A

PROJECT REPORT

ON

AERODYNAMIC COMPARATIVE STUDY OF DIFFERENT VEHICLE MODELS

A dissertation submitted in partial fulfilment of the requirements of the Degree of Bachelor of Technology in Mechanical Engineering of the Maulana Abul Kalam Azad University of Technology, West Bengal.

By

Rajdeep Poulik (Roll – 10200717043)

Souradeep Koley (Roll – 10200717031)

Shubham Alvin Mondal (Roll – 10200717032)

Alapan Das (Roll – 10200717067)

Under the guidance of

Dr. Arijit Dutta

Assistant Professor of Department of Mechanical Engineering
Kalyani Government Engineering College, Kalyani

DEPARTMENT OF MECHANICAL ENGINEERING
KALYANI GOVERNMENT ENGINEERING COLLEGE
Kalyani, Nadia – 741235, West Bengal.

Year of Examination 2021

কল্যাণী - ৭৪১ ২৩৫
নদীয়া, পশ্চিমবঙ্গ

Kalyani 741 235
Nadia, West Bengal, India

পত্রাঙ্ক /Ref. No.:
তারিখ / Date :

কল্যাণী গভঃ ইঞ্জিনিয়ারিং কলেজ
Kalyani Government Engineering College
(Govt. of West Bengal)

Certificate of Approval

I hereby recommended that the project done under my supervision by **Rajdeep Poulik (10200717043), Souradeep Koley (10200717031), Shubham Alvin Mondal (10200717032)** and **Alapan Das (10200717067)** entitled "**AERODYNAMIC COMPARATIVE STUDY OF DIFFERENT VEHICLE MODELS**" be accepted in partial fulfilment of the requirements of the degree of Bachelor of Technology in Mechanical Engineering of the Maulana Abul Kalam Azad University of Technology, West Bengal.

(DR. ARIJIT DUTTA)
Assistant Professor
Department of Mechanical Engineering
Kalyani Govt. Engineering College

(DR. SANTANU DAS)
H.O.D., Department of Mechanical Engineering
Kalyani Govt. Engineering College

OBJECTIVES OF THE PROJECT

- To develop a model of passenger car using standard dimensions.
- To develop a model of racing car using standard dimensions.
- To determine drag and lift coefficients of various car models.
- Comparison of pressure and velocity field distributions of airflow of different car models using same input parameter.
- To optimize design parameters of car models to minimize drag.

ACKNOWLEDGEMENT

First of all, we would like to convey our sincere gratitude to our project guide **Dr. Arijit Dutta, Assistant Professor of the Mechanical Engineering Department of Kalyani Govt. Engineering College** for assisting us throughout our work with their advice and kindly help without any sort of hesitation at any time. Not only have he given us the roots to design everything regarding this project but also the wings to fly and proceed with any technology as we wished whenever.

We would feel our efforts more than rewarded if student and teachers of engineering aspect this report and would welcome and appreciate their comments for this project work.

We may have missed out a few names, our sincere apologies are due for any such inadvertent oversight. We would also like to thank professors of our faculty for their cooperation and help. At the end I wish to thank our friends who have helped us during our tenure of this project.

If there is any driving force that kept us going during our most troubled and anxious moments, it is encouragement, motivation and inspiration of our classmates. So, we extend our heartiest wishes and deep love to them.

We thank all for endorsing our work.

RAJDEEP POULIK	SOURADEEP KOLEY	SHUBHAM ALVIN MONDAL	ALAPAN DAS
10200717043	10200717031	10200717032	10200717067
©7076235730	©9163845063	©9064563191	©8240516253

ABSTRACT

Aerodynamic characteristic of vehicles are of significant interest in reducing air resistance due to drag and lift, and in reducing fuel consumption. Most of the vehicles have a wide steep downward angle going from the rear edge of the roof down to the trunk of the vehicle. Air flowing across the roof at higher speed causes flow separation. The air flow becomes turbulent and a low pressure zone is created and hence increases drag. Reducing flow separation decreases drag, which increases fuel efficiency.

The objective of this project is to identify drag and lift coefficient of vehicle. Process simulation and analysis for the models of vehicle's design was conducted with Solidworks 2016 software and Analyzed using software Ansys fluent. The assessment and differences among the design is very meaningful for determining the design of more efficient design in the vehicle now. Results are also able to enrich the design of the vehicle in early development in the near future.

CONTENT	PAGE NO

1. INTRODUCTION 1-4
   1.1. BACKGROUND 1-2
   1.2. AERODYNAMIC DESIGN 3
   1.3. PROBLEMS STATEMENT 3
   1.4. THEORY OF AERODYNAMIC 3-4
   1.5. MATHEMATICAL MODELLING 4
2. LITERATURE SURVEY 5-7
   2.1. INTRODUCTION 5
3. THEORITICAL CONCEPT 8-12
   3.1. AERODYNAMIC FORCES 8-9
      3.1.1. DRAG 8-9
         3.1.1.1. DIFFERENT TYPES OF DRAG FORCE 8-9
            3.1.1.1.1. PRESSURE DRAG 8
            3.1.1.1.2. SURFACE DRAG 9
      3.1.2. LIFT 9
   3.2. TURBULENT AND LAMINAR BOUNDARY LAYER 9-10
   3.3. SREAMLINED BODY 10
   3.4. BLUFF BODY 10-11
   3.5. WORKING OF CFD 11-12
4. DESIGN APPROACH 13-23
   4.1. DESIGN PROCEDURES OF MODELS OF CARS 13
      4.1.1. SKETCH PROCEDURE 13
      4.1.2. DIMENSION PROCEDURE 13
   4.2. DIFFERENT TYPES OF MODELS OF CARS 14-23
      4.2.1. CAR MODEL 14-15
      4.2.2. COMMERCIAL CAR MODEL 15-17
      4.2.3. RACING CAR MODEL 17-19
      4.2.4. OMNI MODEL 19-21
      4.2.5. DRAG MINIMIZED MODEL 21-23
5. COMPUTATIONAL SIMULATION SETUP 24-51
   5.1. DEFINE THE PROBLEM 24
   5.2. FLOW DOMAIN OR COMPUTATIONAL MODEL 24-26
   5.3. CREATING MESH 27-34
   5.4. INITIAL BOUNDARY CONDITION 35
   5.5. ESTABLISHING THE GOAL 35

5.6. RESULT OF SIMULATION 35-51

6. DRAG COEFFICIENT MINIMIZATION 52-59

6.1. FLOW DOMAIN OR COMPUTATIONAL MODEL 52

6.2. CREATING MESH 53-54

6.3. INITIAL BOUNDARY CONDITION 55

6.4. ESTABLISHING THE GOAL 55

6.5. RESULT OF SIMULATION 55-59

7. RESULT AND DISCUSSION 60-61

8. CONCLUSION 62

9. FUTURE ASPECT 63

10. REFERENCES 64-65

1.INTRODUCTION

Motor vehicle aerodynamics is a complex subject because of the interaction between the air flow and the ground, and the complicated geometrical shapes that are involved. (R Stone 2004). Dr. Dominey of Durham University points out that the stability of regulations has led to a convergence of design which means that competitive circuit performance now depends on fine tuning of aerodynamic design (Fenton J 1999).

The magnitude of the aerodynamic forces is depending upon the velocity of the vehicle and density of the air. These two factors can affect the performance of the vehicle. At different velocity we would get different amount of drag force and also at different density of the air we would get different amount of force. Density of the air is depending upon the ambient temperature. If the temperature is too low then the density of the air become high which cause drag and lift forces will increase. If temperature is gradually reduces so density of the air will also reduce which cause decreasing drag and lift force.

To study the behaviour of the aerodynamics the Solid work flow simulation were chosen and design of the bluff body the car model were made using solid-work and the flow simulation was done using Ansys fluent. This model is a popular bluff body with different backlight angle used by the different researchers and in different literature to explain the theory of coefficient of drag and the coefficient of lift.

1.1 BACKGROUND

The vehicle shapes and designs until the recent years were just only the symbol of fashion which normally doesn't have any concern with the aerodynamics of the car. In early 1st 70's the step was taken for the aerodynamics in the production for the cars because of the price increase in the fuel prices. Since then the only factors which used to consider were the vehicle mass and the engine efficiency apart from the road vehicle aerodynamics properties. In the starting the research on aerodynamics mostly focused on the drag reduction while on the other hand the lift coefficients were only practiced for the race cars. So with the advancement of engines in the automotive industry the speed of the car goes on increasing. The researchers also staring to analyse the drag and the lift reduction, and how they can minimize these forces which effect the efficiency and the stability of the on the road while it's running with the speed. Low drag model was built in early 1980's which is also known as the 'jelly-mould' shape. These type model cars usually have the less drag acting on the car. However, it has the low stability in the crosswinds conditions. The following figure shows the shape of the car. Those cars have the slop base so in these kinds of slopes the flow doesn't have the large-separations.

Fig. 1.1: Jelly Mould Shape (Centurion Magazine, 2011)

So as the time passing the speed of the car goes on increasing so the driver safety and the car performance, it is now very important to that all the aerodynamics and the moments acting on the surfaces of the car are considered.

The wind tunnel testing and the CDF flow simulation were carried out to study the aerodynamics of the car all the other moment forces. Wind tunnel is actually the large tunnel like apparatus which is able to produce the airstream with the known velocity of running car, airplanes etc. This is normally used to investigate the air flow over the cars, airplane etc. the following figure shows the structure of the wind tunnel.

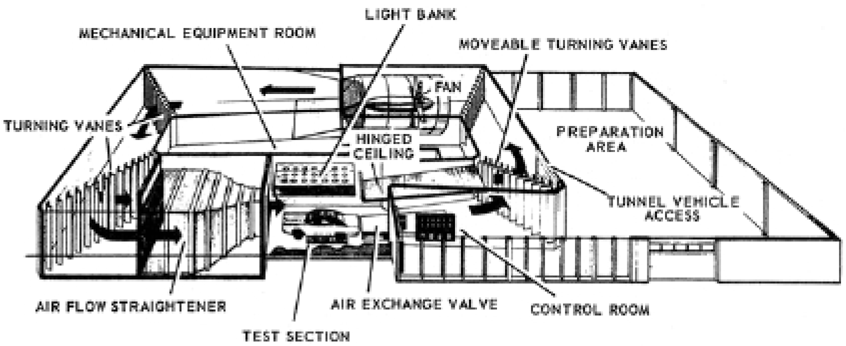

Fig. 1.2: Wind Tunnel (Allpar, 2014)

The standard way to study the flow over the car and the airplane surfaces are the CFD flow simulation over the car 3d models. This way is way cheaper than the wind tunnel because this involves the designing part and then CFD flow simulation which saves lots of times and the effort.

1.2 AERODYNAMIC DESIGN

It is the design of the car which has the low drag and the low coefficient of lift. Cars are like the bluff bodies which has the drag coefficient of ranging from 0.3-0.4.

The researches have trying to design which has the low number of drag coefficient and the coefficient of lift and other side moments of the cars. This increases the stability of the car while it's moving and also gives the driver the confidence in handling properly. The back shape has more effect on reducing the drag and the lift.

1.3 PROBLEM STATEMENT

Drag and lift will cause many problems on the performance of car model like instability, noise and fuel consumption. Thus in this project the CAD model of sedan and setback bodies was simulated and analyse of their aerodynamic especially on the drag and lift estimation. In addition using CFD and FEM analysis as a possible procedure were develop the drag estimation and aerodynamic studies on the body due to no wind tunnel in UMP.

1.4 THEORY OF AERODYNAMICS

In this section, the fundamental of aerodynamic is discussed to gain understanding in doing analysis of the project. The basic equation and term in aerodynamic field or fundamental of fluid mechanics such as Bernoulli's Equation, pressure, lift and drag coefficient boundary layer separation and shape dependence are studied.

BERNOULLI'S EQUATION:-

Aerodynamics play main role to define road vehicle's characteristic like handling, noise performance and fuel economy. The improvement on the characteristic related through the drag force which is ruled by Bernoulli's Equation. Basic Assumption of Bernoulli's Equation is as follows:

1. Viscous effects are assumed negligible.
2. The flow is assumed to be steady.
3. The flow is assumed to be incompressible.
4. The equation is applicable along streamline.

$$p + \frac{1}{2}\rho v^2 = constant$$

This equation depicts that the increasing of velocity will case the decrease in static pressure and vice versa. On the movement of road vehicle will produce a distribution velocity that's create the skin friction due to viscous boundary layer which act as tangential forces (shear stress) then contribute drag. Besides that, force due to pressure also created which acts perpendicular to the surface then contribute both lift and drag forces. The Bernoulli's Equation from equation (2.1) gives the important result which is:

Static pressure + Dynamic Pressure = Stagnation Pressure.

1.5 MATHEMATICAL MODELLING

In system Aerodynamic Vehicle been analyzed with different parameter like Force velocity and temperature. The wall shear stress is analyzed during running process and the effect of skin friction coefficient of external body of the car has also examined, in additional to varying the orientation also the heat transfer and shear stress is taken in consideration with the wall surface of the car body. The equations governing this problem are those of Navier-Stokes along with the energy equation. The Navier-Stokes equations are applied to incompressible flows and Newtonian fluids, including the continuity equation and the equations of conservation of momentum on the x and y:

According to equations

$$\frac{\partial u_2}{\partial t}+u_1\frac{\partial u_1}{\partial x_1}+u_2\frac{\partial u_1}{\partial x_2}=$$

$$-\frac{1}{\rho}\frac{\partial \rho}{\partial x_2}+\nu\left(\frac{\partial^2 u_1}{\partial x_1^2}+\frac{\partial^2 u_1}{\partial x_2^2}\right)+g\beta(T-T\infty)$$

$$\frac{\partial u_1^*}{\partial x_1^*}+\frac{\partial u_2^*}{\partial x_2^*}=0$$

x1 momentum equation

$$\frac{\partial u_1^*}{\partial t^*}+u_1^*\frac{\partial u_1^*}{\partial x_1^*}+u_2^*\frac{\partial u_1^*}{\partial x_2^*}=-\frac{\partial p^*}{\partial x_1^*}+\Pr\left(\frac{\partial u_1^*}{\partial x_1^*}+\frac{\partial u_1^*}{\partial x_2^*}\right)$$

x2 momentum equation

$$\frac{\partial u_2^*}{\partial t^*}+u_1^*\frac{\partial u_2^*}{\partial x_1^*}+u_2^*\frac{\partial u_1^*}{\partial x_2^*}=-\frac{\partial p^*}{\partial x_2^*}+\Pr\left(\frac{\partial u_2^*}{\partial x_1^*}+\frac{\partial u_2^*}{\partial x_2^*}\right)$$

$$+Gr\Pr^2 T^*$$

Energy equation

$$\frac{\partial T^*}{\partial t^*}+u_1^*\frac{\partial T^*}{\partial x_1^*}+u_2^*\frac{\partial T^*}{\partial x_2^*}=\left(\frac{\partial^2 T^*}{\partial x_1^{*2}}+\frac{\partial^2 T^*}{\partial x_2^{*2}}\right)$$

2. LITERATURE SURVEY

2.1 INTRODUCTION

In early years there were no such considerations in the time of drawing the designs of the cars. It was in 1970s the increase in fuel consumption and the fuel prices put stress on the engineering to actually think about the body design. The aerodynamics researchers initially focused on the reduction of the drag force; on the other hand, there was the lift problem which caused the low vehicle stability. So there was great need of considering all the forces while designing the vehicle body perfectly. Computational Fluid Dynamics (CFD) played its vital role in determining the vehicle body shaping. To understand the clear explanation of the theory and the subject area of the topic the literature review has been split into different sections. The topics cover all the basics of aerodynamics of the car body, drag, lift and other different parameters for example height and the pitch which causes on the moving vehicle body. Then, the main area will be covered which is CFD simulation. The main model for the research is the Ahmed model to have the better understanding of the aerodynamics forces at the back light angle of the car. A moving object which is exposed to air always experience three main forces and movements respectively.

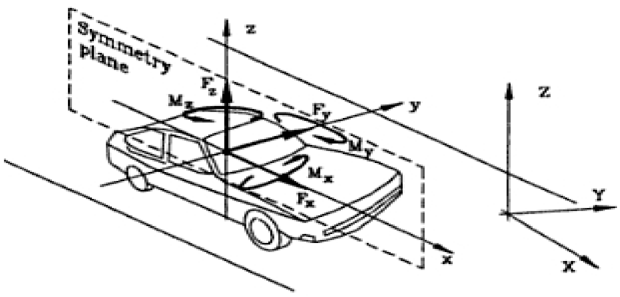

Fig. 2.1 : Forces and moments acting on the Car. (Genta, 2008)

These moments and forces which are acting on a car in the above figure are as follows:

- Earth fixed axis system XYZ : It can be described as the right angled frame fixed on the road. In which X and Y coordinates are in the horizontal plane however the Z is in the vertical direction pointing upwards. All these coordinates can be divided as there is 90 degrees angle between them. Figure 6: Forces and moments acting on the vehicle.

- Vehicle axis frame xyz. This is the right angled frame but this time it is assumed to be fixed with the vehicle and also it moves along with it. The z force called the lift force which is acting on upwards and x is the horizontal force which is moving opposite to the vehicle body called drag. At the last force on the side horizontally called the side force which causes the momentum force.

In the journal "Aerodynamic characteristics of automobile models" R.H. Heald told that:

"The drag coefficients were found to vary from 0.0018 lb./ft.*/m.p.h.to 0.0014 for the model representing an automobile over a span of 10 years in 1930s. Elimination of the fenders and other projections together with pronounced fairing of the body of one model reduced the drag coefficient to 0.0006 Lateral and longitudinal forces were also measured. The lateral force was found to vary approximately as the angle of the relative wind if this was less than 20° to the direction of motion of the automobile. Very little variation in longitudinal force coefficient was observed with this range of angles."

In the journal "Aerodynamic design of F1 and normal cars and their effect on performance"

Shobhit Senge et al. told that :

"Aerodynamics is a branch of dynamics concerned with studying the motion of air, particularly when it interacts with the solid object. Aerodynamic is a subfield of fluid dynamics and gas dynamics, with much theory shared between them. In this paper we are presenting the different forces acting on a car (drag force, lift force). The measurement of forces (computational fluid dynamics CFD and wind tunnel testing WTT). On the basis of forces and measurement the comparison has been done over Hindustan ambassador, Lamborghini Aventador LP 700-4 and the F1 car. After detailed observation and tests performed we obtained that F1 car has most aerodynamic of all the vehicles. The design is made in such a way that it cuts through the air with ease and channelize the air flowing over it to the rear wings. This results in a highly reduced drag and lift force acting on the car body. It in turn, generates more amount of down force making the car stable at high speeds. It is the pinnacle of racing technology. On the other hand, the Lamborghini Aventedor LP 700-4, is a full on super car. It was designed to give speed and performance in a coupe car, thus the body had to be designed such that there is minimum air resistance at high speed and proper cornering stability as well as drivability."

In the journal "Aerodynamic drag reduction for a generic truck using geometrically optimized rear cabin bumps" Abdellah Ait Moussa et al. told that:

"The continuous surge in gas prices has raised major concerns about vehicle fuel efficiency, and drag reduction devices over a promising strategy. In this paper, we investigate the mechanisms by which geometrically optimized bumps, placed on the rear end of the cabin roof of a generic truck, reduce aerodynamic drag. The incorporation of these devices requires proper choices of the size, location, and overall geometry. In the following analysis we identify these factors using a novel methodology. The numerical technique combines automatic modelling of the add-ons, computational fluid dynamics and optimization using orthogonal arrays and probabilistic restarts. Numerical results showed reduction in aerodynamic drag between 6% and 10%."

In the journal “Aerodynamic drag reduction of emergency response vehicles,” Taherkhani AR et al. told that :

“This paper presents the first experimental and computational investigation into the aerodynamics of emergency response vehicles and focuses on reducing the additional drag that results from the customary practice of adding light-bars onto the vehicles’ roofs. A series of wind tunnel experiments demonstrate the significant increase in drag that results from the light bars and show these can be minimized by reducing the flow separation caused by them. Simple potential improvements in the aerodynamic design of the light bars are investigated by combining Computational Fluid Dynamics (CFD) with Design of Experiments and meta modelling methods. An aerofoil-based roof design concept is shown to reduce the overall aerodynamic drag by up to 20% and an analysis of its effect on overall fuel consumption indicates that it offers a significant opportunity for improving the fuel economy and reducing emissions from emergency response vehicles. These benefits are now being realised by the UK’s ambulance services.”

In the journal “Calculation and optimization of aerodynamic drag of an open wheel car, ” Abdulkareem SH et al. told that:

“Aerodynamic drag reduction is one of the important factors to make a race car achieve a faster lap time. Additional drag is produced due to the air channel for radiator cooling of the student designed open-wheel race car. This paper presents the aerodynamic drag optimization of the race car through studying the effect of the angle of the radiator air channel numerically using ANSYS Fluent and experimentally using wind tunnel. A reduction of 12.7% in drag coefficient compared to the current setup is achieved by tilting the angle of cooling channel to 72.5 degree. Numerical results and experimental results show good agreement, a maximum deviation of 7.7% between numerical and experimental drag coefficient for case of the race car with driver included.”

3. <u>THEORETICAL CONCEPTS</u>

3.1 AERODYNAMIC FORCES

3.1.1 DRAG:

Drag can generate by two main perspectives:-

1) From the vehicles (body)
2) From the moving fluid

From the two perspectives, three major coefficients were produced from the two basic of aerodynamics forces.

The first force is pressure distributions that normal (perpendicular) force to the body which is will produce pressure, drag and lift coefficient.

The second force is shear force that tangential (parallel) to the surface of body's vehicle where is contribute drag coefficient only.

The net drag is produced by both pressure and shear forces, thus the drag coefficient (C_D) for a vehicle body can define as :-

$C_D = 2 * D / \rho * V^2 * A$;

Where D= Drag , A= Frontal Area , V= Velocity and ρ= Air Density ;

3.1.1.1 DIFFERENT TYPES OF DRAG FORCE

3.1.1.1.1 PRESSURE DRAG

This is the component which is identified on the external surface of the car. As when the vehicle moves with the forward direction of the air then the surface of the car experience the pressure which is vary over the different points of the car as shown in the following figure. To have a look it very closely the small area of the flat surface is considered then the force which is acting on the axis of the car the drag force depends on the magnitude of the pressure.

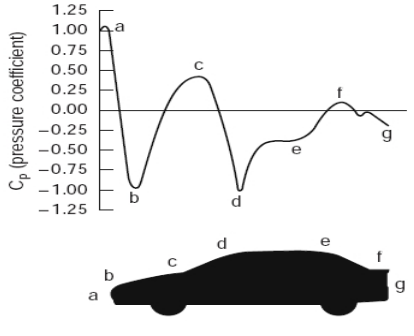

Fig. 3.1: Pressure distribution over a vehicle body.

3.1.1.1.2 SURFACE DRAG

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

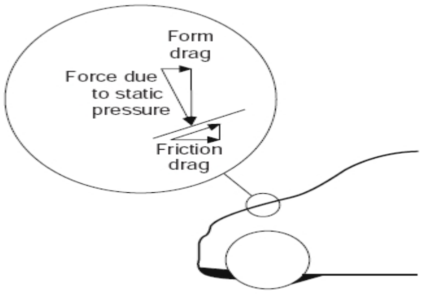

Fig. 3.2: Force acting on one surface element of the vehicle

3.1.2 LIFT:

The lift force can be determined if the distribution of dynamic pressure and shear force on the entire body are known. Therefore the lift coefficient (C_L) can indicate as:

$C_L = 2 * L / \rho * V^2 * A$;

Where L= Lift , A= Frontal Area , V= Velocity and ρ= Air Density ;

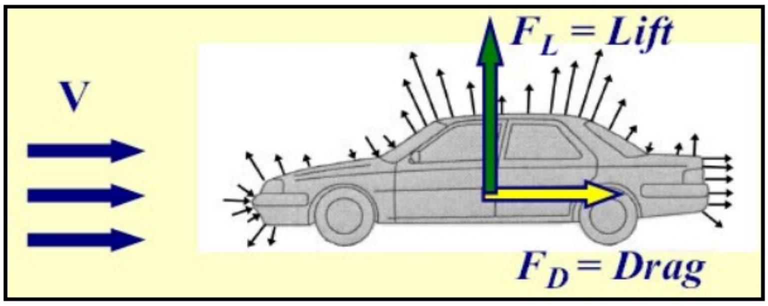

Fig. 3.3: Drag and Lift force due to pressure from velocity distribution

3.2 TURBULENT AND THE LAMINAR BOUNDARY LAYERS

Boundary layer flow has two distinguishable types when the flow layer passes over the surface of the vehicle. The following figure 5 shows the flow layers on the top of the bus. Smooth air flow can be seen in the front side. In this as it is clear that the moment of the outer layer is faster than the inner layer because the friction is effecting on the inside layer which has the direct contact to the surface. This type of flow is known as the Laminar flow**.** Mostly the turbulent body flow is streamlined.

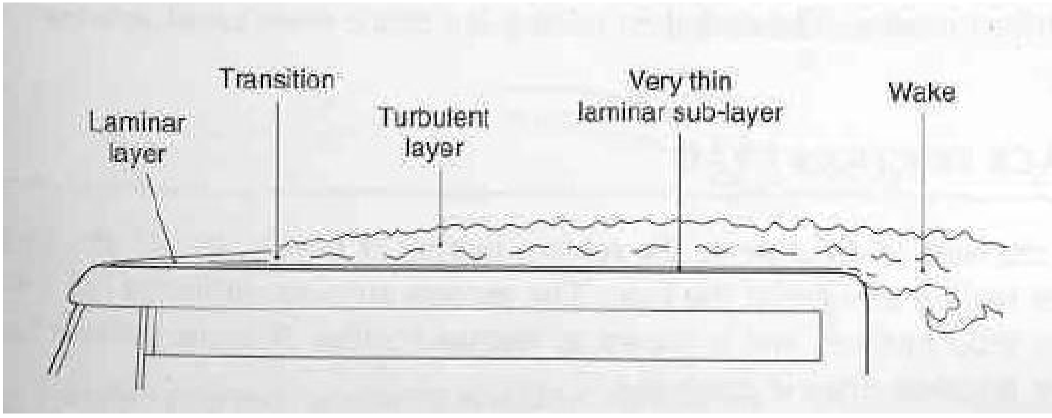

Fig. 3.4: Flow layers on the top of the bus.

3.3 STREAMLINED BODY

A streamlined body is a shape that decreases the friction drag between a fluid, such as air and water, and an object that passes through that fluid. Drag is the force which reduces the speed of the motion.

Streamlining in aerodynamics, the contouring of an object, such as an aircraft body, to reduce its drag, or resistance to motion through a stream of air:

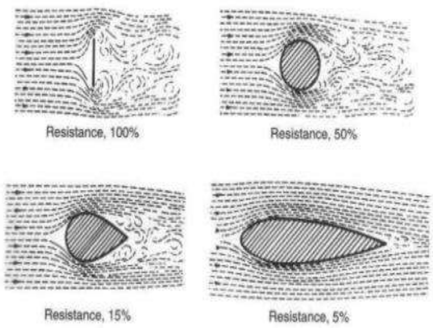

Fig. 3.5: Streamlined Body

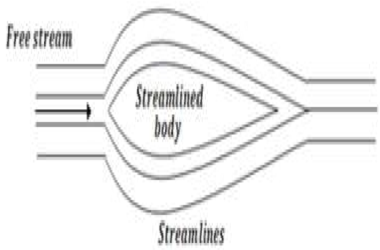

Fig. 3.6: Streamlined Body

3.4 BLUFF BODY

A bluff body can be defined as a body that, as a result of its shape, has separated flow over a substantial part of its surface. A body which when kept in fluid flow, the fluid does not touch the whole boundary of the object.

An important feature of a bluff body flow is that there is a very strong interaction between the viscous and non-viscous regions.

When the flow separates from the surface and the wake is formed, the pressure recovery is not complete. The larger the wake, the smaller is the pressure recovery and the greater the pressure drag.

The art of streamlining a body lies, therefore, in shaping its contour so that separation, and hence the wake, is eliminated, or at least in confining the separation to a small rear part of the body and, thus, keeping the wake as small as possible. Such bodies are known as streamlined bodies. Otherwise a body is referred to as bluff and a significant pressure drag is associated with it.

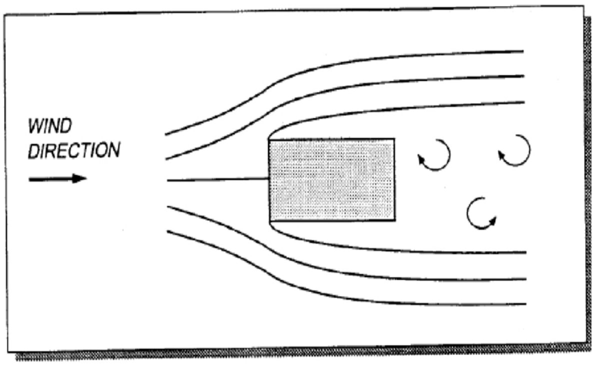

Fig. 3.7: Flow over a Bluff Body

3.5 WORKING OF CFD

CFD simulations create cells which is the division of fluid volume in a finite number of blocks. To divide the fluid into number of blocks or to create the finite number of cells a finite volume around the selected model is created which is known as the Computational domain. For the accuracy it all depends on the structure and the size of the cells. Following is the figure showing the computational domain.

