Aerodynamic Optimization of Formula Student Car

[1]Pinkesh Parmar, [2]Maulesh Parikh, [3]Nelvin Johny, [3]Suruchi Sharma

[1]PG Scholar, [2]UG Scholar, [3]Assistant Professor
[1]Mechanical Engineering
[1]PIET, Parul University, Vadodara, India

_____

Abstract: From the recent years, research on aerodynamic for cars is increasing day by day to enhance the stability of a vehicle and reducing drag. This paper shows the design and CFD FLUENT analysis of formula student car. A study for the rear engine race car is presented here. The objective of the recent study is to optimizing the base model by investigating the aerodynamics characteristics of student race car with side road, fire wall vent and the front spoiler and without it. Student car is used to participated in SAE competition every year where students from all the territories takes part to build and compete with open wheel formula style race car. The Society of Automotive Engineers SAE INDIA car focuses on developing a simple and open chassis vehicle with light weight. The main focus on this paper is to optimize the race car with minimum drag and vehicle stability by using ANSYS CFD fluent software for performing the turbulent simulation. The analysis results are shown through graph with coefficient of drag, velocity and pressure counter.

Index Terms – **Reduction of drag, Optimization of race car, CFD Fluent analysis, Spoiler in car**
_____

I. INTRODUCTION

This paper describes the analysis of Formula Student Car. The design of a base model with Chassis is shown and study is conducted on it. In the past several decades the study of car aerodynamics is conducted to determine the various parameters like coefficient of drag and lift and downforce to stabilize the vehicle. As this car is working at the high speed the stability of vehicle is more concerned which is analyzed by the CFD Ansys Fluent analysis. The drag force is normally acts against the direction of moving car and lift is due to the movement of around an object and opposite to it is called downward force of the vehicle which is relatively important in formula student car.

Aerodynamics is the study of an object moves through air. It is divided into two sub categories like external and internal aerodynamics. External aerodynamics is normally the study of air flow around solid objects of different shapes and on the other hand Internal aerodynamics is the study of flow through passages inside the solid objects.

For the improvement in design of the car the SOLID WORKS software is used to modified the different shapes of the car. Then the CFD Fluent is used to find out the Aerodynamics results likewise coefficient of drag, coefficient of lift, downforce and velocity and pressure counter for the optimization of working model.
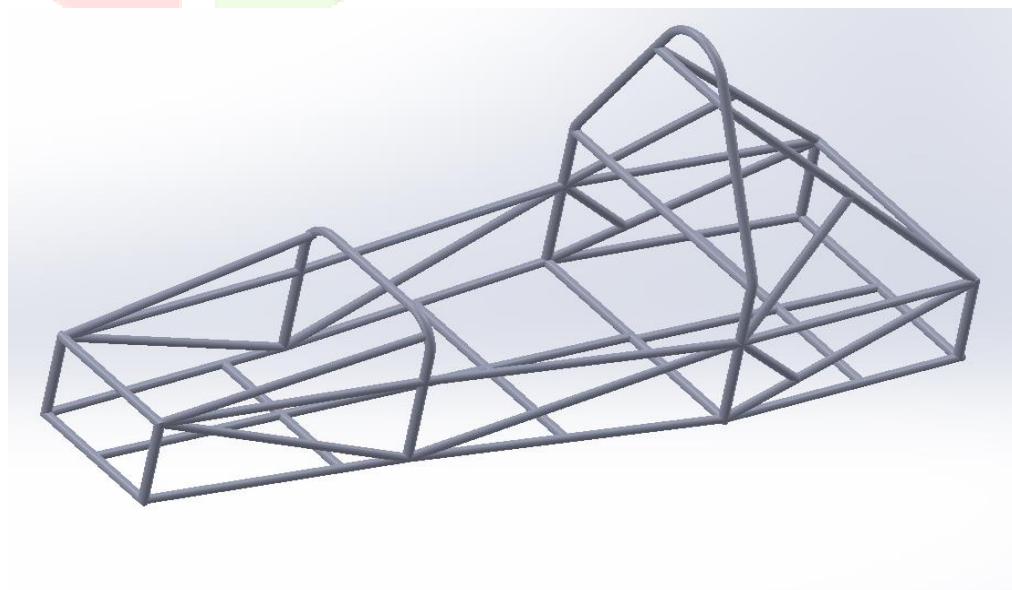
II. DESIGN OF FORMULA STUDENT CAR
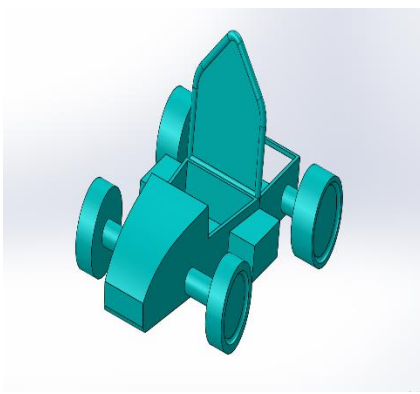

