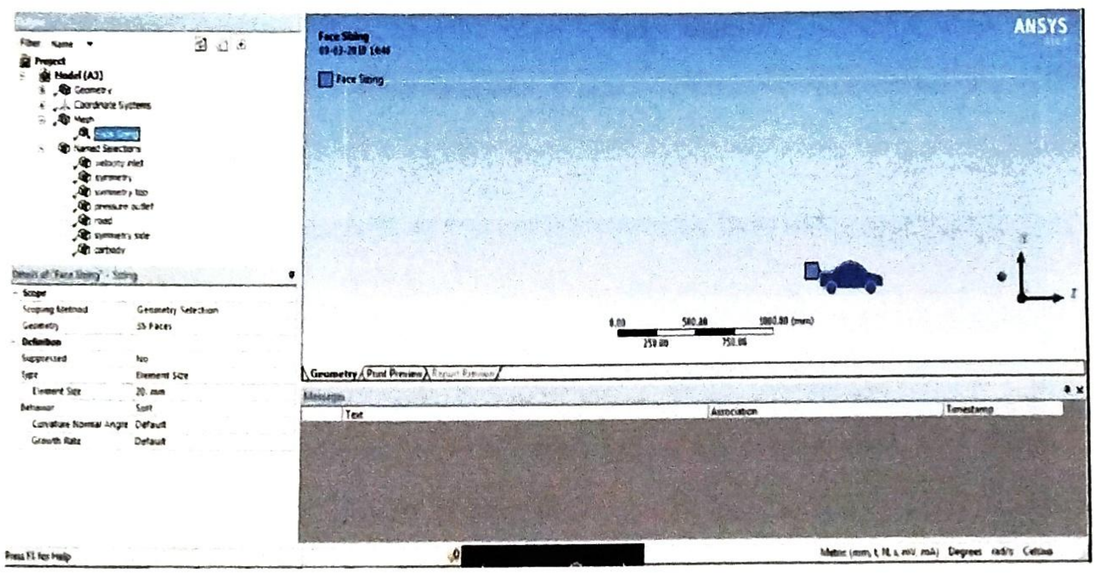


Fig 5.6.4 FACE SIZING

5.7 NAMED SELECTIONS:

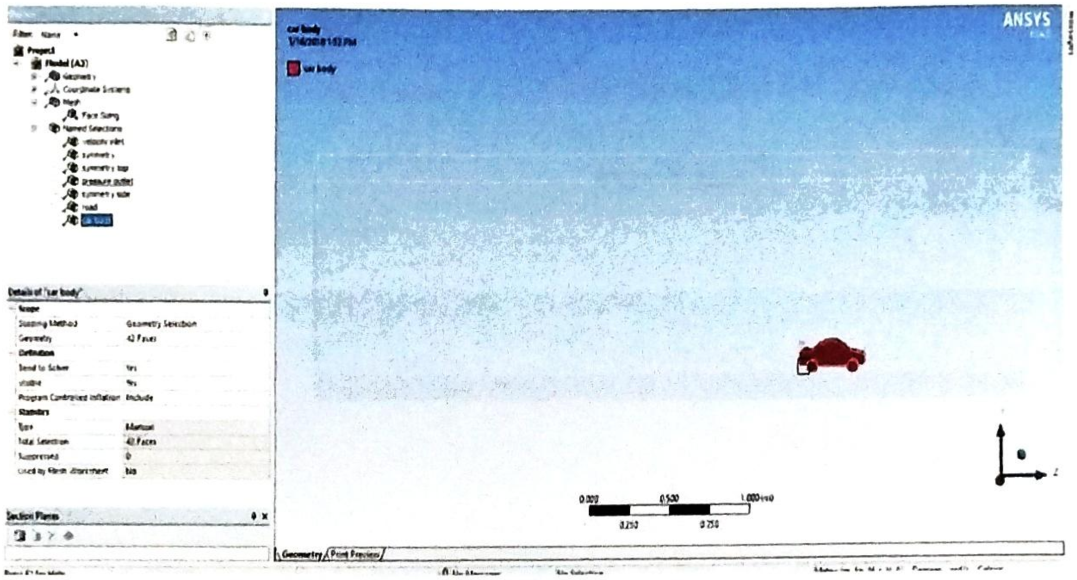


Fig 5.7.1 CAR BODY

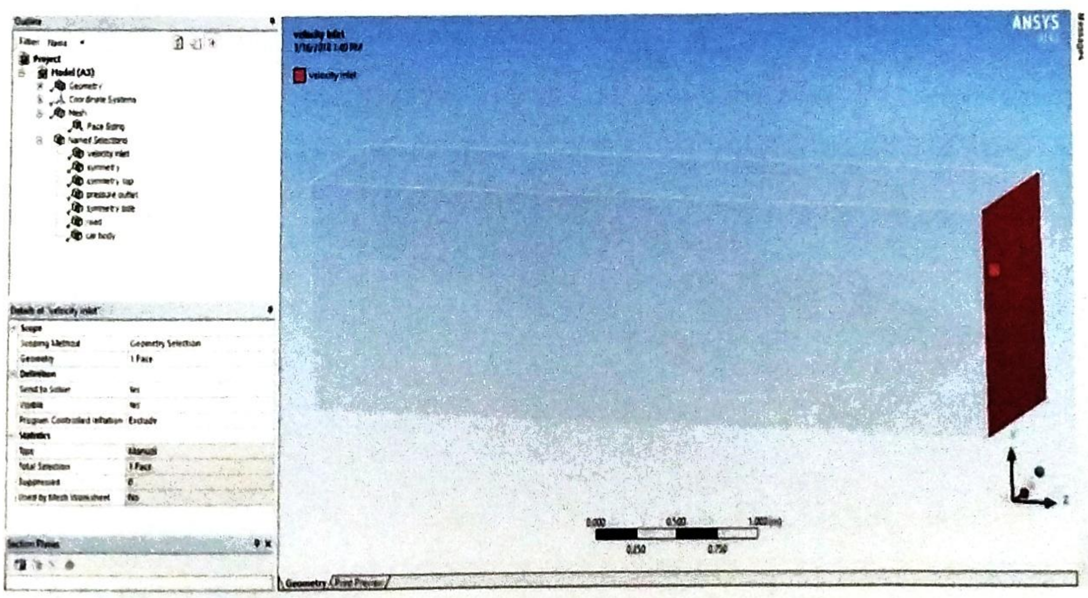


Fig 5.7.2 INLET VELOCITY

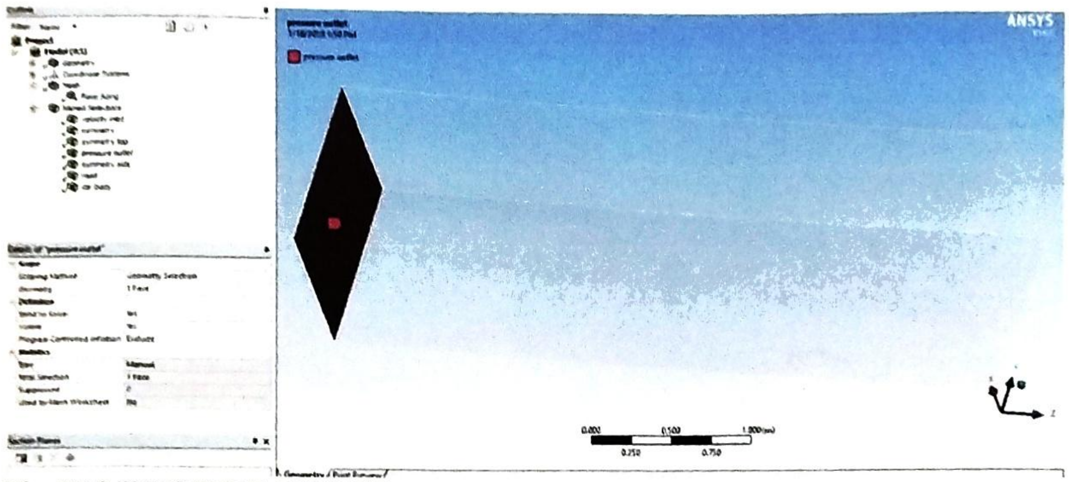


Fig 5.7.3 PRESSURE OUTLET

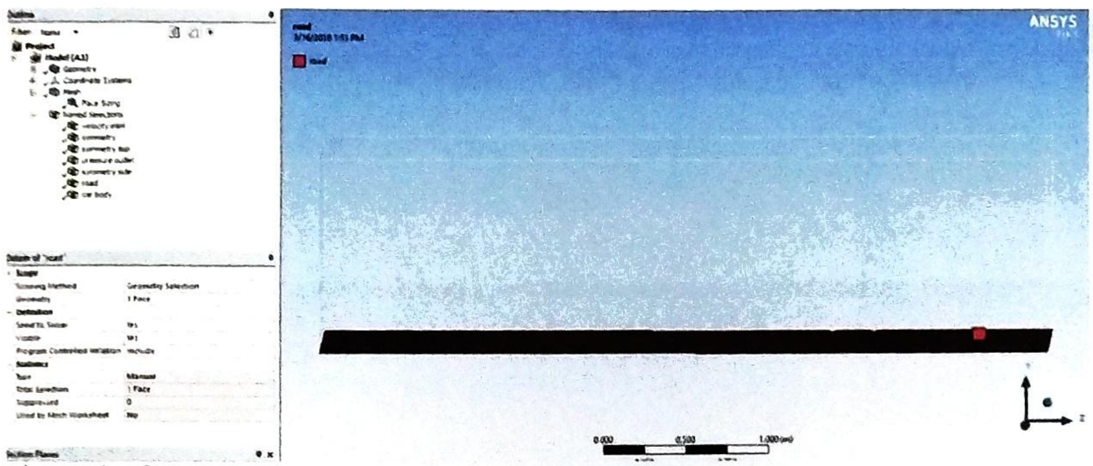


Fig 5.7.4 ROAD

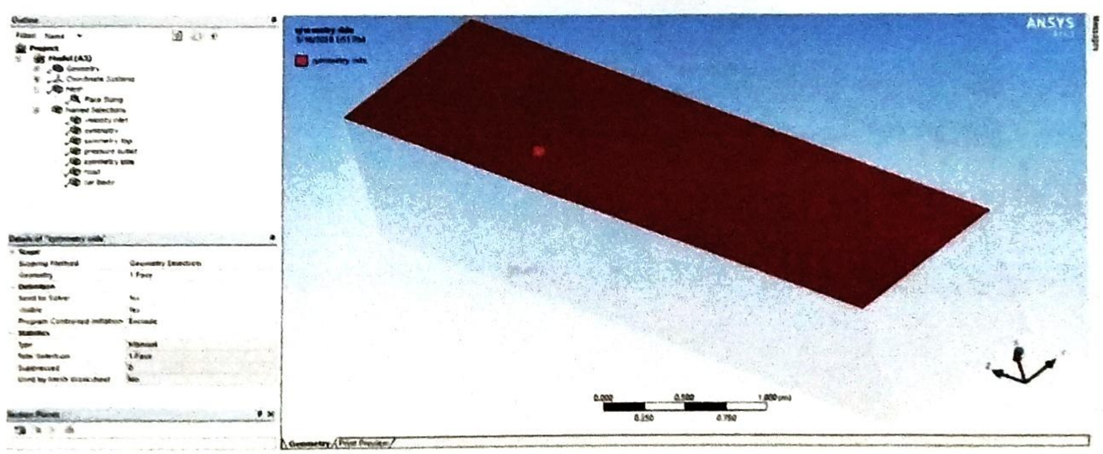


Fig 5.7.5 SYMMETRY SIDE

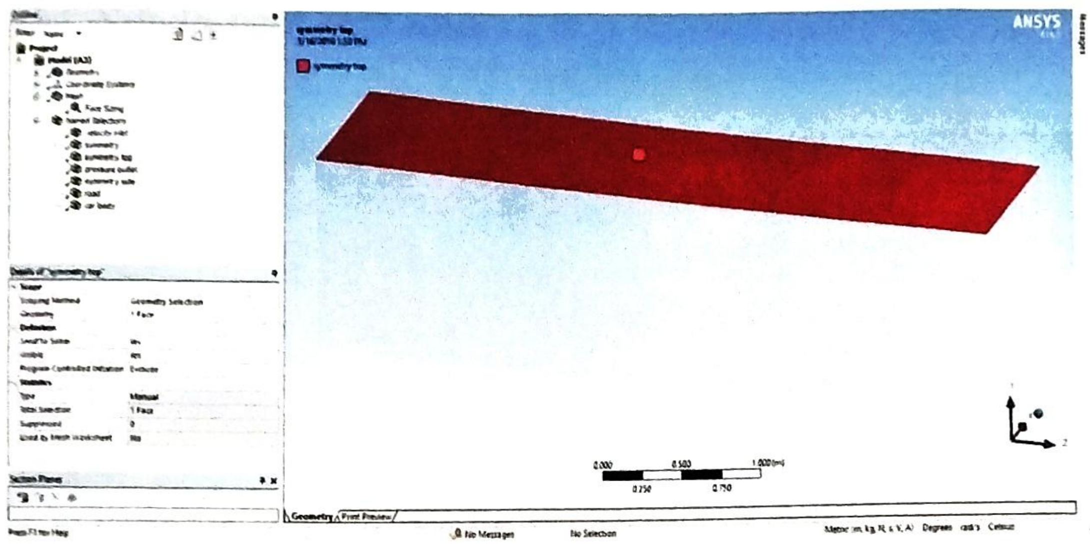