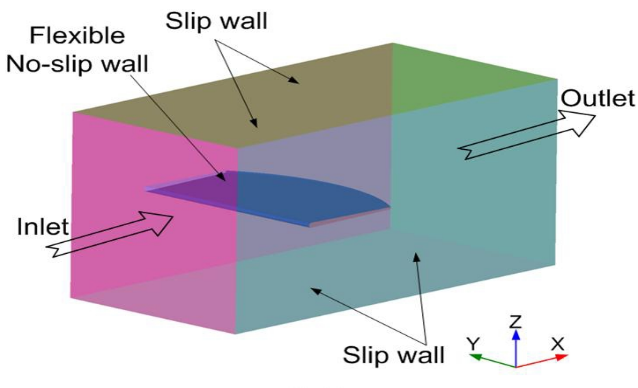

Fig. 3.8: Computational domain

For this project the Solidworks has been chosen to study the aerodynamics acting on the different vehicle models. After the solid model is formed then meshing is applied then the software automatically generates the mixture of contact, shells, solid and spring elements which are completely based on the geometry of the model. The following meshes will be automatically be created by the Solidworks CFD software.

SHELL MESH :

These types of meshes are for the sheet metals. The program automatically creates these types of meshes for the sheet metals which has the uniform thickness. For these kind of models the mesh is generated from the mid-surface. Following are the two examples of the Shell mesh

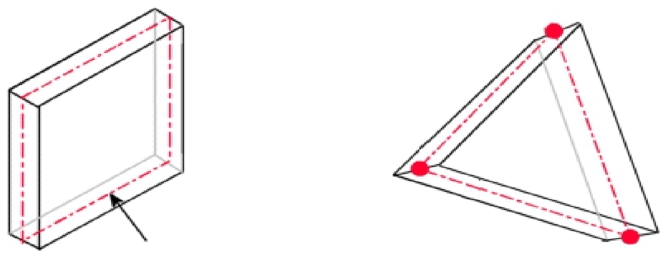

Fig. 3.9: Highlighted is a mid-surface sheet metal (left), nodes shown and the shell element created at mid surface (right) (SolidWorks, 2016)

MIXED MESH:

This mesh is for the complex geometries model.

SOLID MESH:

This mesh is created for the tetrahedral 3d mesh solid elements for all solid components for all the solid components in the parts. For the bulky objects tetrahedral elements are fitting.

So the process includes the formation of the model with all the parameters and geometry configuration. Then the next step will be making the conceptual domain around the model. This will prescribe the area where the tests have to perform. The end step is to perform the flow simulation. This will give the aerodynamics flow over the desire body or shape of the model.

4. DESIGN APPROACH

4.1 Design Procedures of Models of Cars

In the initial stage, the car models were completed in the CAD software Solidworks 2016. This software is basically chosen as it is faster to perform the CAD flow simulation tests and even better for the designing.

4.1.1 Sketch Procedure:-

- Open SOLIDWORKS part program.
- Select a top plane and sketch using different functions like line, centerpoint arc, rectangle, ellipse etc.

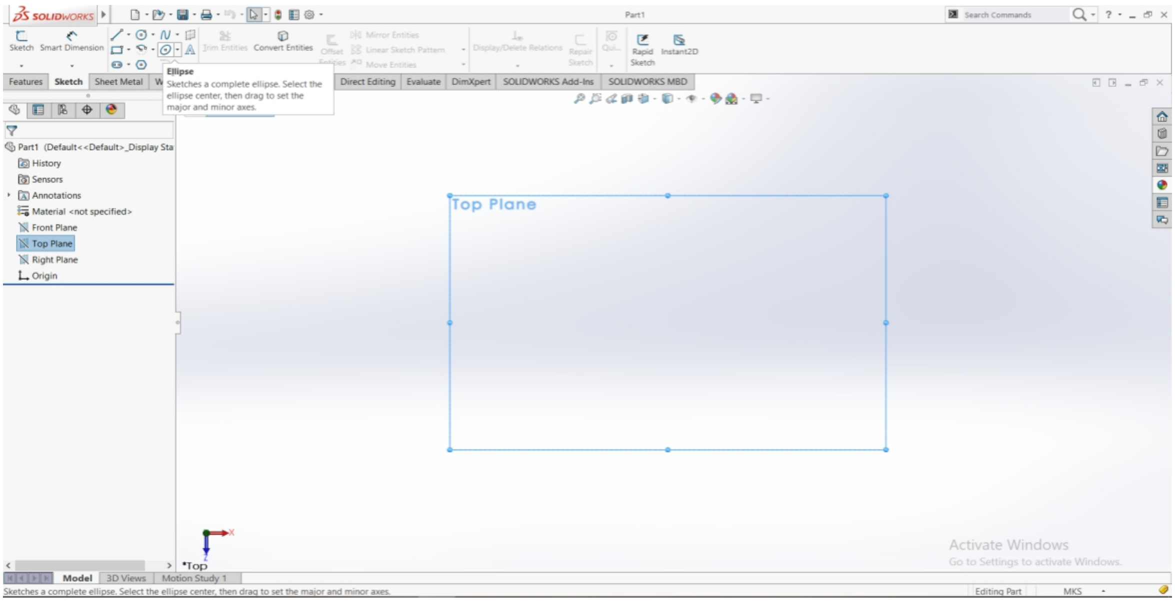

Fig. 4.1: Solidworks program opening with Top-plane

- Use Trim functions to remove sharp corners.
- The wheels are drawn on different parallel plane with the body drawing.
- Extrude the final sketch so as to create a 3D Model.

4.1.2 Dimension Procedure :-

- First open a file of model in SOLIDWORKS.
- Then click Make Drawing from Parts/Assembly right below New.
- Select A3 (Isometric) for making the sheet.
- Click Drawing View in left side and select hidden line option in Display style section.
- After selecting the view, select Smart Dimension then click Auto dimension and select the horizontal and vertical edges for giving dimension.