Fig 1: Chassis model of Formula SAE Car

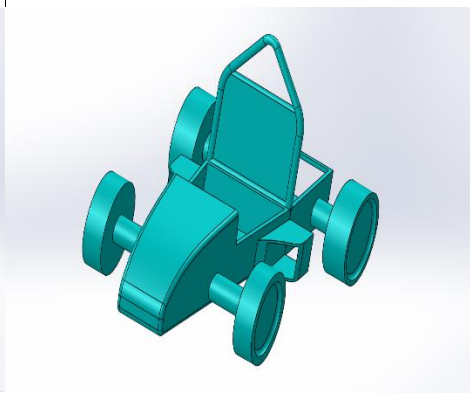*Fig 2: Base Model*   *Fig 3: Modified model 1*   *Fig 4: Modified model 2*
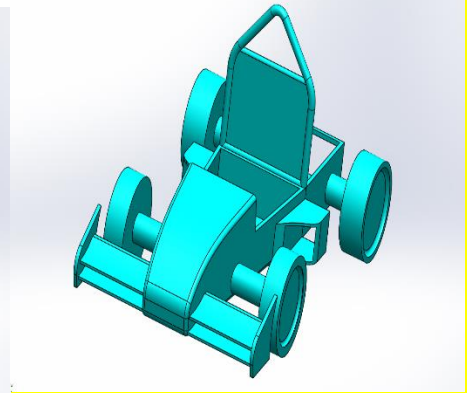
Technical Specification

Frame type   : Space Frame
Engine   : Single Cylinder; 373CC; Liquid cooled engine
Power   : 43 bhp @9800rpm
Torque   : 35 Nm @9800rpm
Transmission   : Constant mesh; Claw shifted
Brakes   : Hydraulic actuated disc brakes; Fixed calipers; X-split
Wheelbase   : 1615mm
Front Track   : 1270mm
Rear Track   : 1209mm

Frame configuration, design and material selection

It is based on the space frame, as it most suited for SAE vehicle because of its light weight properties. Chassis is constructed cylinder AISI 1018 steel tubes of various thickness as suggested in rulebook which is strong and rigid enough to withstand high impact forces and welded together with arc welding technology.

Impact Analysis

Mass of the vehicle   : 2500N
Acceleration   : 9 m/s^2
Force   : 22500N
Taking 2G Force   : 45000N

Calculation of equations

Drag Force   : ½ ρv^2 A Cd
Lift Force   : ½ ρv^2 A Cl
where;
       D : Drag force
       Cd   : Drag coefficient
       Cd   : Drag coefficient
       A   : Frontal area of the vehicle
       ρ : Air density
       V : Vehicle velocity

Navier - Strokes Equation for flow separation

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i$$

III. ANSYS CFD FLUENT CALCULATIONS

CFD also known as Computational Fluid Dynamics is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer.

NUMERICAL METHOD having basic three elements
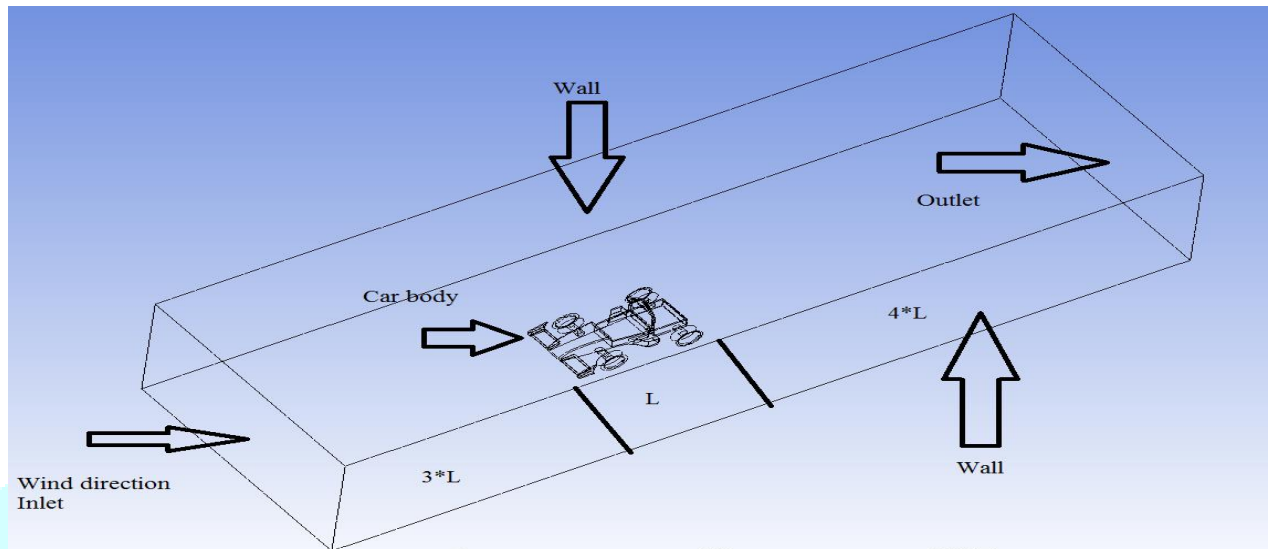   a. Pre Processor
   b. Solver
   c. Post Processor