Fig 5.7.6SYMMETRY TOP

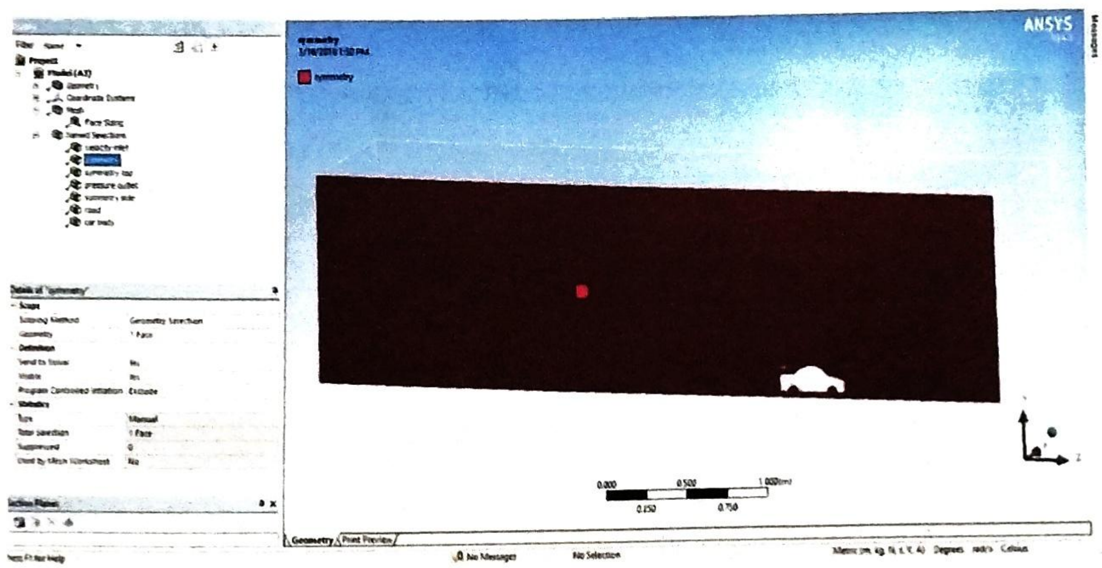


Fig 5.7.7 SYMMETRY

5.8 DIFFERENT INLET VELOCITIES FOR WITH SPOILER:

5.8.1 Velocity inlet: 22.22 m/sec

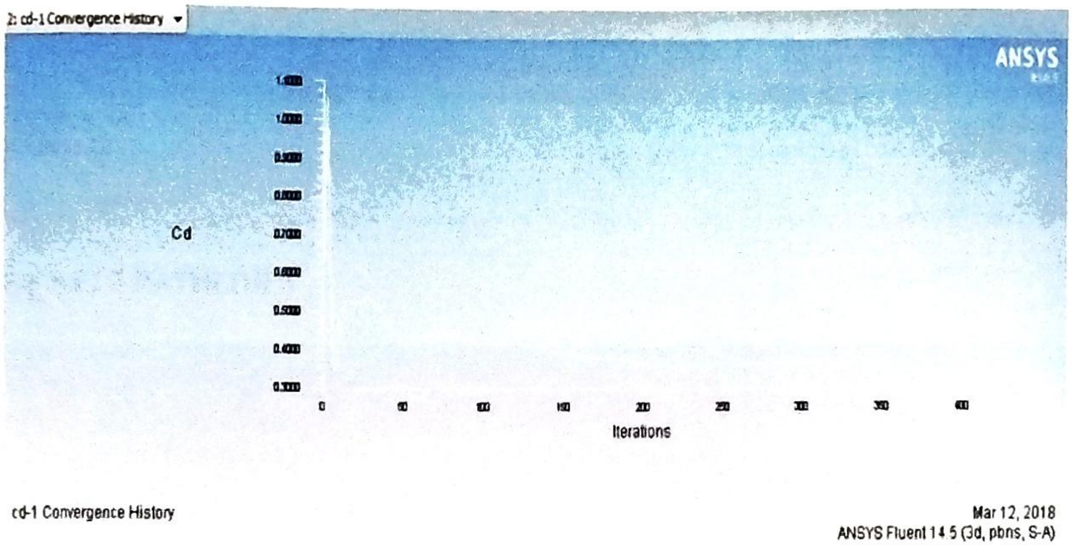


Fig-5.8.1.1 FOR CD

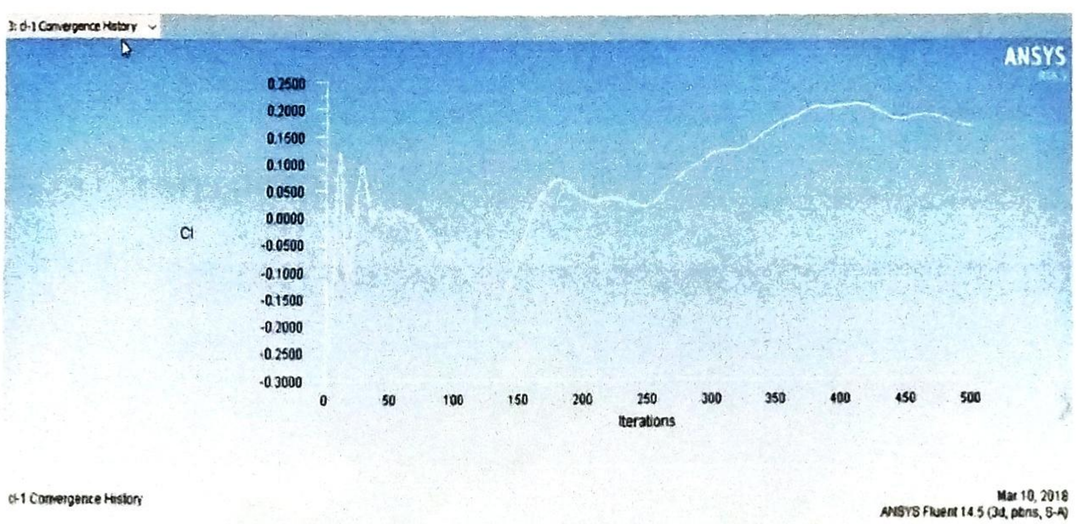


Fig -5.8.1.2 FOR CL

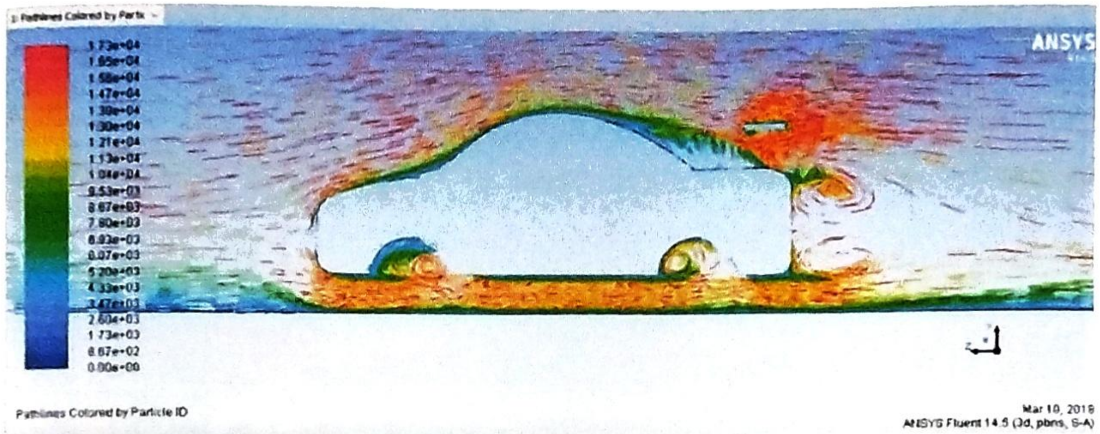


Fig 5.8.1.3 PATHLINES

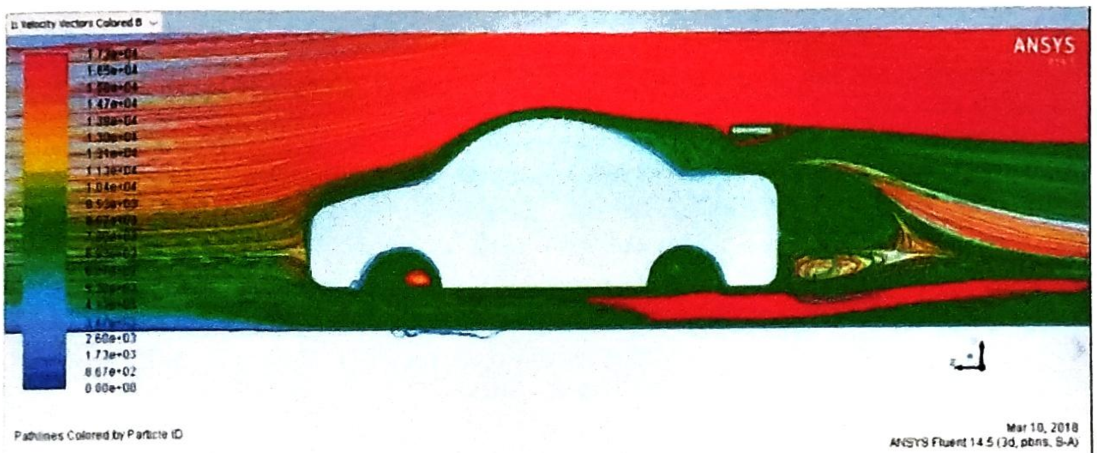


Fig 5.8.1.3 PATHLINES(2)