4.2. DIFFERENT TYPES OF MODELS OF CAR

4.2.1. Car Model

Car Model is shown in the following figures -

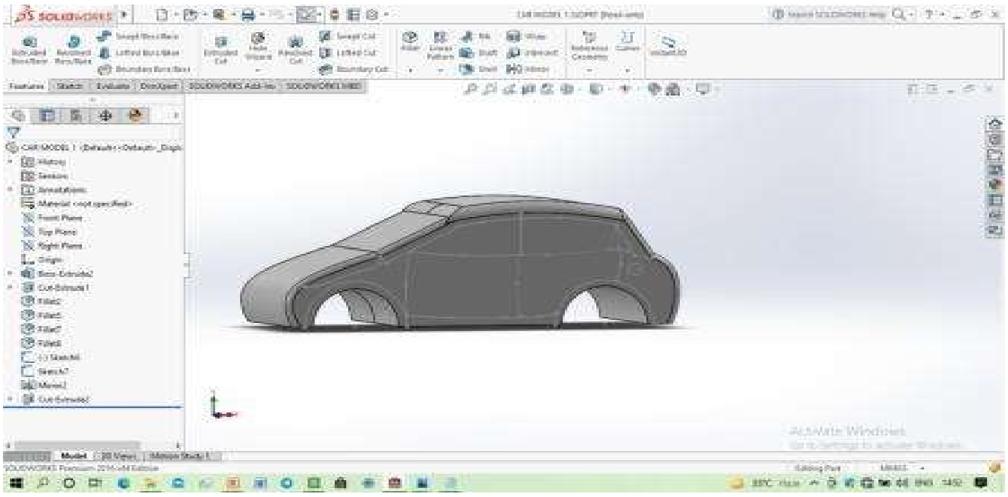

Fig. 4.2: Car Model 3D view

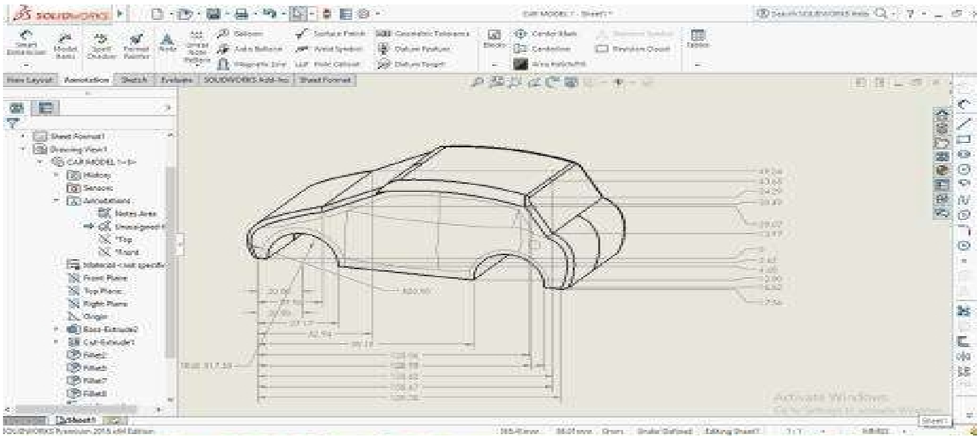

Fig. 4.3: Car Model Dimension Isometric View

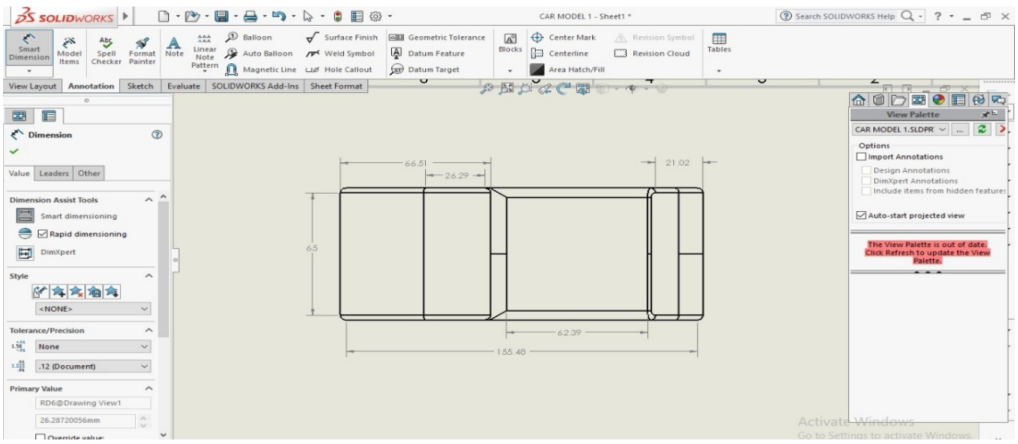

Fig. 4.4: Car Model Top View

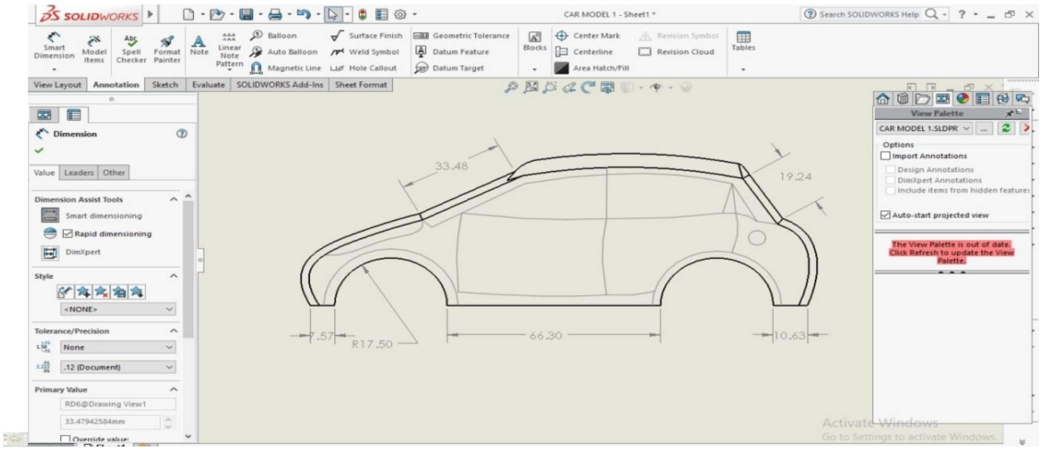

Fig. 4.5: Car Model with Dimensions Front View

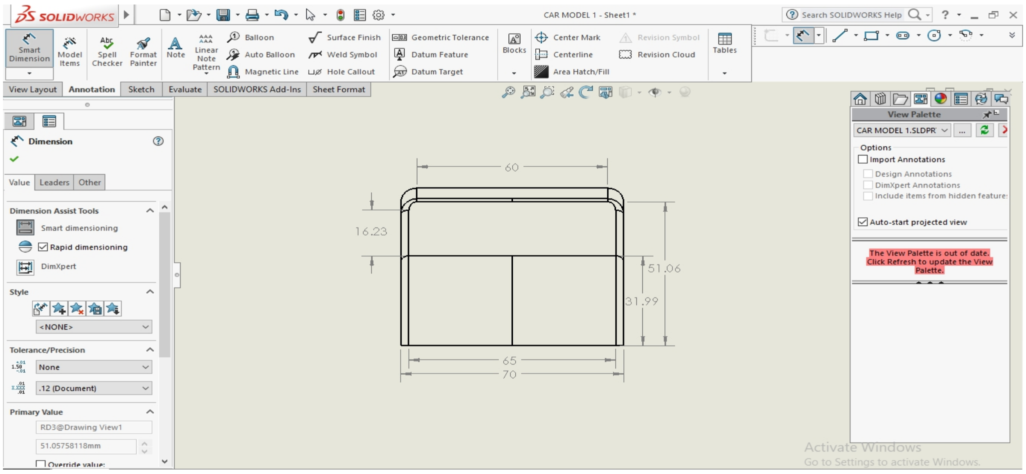

Fig. 4.6: Car Model with Dimensions Back View

4.2.2 Commercial Car Model

Commercial Car Model is shown in the following -

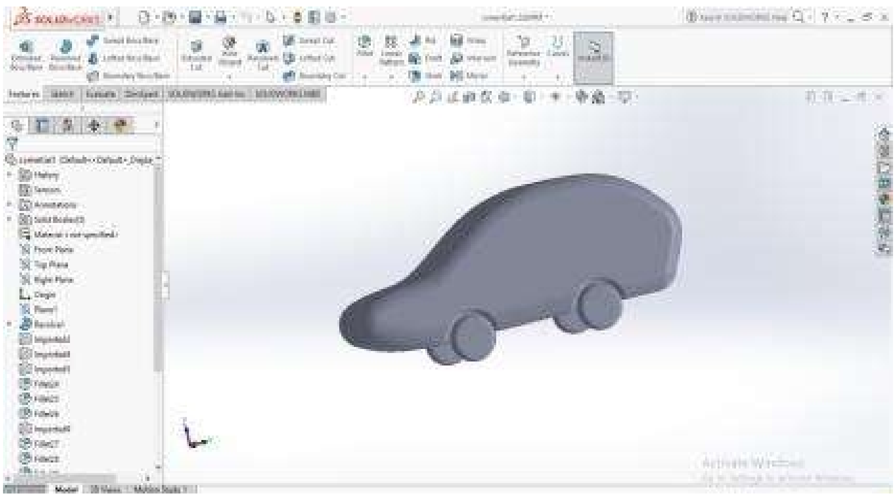

Fig. 4.7: Commercial Car Model 3D View

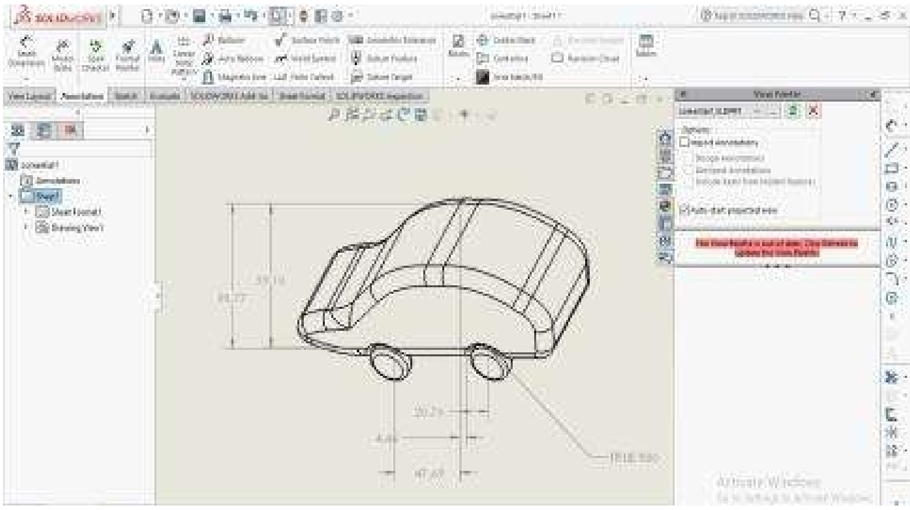

Fig. 4.8: Commercial Car Model with Dimensions Isometric View

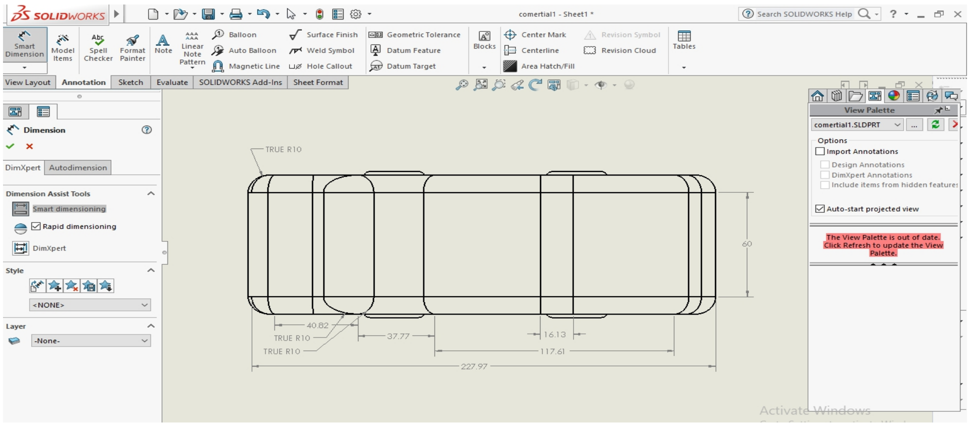

Fig. 4.9: Commercial Car Model with Dimensions Top View

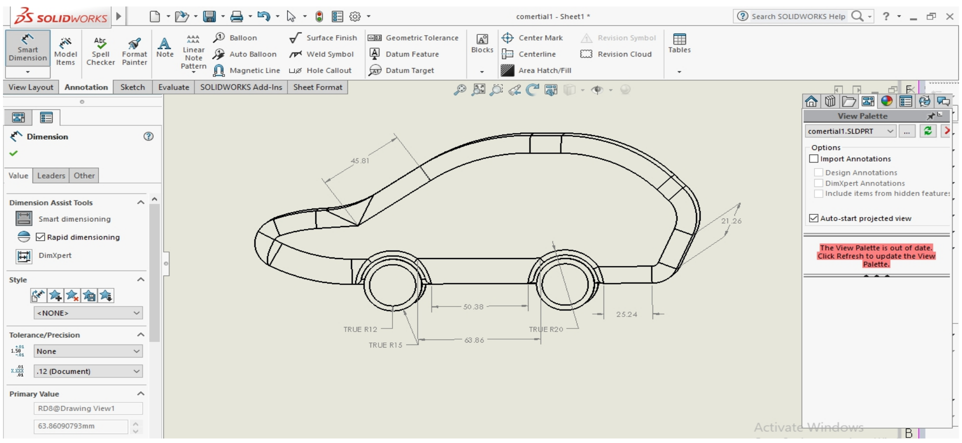

Fig. 4.10: Commercial Car Model with Dimensions Front View

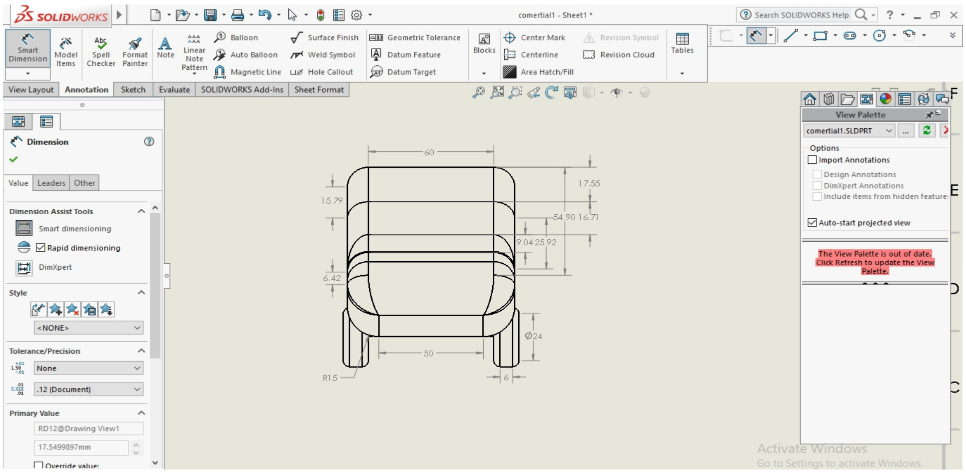

Fig. 4.11: Commercial Car Model with Dimensions Front View

4.2.3 Racing Car Model

Racing Car Model is shown in the following -

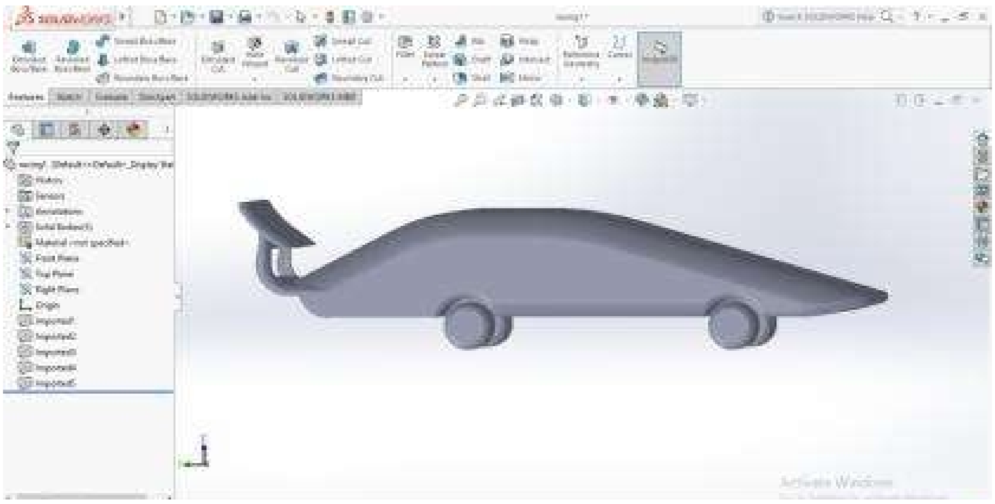

Fig. 4.12: Racing Car Model with 3D View

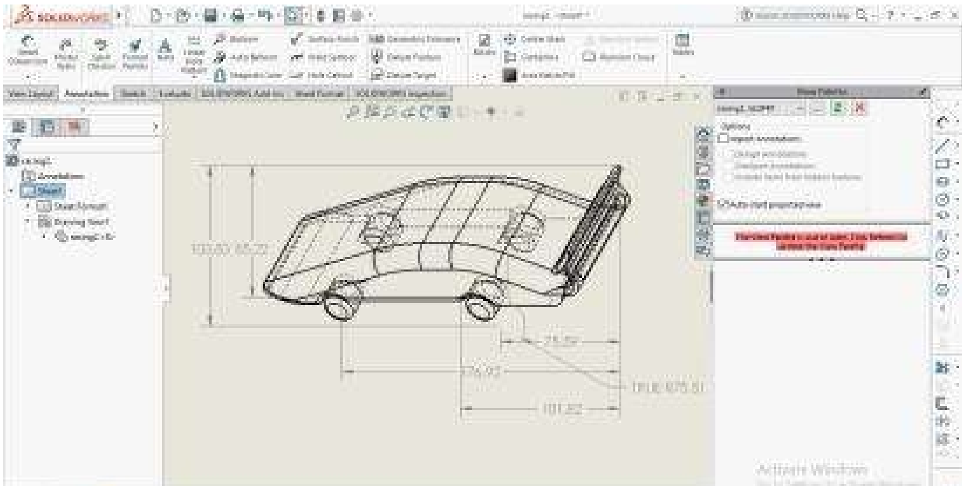

Fig. 4.13: Racing Car Model Dimensions with Isometric View

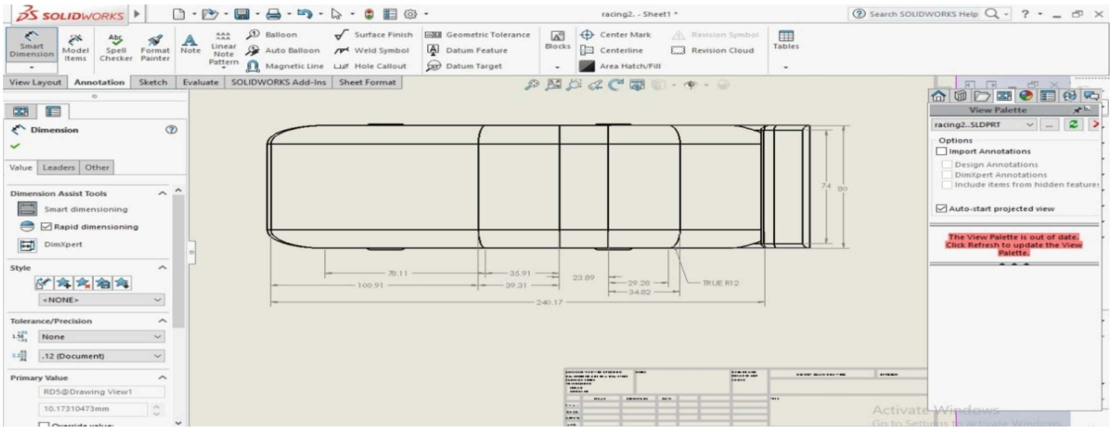

Fig. 4.14: Racing Car Model Dimensions with Top View

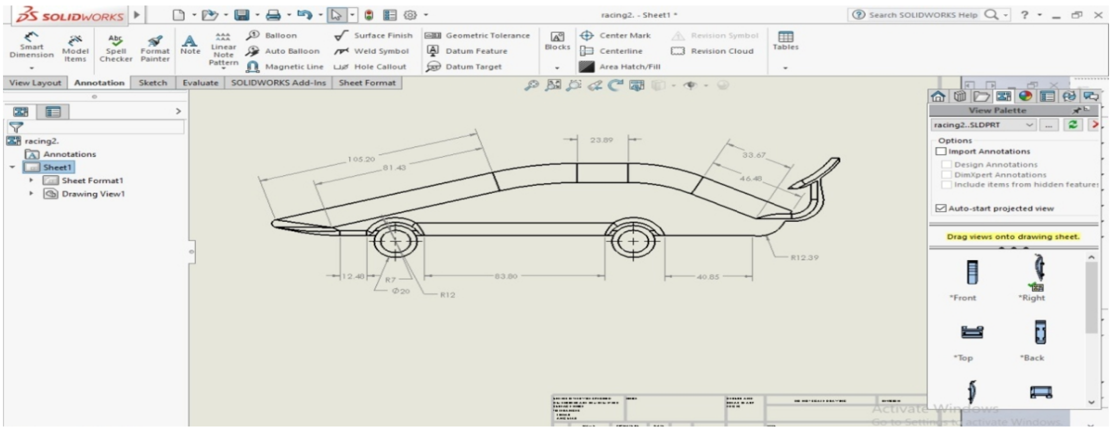

Fig. 4.15: Racing Car Model Dimensions with Front View

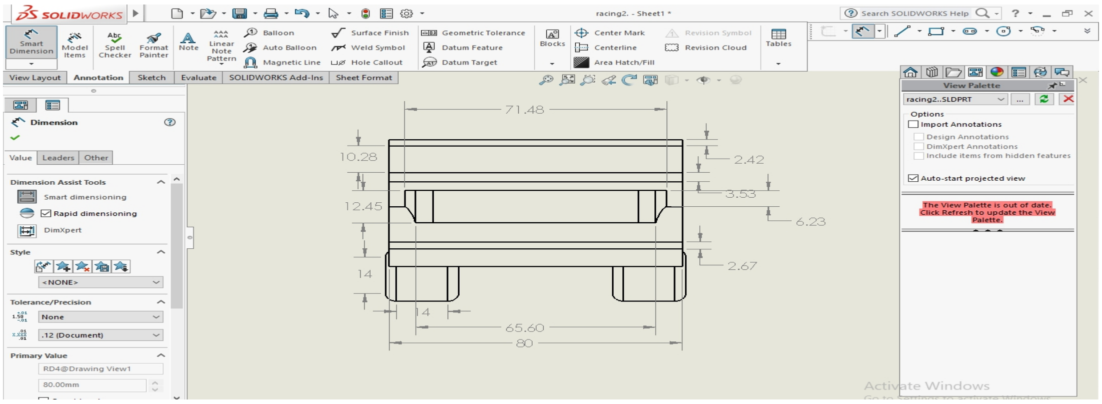

Fig. 4.16: Racing Car Model Dimensions with Back View

4.2.4 Omni Model

Omni Model is shown in the following figures -

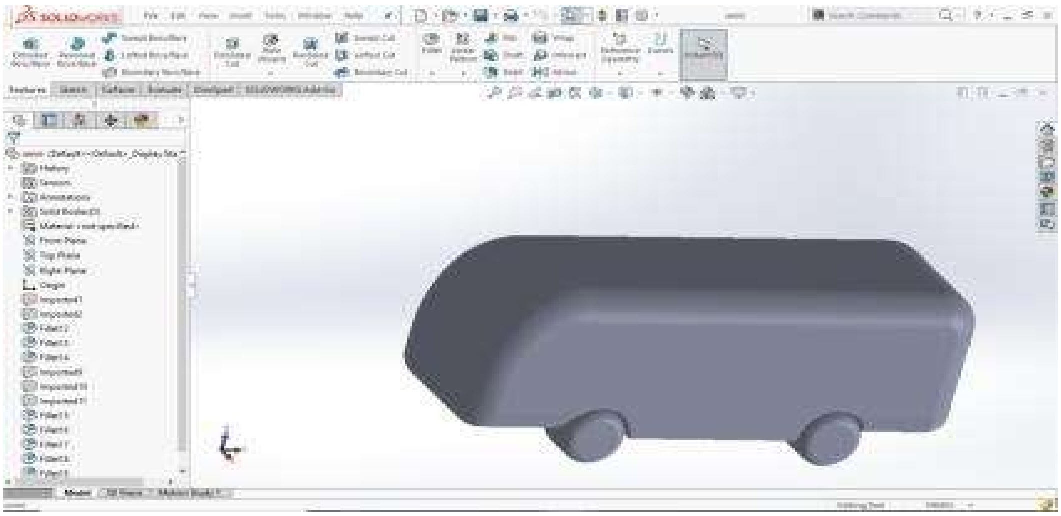

Fig. 4.17: Omni Model with 3D View

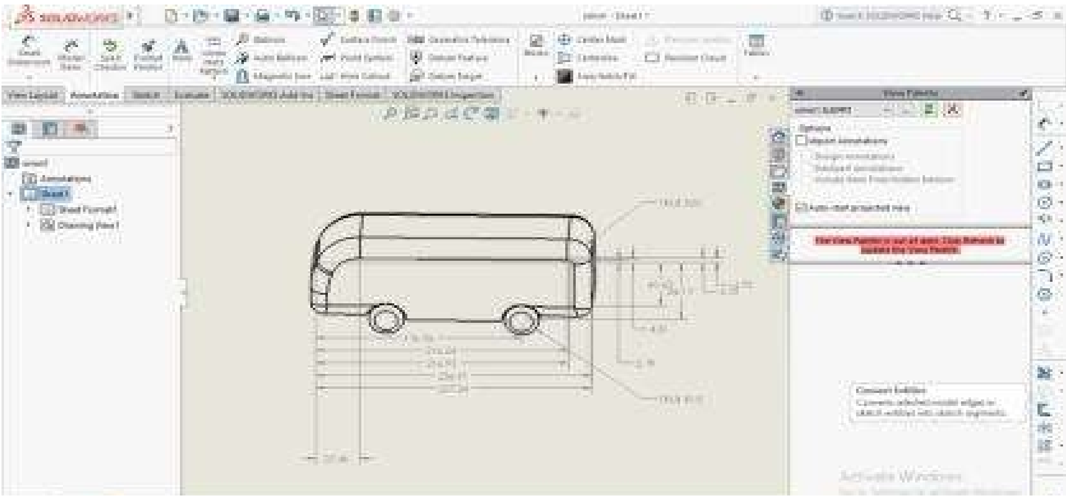

Fig. 4.18: Omni Model Dimensions with Isometric View

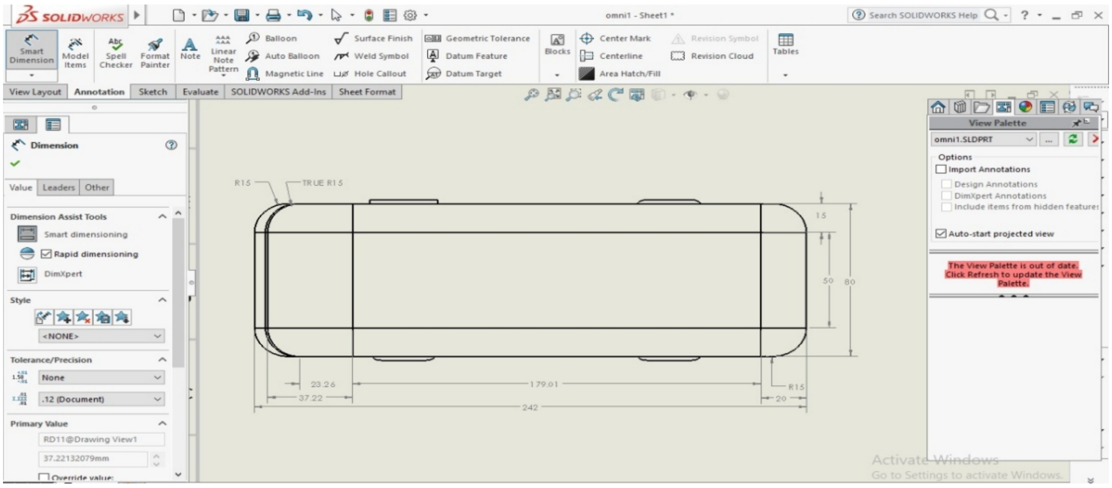

Fig. 4.19: Omni Model Dimensions with Top View

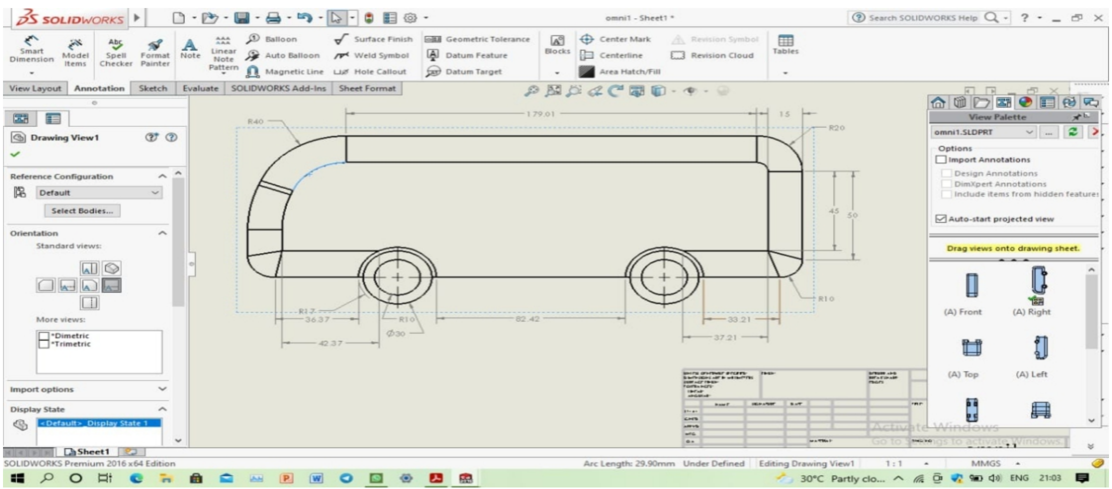

Fig. 4.20: Omni Model Dimensions with Top View

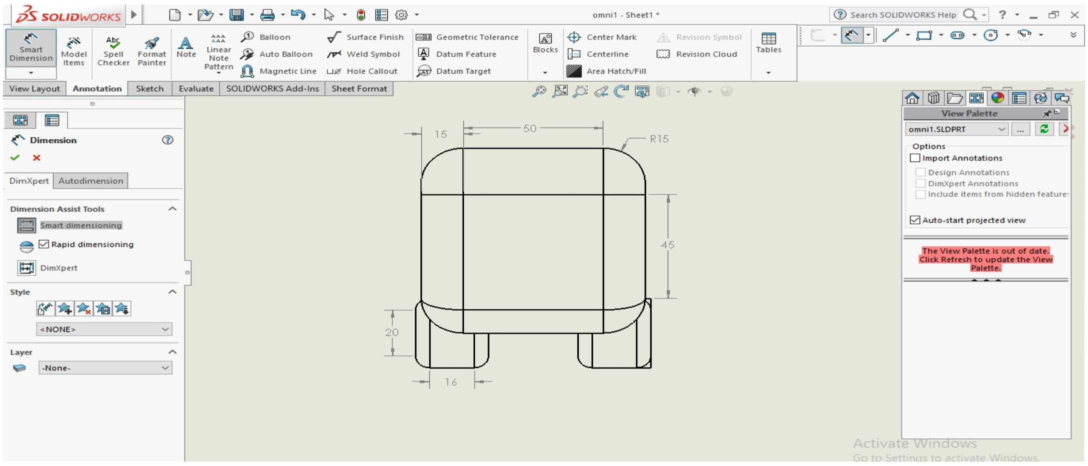

Fig. 4.21: Omni Model Dimensions with Back View

4.2.5 Drag Minimised Model

Drag Minimised Model is shown in the following figures -

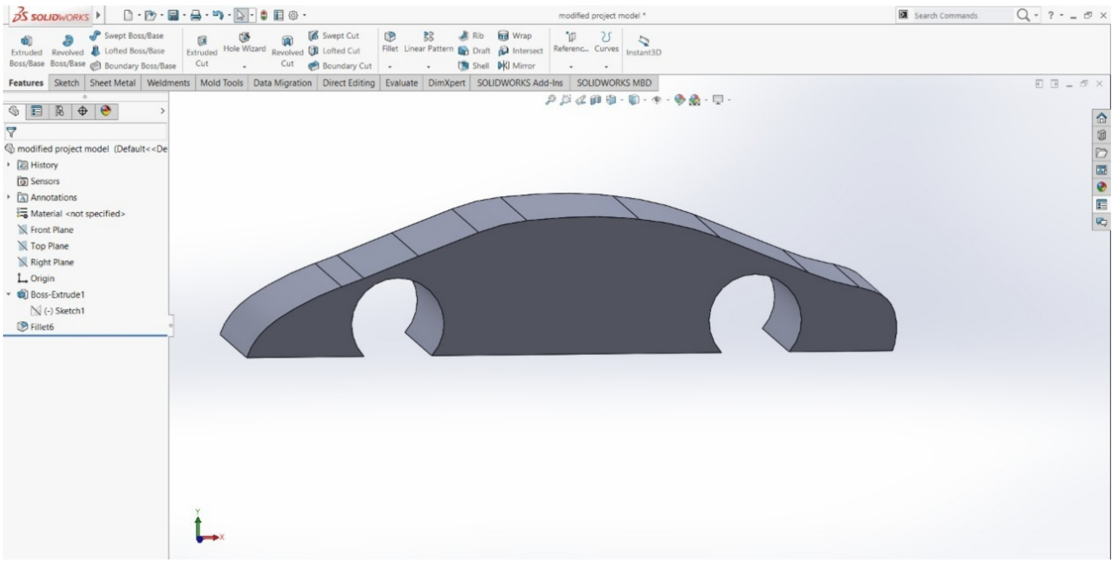

Fig 4.22: Drag Minimised Model Dimensions with 3D View

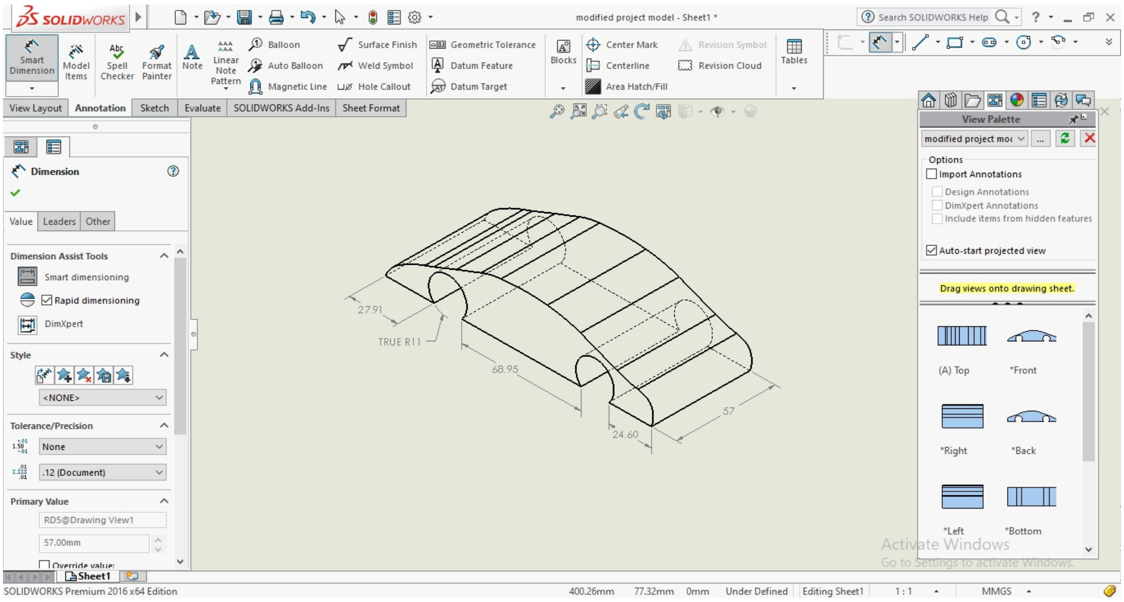

Fig. 4.23: Drag Minimised Model Dimensions with Isometric View

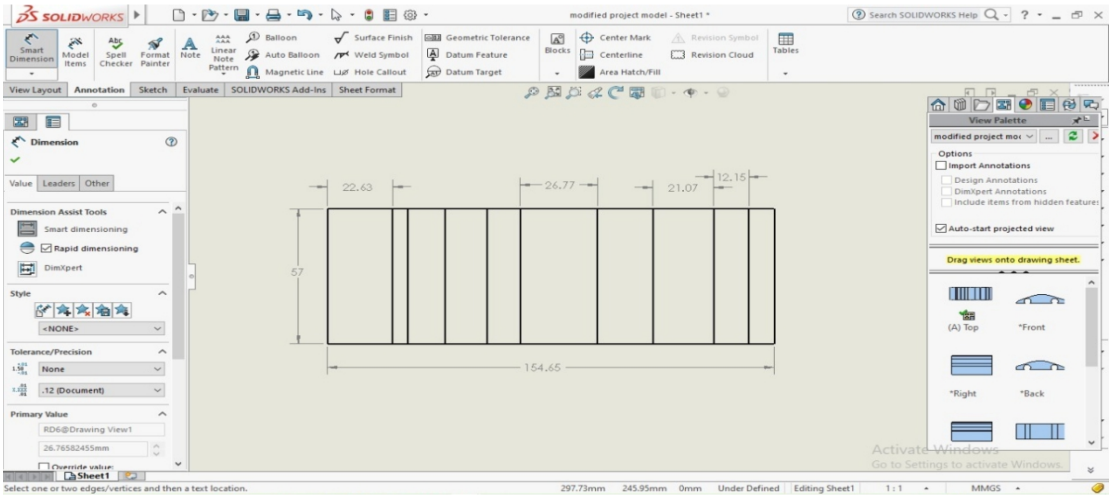

Fig. 4.24: Drag Minimised Model Dimensions with Top View

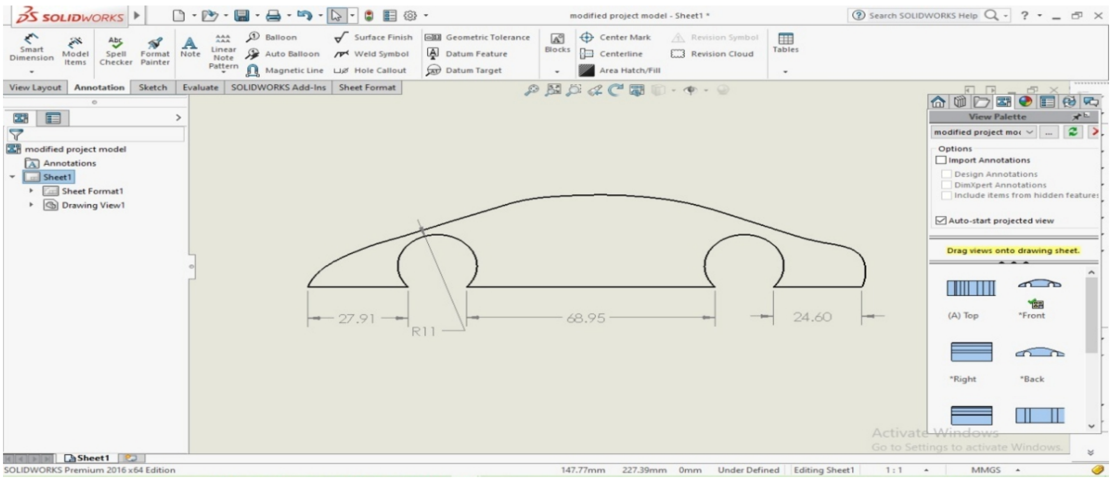

Fig. 4.25: Drag Minimised Model Dimensions with Front View

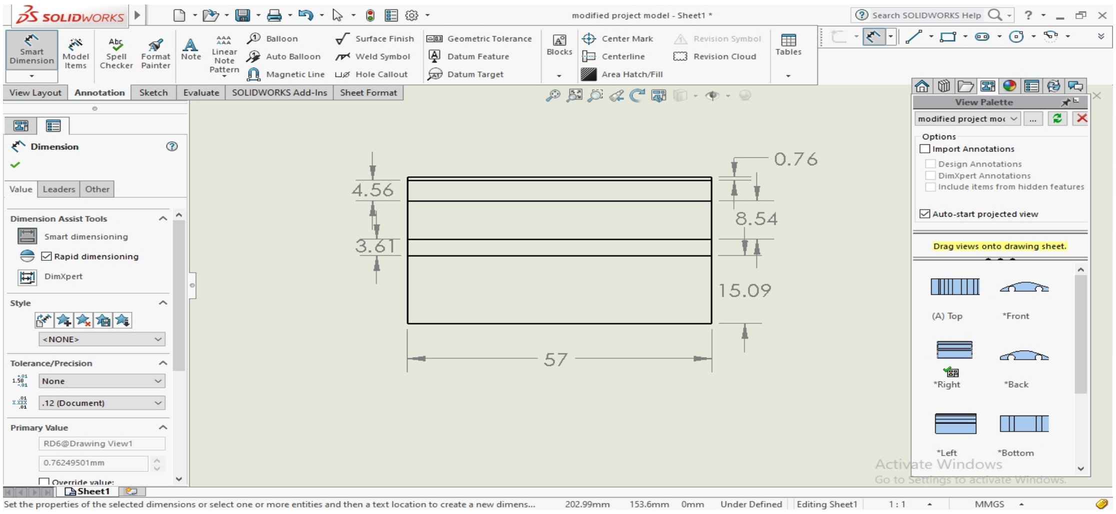

Fig. 4.27 Drag Minimised Model Dimensions with Back View

5. COMPUTATIONAL SIMULATION SETUP

5.1. DEFINE THE PROBLEM

So for this project, vehicle models there are some process which involved in the CFD (computational fluid dynamics). The CFD (computational fluid dynamics) simulation analysis is explained below.

These models were designed keeping in view the actual design of the models in practical life. The next step is to create the flow domain or the computational domain. This defines the flow around the 3d model and which area the flow is restricted to. The nature of the viscous flow is determined. Inlet air flow is determined and the outlet.

5.2. FLOW DOMAIN OR COMPUTATIONAL DOMAIN

The area around the 3d model, which defines the flow simulation the restricted area, is known as the computational domain. This defines the boundary under which the flow simulation will take place.

The general figure 5.1 shows the basis of the computational domain. The following figure 5.2 shows that actual computational domain for the experiment.

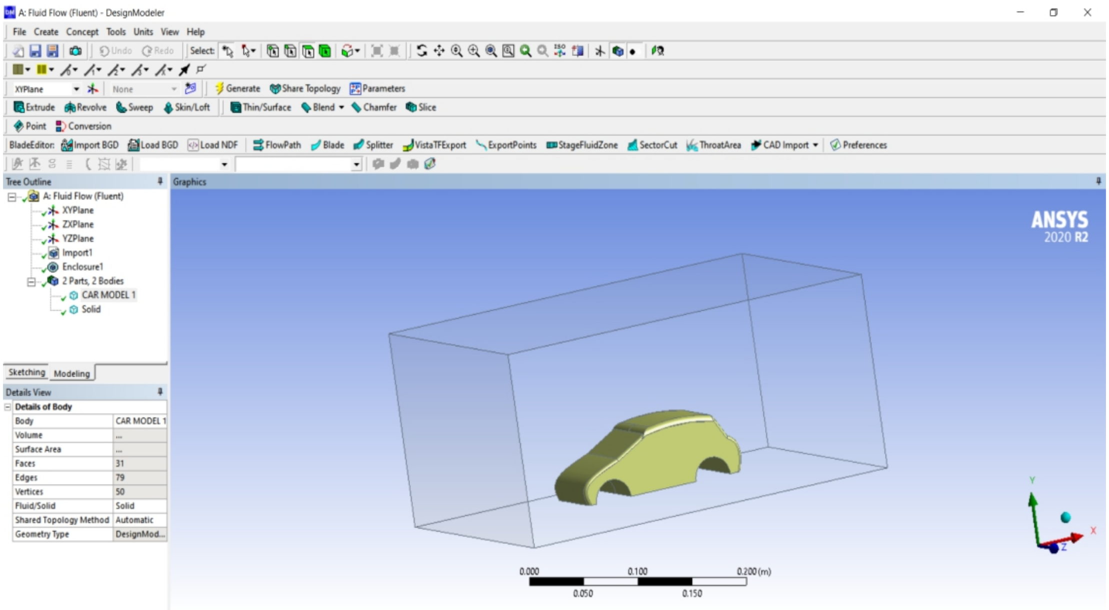

Fig. 5.1: Computational domain of Car Model

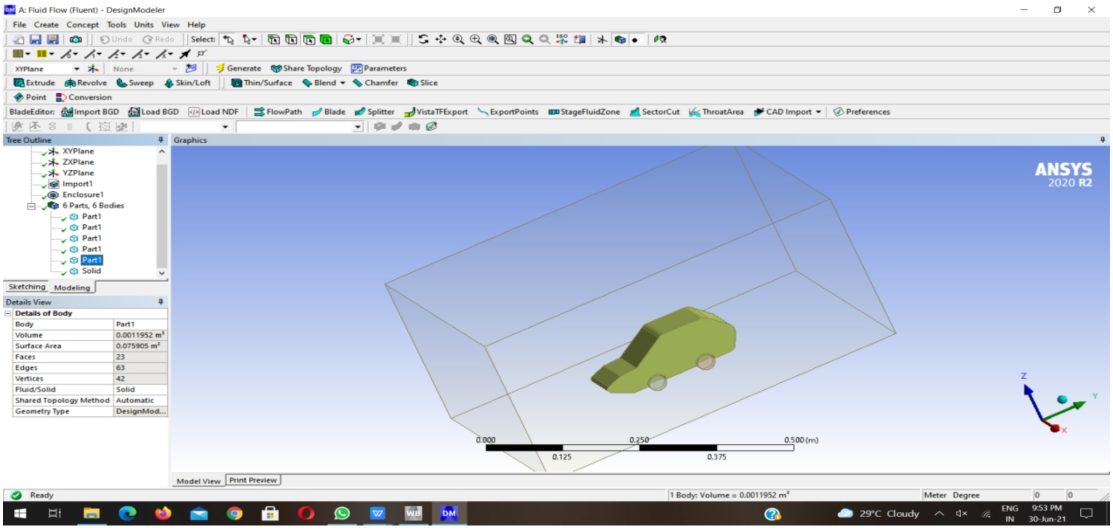

Fig. 5.2: Computational domain of Commercial Model

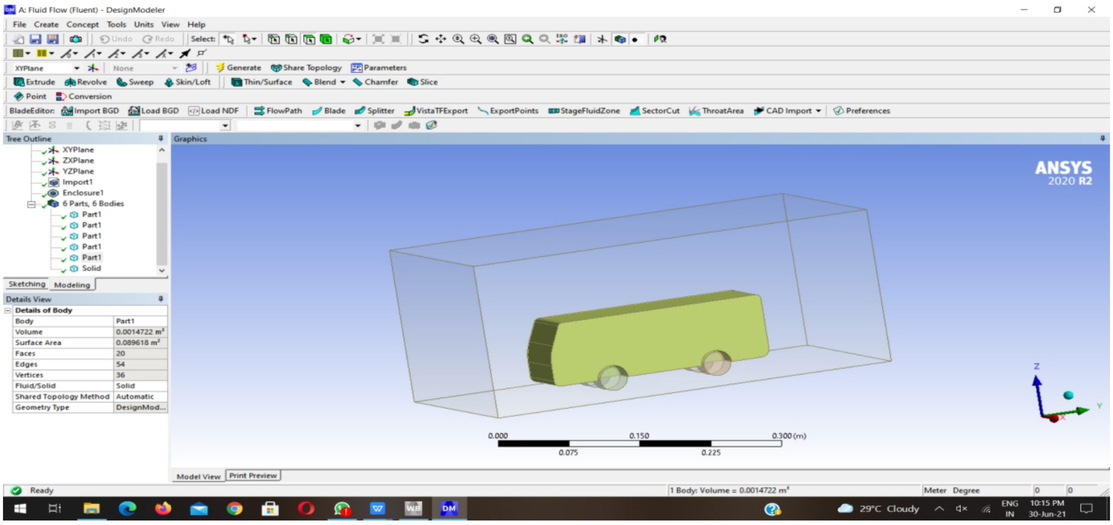

Fig. 5.3: Computational domain of Omni Model

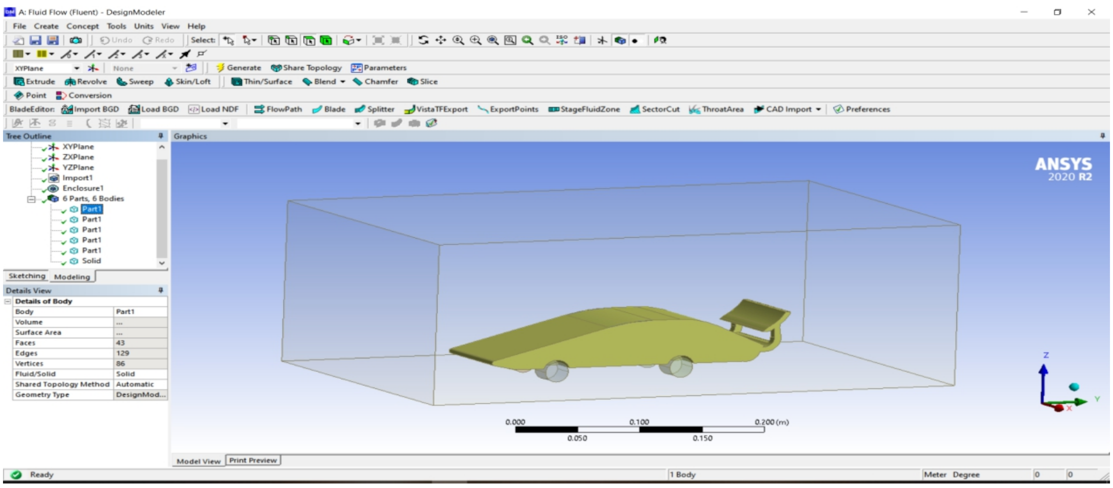

Fig. 5.4: Computational domain of Racing Model

COMPUTATIONAL DOMAIN SIZE USED IN THIS PROJECT:-

1. **FOR CAR MODEL**

X min	-0.1 m
X max	0.1 m
Y min	-0.01 m
Y max	0.1 m
Z min	-0.1 m
Z max	0.1 m

2. **FOR COMMERCIAL CAR MODEL**

X min	- 0.1 m
X max	0.1 m
Y min	- 0.1 m
Y max	0.1 m
Z min	- 0.001m
Z max	0.1 m

3. **FOR OMNI MODEL**

X min	-0.1 m
X max	0.1 m
Y min	-0.1 m
Y max	0.1 m
Z min	-0.001 m
Z max	0.1 m

4. **FOR RACING CAR MODEL**

X min	-0.1 m
X max	0.1 m
Y min	-0.1 m
Y max	0.1 m
Z min	-0.001 m
Z max	0.1 m

5.3 CREATING MESH

The basic mesh is used in this project. It can be say that the system auto generated meshing was used. However in same models during drag and lift analysis the hex dominant mesh system is used.

As the Flow domain or the computational domain was established then physical conditions are needed for the boundary of the computational domain.