Fig 5: Virtual Wind Tunnel and Vehicle Orientation

CFD Simulation Meshing

The meshing is done triangular one for obtaining the unstructured grid. For obtaining the boundary layer effect, Automatic inflation layer Program controlled is used having the growth rate of 20%.

SIMULATION Boundary conditions

Analysis has to be done like the testing of car model in the wind tunnel test. For doing this, the domain around the body is considered as the wind tunnel size. The domain contains velocity inlet, pressure outlet, walls around the car body as shown in figure 5. The inlet velocity is given 30 m/s

Table 1: Boundary Conditions Settings

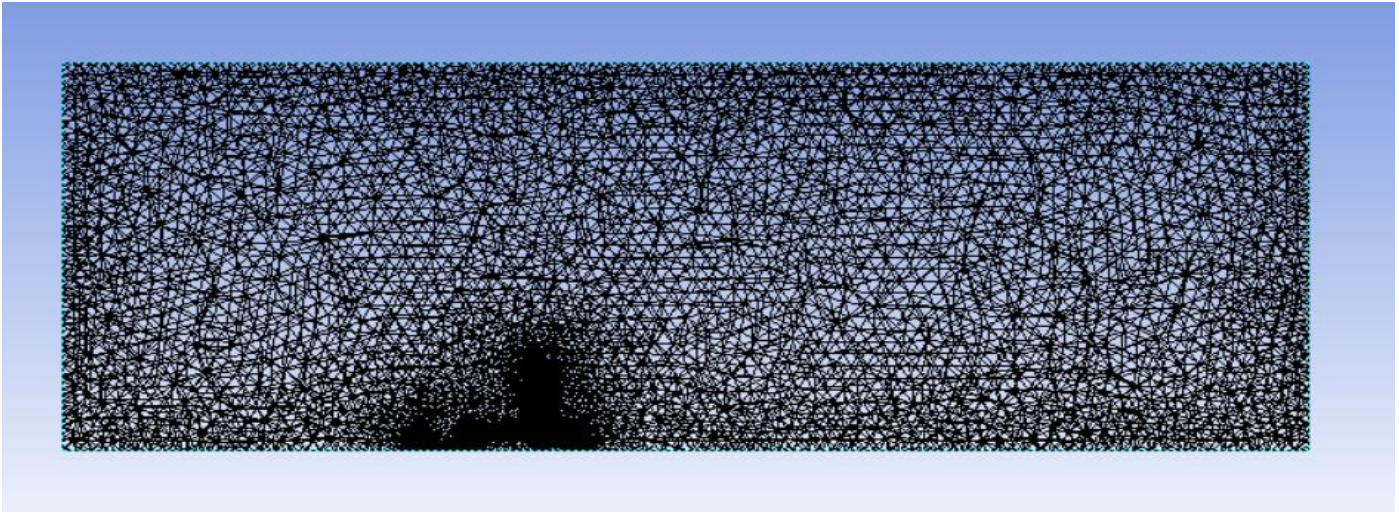| Inlet	Velocity Inlet	30 m/s
Outlet	Pressure Outlet	0 Pa
Top, Side, Bottom	Domain	Wall
Car body	In Domain	Solid Surface
Inlet Turbulence Intensity	-	1.00%
Backflow Turbulence Intensity	-	10%
Viscosity Ratio	-	10
Fluid Type	Fluid	Air
Density	ρ	1.175 kg/m^3
Kinematic Viscosity	V	$1.8247*10^{-5} \text{ kg/m s}$