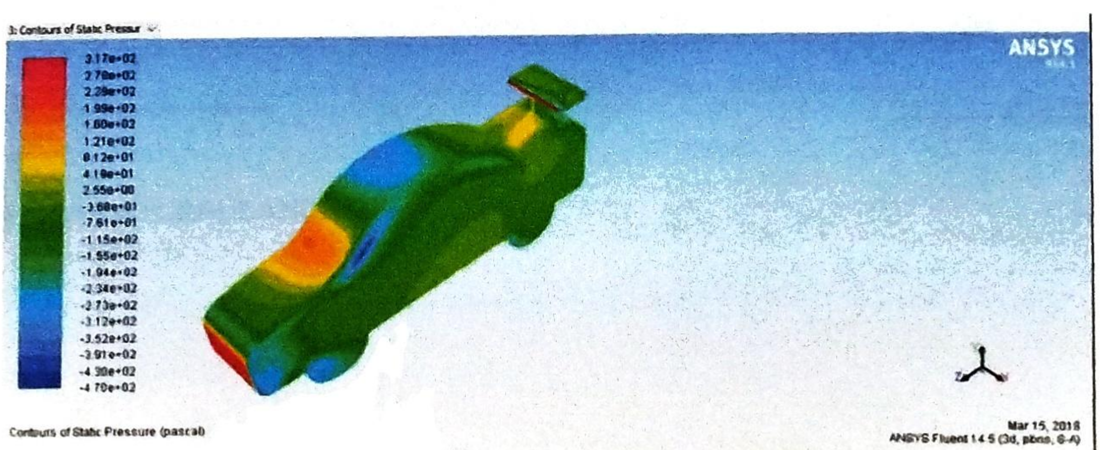


Fig 5.8.1.4 STATIC PRESSURE

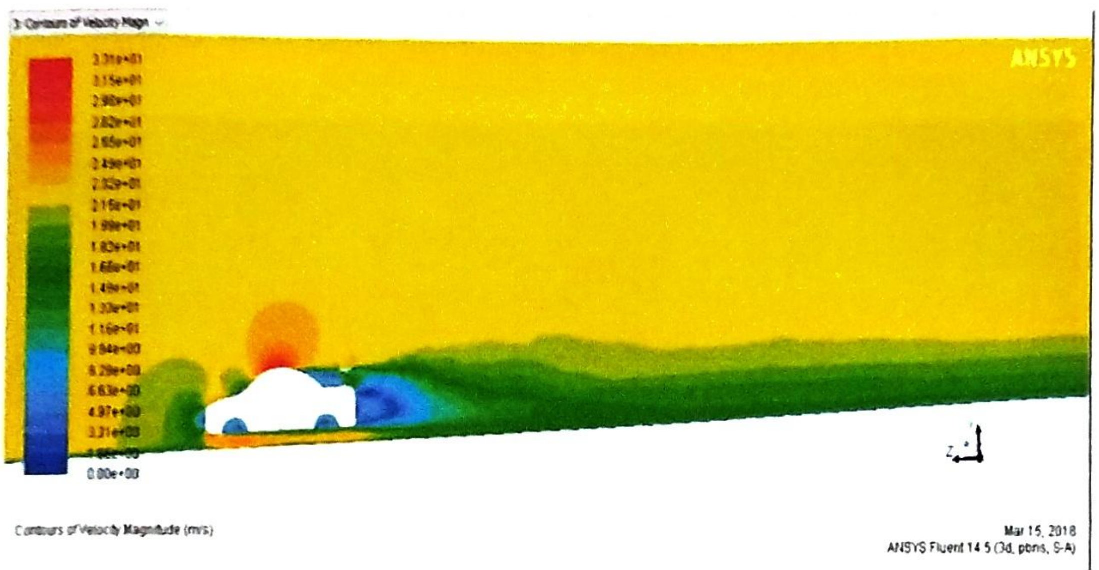


Fig-5.8.1.5 VELOCITY CONTOUR

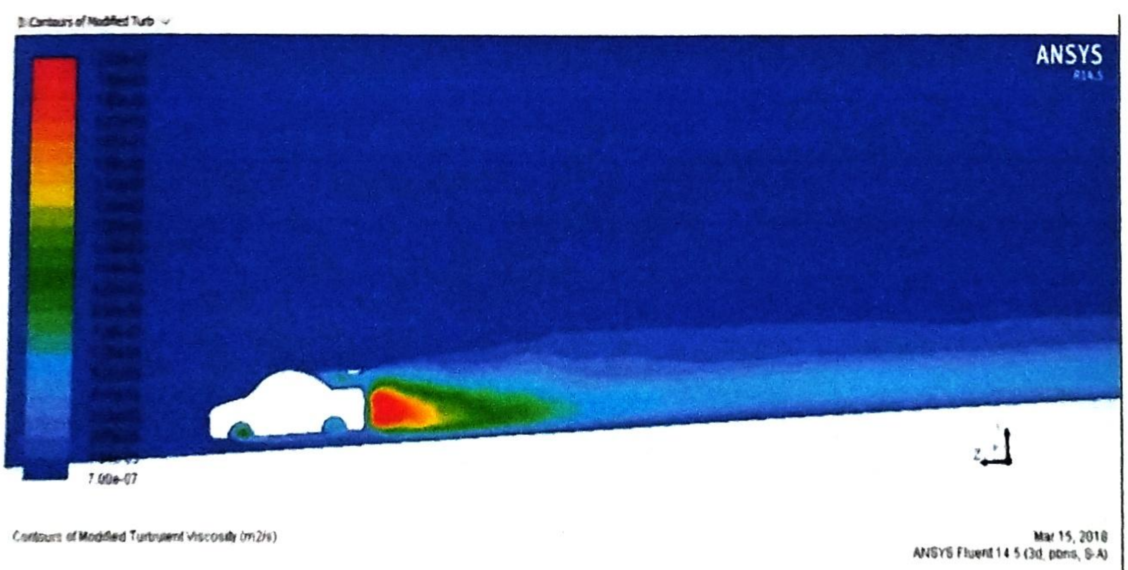


Fig-5.8.1.6 TURBULANCE

5.8.2 Velocity inlet: 33.33 m/sec

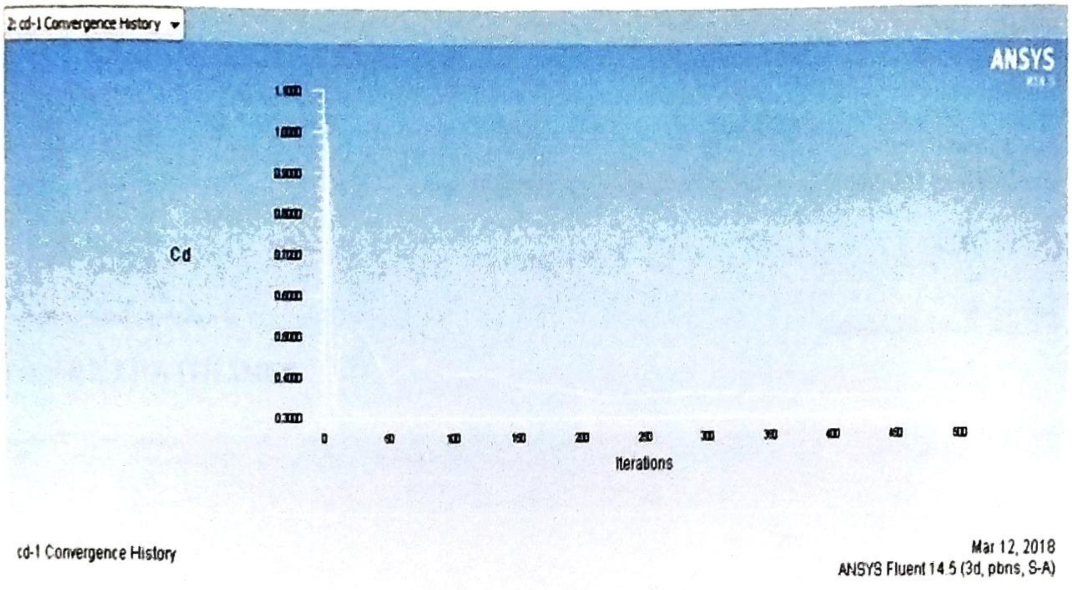


Fig 5.8.2.1 FOR CD

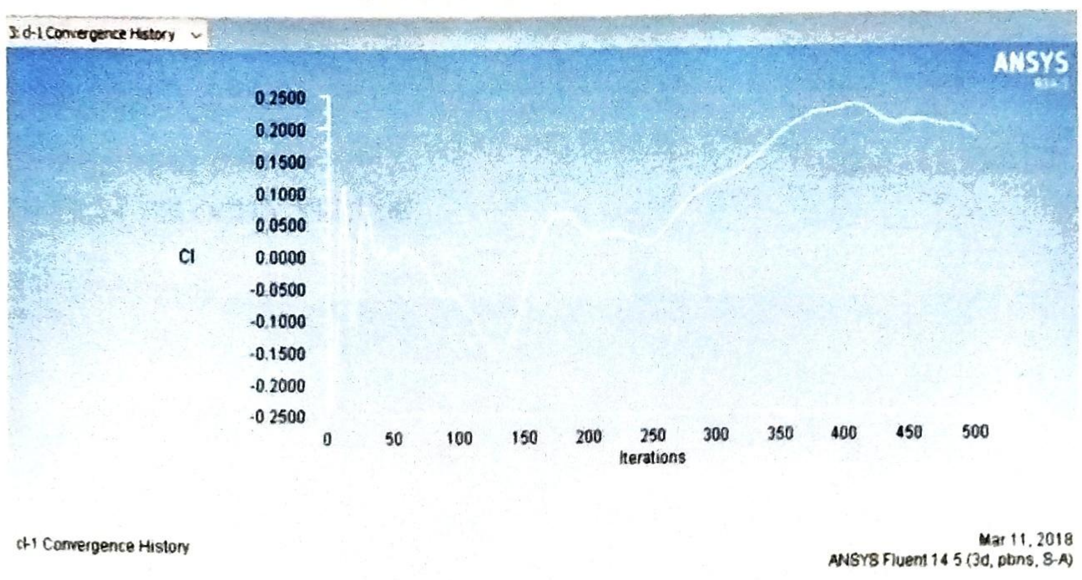


Fig 5.8.2.2 FOR CL

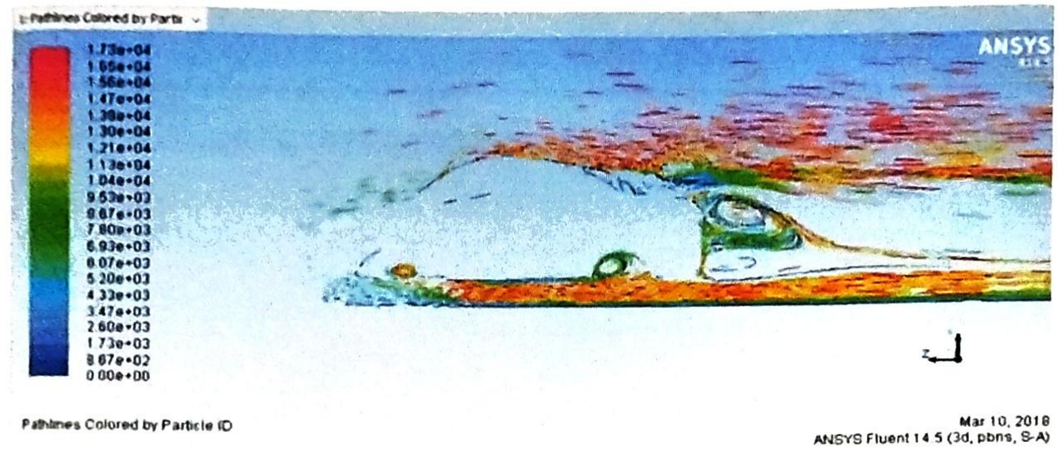


Fig 5.8.2.3 PATHLINES

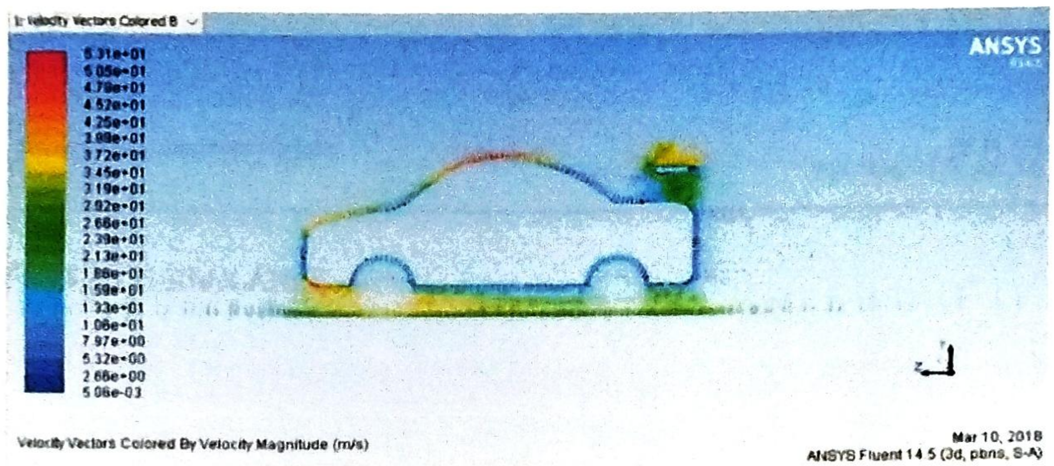


Fig 5.8.2.4 VELOCITY VECTOR

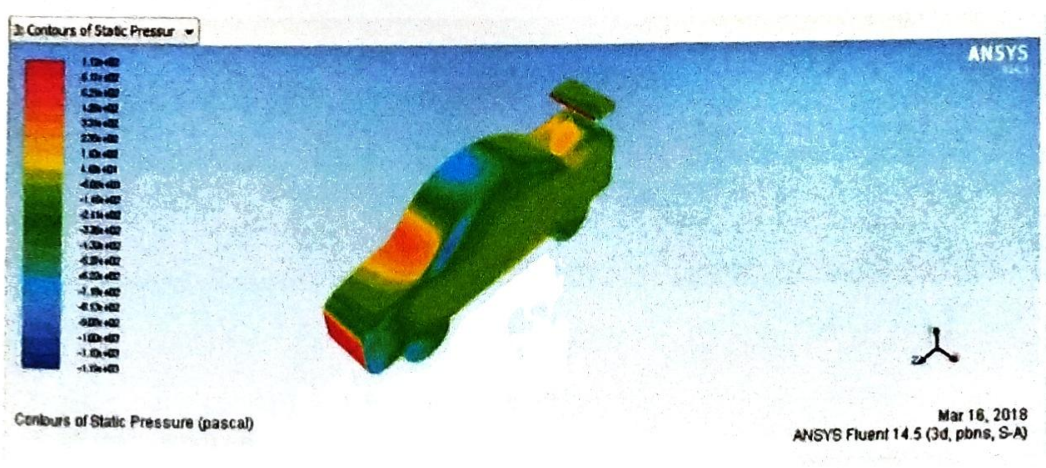


Fig 5.8.2.5 STATIC PRESSURE

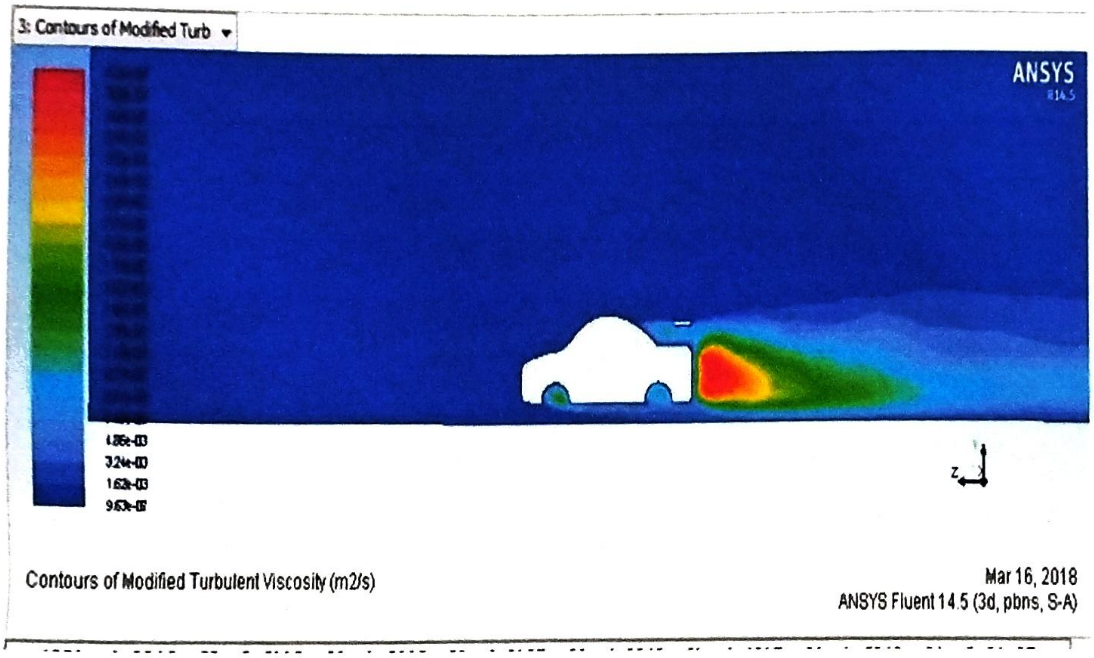


Fig 5.8.2.6 TURBULANCE

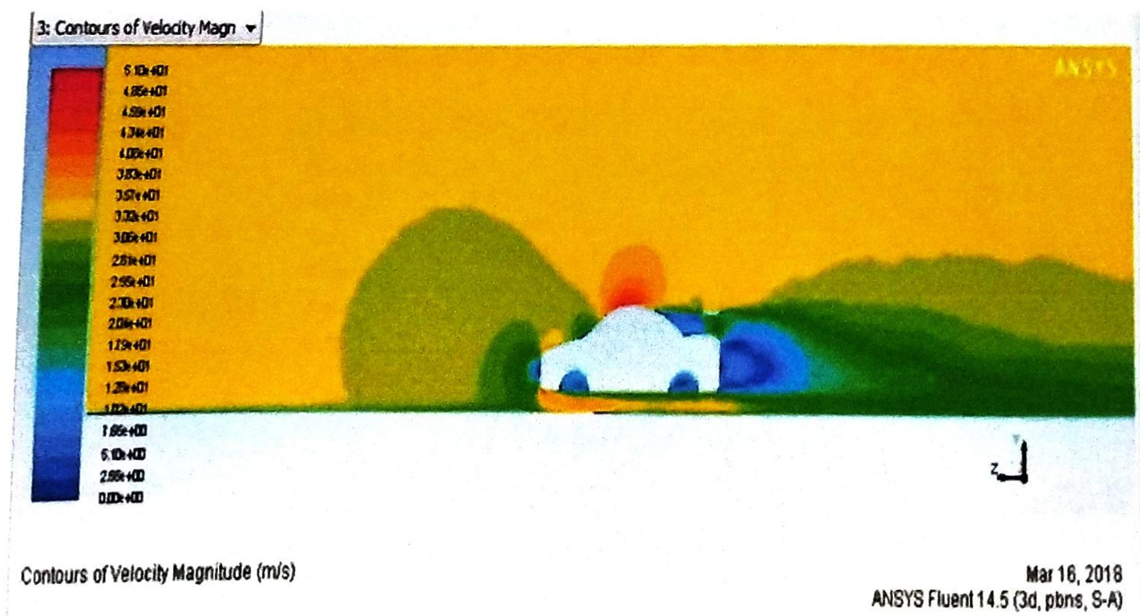


Fig 5.8.2.7 VELOCITY CONTOURS

5.8.3 Velocity at 38.3 m/sec

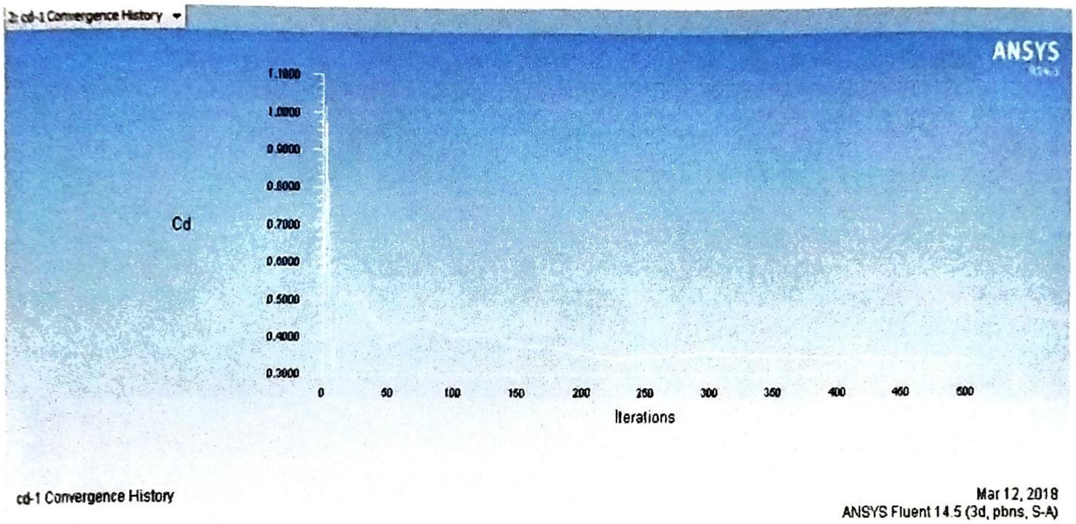


Fig 5.8.3.1 FOR CD

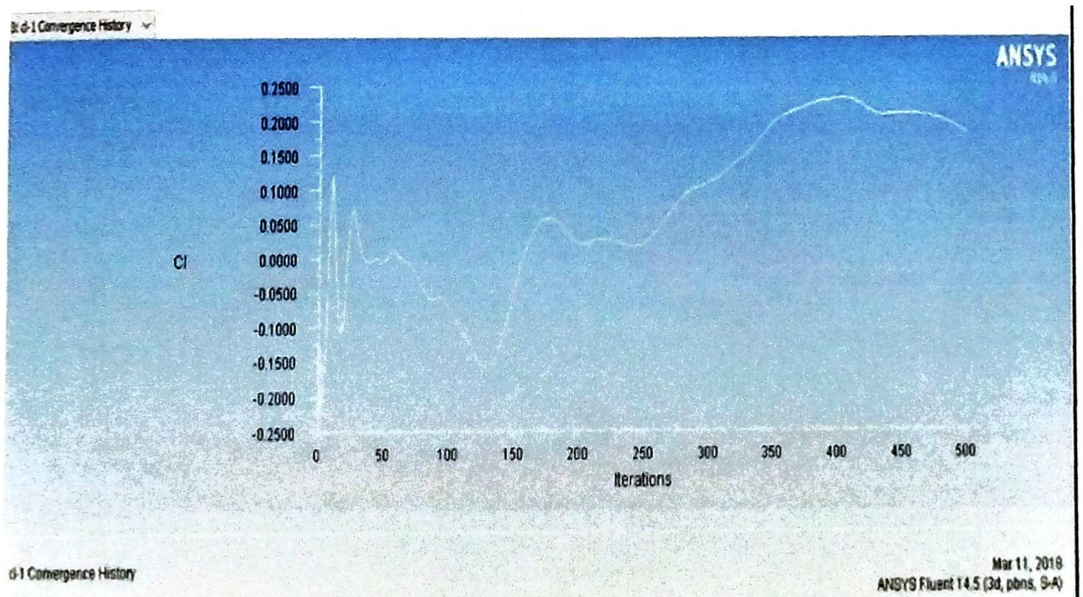


Fig 5.8.3.2 FOR CL

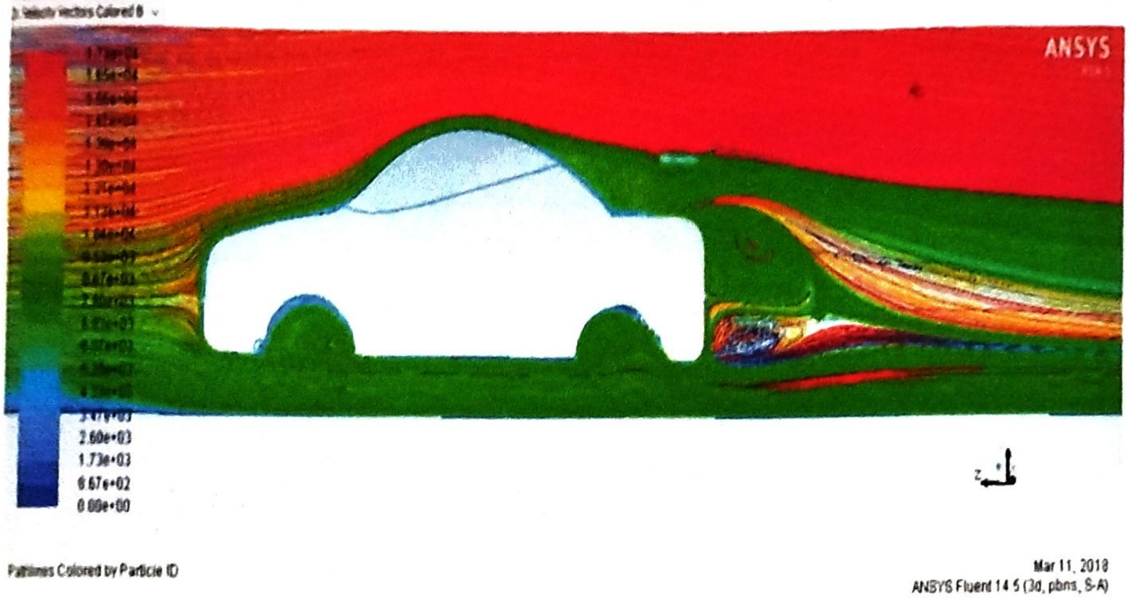


Fig 5.8.3.3PATHLINES-1

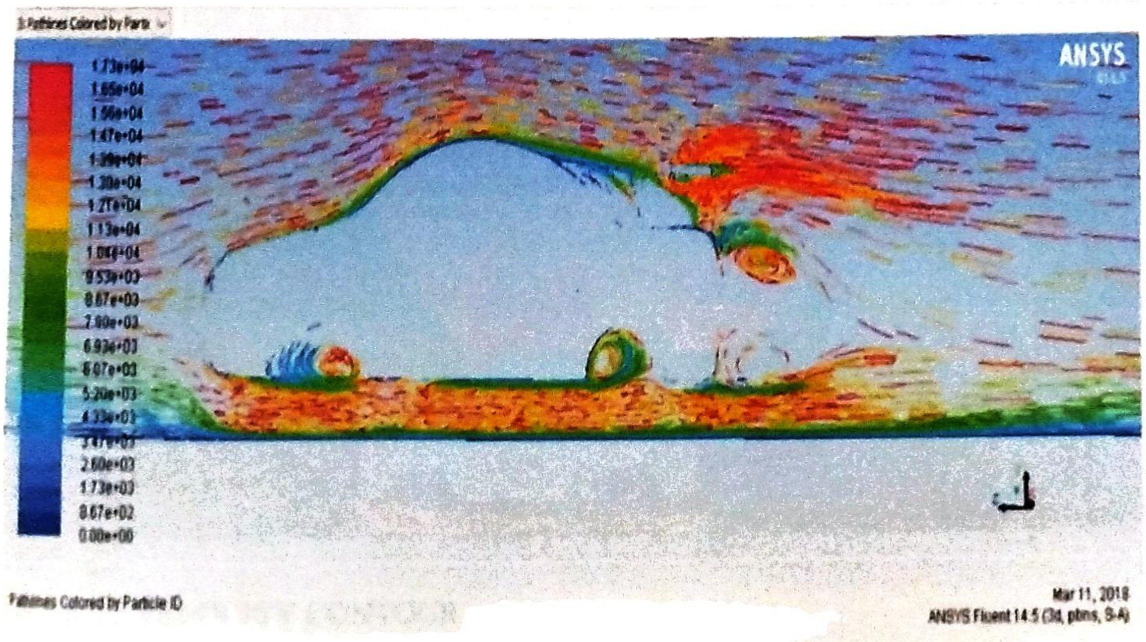


Fig 5.8.3.4 PATHLINES

Fig 5.8.3.5 STATIC PRESSURE

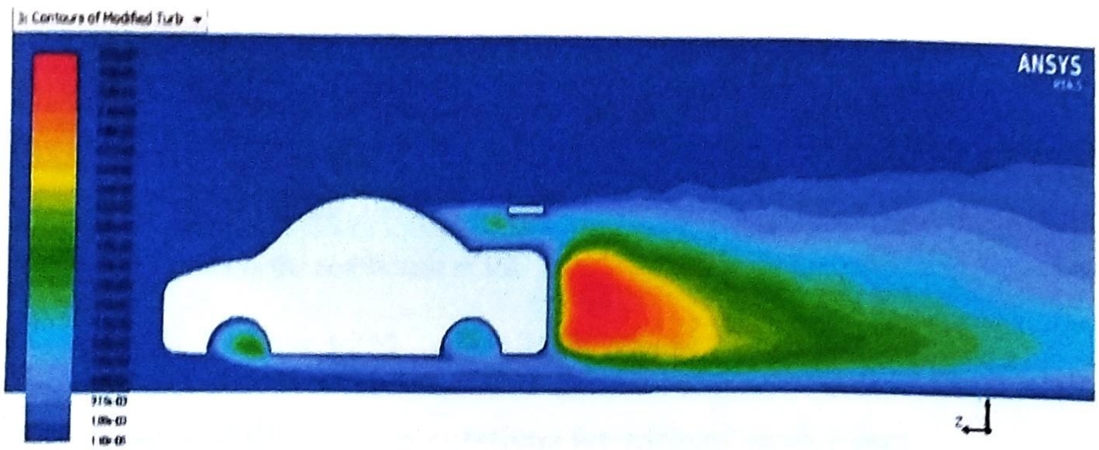


Fig 5.8.3.6 TURBULANCE

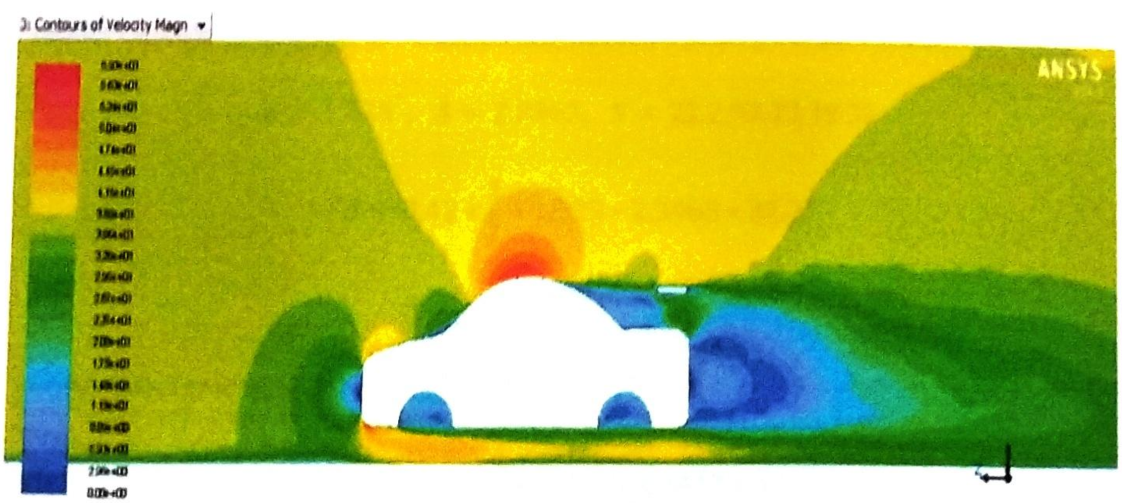


Fig 5.8.3.7 VELOCITY CONTOUR

6 CALCULATIONS

- **Formula for finding the drag force:**

$$Fd = Cd\frac{1}{2}\rho A\,V^2 \text{ (N)}$$

$$Fl = Cl\frac{1}{2}\rho A\,V^2 \text{ (N)}$$

$where =$ ρ $is\ the\ density\ of\ the\ Air\ in\ kg/m^3.$

$A\ is\ the\ frontal\ area\ of\ the\ car\ in\ m^2.$

$V\ is\ the\ flow\ velocity\ in\ m/sec.$

Cd is the coefficient of drag

Cl is the coefficient of lift

$$\rho = 1.225\ ,\ A = 2.3463,\ \ V = 22.2,33.3,38.3$$