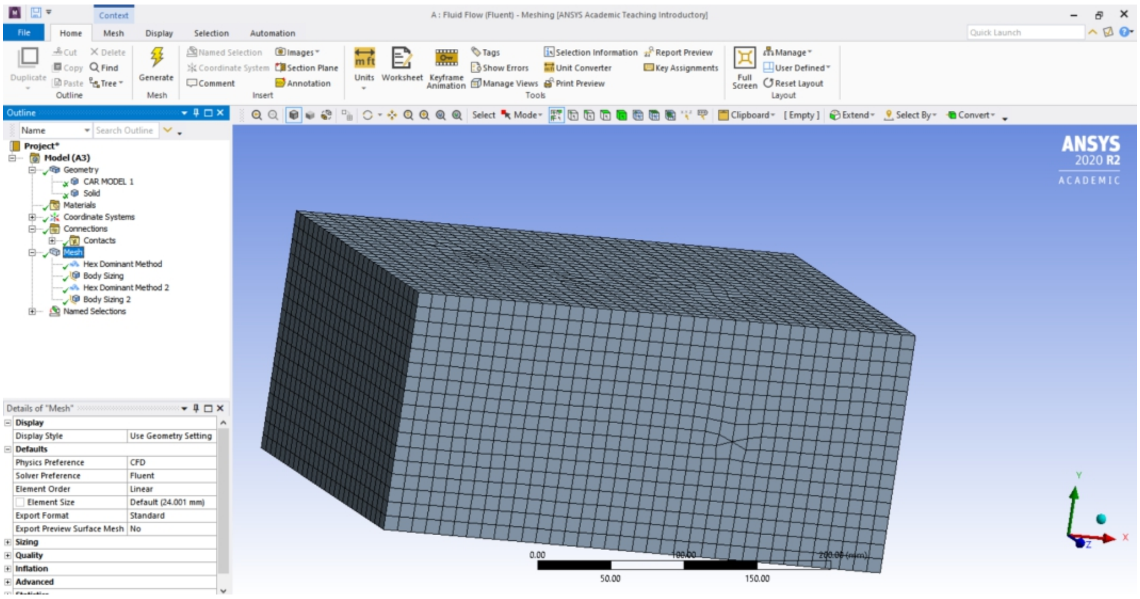

Fig. 5.5: Enclosure mesh of Car model

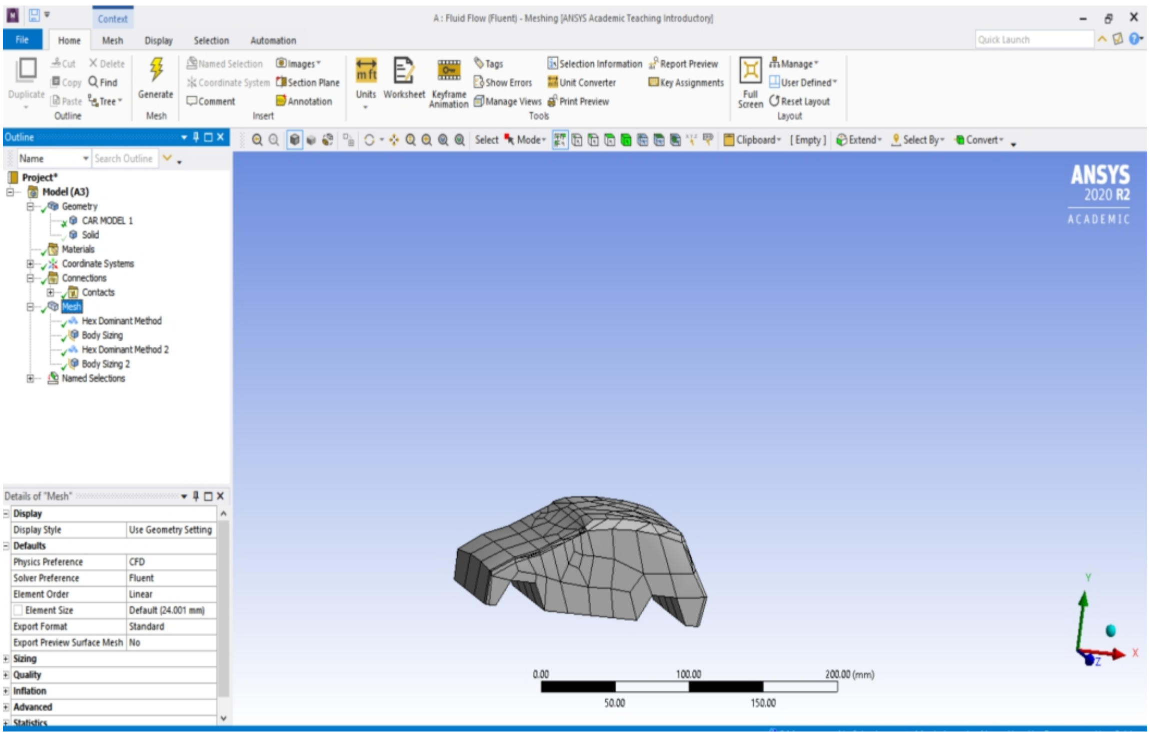

Fig. 5.6: Mesh of Car model

Mesh Report of Car model

Domain	Nodes	Elements
car_model_1	5129	5910
solid	130155	131720
All Domains	135284	137630

Physics Report of Car model

Domain - car_model_1	
Type	solid
Domain - solid	
Type	cell

Boundaries of Car model

Domain	Boundaries	
car_model_1	**Boundary - wall car_model_1**	
	Type	WALL
solid	**Boundary - fluid_side_wall**	
	Type	WALL
	Boundary - inlet	
	Type	VELOCITY-INLET
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Boundary - wall solid	
	Type	WALL

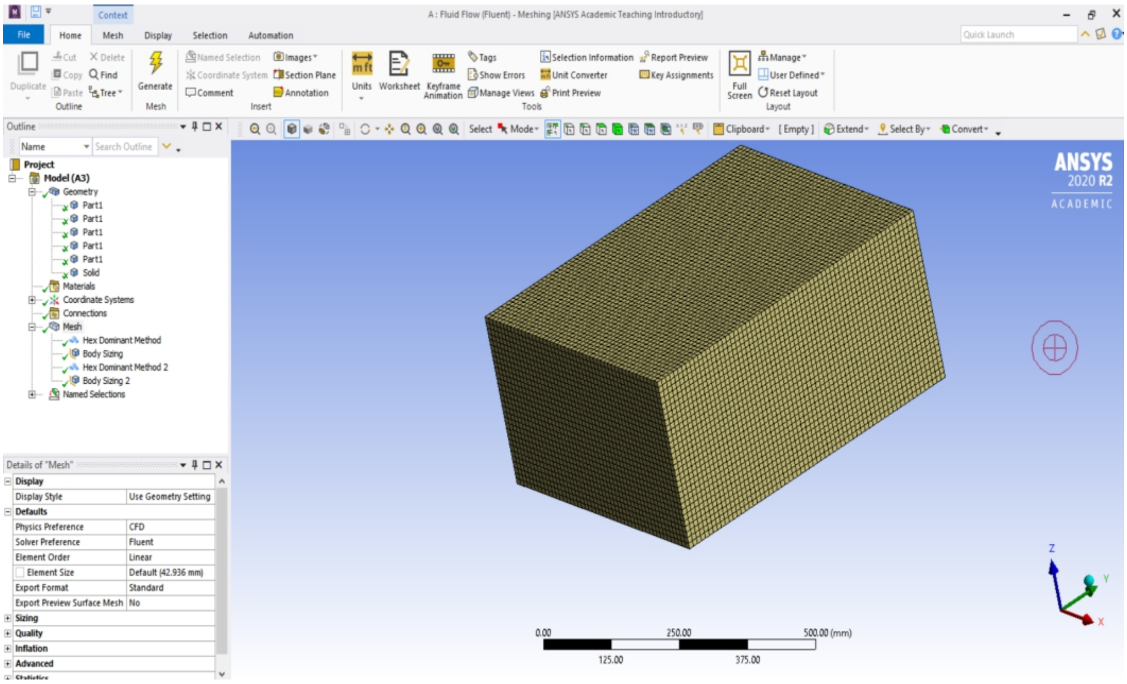

Fig. 5.7: Enclosure mesh of Commercial model

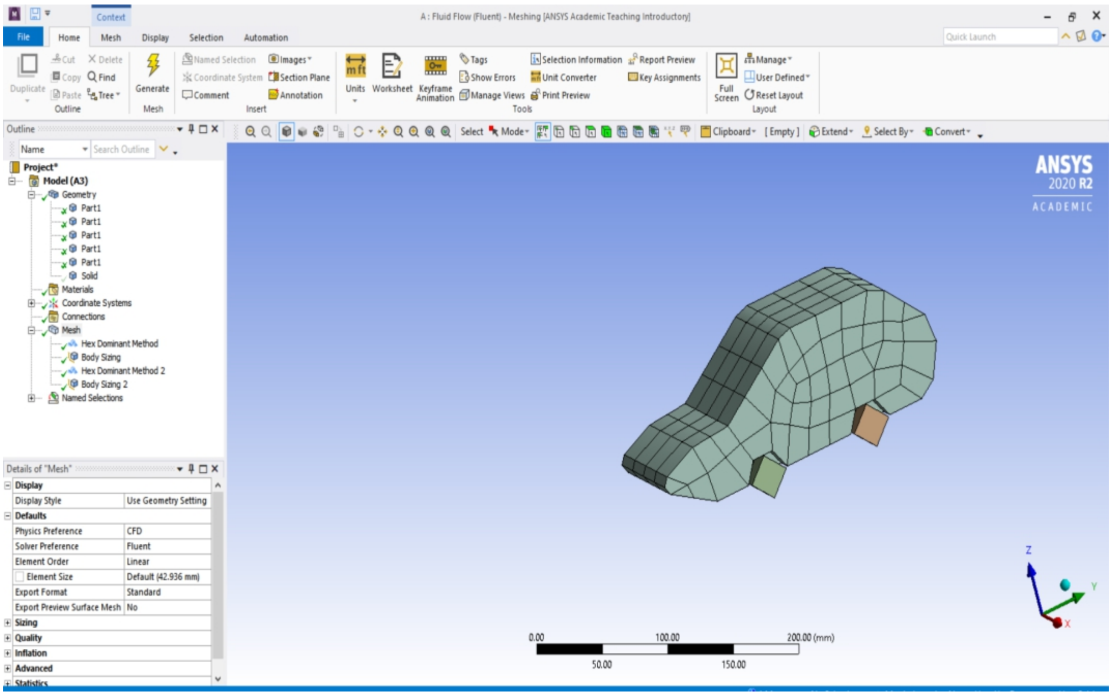

Fig. 5.8: Mesh of Commercial model

Mesh Report of Commercial model

Domain	Nodes	Elements
part1	13527	14375
solid	16073	22507
All Domains	29600	36882

Physics Report of Commercial model

Domain - part1	
Type	solid
Domain - solid	
Type	cell

Boundaries of Commercial model

Domain	Boundaries	
part1	**Boundary - wall part1**	
	Type	WALL
solid	**Boundary - inlet**	
	Type	VELOCITY-INLET
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Boundary - wall solid	
	Type	WALL

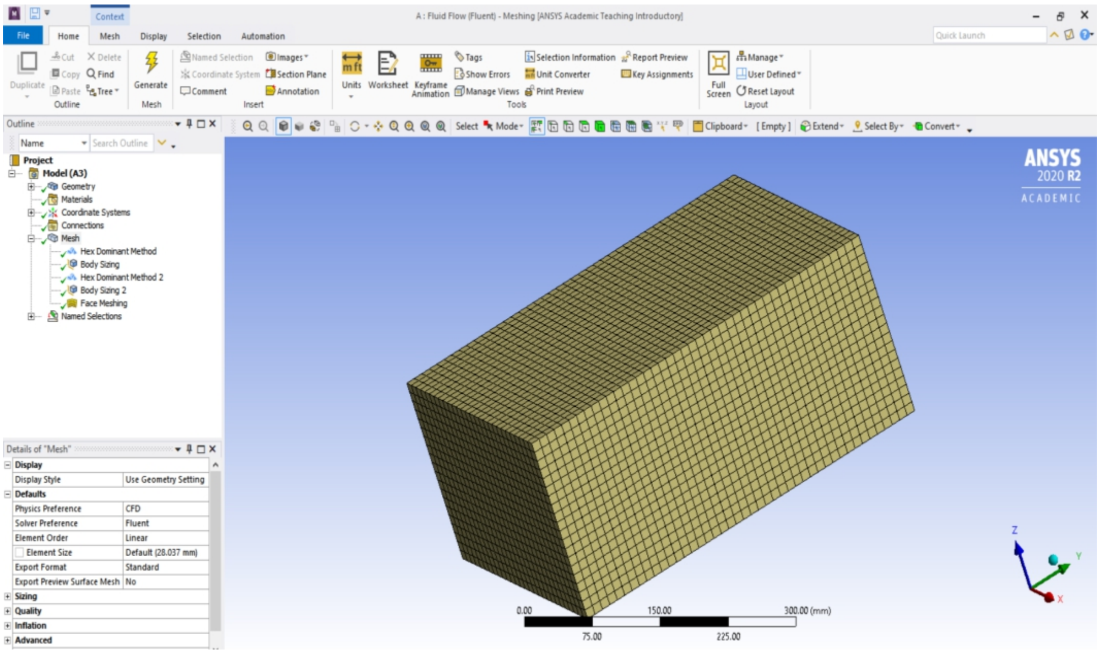

Fig. 5.9: Enclosure mesh of Omni model

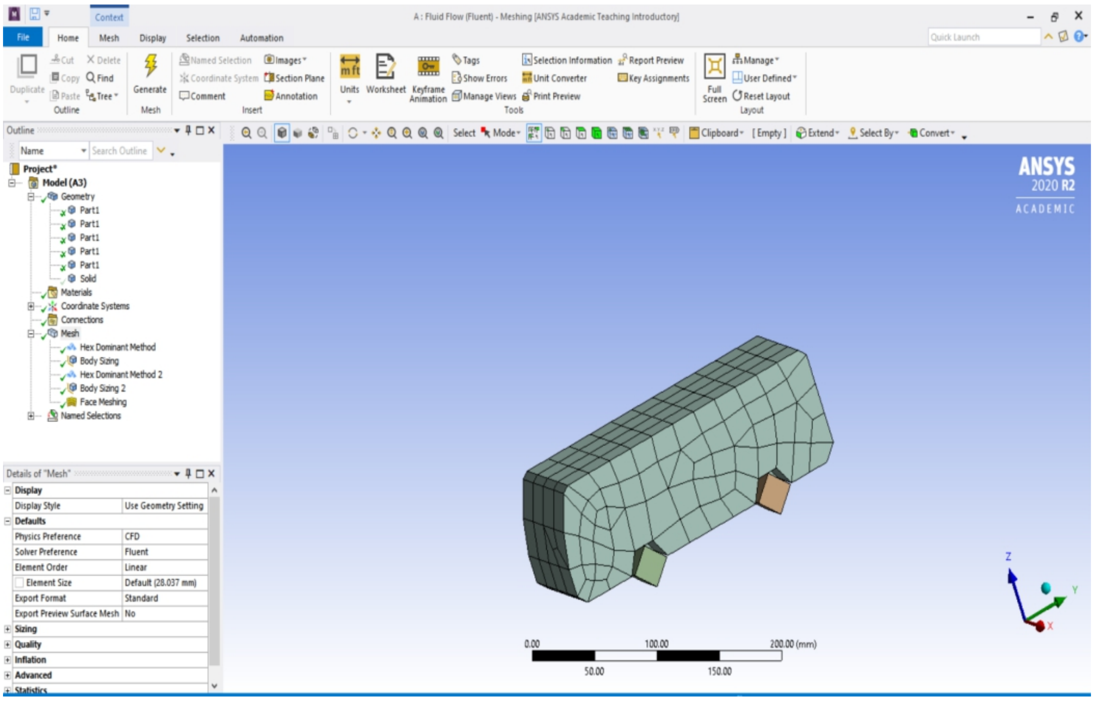

Fig. 5.10: Mesh of Omni model

Mesh Report of Omni model

Domain	Nodes	Elements
Omni model	15221	16250
solid	179538	181458
All Domains	194759	197708

Physics Report of Omni model

Domain - Omni model	
Type	solid
Domain - solid	
Type	cell

Boundaries of Omni model

Domain	Boundaries	
Omni model	**Boundary - wall Omni model**	
	Type	WALL
solid	**Boundary - inlet**	
	Type	VELOCITY-INLET
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Boundary - wall solid	
	Type	WALL

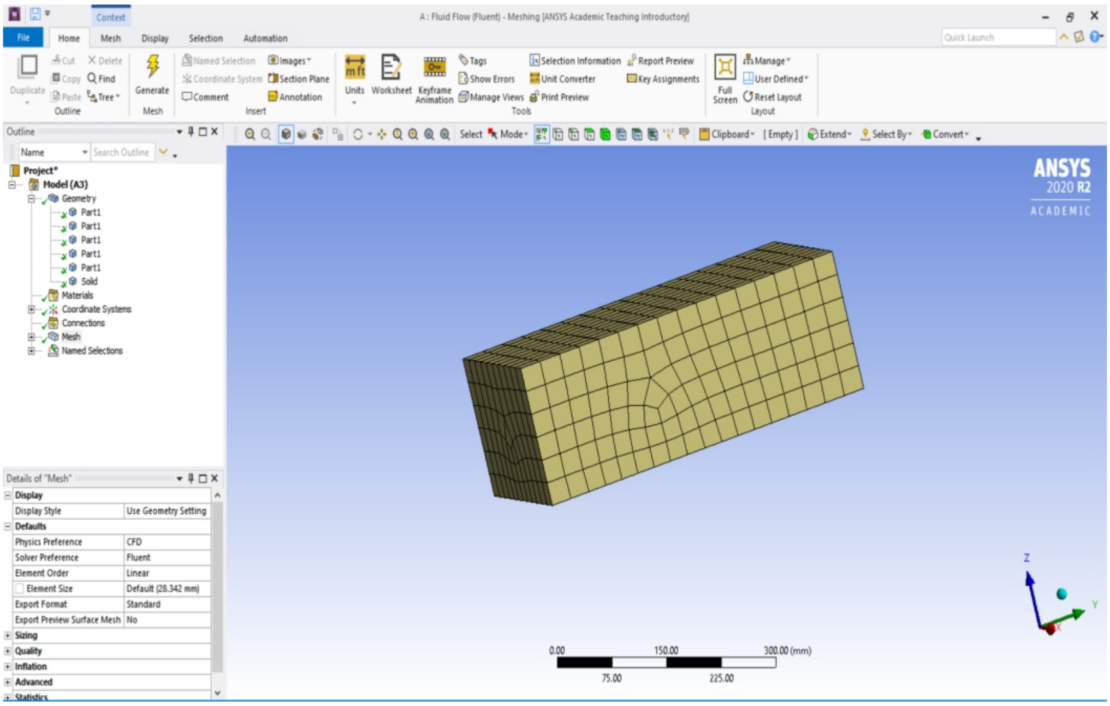

Fig. 5.11: Enclosure mesh of racing model

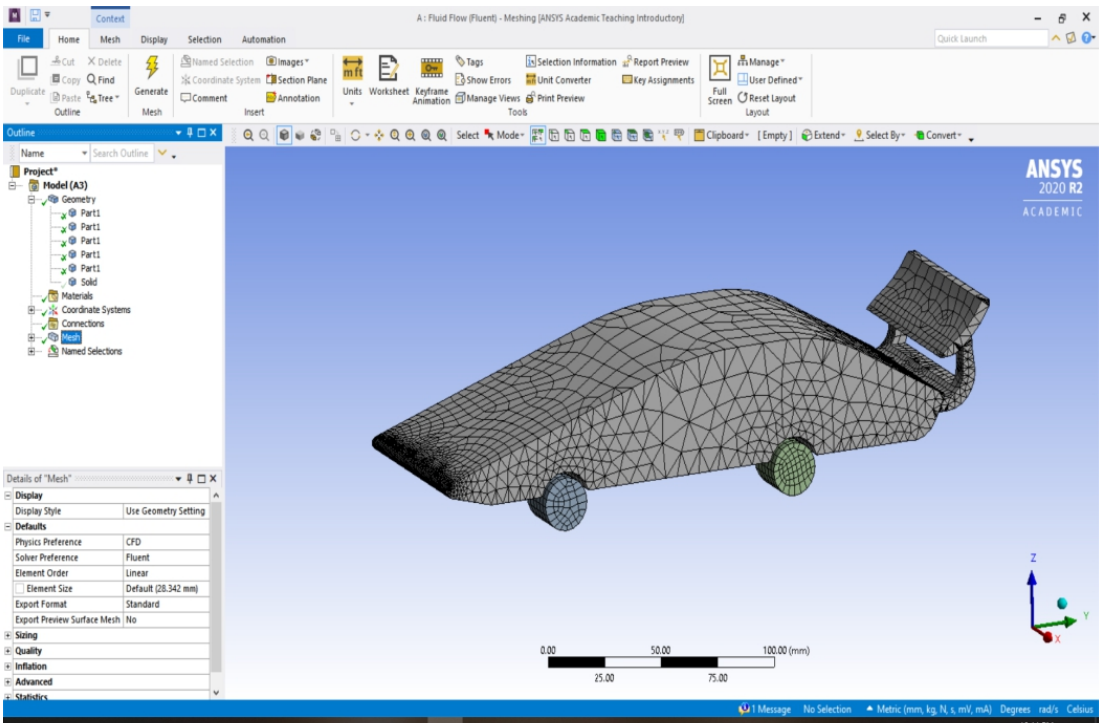

Fig. 5.12: Mesh of Racing model

Mesh Report of Racing model

Domain	Nodes	Elements
part1	7886	7643
solid	25245	26868
All Domains	33131	34511

Physics Report of Racing model

Domain - part1	
Type	solid
Domain - solid	
Type	cell

Boundaries of Racing model

Domain	Boundaries	
part1	**Boundary - wall part1**	
	Type	WALL
solid	**Boundary - inlet**	
	Type	VELOCITY-INLET
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Boundary - wall solid	
	Type	WALL

5.4. INITIAL CONDITION

As the Flow domain or the computational domain was established then physical conditions are needed for the boundary of the computational domain.

INLET VELOCITY USED =50 m/s

5.5. ESTABLISHING THE GOALS

After completing all the steps for pressure and velocity distribution we will define the nature of the flow simulation over the body.

5.6. RESULTS OF SIMULATION

After completing the simulation the results are ready to view. The result obtained in the form of graphs is between the coefficient of lift and coefficient of drag.