Fig 6: Meshing of car body

IV. RESULTS AND CONCLUSION

CFD Fluent analysis of air flow over the race car is carried out at speed of 30 m/s for all three models. Results are taken by the three models and graphs are plotted.
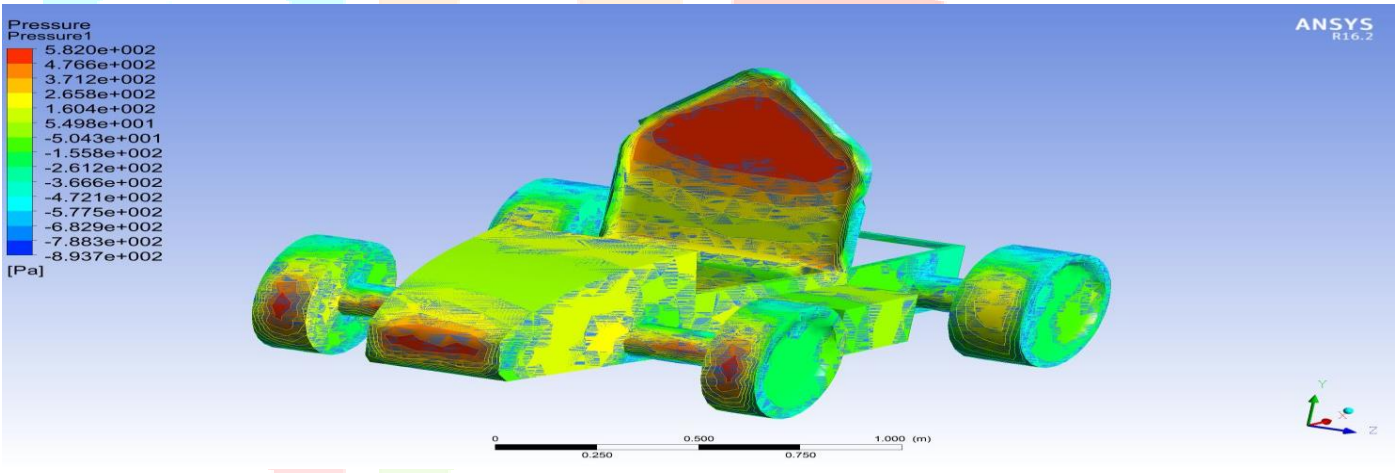
Pressure counters