6.1 Drag and lift force calculations for without spoiler car:

Drag Force:

Model Calculations: 1

$$\rho = 1.225\ ,\ A = 2.3463,\ \ V = 22.2,33.33,38.33$$

$$Fd = 0.42 * \frac{1}{2} * 1.225 * 2.3463 * 22.2^2$$

$$= 297.47 \text{ N}$$

Model Calculations: 2

$$Fd = 0.425 * \frac{1}{2} * 1.225 * 2.3463 * 33.3^2$$

$$= 678.49 \text{ N}$$

Model Calculations: 3

$$Fd = 0.43 * \frac{1}{2} * 1.225 * 2.3463 * 38.33^2$$

$$= 907.89N$$

Lift Force:

Model Calculations: 1

$$\rho = 1.225 \;,\; A = 2.3463, \;\; V = 22.2,33.3,38.3$$

$$Fl = 0.257 * \frac{1}{2} * 1.225 * 2.3463 * 22.2^2$$

$$= 182.02 \;\; N$$

Model Calculations: 2

$$Fl = 0.258 * \frac{1}{2} * 1.225 * 2.3463 * 33.3^2$$

$$= 411.88 \; N$$

Model Calculations: 3

$$Fl = 0.275 * \frac{1}{2} * 1.225 * 2.3463 * 38.33^2$$

$$= 580.63 \; N$$

6.2 Drag and lift force calculations for with spoiler car:

Drag Force:

Model Calculations: 1

$$\rho = 1.225 \ , \ A = 2.3463, \ \ V = 22.2,33.3,38.3$$

$$Fd = 0.36 * \frac{1}{2} * 1.225 * 2.3463 * 22.2^2$$

$$= 254.97 \ \ N$$

Model Calculations: 2

$$Fd = 0.346 * \frac{1}{2} * 1.225 * 2.3463 * 33.3^2$$

$$= 542.79 \ \ N$$

Model Calculations: 3

$$Fd = 0.345 * \frac{1}{2} * 1.225 * 2.3463 * 38.33^2$$

$$= 728.42 \ \ N$$

Lift Force:

Model Calculations: 1

$$\rho = 1.225 \ , \ A = 2.3463, \ \ V = 22.2,33.3,38.3$$

$$Fl = 0.175 * \frac{1}{2} * 1.225 * 2.3463 * 22.2^2$$

$$= 123.94 \ \ N$$

Model Calculations: 2

$$Fl = 0.18 * \frac{1}{2} * 1.225 * 2.3463 * 33.3^2$$

$$= 287.36 \quad N$$

Model Calculations: 3

$$Fl = 0.19 * \frac{1}{2} * 1.225 * 2.3463 * 38.33^2$$