Number of streamlines used to depict the air flow =200

- <u>Car model 1</u> :-

Giving some images from the analysis

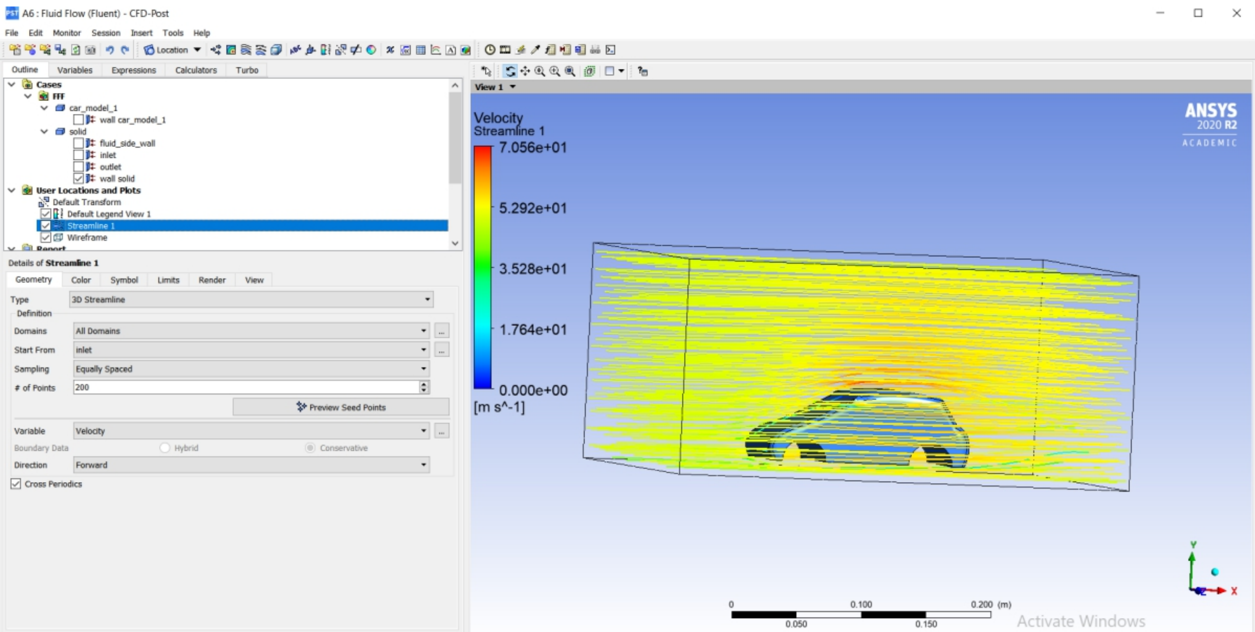

Fig. 5.13

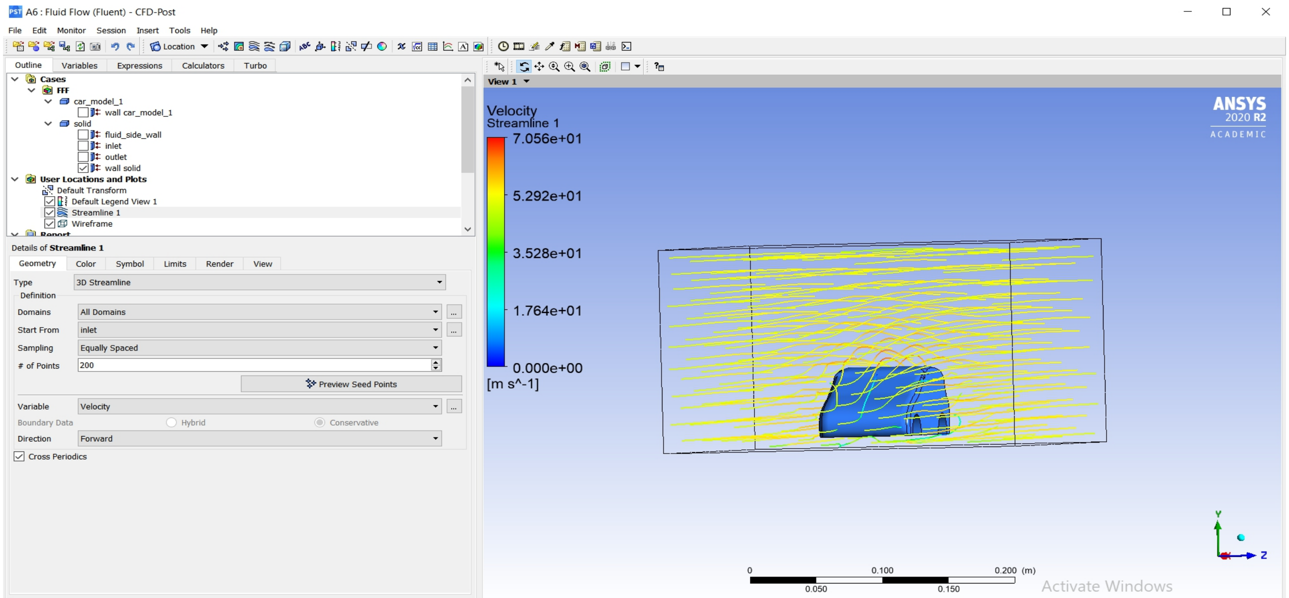

Fig. 5.14

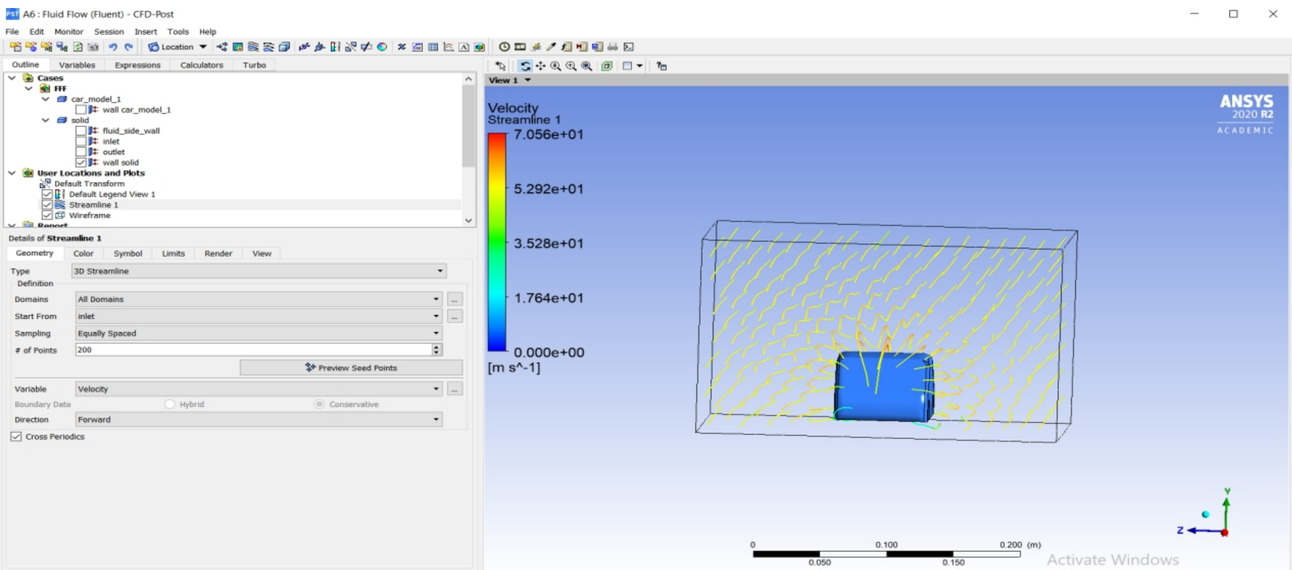

Fig. 5.15

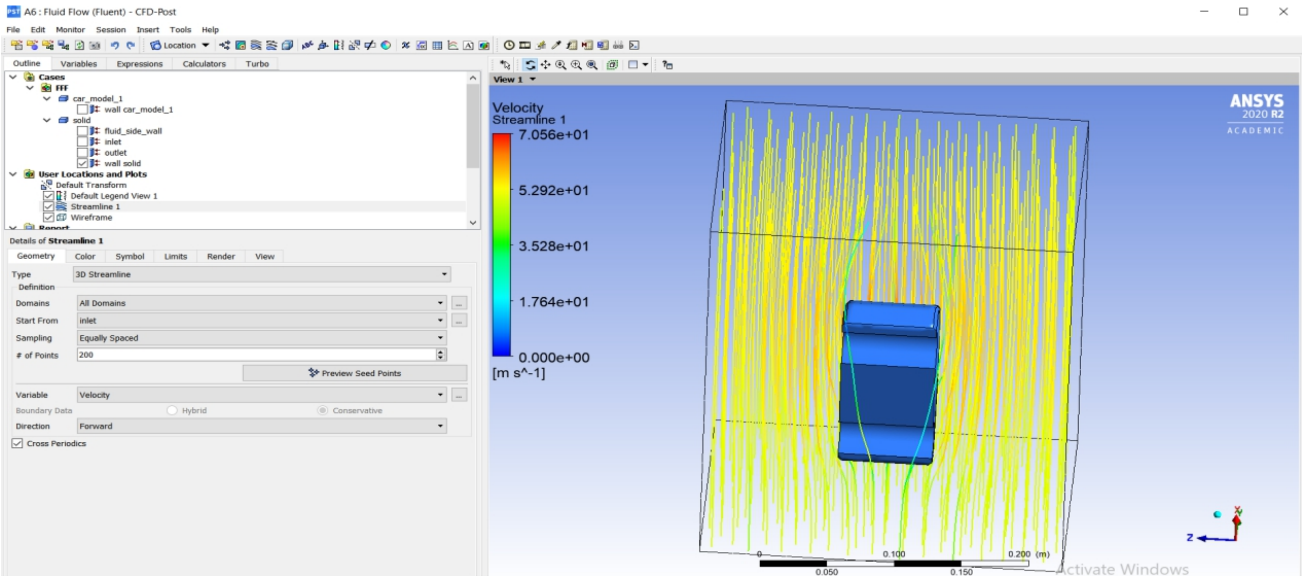

Fig. 5.16

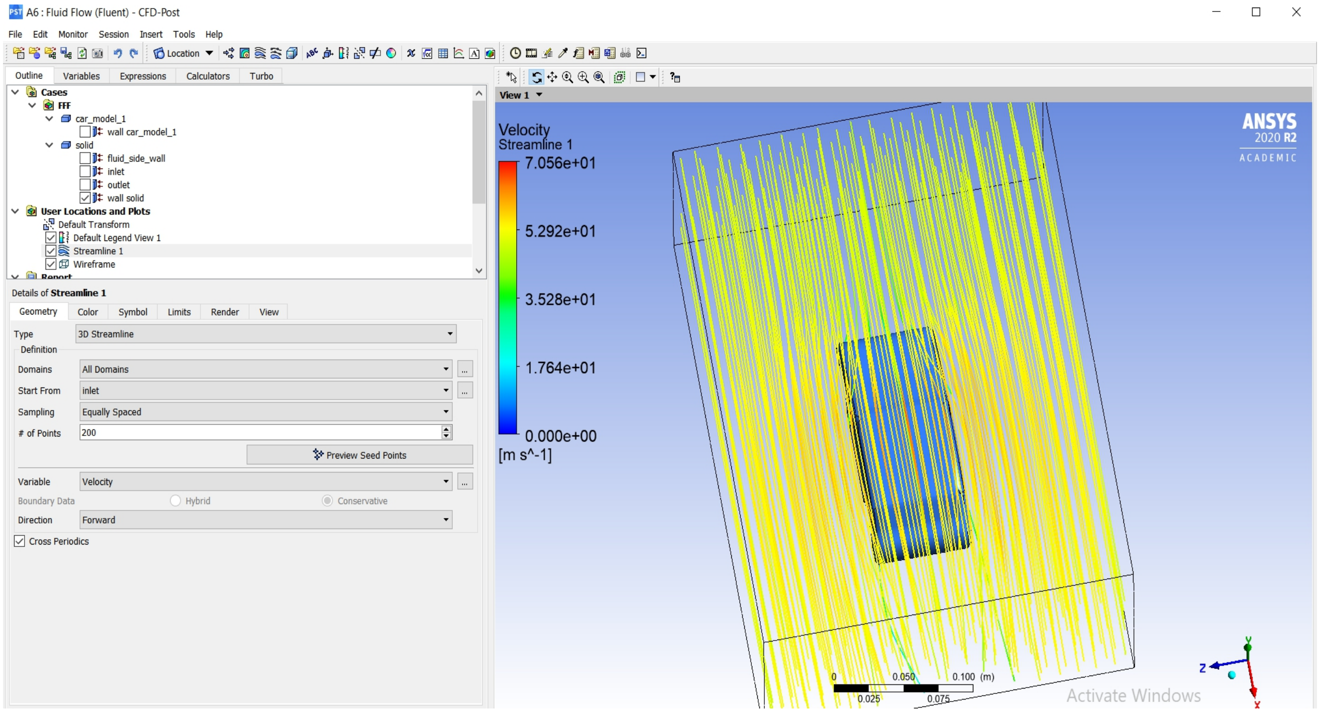

Fig. 5.17

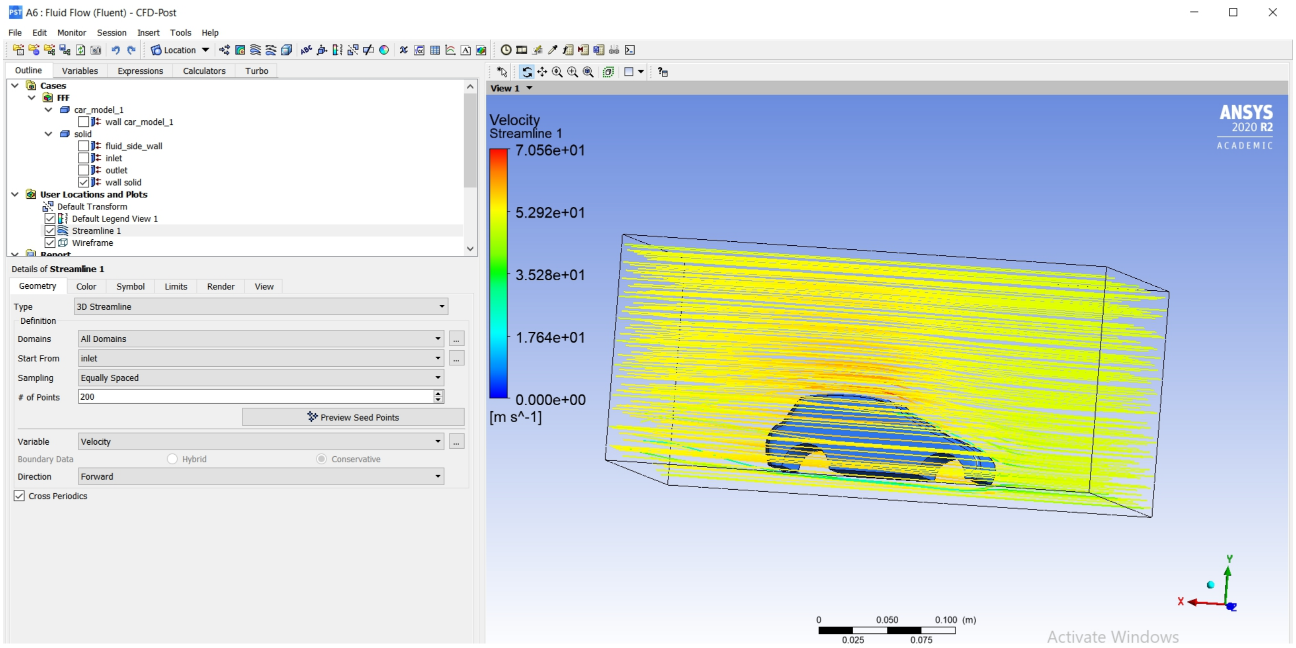

Fig. 5.18

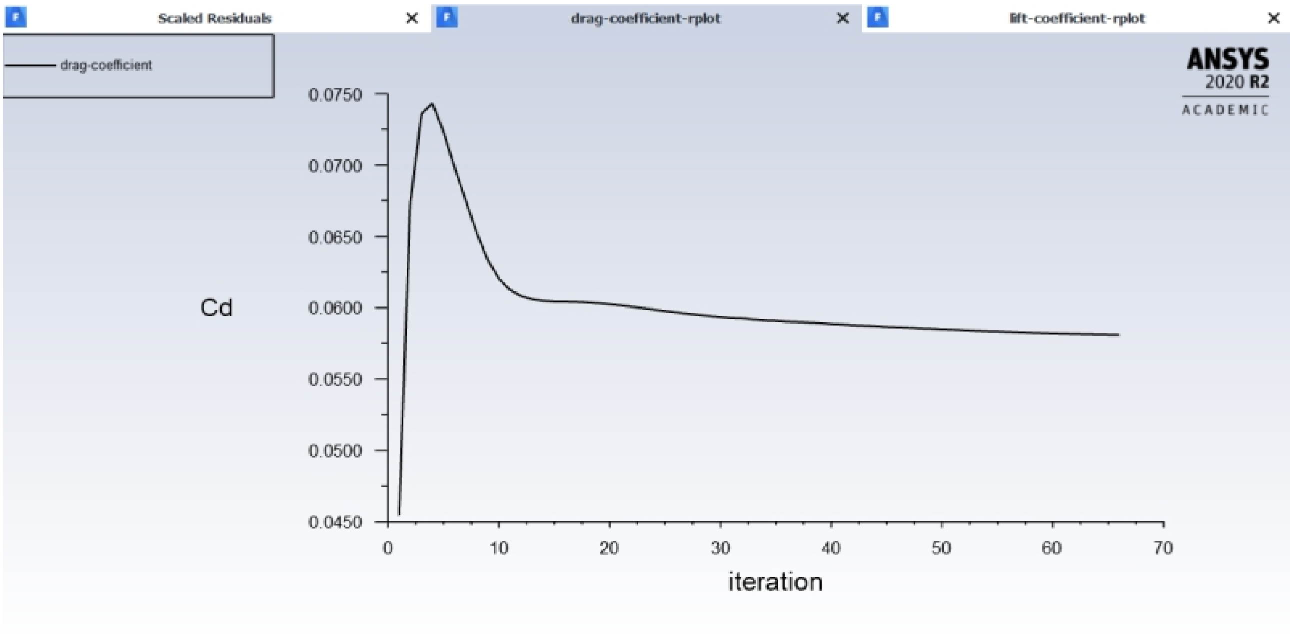

Fig. 5.19: Car model Drag-coefficient plot

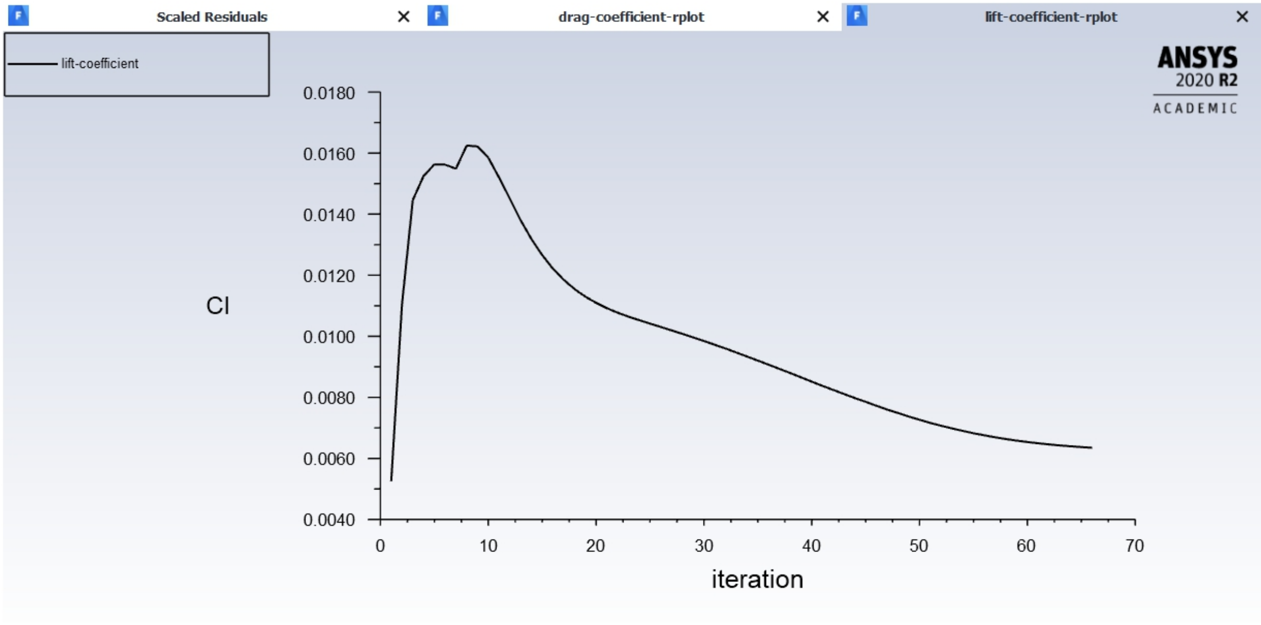

Fig 5.20: Car model Lift-coefficient plot

- <u>Commercial Car Model ;-</u>

Giving some images from the analysis

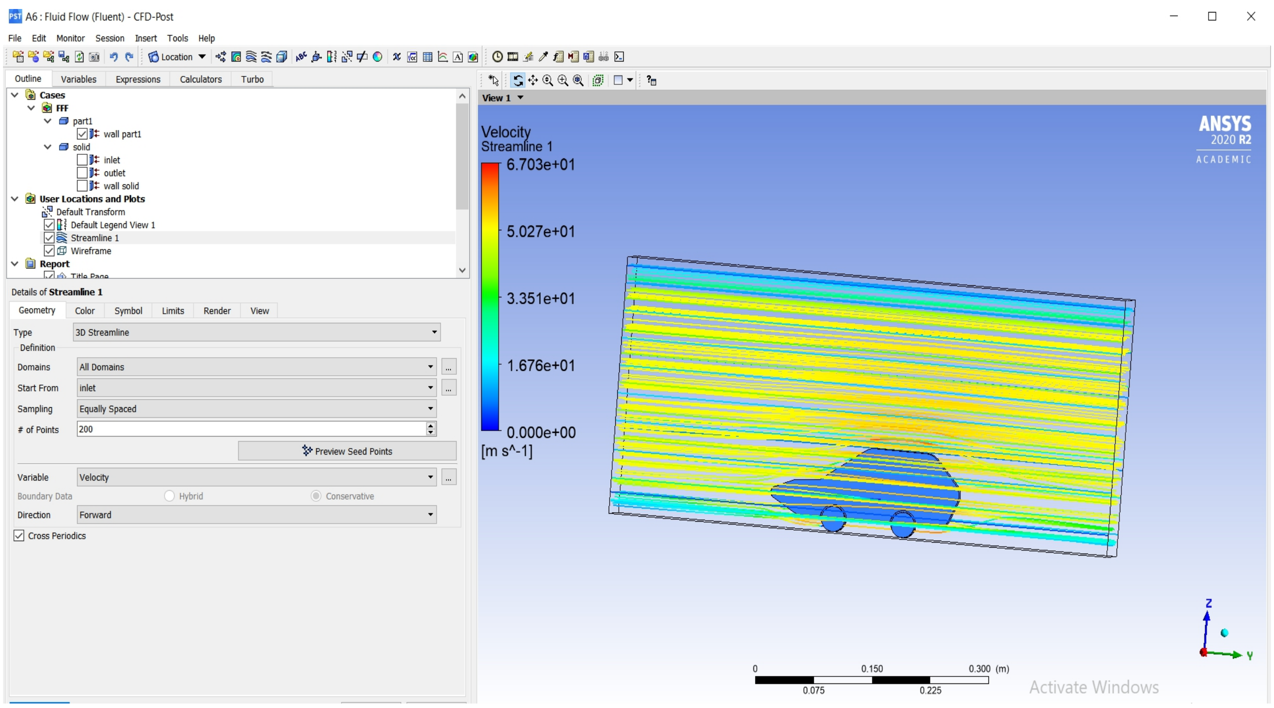

Fig. 5.21

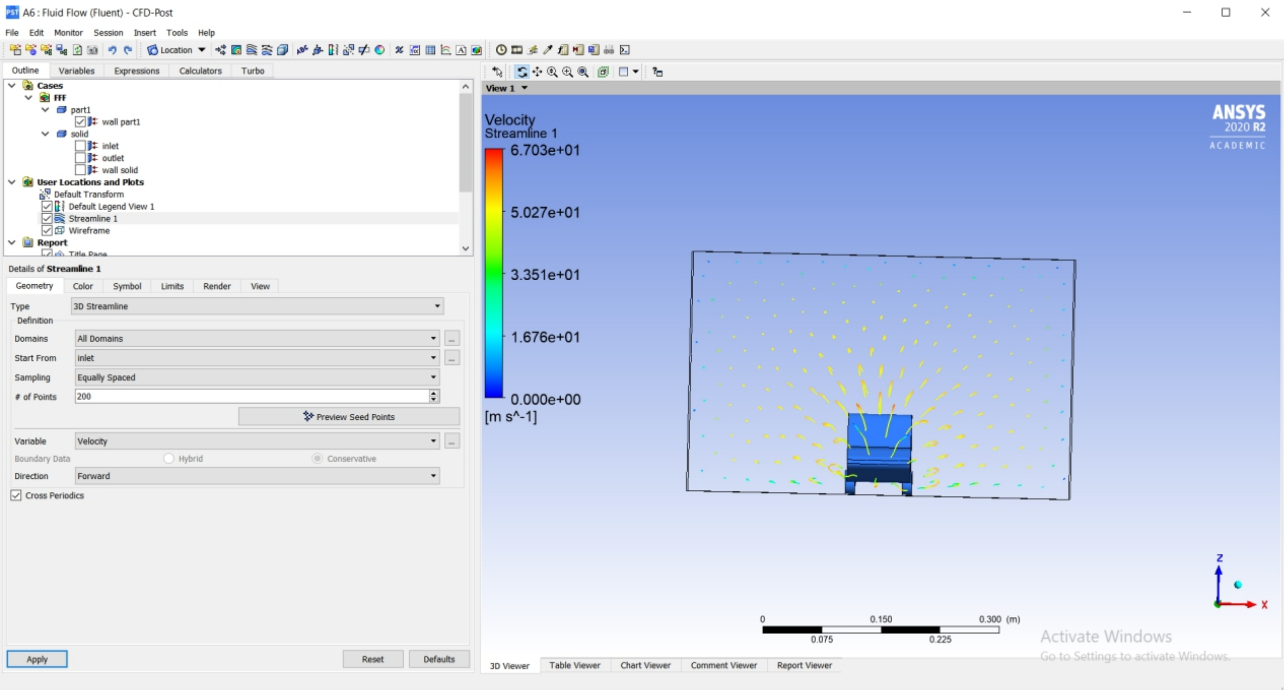

Fig. 5.22

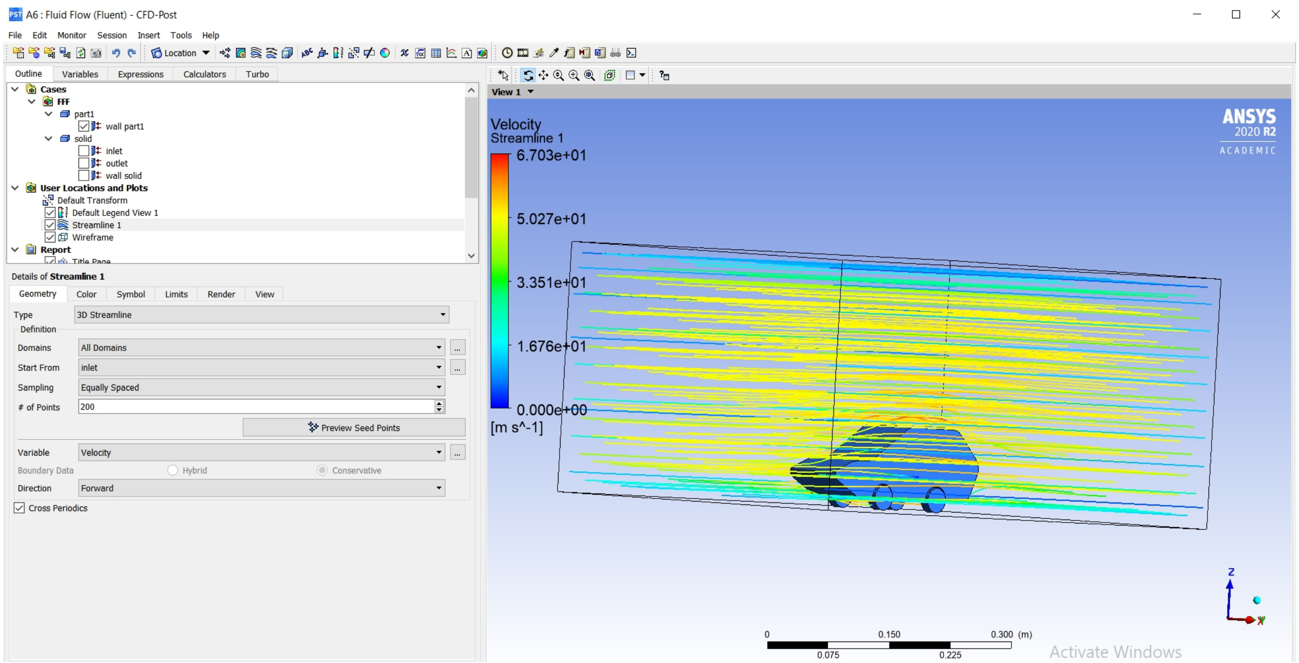

Fig. 5.23

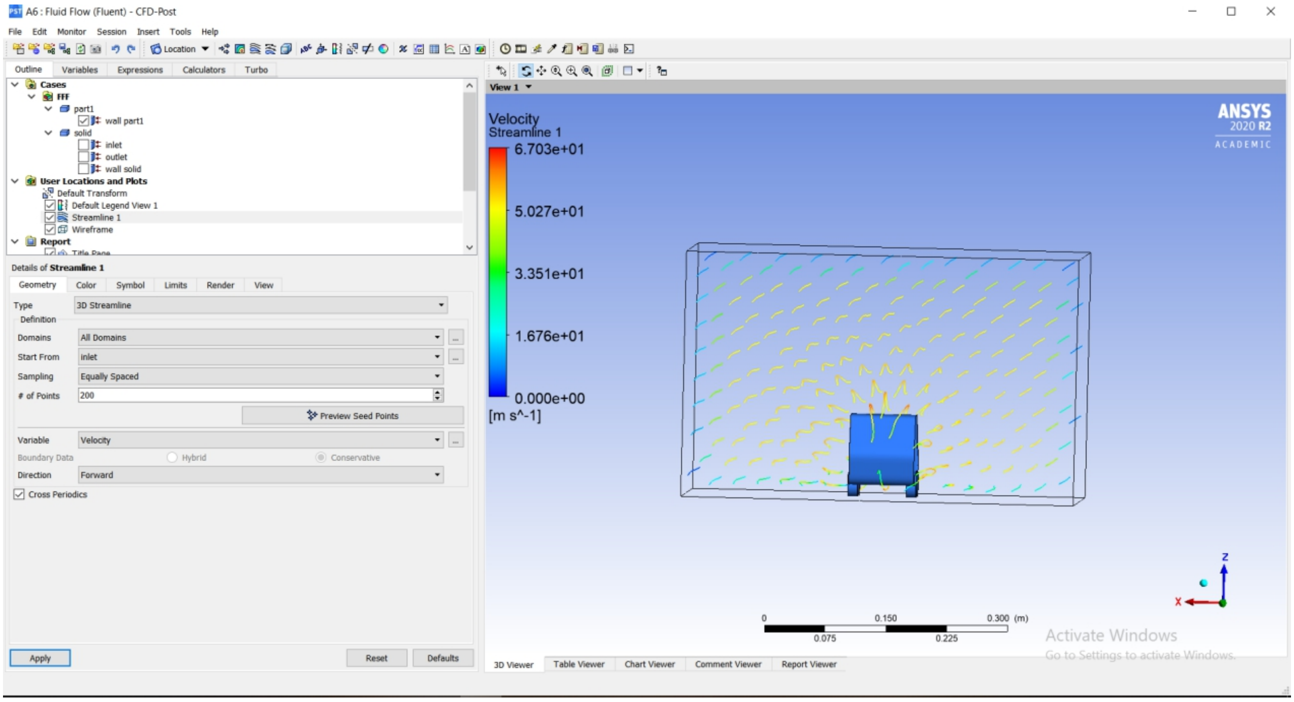

Fig. 5.24

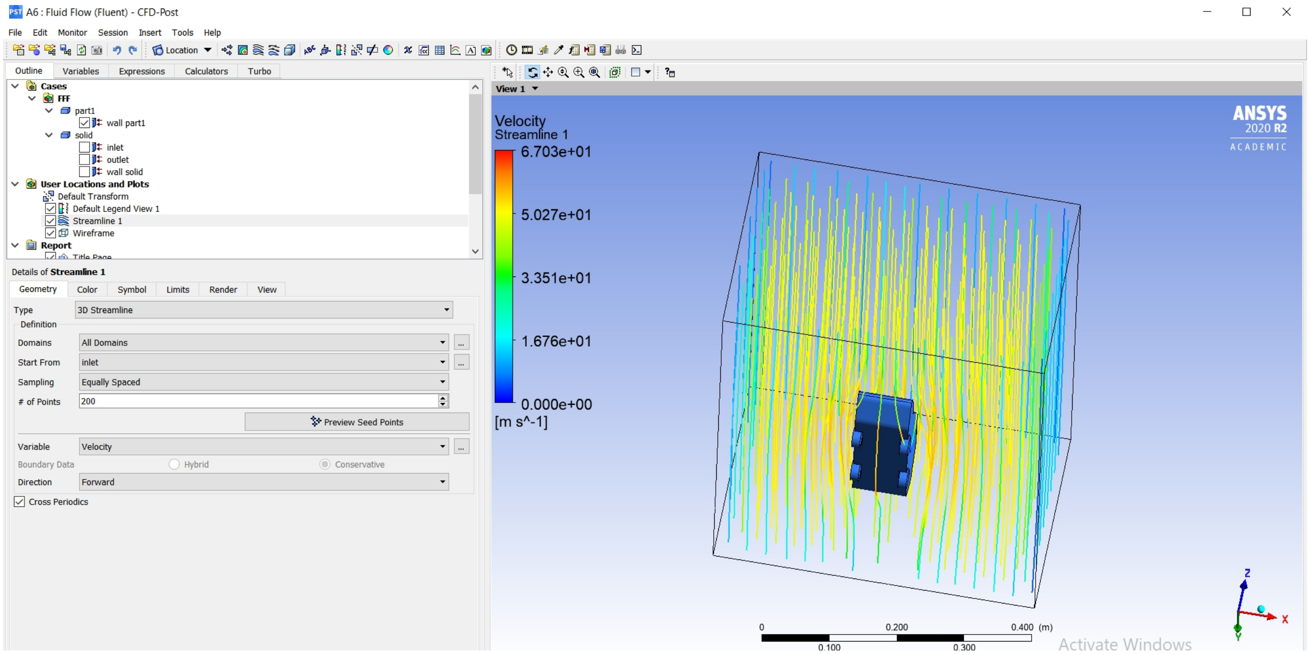

Fig. 5.25

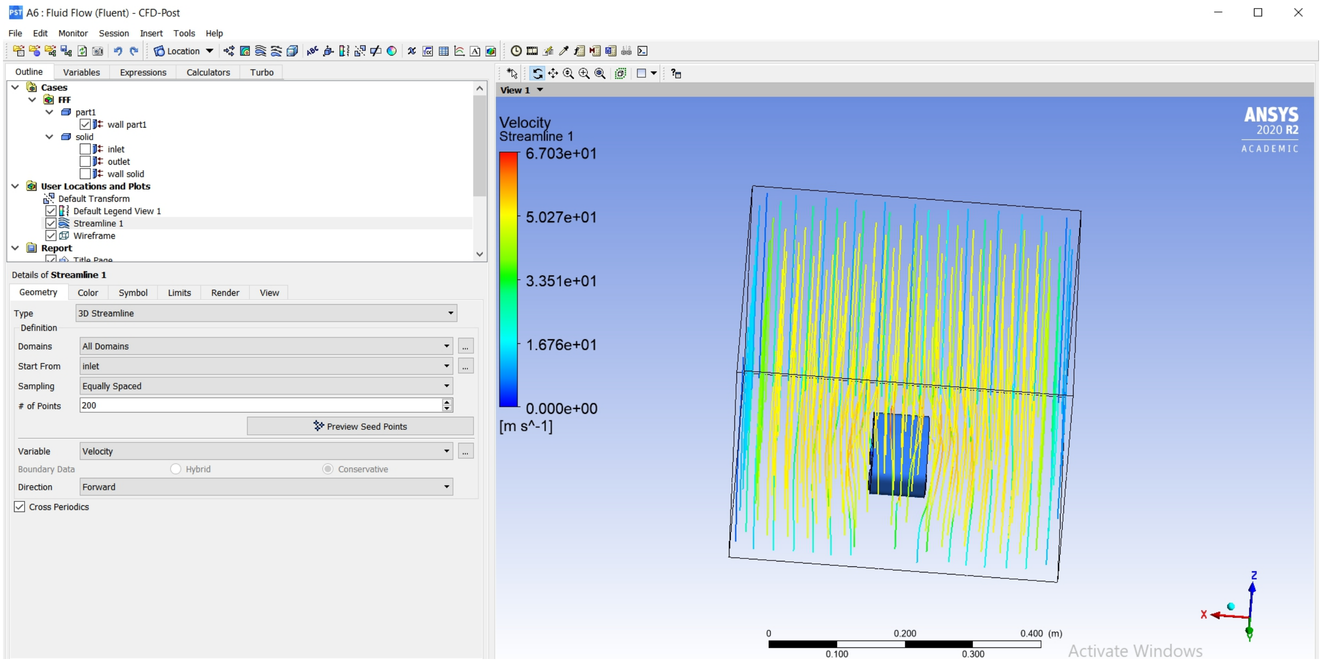

Fig. 5.26

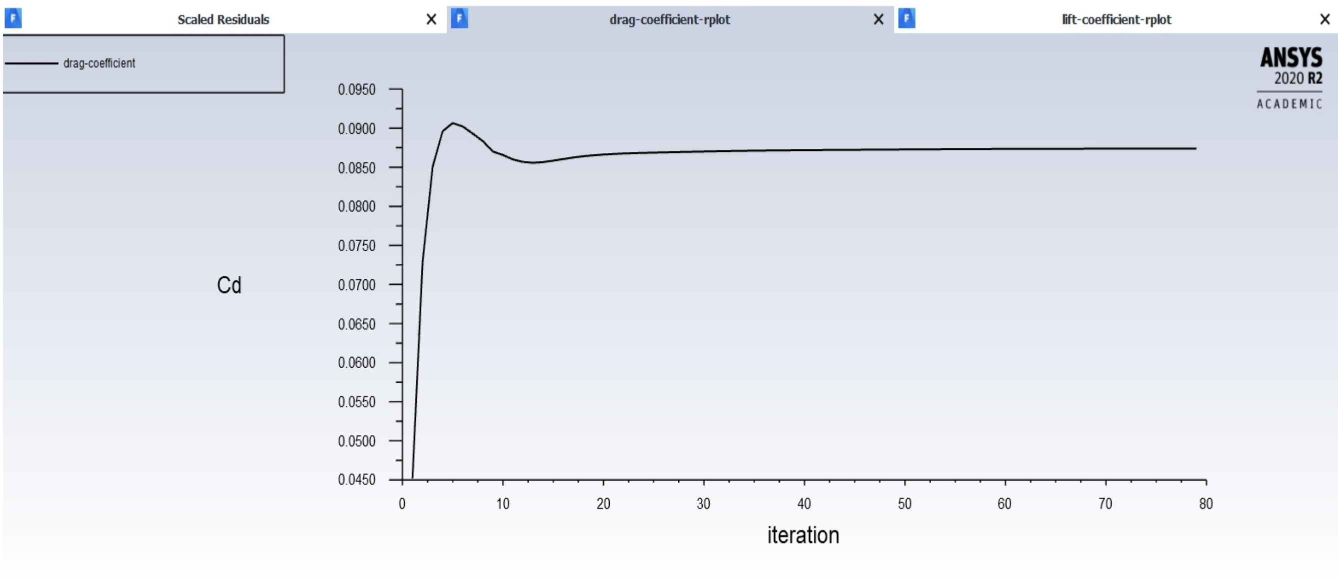

Fig. 5.27: Commercial model Drag-coefficient plot

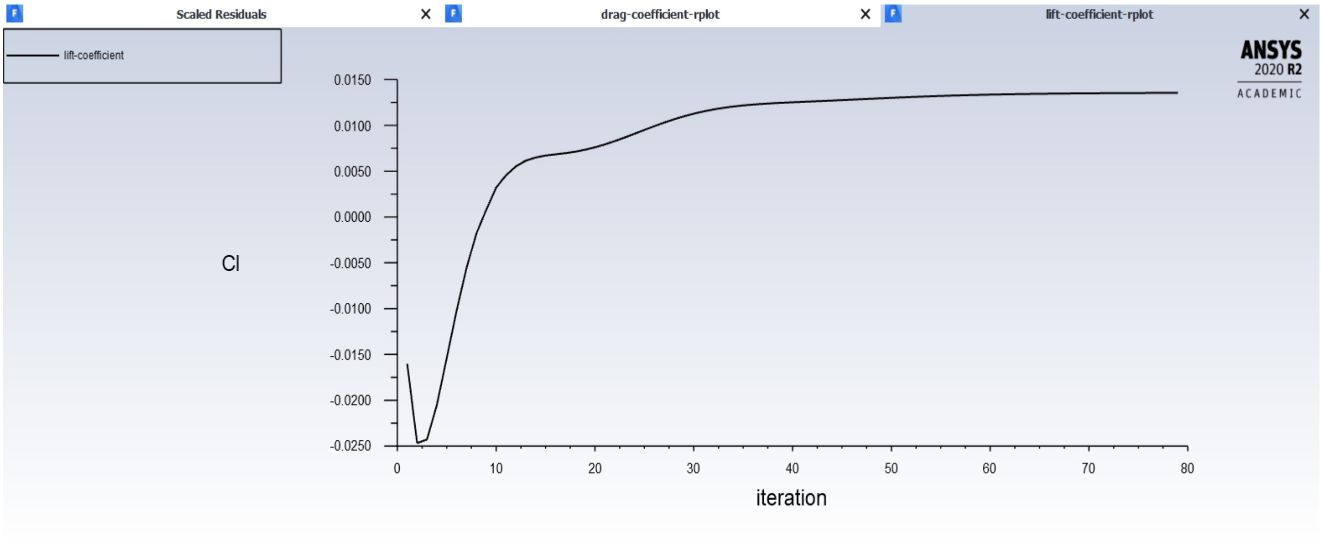

Fig. 5.28: Commercial model Lift-coefficient plot

- <u>Racing Car Model</u> ;-

Giving some images from the analysis

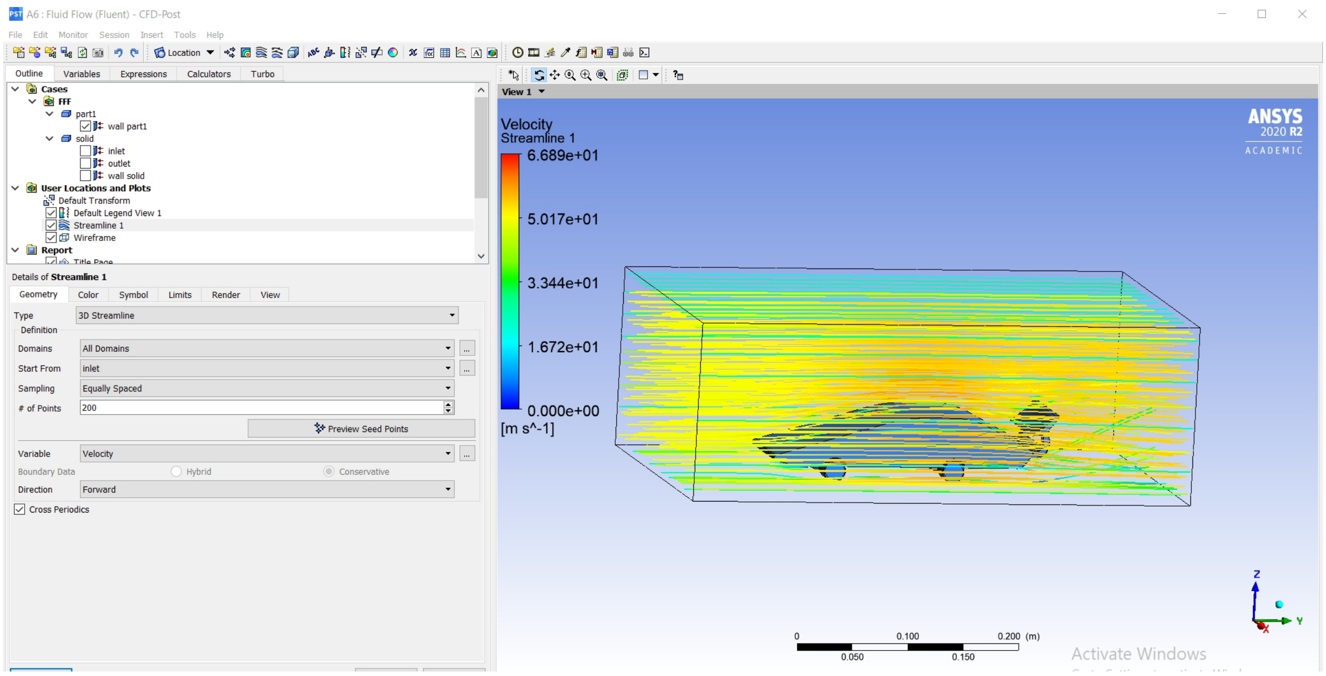

Fig. 5.29

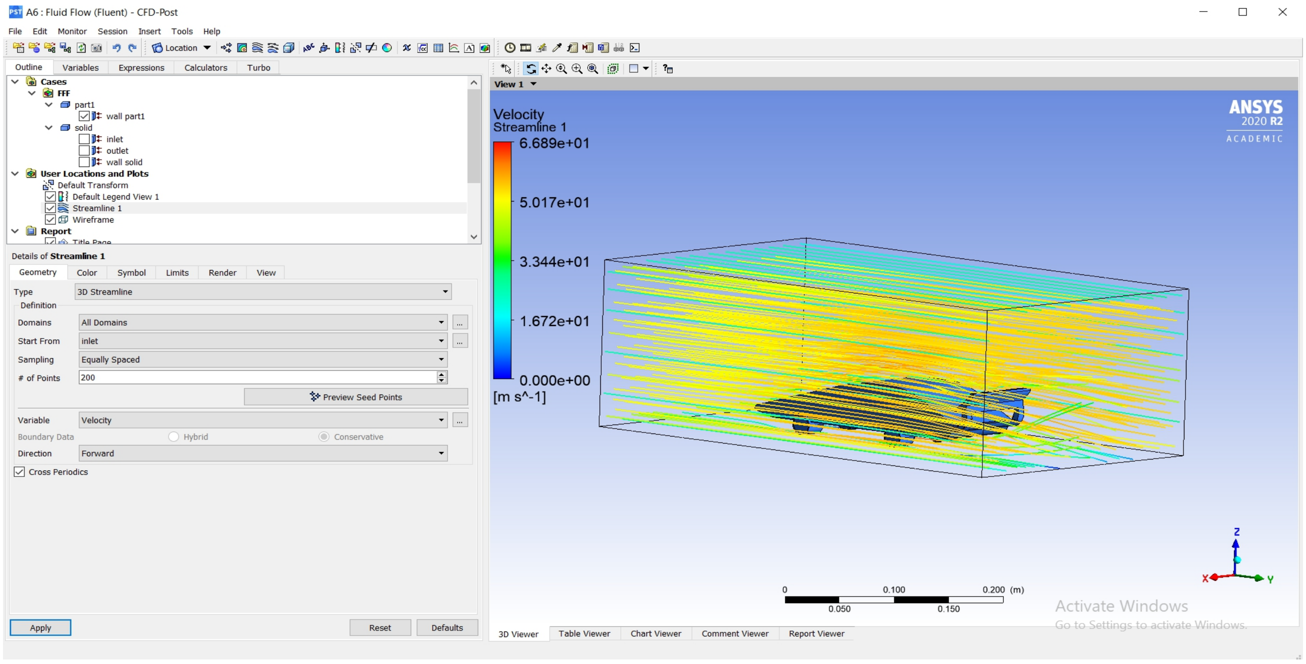

Fig. 5.30

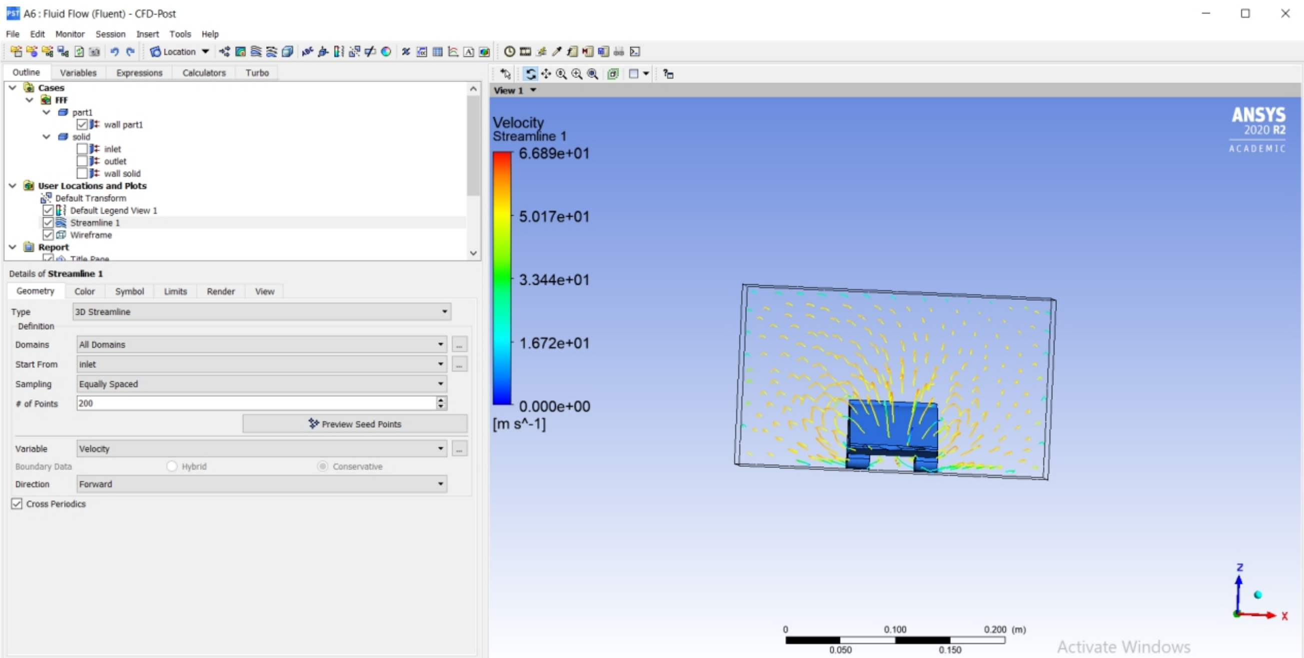