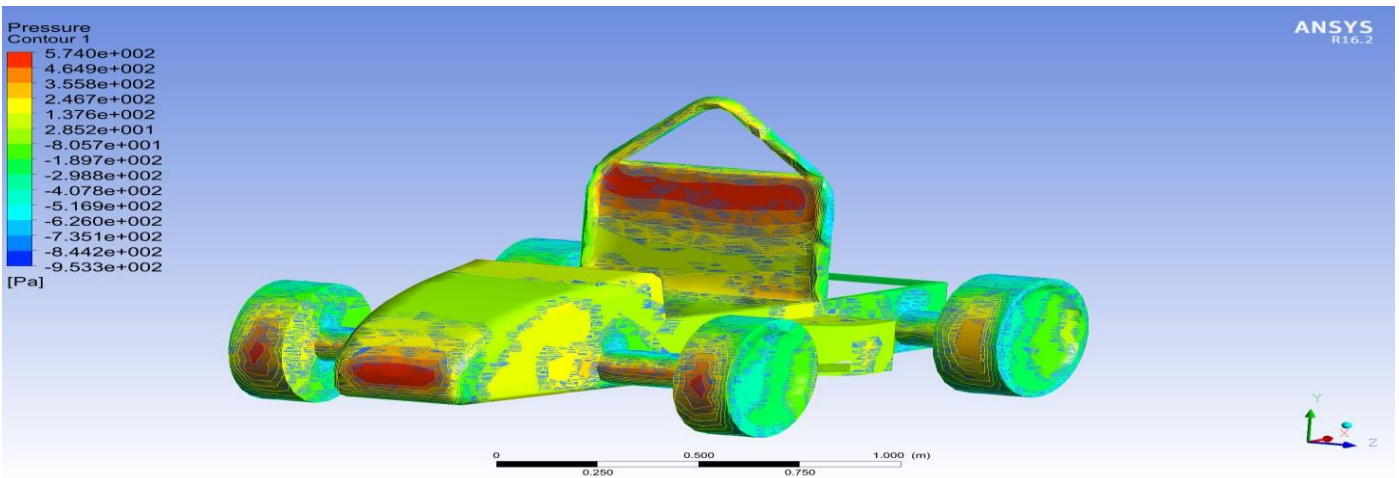*Fig 7: Pressure Counter for Base Model*
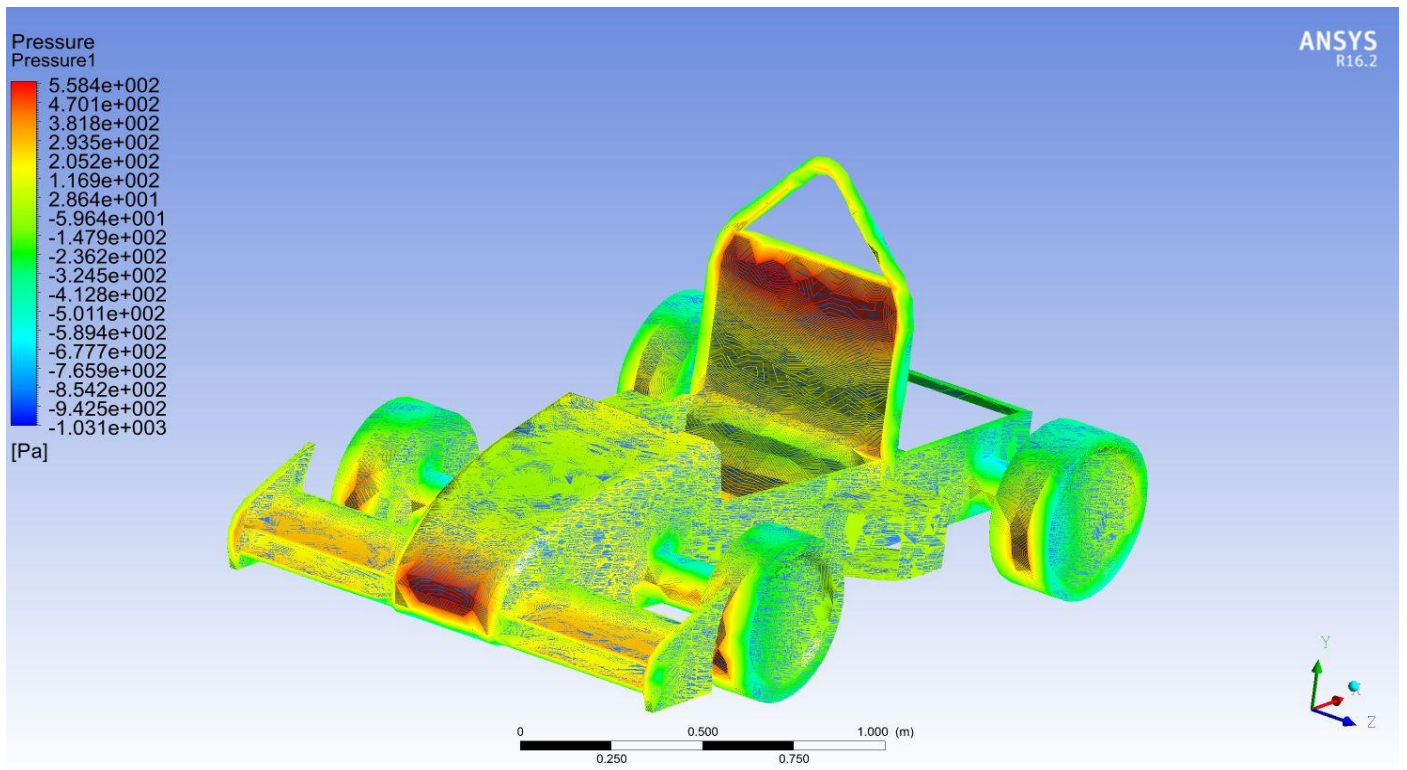