$$= 401.16 \quad N$$

7 RESULTS AND DISCUSSIONS

The exterior profile design of a hatch back car has developed and analysis is carried out to find different performance characteristics of aerodynamics such as Drag coefficients, Lift coefficients, Drag forces, Lift forces, Contours and Vectors, Turbulent kinetic energy velocity streamline flows at various speed limits such as 80kmph, 120kmph and 140kmph. The results that observed in different cases are as follows;

- **Co-efficient of drag for both cases:**

Co-efficient of drag(Cd)			
No	**Speed**	**Without spoiler**	**With spoiler**
1	**80 kmph**	0.42	0.36
2	**120kmph**	0.425	0.346
3	**140 kmph**	0.43	0.345

Fig 7.1 Co-efficient of drag for both cases:

a. From the above results, the Cd value of with-spoiler car is less than the value of without spoiler car.

b. On increasing the velocity of the vehicle, the drag is increasing for without spoiler car.

c. Hence, there is a reduction in the drag co-efficient value, which has a considerable advantage in the aerodynamic performance.

d. Reduction of drag value indicates the reduction of drag force, which leads to decrease the fuel consumption in which we mainly concentrated.

- **Co-efficient of lift for both cases:**

Co-efficient of lift(Cl)			
No	**Speed**	**Without spoiler**	**With spoiler**
1	**80 kmph**	0.257	0.175
2	**120kmph**	0.258	0.18
3	**140 kmph**	0.275	0.19

Fig 7.2 Co-efficient of lift for both cases:

a. From the above results, the Cl value of with-spoiler car is less than the value of without spoiler car.