Fig. 5.31

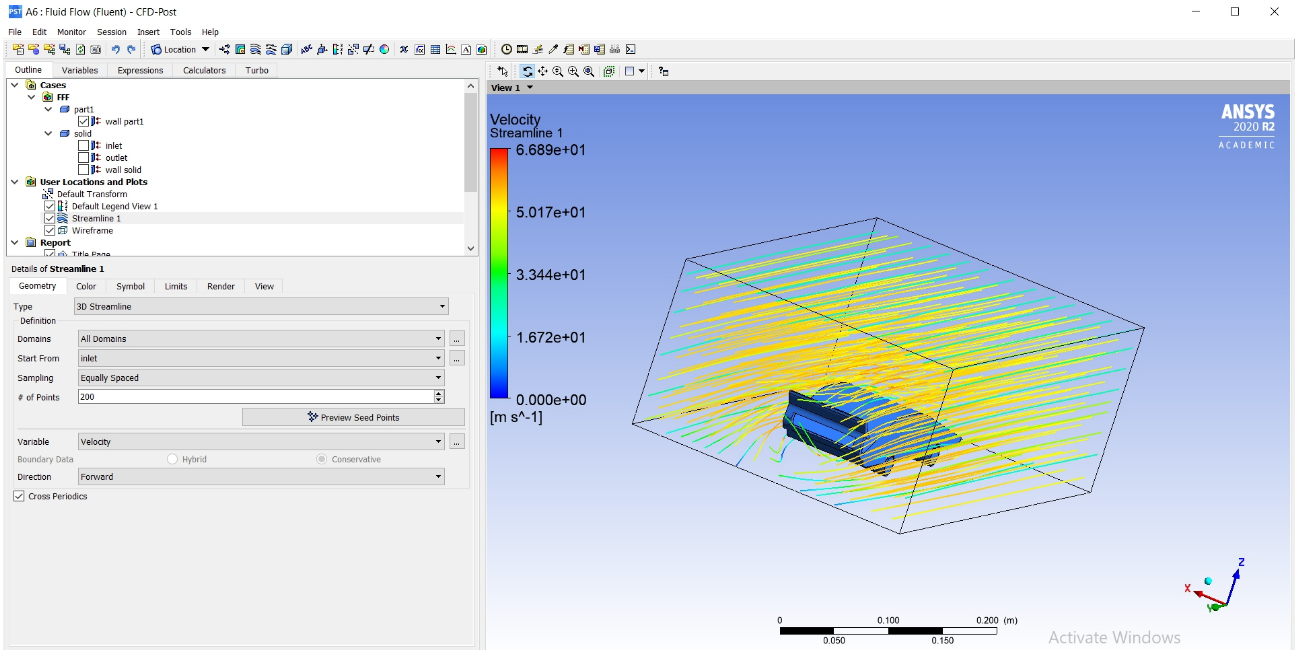

Fig. 5.32

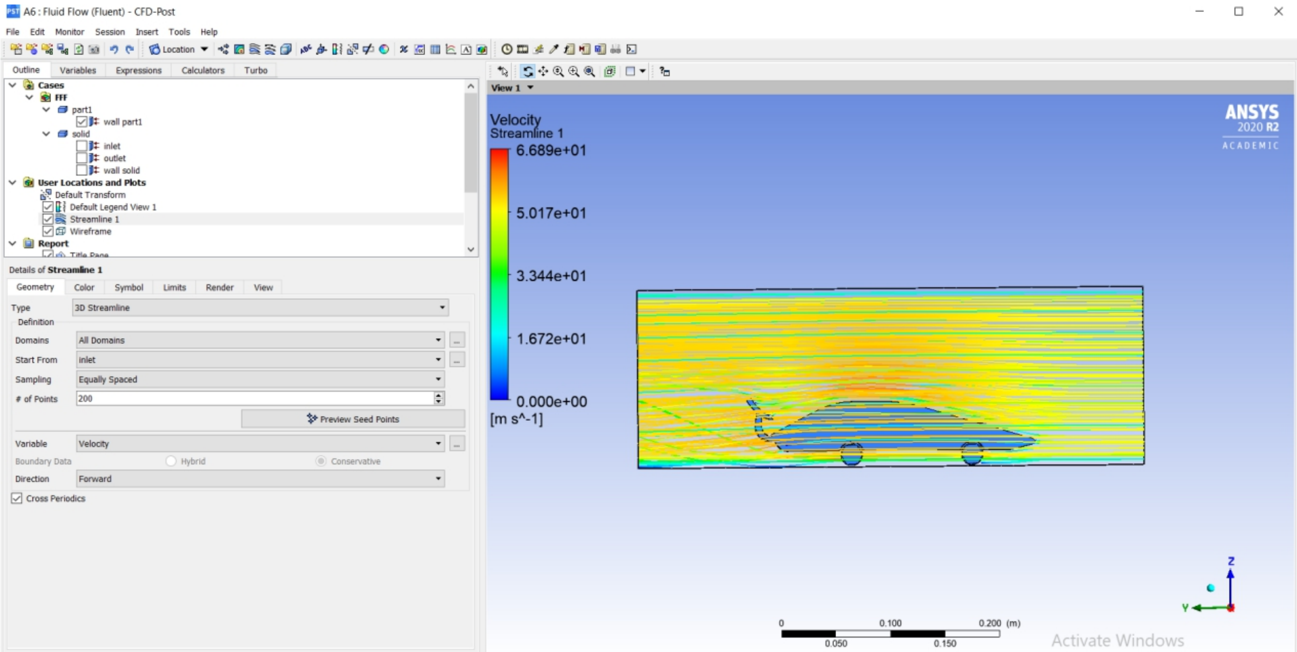

Fig. 5.33

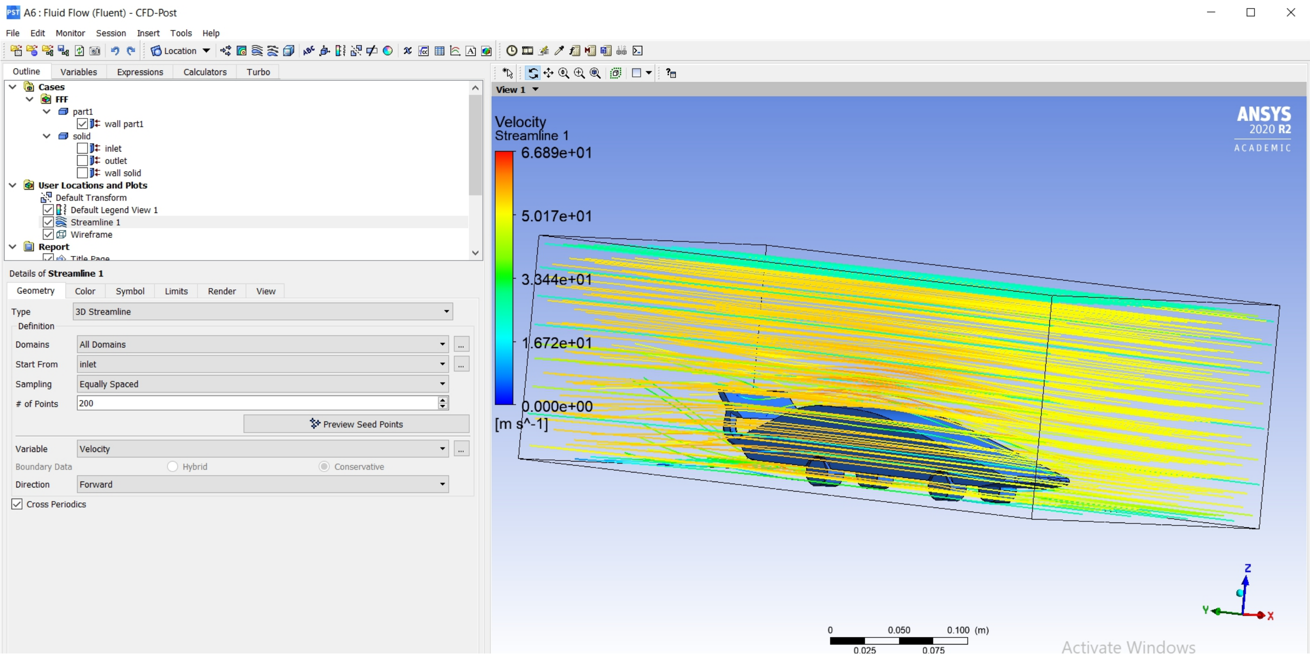

Fig. 5.34

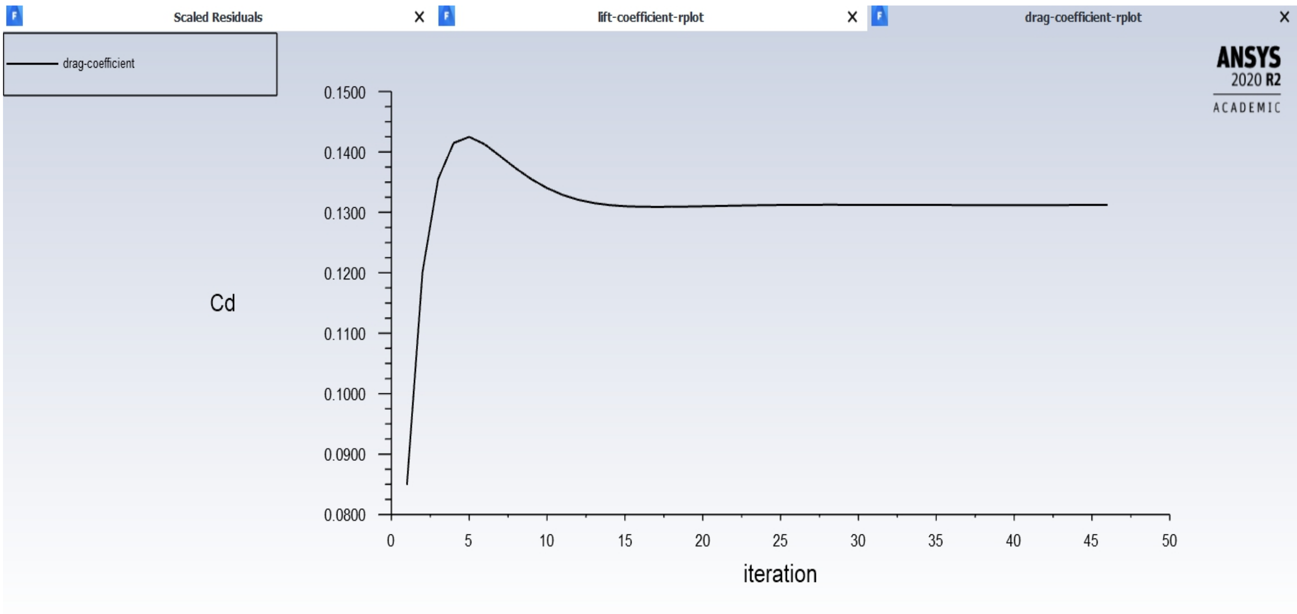

Fig. 5.35: Racing car model Drag-coefficient plot

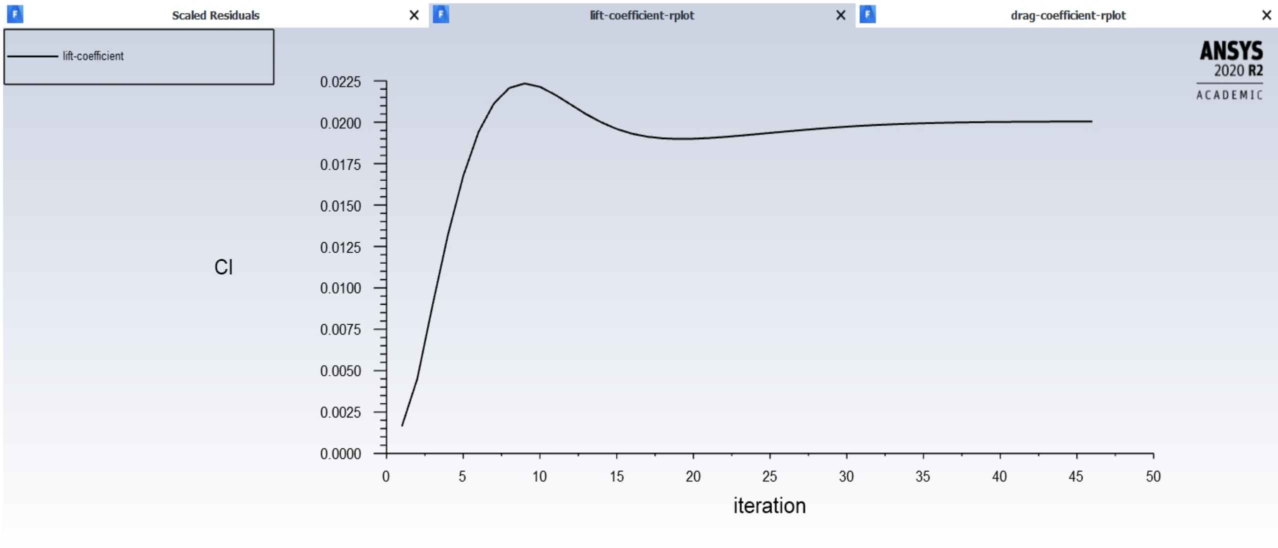

Fig. 5.36: Racing car model Lift-coefficient plot

- <u>Omni Model</u> :-

Giving some images from the analysis

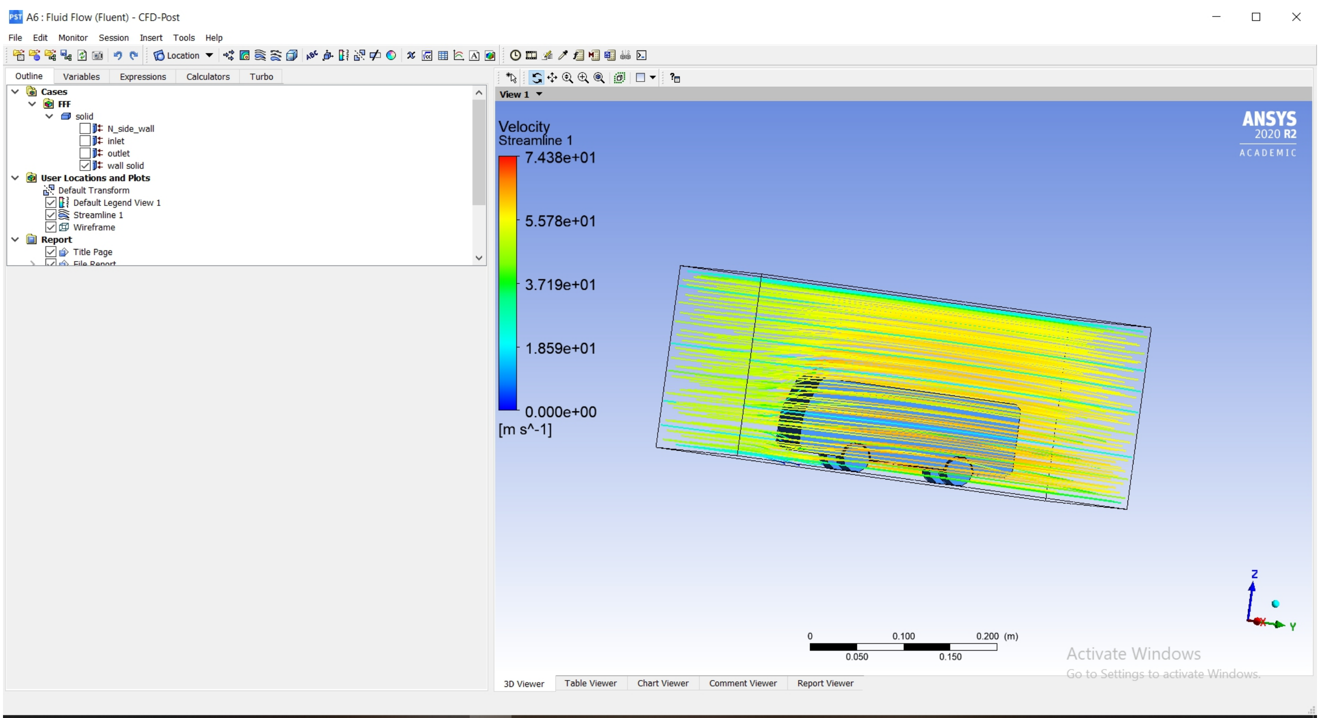

Fig. 5.37

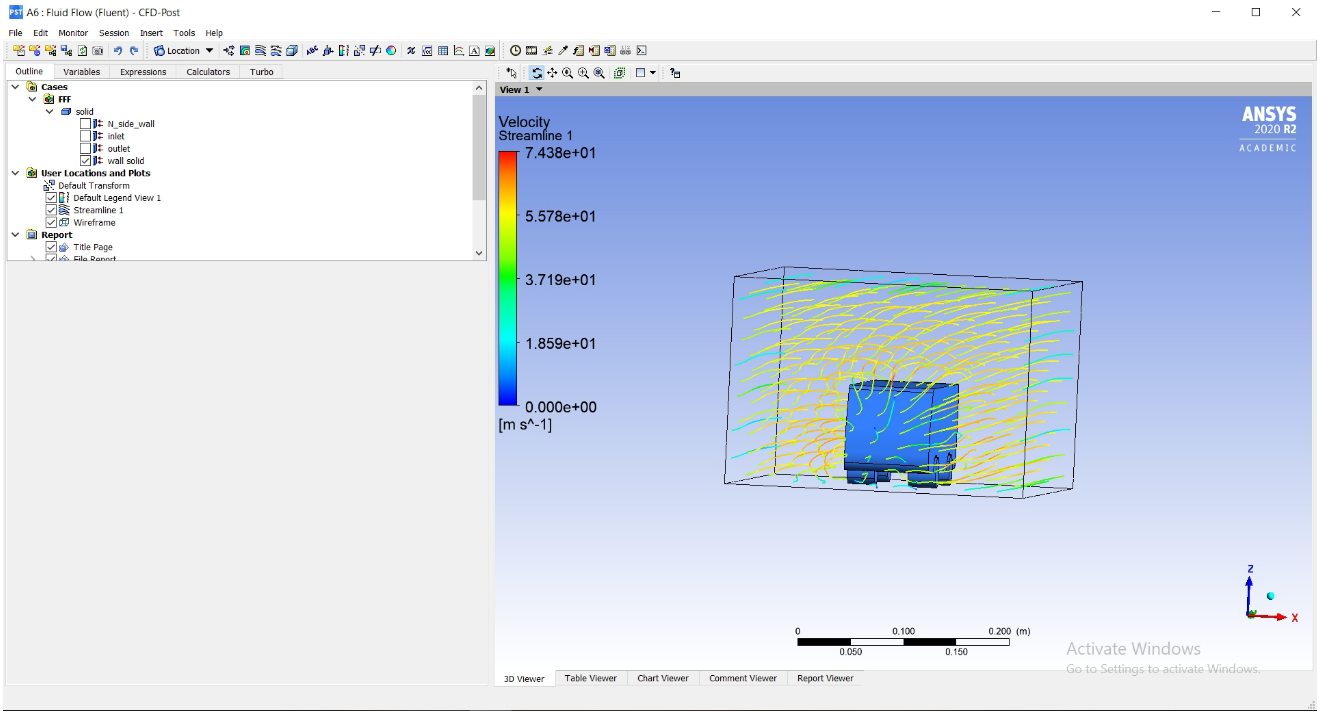

Fig 5.38

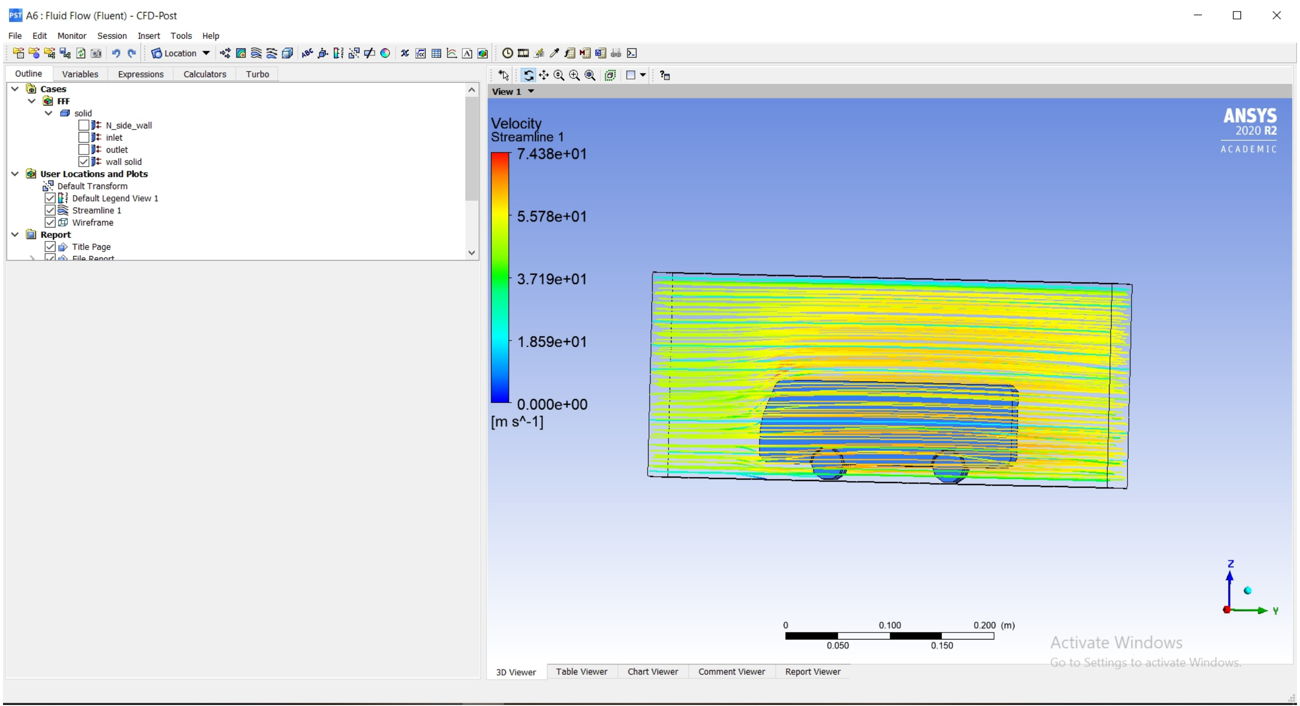

Fig. 5.39

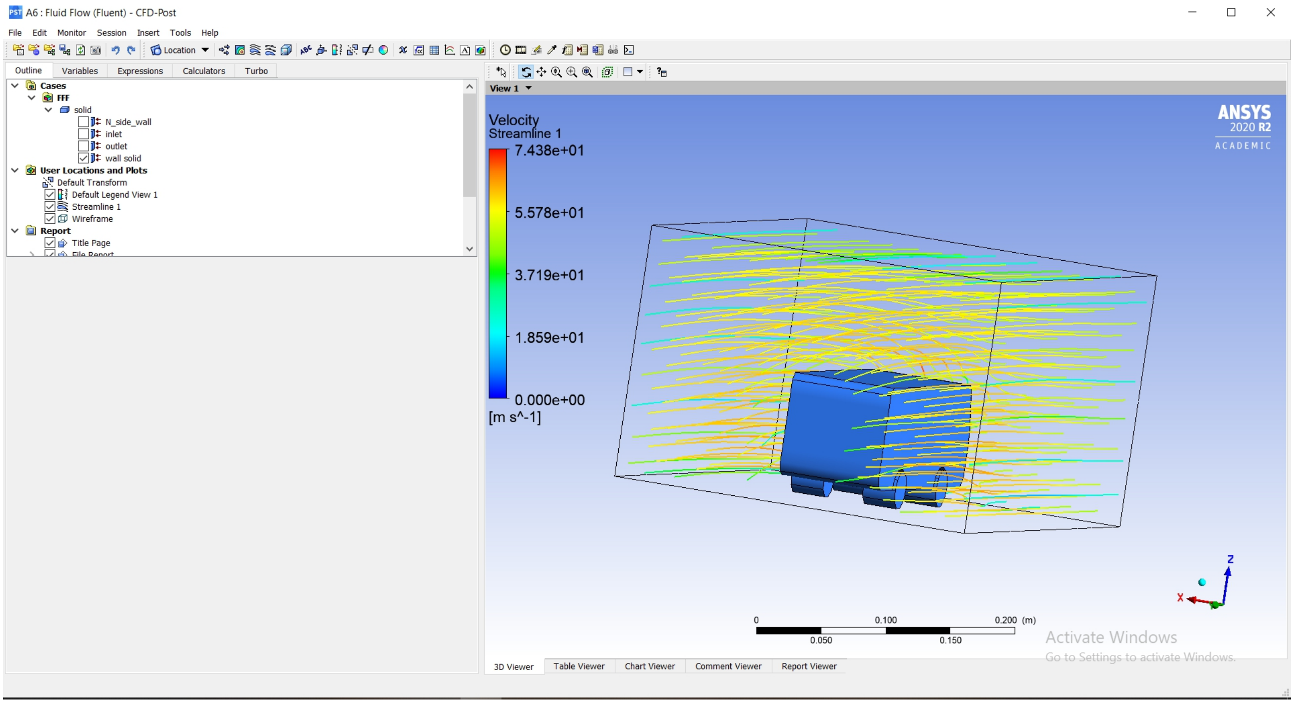

Fig. 5.40

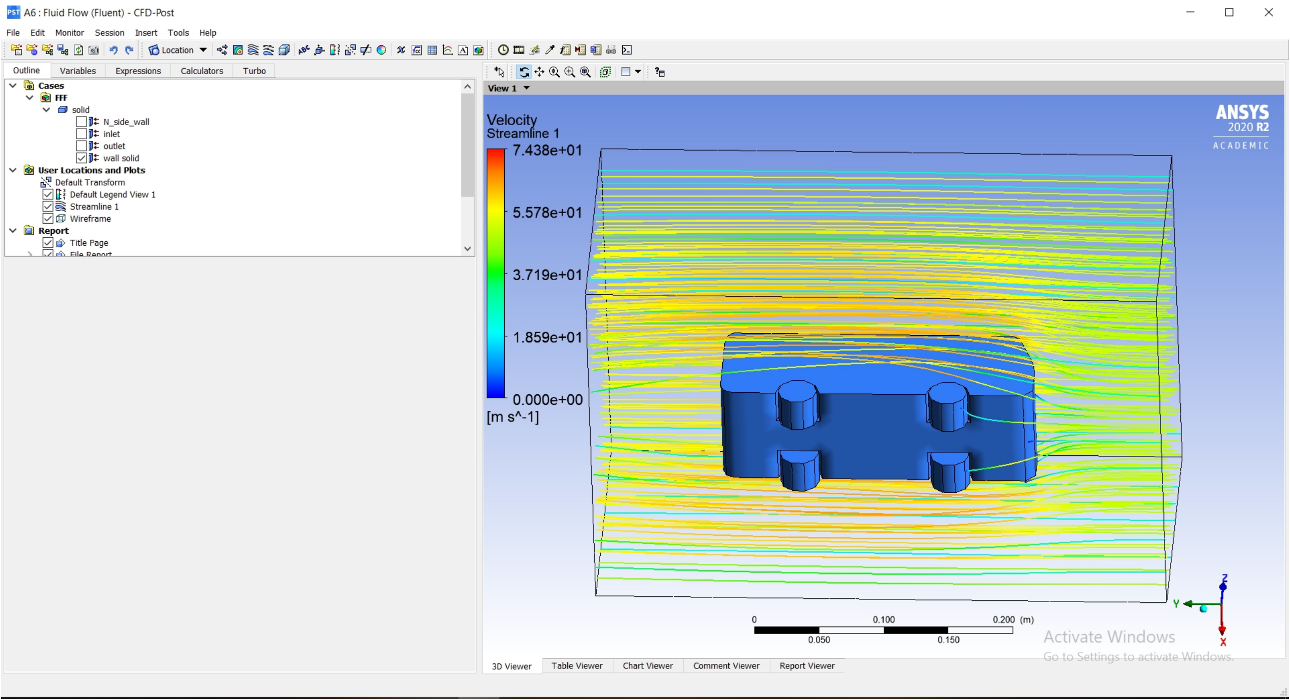

Fig. 5.41

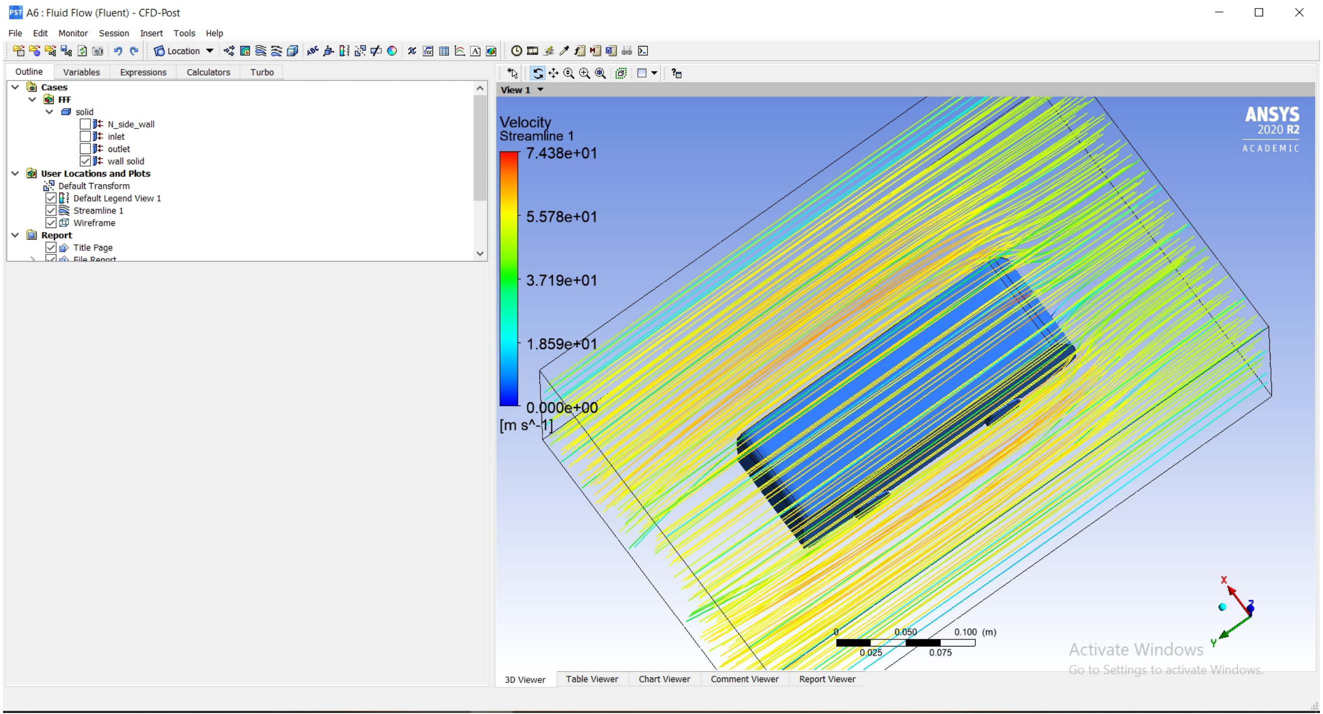

Fig. 5.42

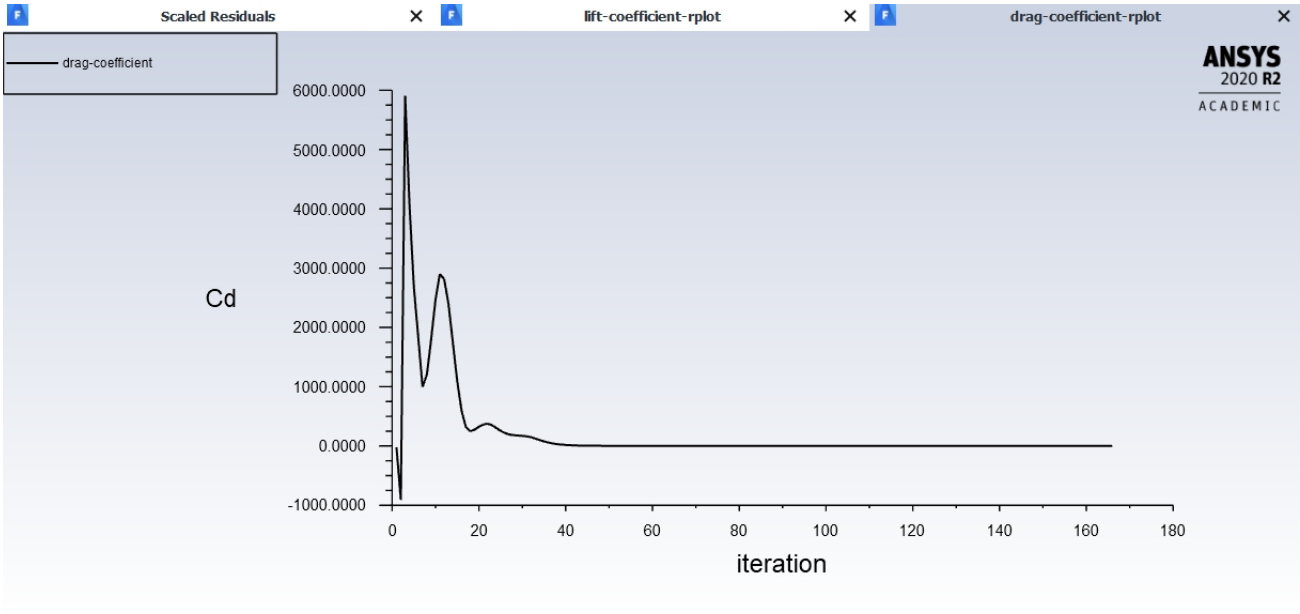

Fig. 5.43: Omni car model Drag-coefficient plot

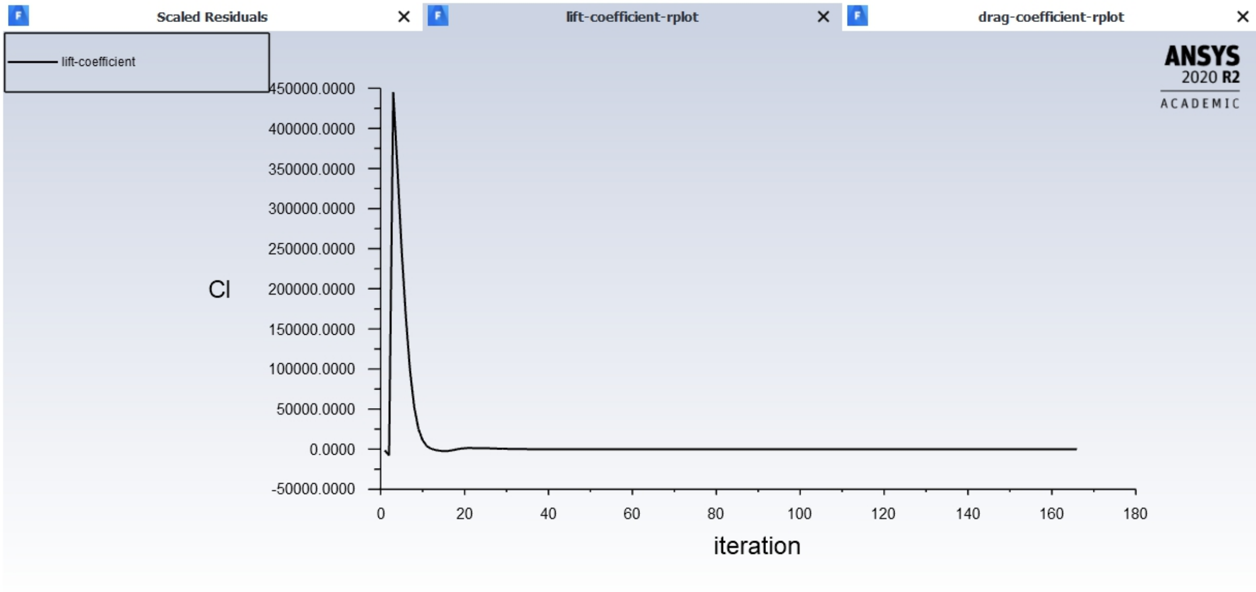

Fig. 5.44: Omni car model Lift-coefficient plot

6. DRAG-COEFFICIENT MINIMIZATION

6.1 FLOW DOMAIN OR COMPUTATIONAL DOMAIN

The area around the 3d model, which defines the flow simulation the restricted area, is known as the computational domain.

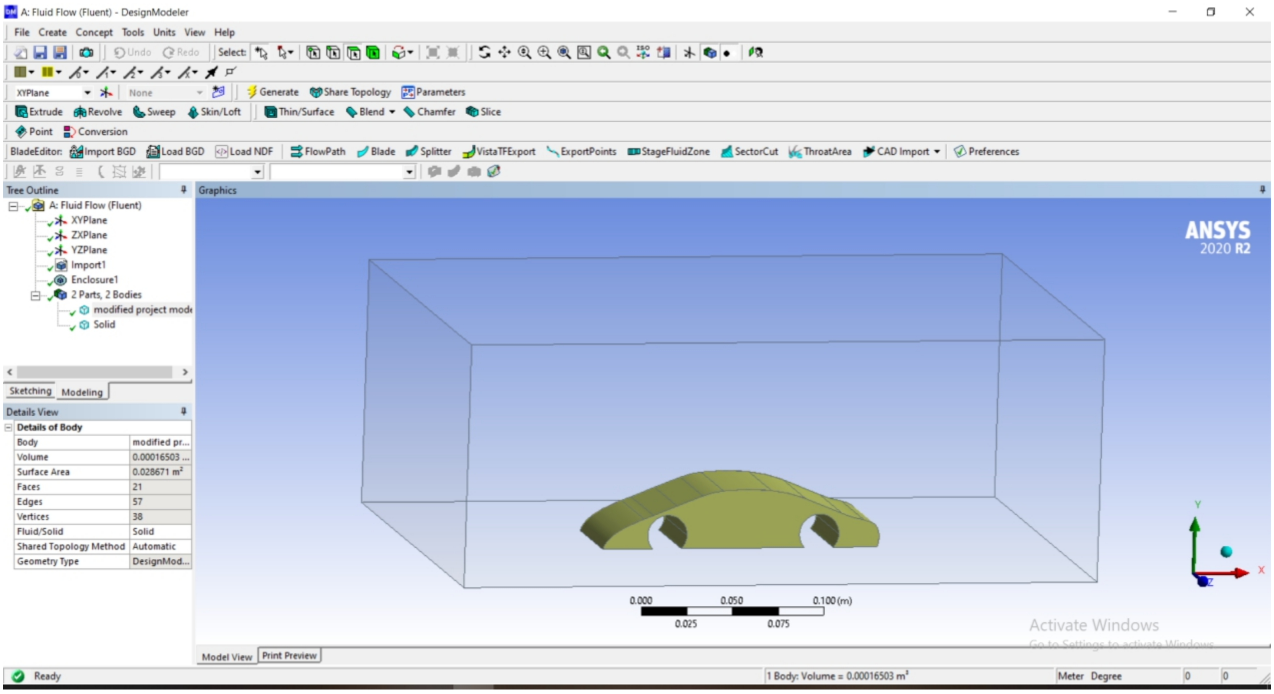

Fig. 6.1: Computational domain of Drag-coefficient minimised Car Model

COMPUTATIONAL DOMAIN SIZE:

X min	-0.1 m
X max	0.1 m
Y min	-0.0025 m
Y max	0.1 m
Z min	-0.1 m
Z max	0.1 m

6.2 CREATING MESH