Fig 8: Pressure Counter for Modified model 1

Fig 9: Pressure Counter for Modified model 2
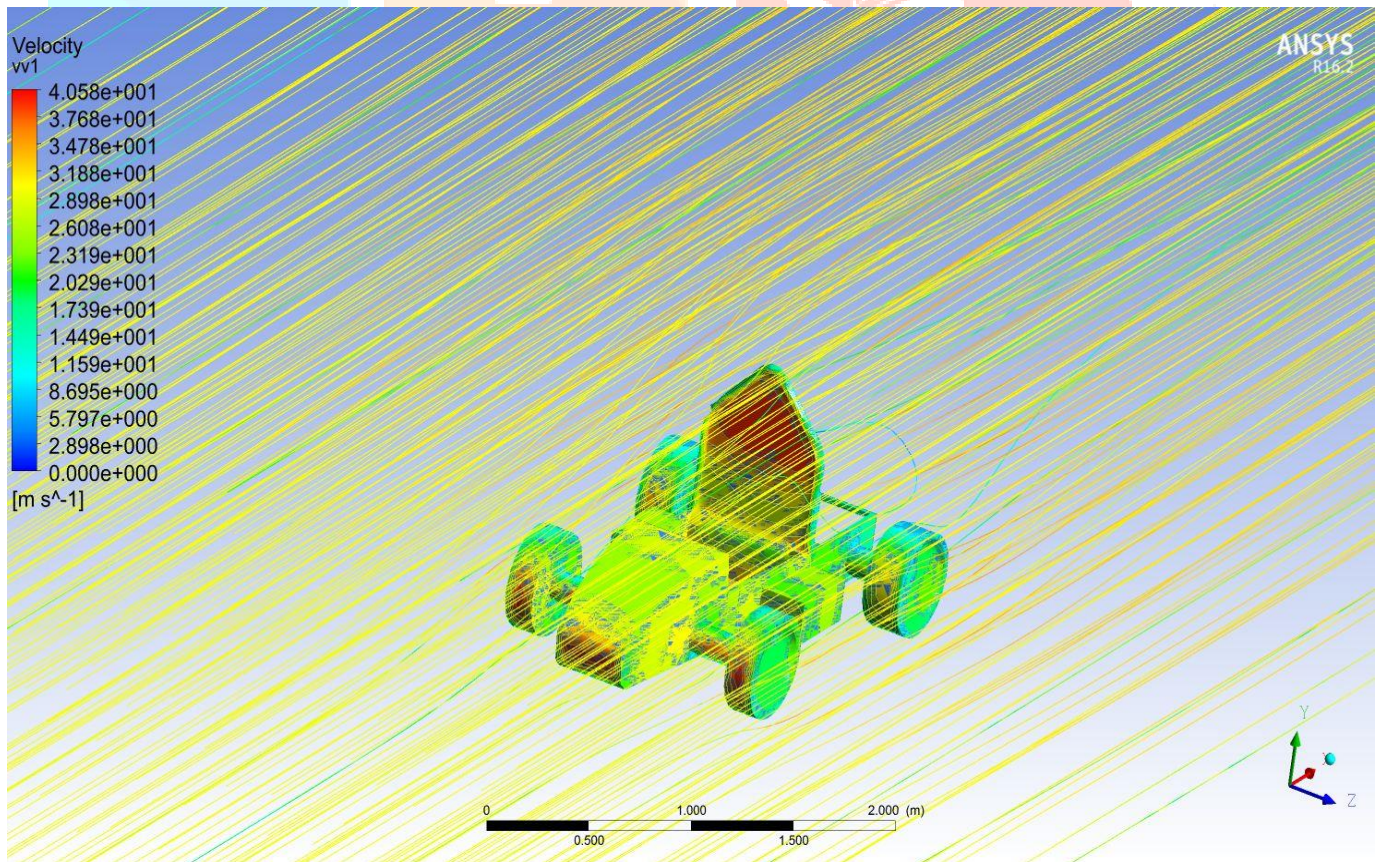
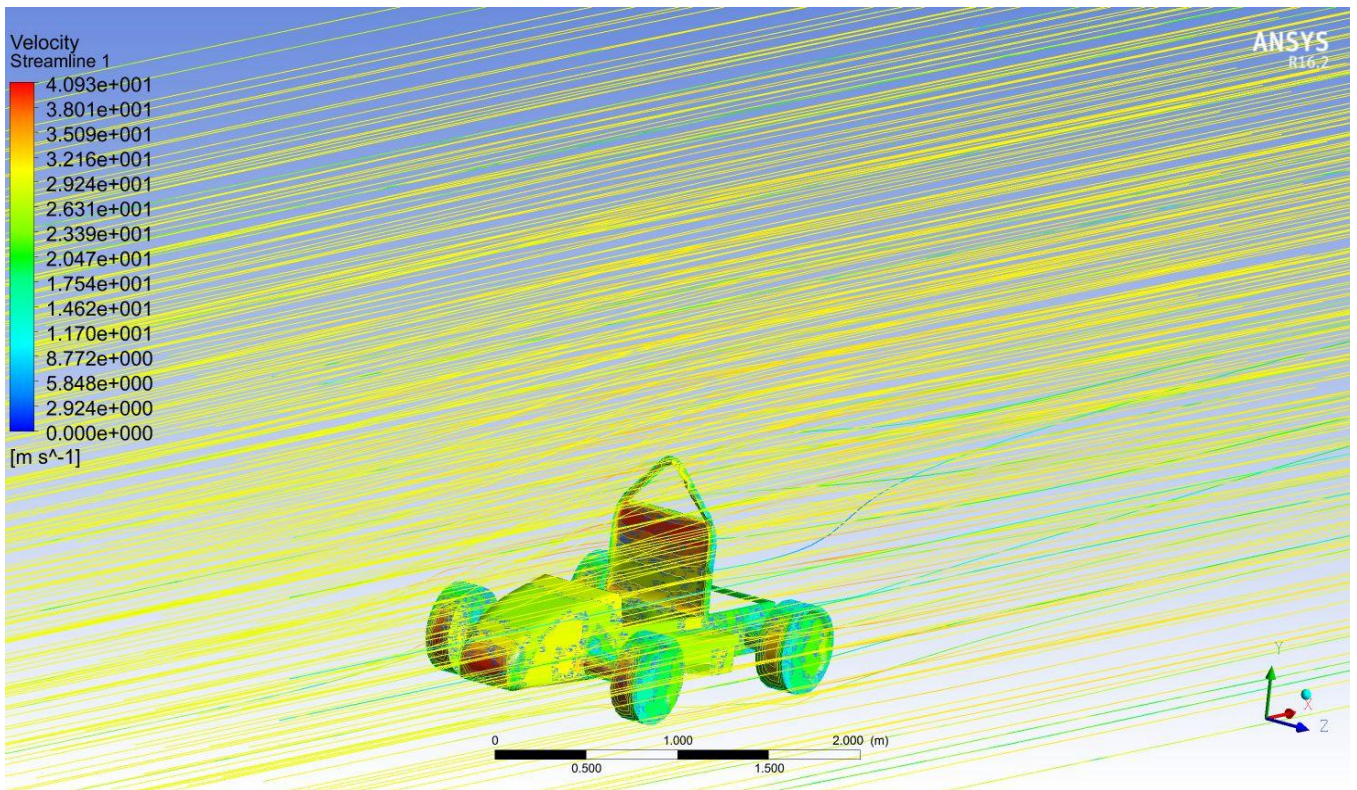**Velocity counters**