b. On increasing the velocity of the vehicle generally the Cl is increasing for both cases but the value of Cl with spoiler is less than the without spoiler.
c. Which is an advantageous for our analysis; decrease in lift co-efficient causes the reduction of the lift forces, which is the major concern in the high-speed cars.
d. Low lift prevents the losing the contact with the ground.
e. Reduction of lift value indicates the reduction of lift force, which leads to decrease the accidents and give the sufficient control over the car.

Drag force and lift forces for various speeds:

Drag force(N)			
Sl No:	**speed in m/s**	**without spoiler**	**with spoiler**
1	22.22	298.007	255.434
2	33.33	678.498	552.377
3	38.33	907.895	728.427
Lift force(N)			
Sl No:	**speed in m/s**	**without spoiler**	**with spoiler**
1	22.22	182.352	124.169
2	33.33	411.888	287.364
3	38.33	580.630	401.163

Fig 7.3 Drag force and lift forces for various speeds

a. Here the forces calculated by the obtained Cd and Cl values by the CFD analysis.
b. The drag force reduced with the addition of the spoiler in this analysis which is the major concern of this project.
c. Major problem in high speed driving is that the vehicle may lose the contact with the ground and causes accidents.
d. In order to avoid losing the contact with the ground lift force should be less, we can get it by adding the spoiler to our new design.

GRAPHS:

1) Cd VS Speed without Spoiler

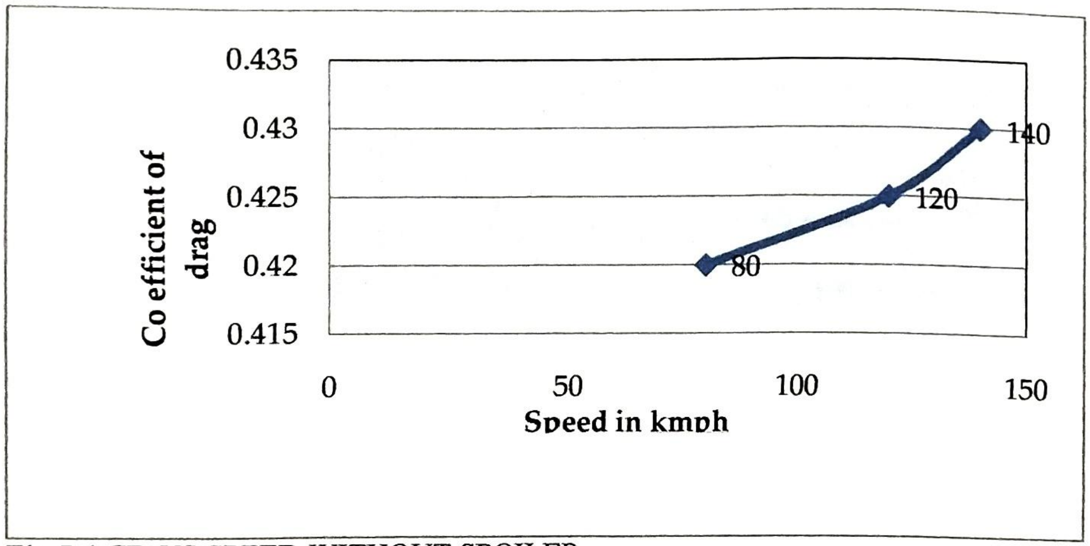


Fig 7.4 CD VS SPEED WITHOUT SPOILER

- On increasing the velocity of the vehicle the drag co-efficient is increasing without spoiler.