During drag and lift analysis of this model the hex dominant mesh system is used.

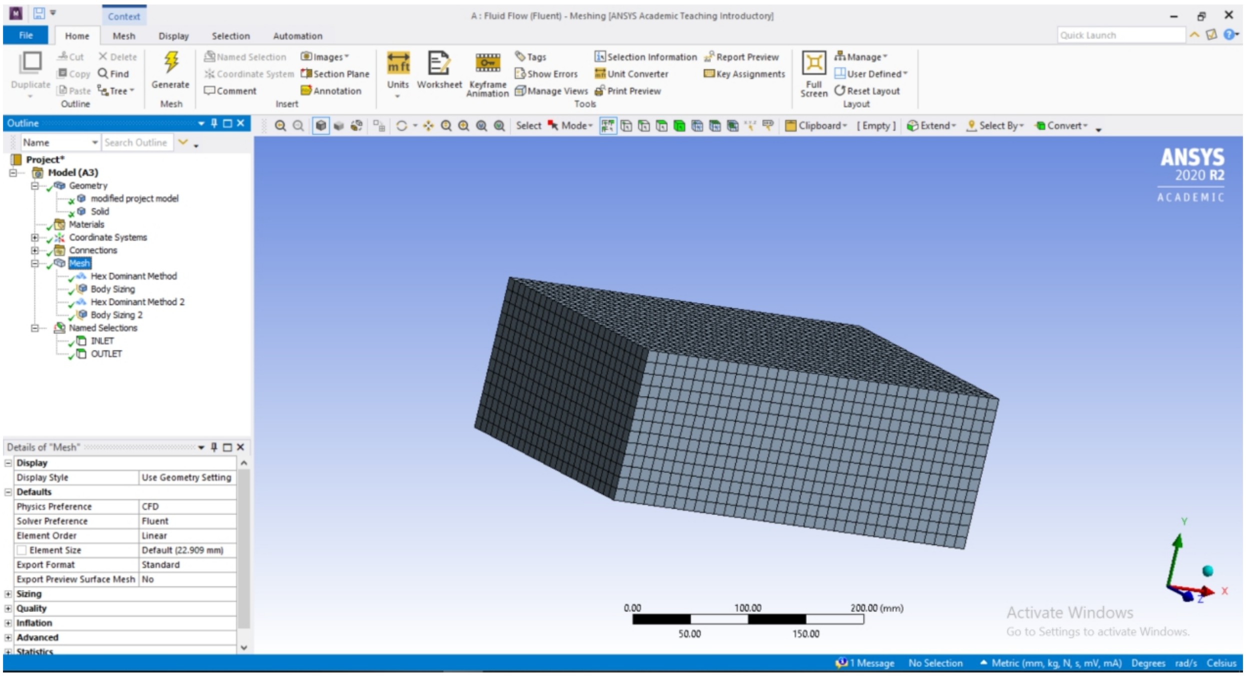

Fig. 6.2: Enclosure mesh of Drag-coefficient minimised car model

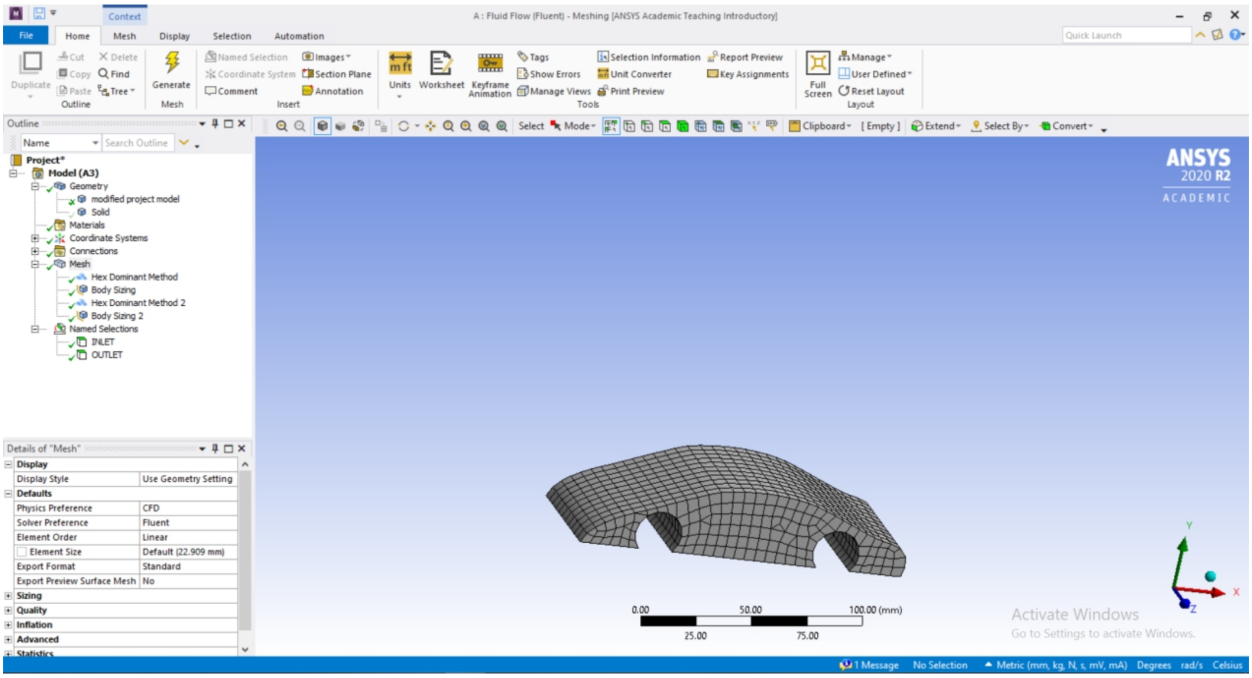

Fig. 6.3: Mesh of Drag-coefficient minimised car model

Mesh Report of Drag-coefficient minimised car model

Domain	Nodes	Elements
Modified project model	2113	2040
solid	14295	13650
All Domains	16408	15690

Physics Report of Drag-coefficient minimised car model

Domain – modified project model	
Type	solid
Domain - solid	
Type	cell

Boundaries of Drag-coefficient minimised car model

Domain	Boundaries	
Modified project model	**Boundary - wall modified_project_model**	
	Type	WALL
solid	**Boundary - inlet**	
	Type	VELOCITY-INLET
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Boundary - wall solid	
	Type	WALL

6.3 INITIAL CONDITION

As the Flow domain or the computational domain was established then physical conditions are needed for the boundary of the computational domain.

INLET VELOCITY USED =50 m/s

6.4 ESTABLISHING THE GOALS

After completing all the steps for pressure and velocity distribution we will define the nature of the flow simulation over the body.

6.5 RESULTS OF SIMULATION

After completing the simulation the results are ready to view. The result obtained in the form of graphs is between the coefficient of lift and coefficient of drag.

Number of streamlines used to depict the air flow =200

Giving some images from the analysis:

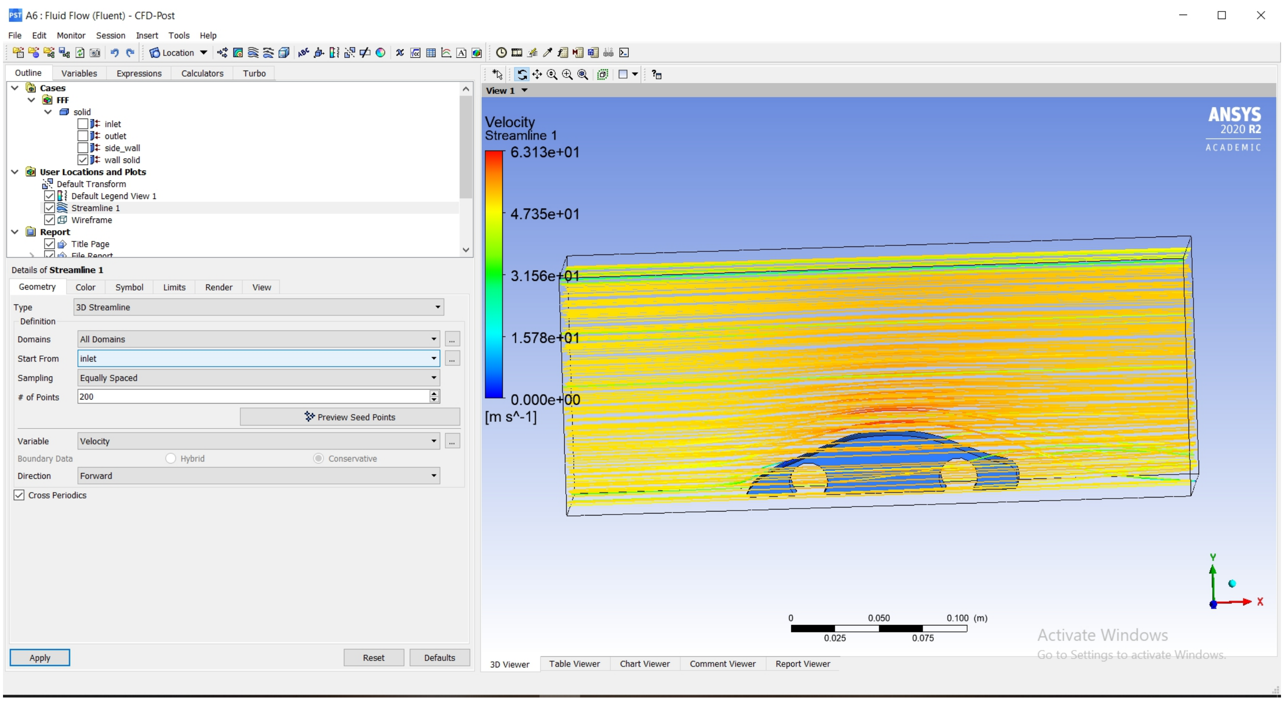

Fig. 6.4

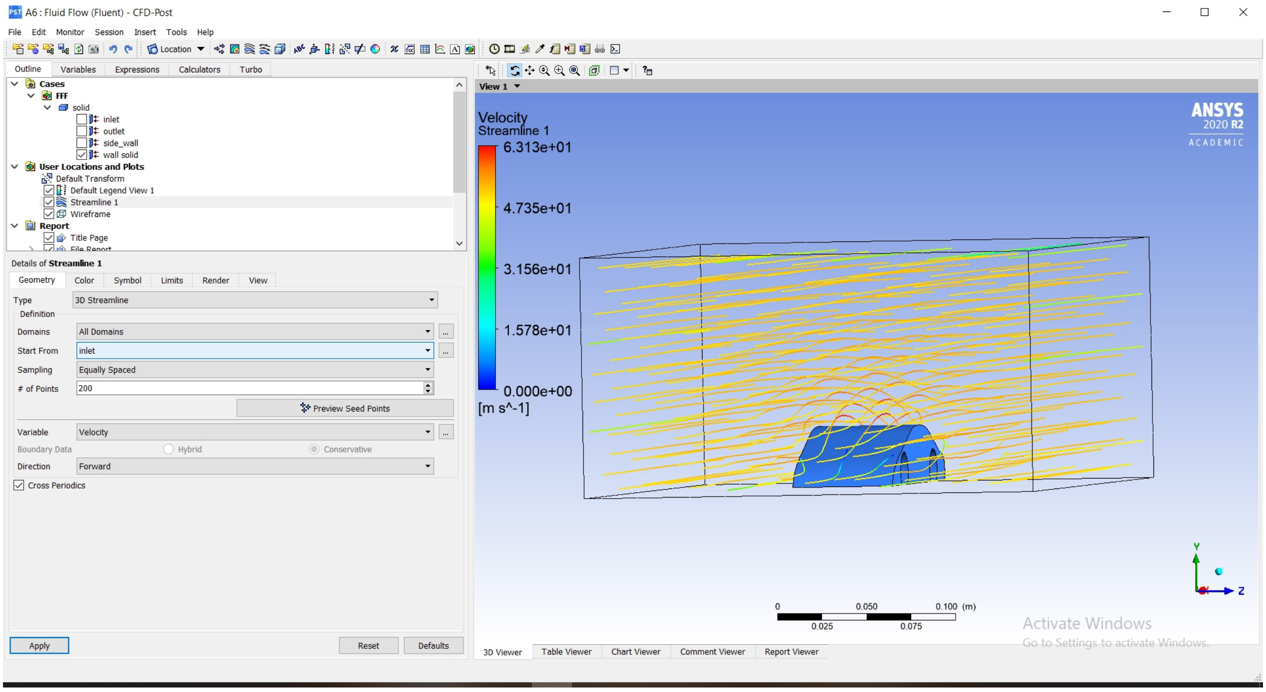
A6 : Fluid Flow (Fluent) - CFD-Post
File Edit Monitor Session Insert Tools Help
Outline Variables Expressions Calculators Turbo
Cases
FFF
solid
inlet
outlet
side_wall
wall solid
User Locations and Plots
Default Transform
Default Legend View 1
Streamline 1
Wireframe
Report
Title Page
Details of Streamline 1
Geometry Color Symbol Limits Render View
Type 3D Streamline
Definition
Domains All Domains
Start From inlet
Sampling Equally Spaced
# of Points 200
Preview Seed Points
Variable Velocity
Boundary Data Hybrid Conservative
Direction Forward
Cross Periodics
Apply Reset Defaults
View 1
Velocity
Streamline 1
6.313e+01
4.735e+01
3.156e+01
1.578e+01
0.000e+00
[m s^-1]
ANSYS
2020 R2
ACADEMIC
0 0.050 0.100 (m)
0.025 0.075
Activate Windows
Go to Settings to activate Windows.
3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer

Fig. 6.5

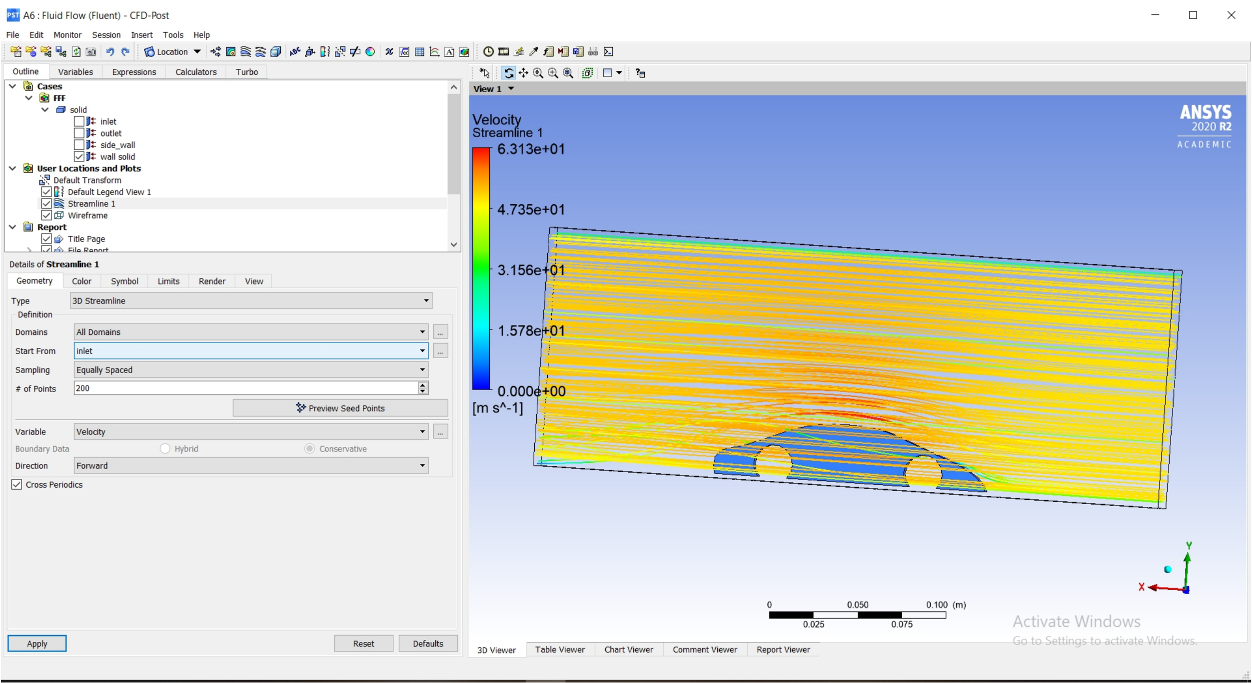
A6 : Fluid Flow (Fluent) - CFD-Post
File Edit Monitor Session Insert Tools Help
Outline Variables Expressions Calculators Turbo
Cases
FFF
solid
inlet
outlet
side_wall
wall solid
User Locations and Plots
Default Transform
Default Legend View 1
Streamline 1
Wireframe
Report
Title Page
Details of Streamline 1
Geometry Color Symbol Limits Render View
Type 3D Streamline
Definition
Domains All Domains
Start From inlet
Sampling Equally Spaced
# of Points 200
Preview Seed Points
Variable Velocity
Boundary Data Hybrid Conservative
Direction Forward
Cross Periodics
Apply Reset Defaults
View 1
Velocity
Streamline 1
6.313e+01
4.735e+01
3.156e+01
1.578e+01
0.000e+00
[m s^-1]
ANSYS
2020 R2
ACADEMIC
0 0.050 0.100 (m)
0.025 0.075
Activate Windows
Go to Settings to activate Windows.
3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer

Fig. 6.6

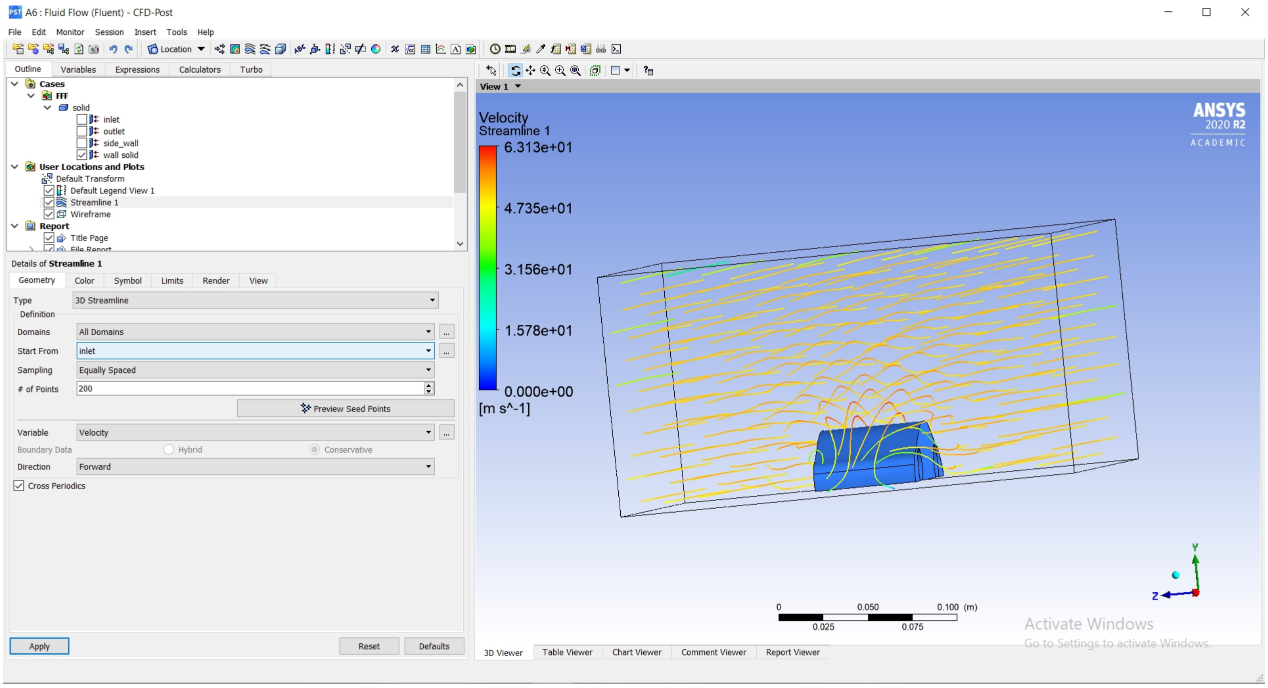

Fig. 6.7

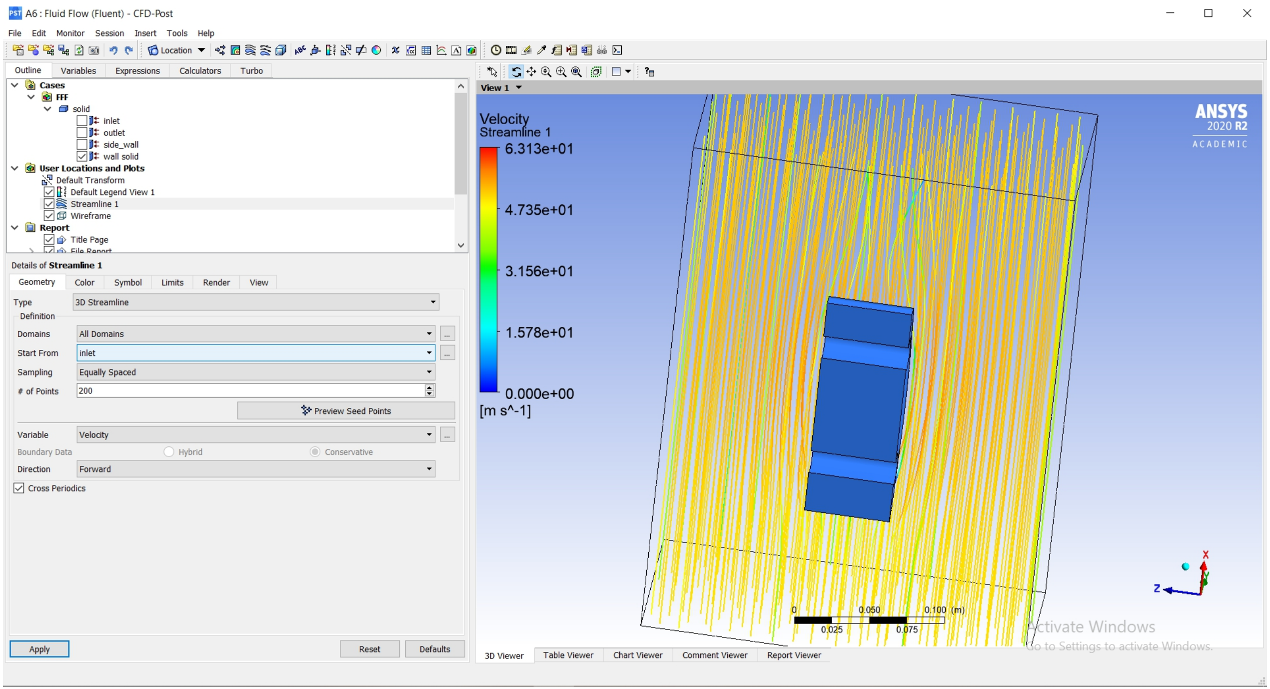

Fig. 6.8

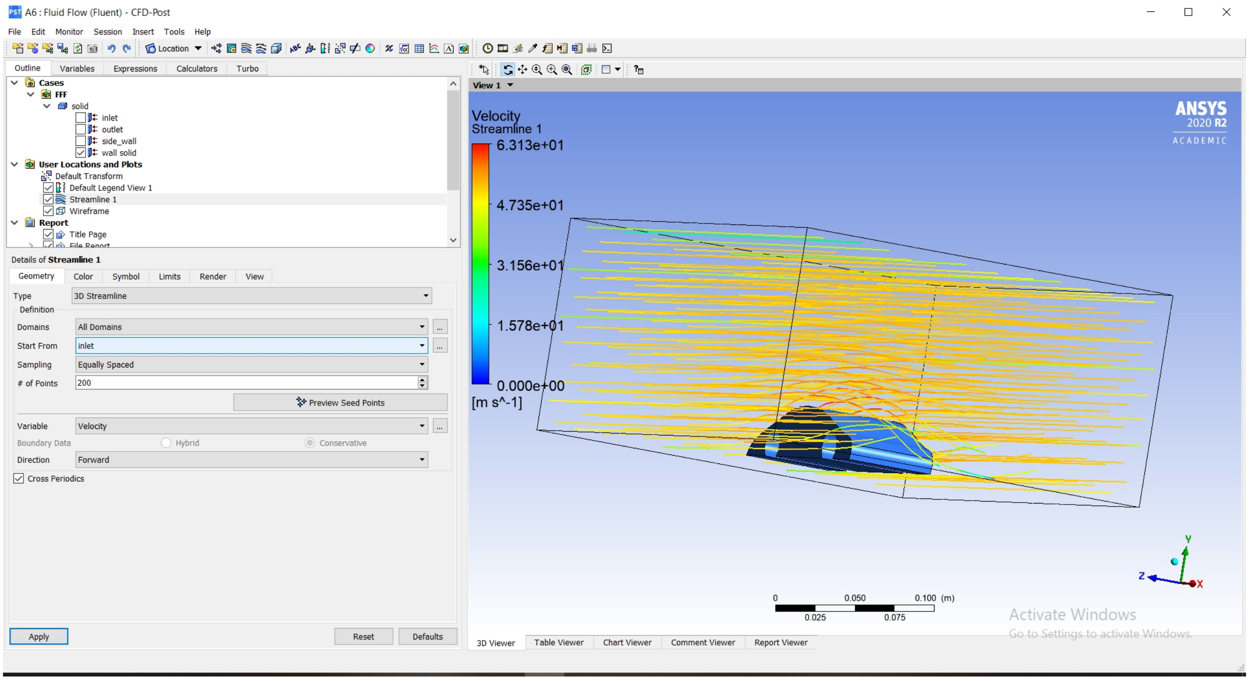

Fig. 6.9

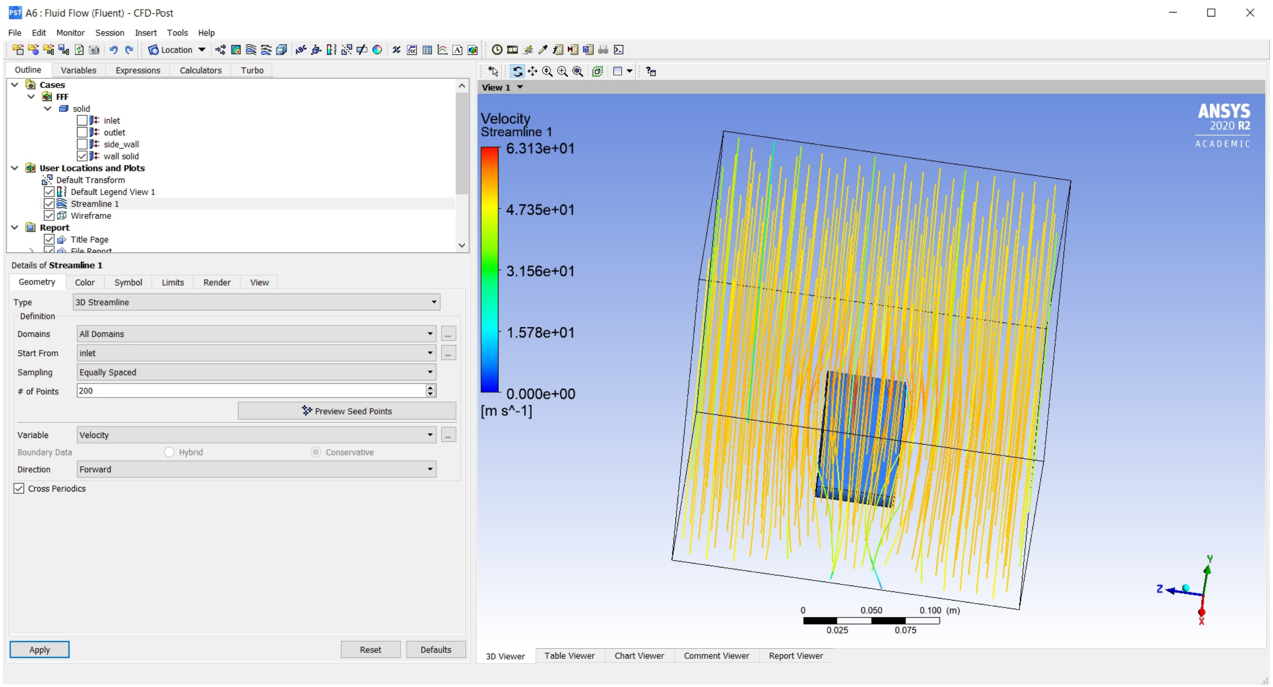

Fig. 6.10

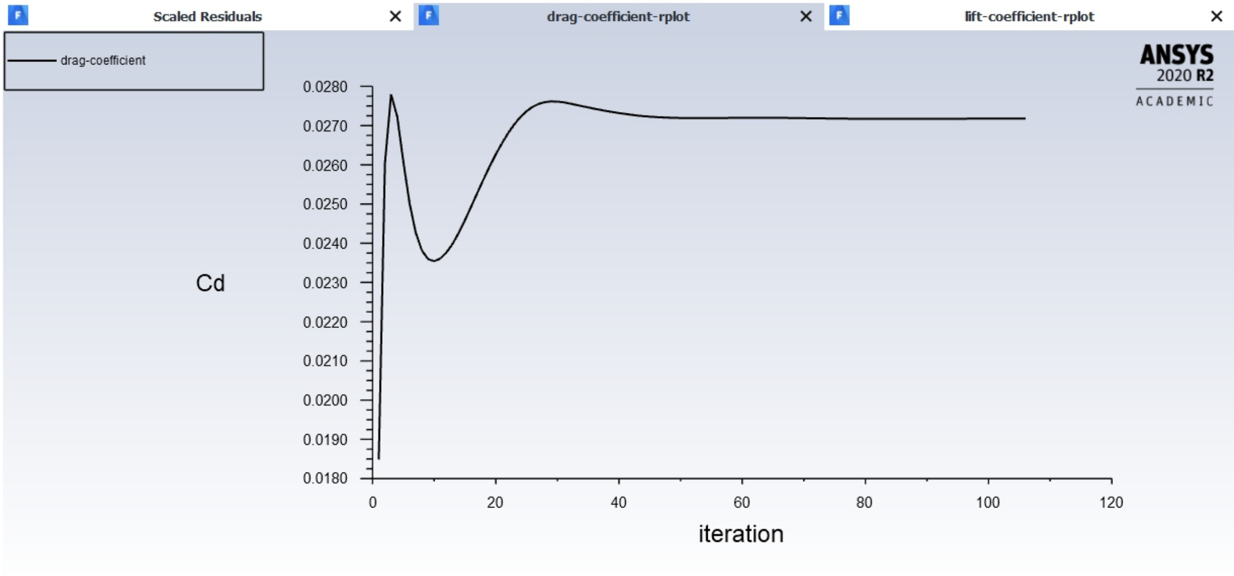

Fig. 6.11: Drag-coefficient minimised car model Drag-coefficient plot

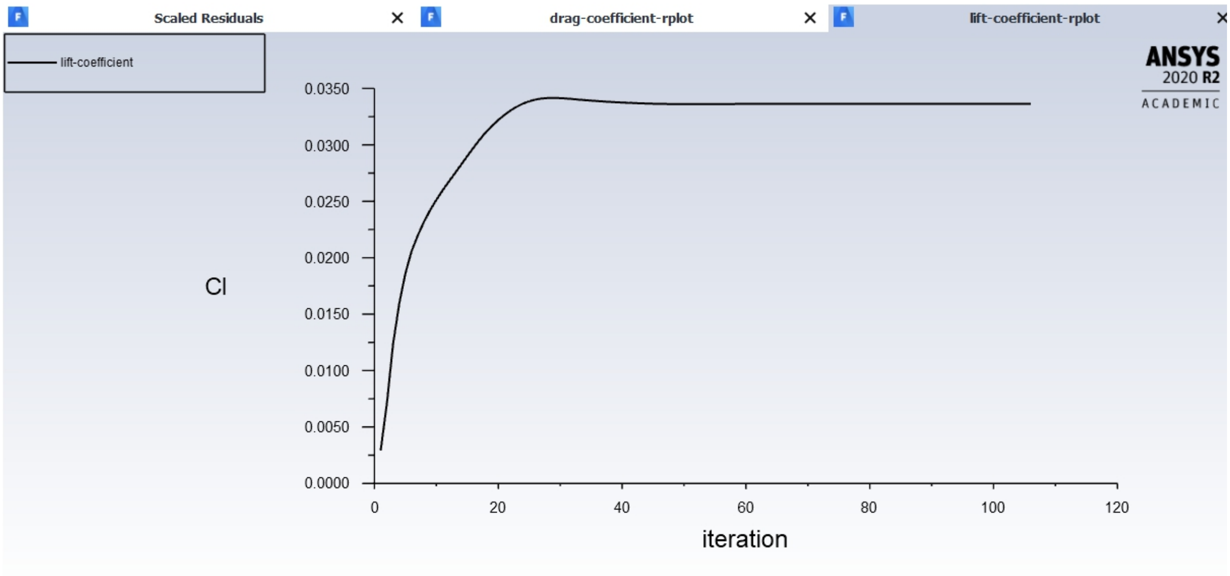

Fig. 6.12: Drag-coefficient minimised car model Lift-coefficient plot

7. RESULTS AND DISCUSSION

1. From the analysis of car model and seeing the pathline of air molecules, we can discuss the streamlining behaviour of the model. The pathlines follow the contour along the entire front and top part. The pathlines do not cover the entire back portion. On the top portion velocity of air is maximum. So, the pressure of air becomes less than the inlet pressure of air. The streamlining behaviour is not observed at around the sharp corners.

Result of drag-coefficient = 0.0581

Result of lift-coefficient = 0.00635

2. From the analysis of racing car model and seeing the pathline of air molecules, we can discuss the streamlining behaviour of the model. The pathlines follow the entire contour; however they bend near sharp corners, increasing the possibility of vortex formation, and turbulent flow. The speed of air is maximum at the top of car. So, the pressure of air becomes less than the inlet pressure of air. Air also passes through bottom of the car with great speed, increasing the tendency of lifting of the front portion of car.

Result of drag-coefficient = 0.1312

Result of lift-coefficient = 0.02005

3. From the analysis of Omni model and seeing the pathline of air molecules, we can discuss the streamlining behaviour of the model. The streamlines cover the top, front and side portions . The speed of air is maximum at the top of car. So, the pressure of air becomes less than the inlet pressure of air. Due to the design, the air, not being able to pass across the front face smoothly, tends to decelerate the car. The air does not reach the back portion, thereby creating a void region. Hence a pressure difference prevails between the front and back portion.

Result of drag-coefficient = 0.1508

Result of lift-coefficient = -0.00875

4. From the analysis of commercial car model and seeing the pathline of air molecules, we can discuss the streamlining behaviour of the model. The streamlines pass though the front, top, side portions. The air, in compare to other previous models, also reaches the back portion, but there are still void portions around the corners at the back. The speed of air is maximum at the top of car. So, the pressure of air becomes less than the inlet pressure of air. The air also passes through underneath the model, which suggests that increasing the speed too much, will increase the tendency of lifting up of the front portions of the vehicle thereby decreasing the stability.

Result of drag-coefficient = 0.0873

Result of lift-coefficient = 0.01354

5. From the analysis of drag-coefficient minimised car model and seeing the pathline of air molecules, we can discuss the streamlining behaviour of the model. The pathlines follow the contour along the entire front and top part, and does not pass through the bottom due less base clearance. The speed of air is maximum at the top of car. So, the pressure of air becomes less than the inlet pressure of air. Due to the design, the air passes across the front face smoothly, does not tend to decelerate the car. However, the streamlining behaviour is not observed around sharp corners , but due to smooth shape and less frontal area in compare to previous models , the drag coefficient is much less .

Result of drag-coefficient = 0.02718

Result of lift-coefficient = 0.0336

8. CONCLUSION

The aerodynamic drag of a vehicle will depend on overall shape of the vehicle and body details such as curved edges, surface roughness, gutter at the edge of the windshield or the wheel trim. However in our project we have analysed the overall shape of the vehicle. The drag co-efficient will depend on how the area is defined. We have designed all the car models keeping in view the designs which are practically available to us. From the result of simulations which were conducted on these various car models it was concluded that the value of drag co-efficient and all aerodynamics forces such as lift and drag and their co-efficients depended on the changes in shape. The pressure and velocity analysis suggests the ways in which air flows across the models and the pressure variations in the controlled volume (enclosure).

So the main conclusion is found that a good streamlined design will have much less drag co-efficient and the model will also experience very less aeroynamic forces.

9. FUTURE ASPECT

The three-dimensional flow around a vehicle is highly complex, and complete numerical solutions are yet to be achieved. Consequently, the wind tunnel testing of models and full-size vehicles remains a vital part of this project .

We can include the temperature variations in air which generally occur due to weather , also , we can inlcude back angle variations of a single model , and analyze the aerodynamic forces.

We can also design the models by including more details of the body , so that the flow trajectories and flow separation points can be analyzed in a better manner .

In future work , these modifications can be used , so as to compare properly with the existing car modelsin real life .

10. REFERENCES

1. Simulation and Analysis Drag and Lift Coefficient Between Sedan and Hatchback car by Mohd Khalil Azinganj Bin Salleh

2. Under International Research Journal of Engineering and Technology (IRJET) ,

A Review paper on Aerodynamic Drag Reduction and CFD Analysis of Vehicles by

Subhasis Sarkar, Kunj Thummar, Neel Shah, Vishal Vagrecha.

3. R.H. Heald, "Aerodynamic characteristics of automobile models," Part of Bureau of Standards Journal of Research, Vol 11, August 1933.

4. Shobhit Senger and S.D. Rahul Bhardwaj, "Aerodynamic design of F1 and normal cars and their effect on performance," International Review of Applied Engineering Research. ISSN 2248-9967 Vol. 4, Number 4 (2014), pp. 363-370.

5. Abdellah Ait Moussa, Justin Fischer, and Rohan Yadav, "Aerodynamic drag reduction for a generic truck using geometrically optimized rear cabin bumps," Hindawi Publishing Corporation Journal of Engineering Vol. 2015, Article ID 789475.

6. Taherkhani AR, deBoer GN, Gaskell PH2, Gilkeson CA, Hewson RW, Keech A, Thompson HM and Toropov VV, "Aerodynamic drag reduction of emergency response vehicles," Advanced Automobile Engineering Vol. 4 Issue 2 - 1000122 ISSN: 2167-7670 AAE, 2015.

7. Abdulkareem SH, Mahdi Al Obaidi and Lee Chung Sun, "Calculation and optimization of aerodynamic drag of an open wheel car," Journal of Engineering Science and Technology EURECA 2013 Special Issue August (2014-15).