2) Cd VS Speed with spoiler

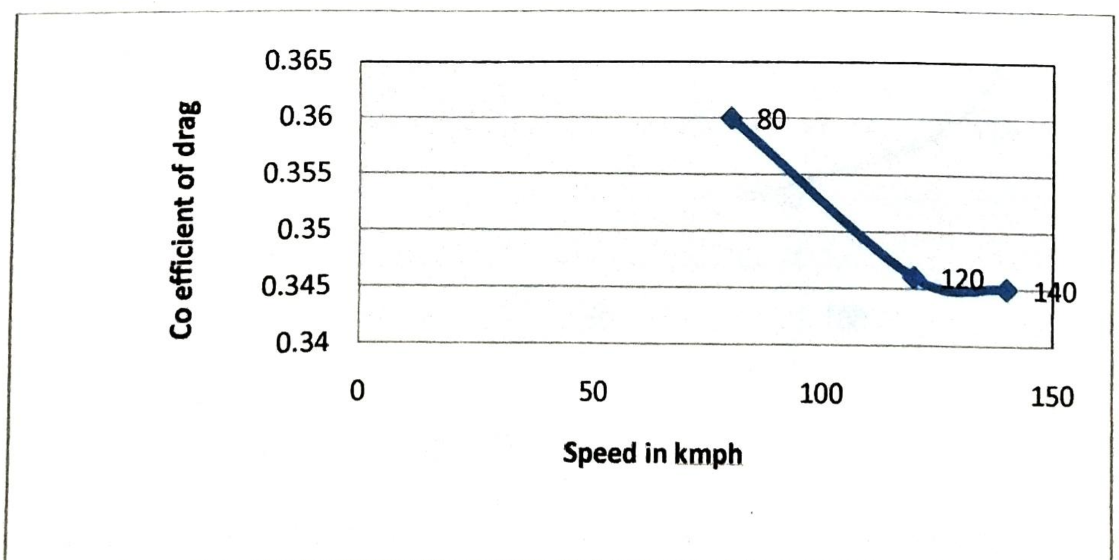


Fig 7.5 CD VS SPEED WITH SPOILER

3) Cl vs Speed without spoiler

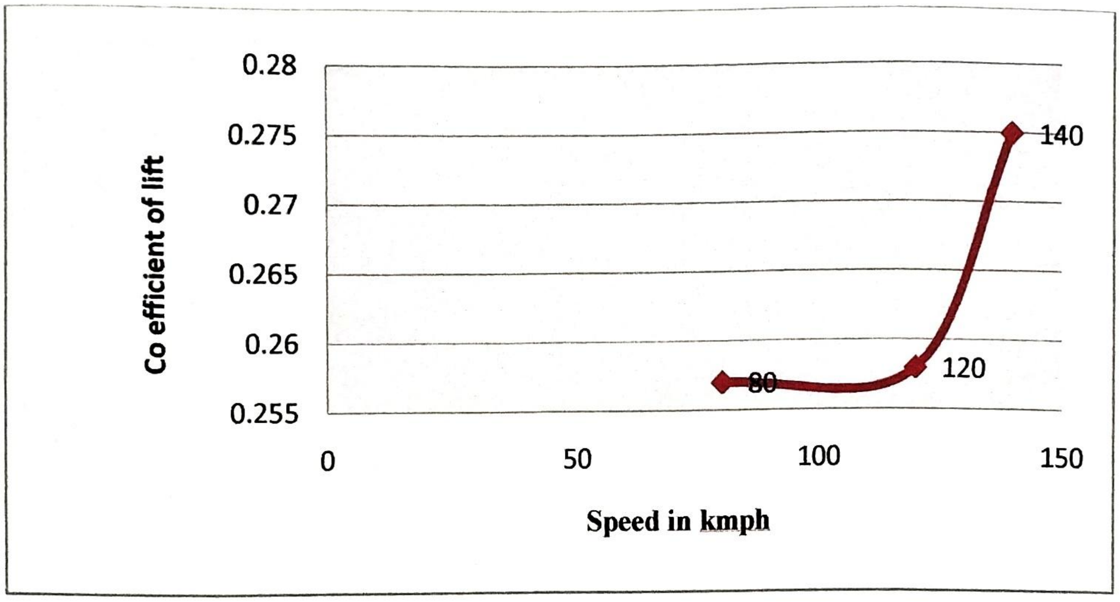


Fig 7.6 CL VS SPEED WITHOUT SPOILER

The lift co-efficient is increasing when the vehicle speed is increasing which is not appreciable.

4) Cl vs Speed With Spoiler

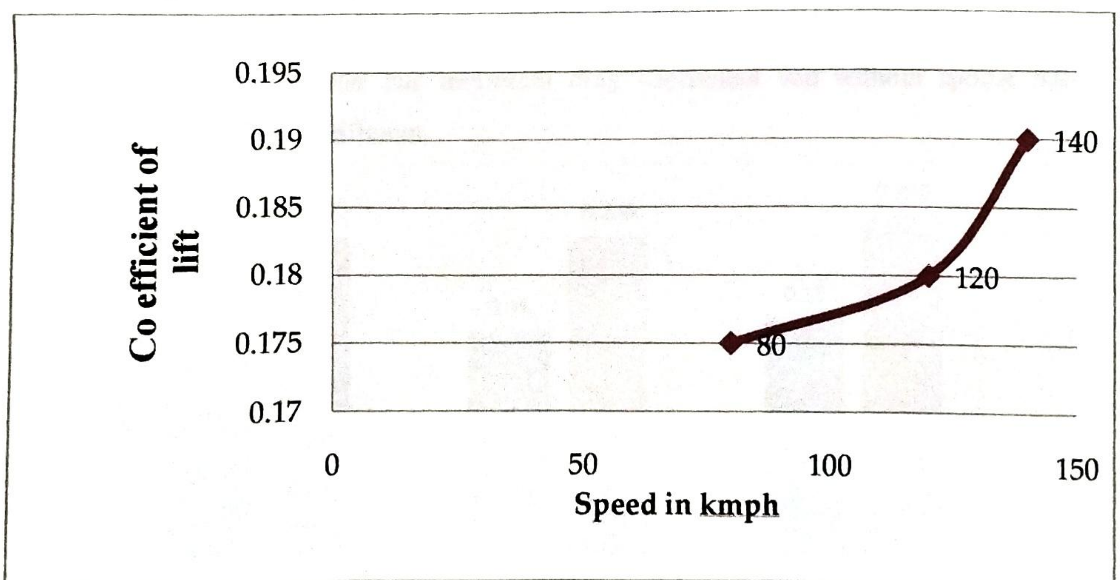


Fig 7.7 CL VS SPEED WITH SPOILER

On increasing the speed of vehicle the Cl value decreasing due to incorporation of the spoiler.

Comparison of Cd between with and without spoiler cars.

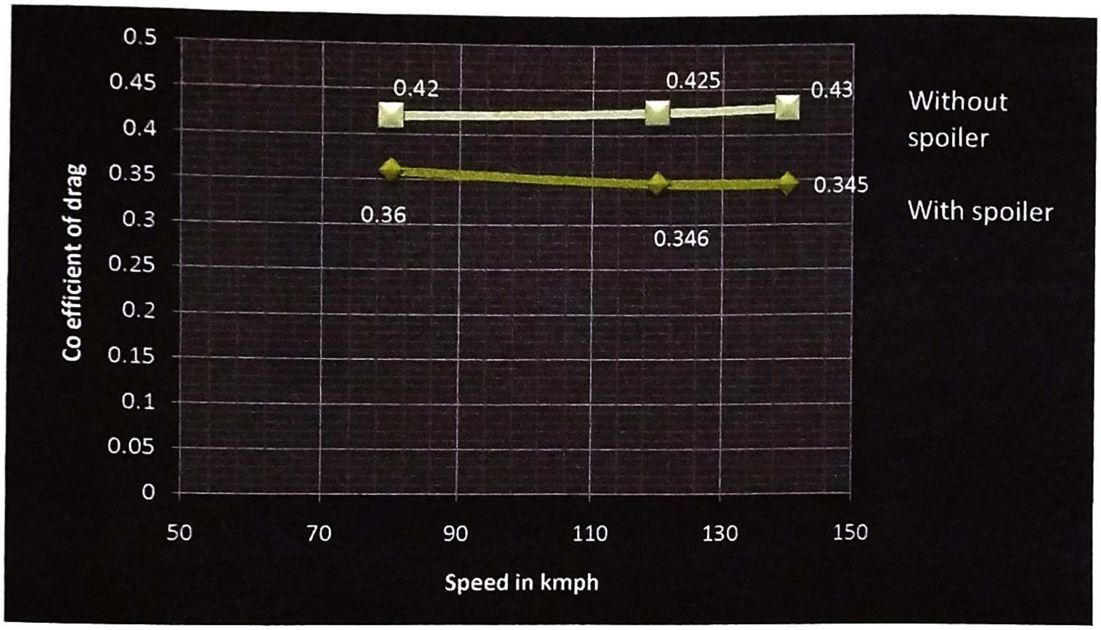
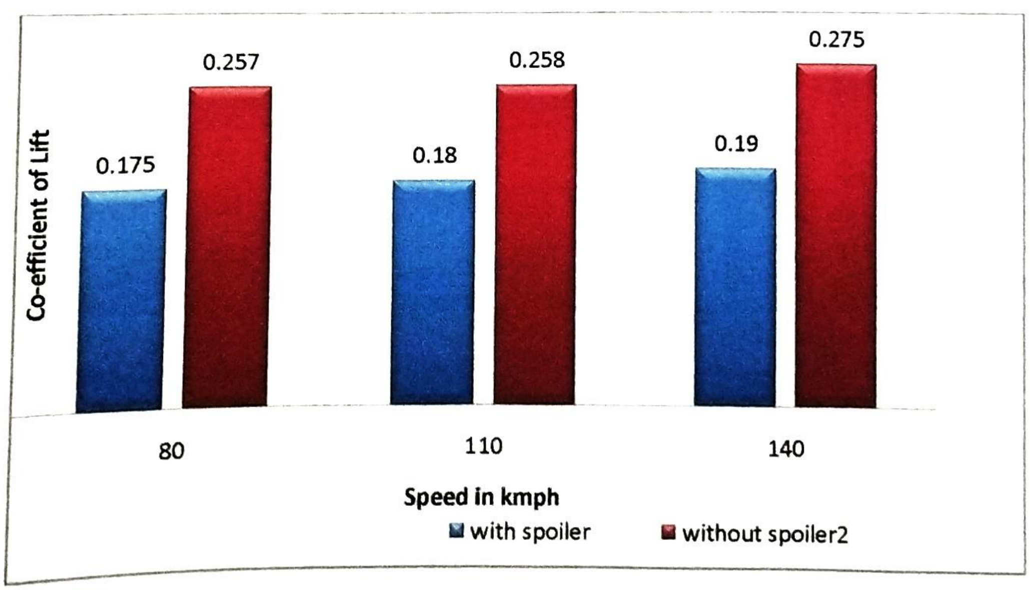


Fig 7.8 COMPARSION OF CD BETWEEN WITH AND WITHOUT SPOILER

a. As we are familiar, that drag coefficient should be 0.1 to 0.5 for Passenger car. The results that we obtained are within the standard.

b. The car with spoiler has minimum drag coefficient and without spoiler has maximum drag coefficient.



Comparison of Cl between with and without spoiler cars.

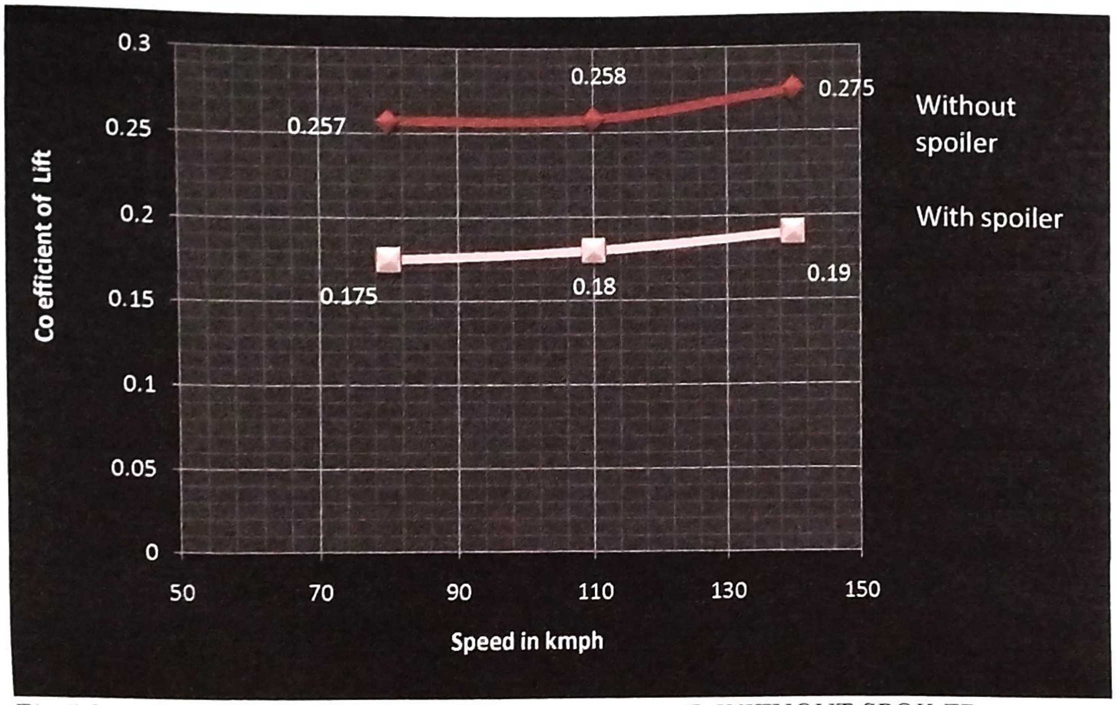
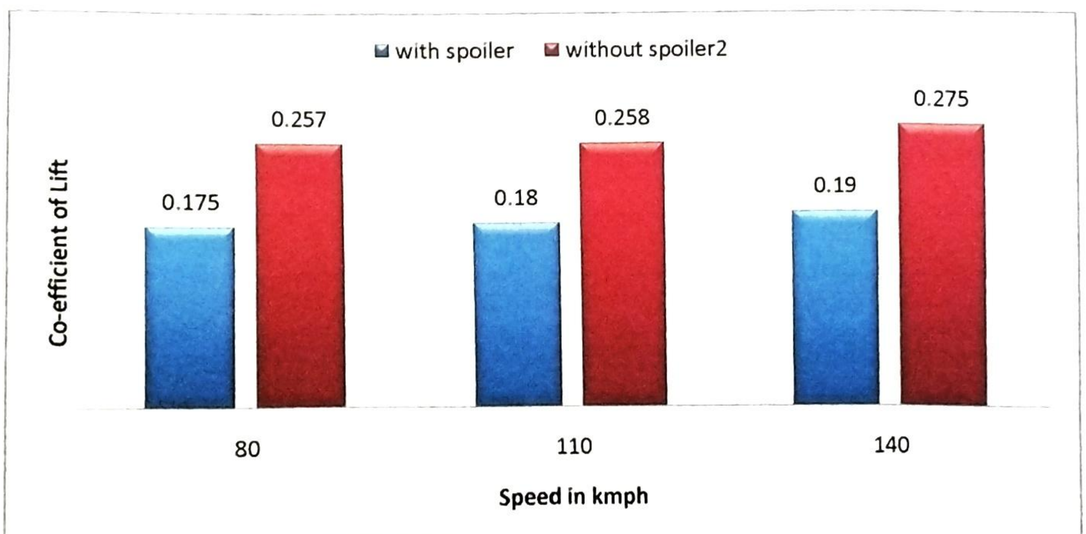


Fig 7.9 COMPARSION OF CL BETWEEN WITH AND WITHOUT SPOILER



The car with spoiler has minimum lift coefficient and without spoiler has maximum lift coefficient.

8 CONCLUSION

The present work aims to reduce the drag force, which improves fuel utilization. Hence, we reduced the drag force on the vehicle by adding the spoiler.The decrement in drag co-efficient is nearly 30%.

Therefore, we are working on a passenger car with spoiler and without spoiler. The spoiler used to reduce the aerodynamic drag force. The design of passenger car with and without spoiler has done on SOLIDWORKS 2017 and the model is imported to ANSYS 14.5 in that we used ANSYS FLUENT for the CFD analysis.

The analysis completed within the stipulated timeand we foundthe drag and lift forces at different highway speeds such as 80kmph, 120kmph and 140kmph. The study of this work proposes an effective passenger car based on CFD approach to obtain the drag and lift forces of the car with and without spoiler that is concerned& we successfully met the requirements of the new designed passenger car.

The results obtained in the form of forces and coefficientsare compared. It has been observed that the passenger car model with spoiler had lower drag and lower lift while for without spoiler has more lift and more drag. In addition of spoiler avoids the accidents.

9 References

1. Simon Watkins, Gioacchino Vino, *"The effect of vehicle spacing on the aerodynamics of a representative car shape"*. Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) p.1232–1239.
2. R.B.Sharma, *"CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction"*. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 7, Issue 5 (Jul. - Aug. 2013), PP 28-35. www.iosrjournals.org.
3. Dan Barbut and EugenMihai, Negrus, *"CFD analysis for road vehicles"* - case study incas bulletin, Volume 3, Issue 3/ 2011, pp. 15 – 22 ISSN 2066 – 8201 DOI: 10.13111/2066-8201.2011.3.3.2.
4. C.J. Baker, N.D. Humphreys, *"Assessment of the adequacy of various wind tunnel techniques to obtain aerodynamic data for ground vehicles in cross wind"*. Journal of Wind Engineering and Industrial Aerodynamics, 60 (1996) p .49-68.
5. Mustafa Cakir, *"Cfd study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle"*. Santa Clara University, Mechanical Engineering Masters Theses. Paper 1, December 2012.
6. Mahmoud Khaled, HichamElHage, FabienHarambat, and HassanPeerhossaini, *"A parametric analysis of aerodynamic forces on a simplified body"*. Journal of Wind Engineering and Industrial Aerodynamics 107–108 (2012) p. 36–47.
7. Johan zaya, *"Aerodynamic Optimization of Ground Vehicles with the Use of Fluent'sAdjoint Solver"*. Master's Thesis in the Masters Programme Automotive Engineering, Department of Applied Mechanics Division of Vehicle Engineering and Autonomous Systems Road Vehicle Aerodynamics chalmers university of technology Göteborg, Sweden 2013, 2013:01.

8. Chien-Hsiung Tsai, Lung-Ming Fu, Chang-Hsien Tai, Yen-Loung Huang, Jik-Chang Leong, *Computational aero-acoustic analysis of a passenger car with a rear spoiler*. Volume 33, Issue 9, September 2009, p. 3661-3673.
9. Xingjun HU, Rui ZHANG, Jian YE, Xu YAN, Zhiming ZHAO, *"Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics"*. 2011 International Conference on Physics Science and Technology, Physics Procedia 22 (2011) p.239 – 245.

10. J.P. Howell, *"The side load distribution on a Rover 800 saloon car under crosswind conditions"*. Journal of Wind Engineering and Industrial Aerodynamics 60 (1996) p.139-153.
11. Bhagirathzala, *"Aerodynamic performance of Sedan and hatchback car by experimental method and simulation by computational fluids dynamics"*. Gujarat Technical University 2011.
12. R.B.Sharma and Ram Bansal, *"CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction"*. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 7, Issue 5 (Jul. - Aug. 2013), PP 28-35.
13. Hugo G. Castro, Rodrigo R. Paz, Mario A. Storti, Victorio, *"Experimental and numerical studies of the aerodynamic behavior of simplified road vehicle Mechanical Computational"*. Vol XXIX, 15-18 November2010.p. 3291-3303