Fig 10: Velocity Counter for Base model
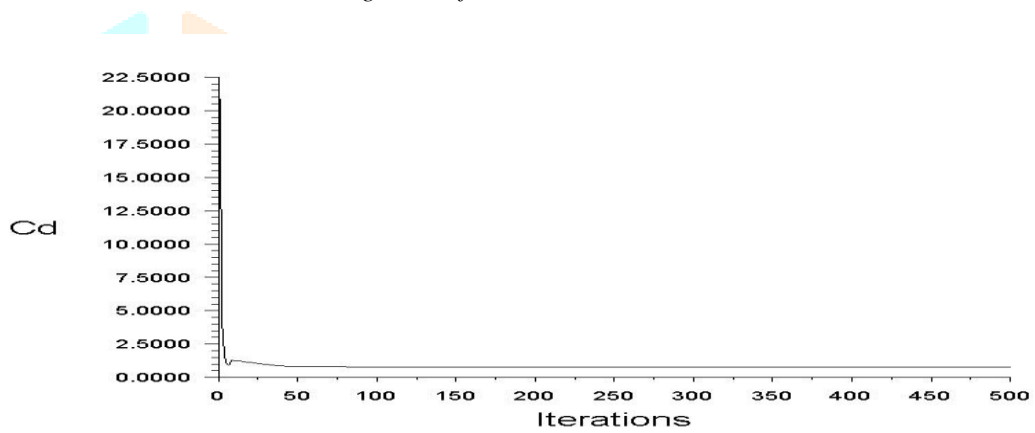
Fig 11: Velocity Counter for Modified model 1
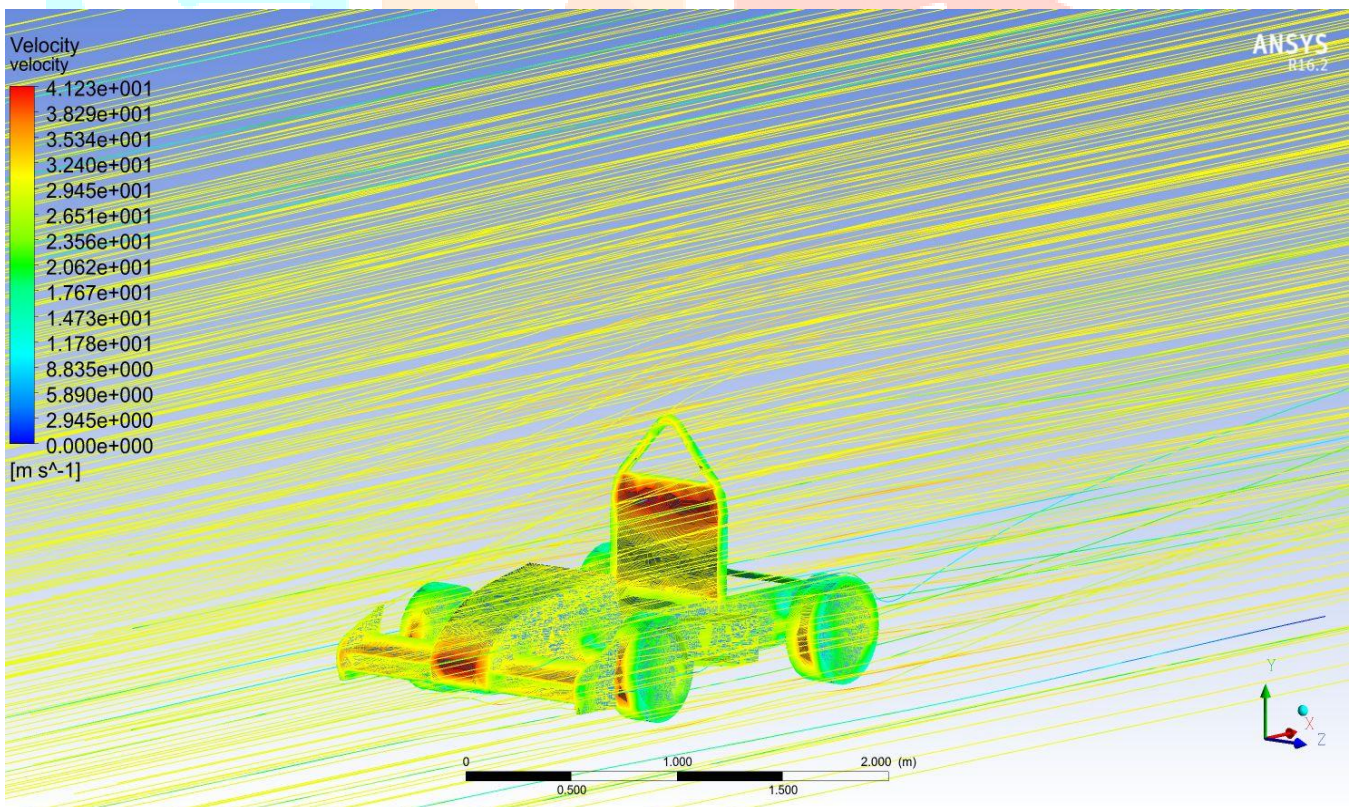
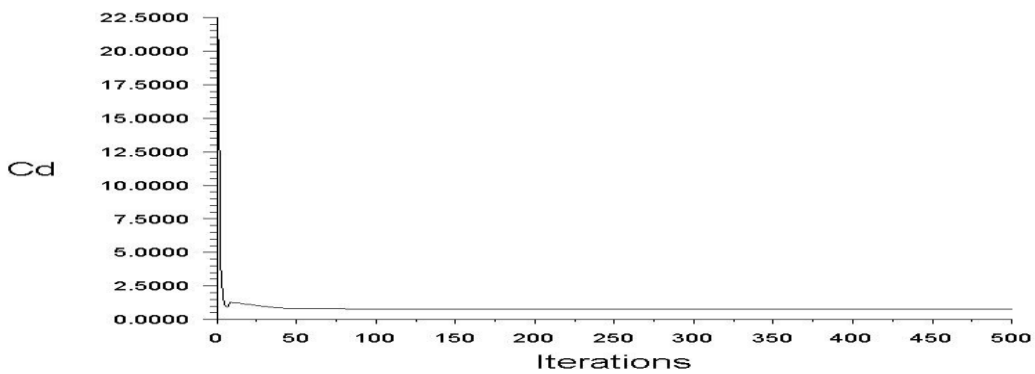*Fig 12: Velocity Counter for Modified model 2*
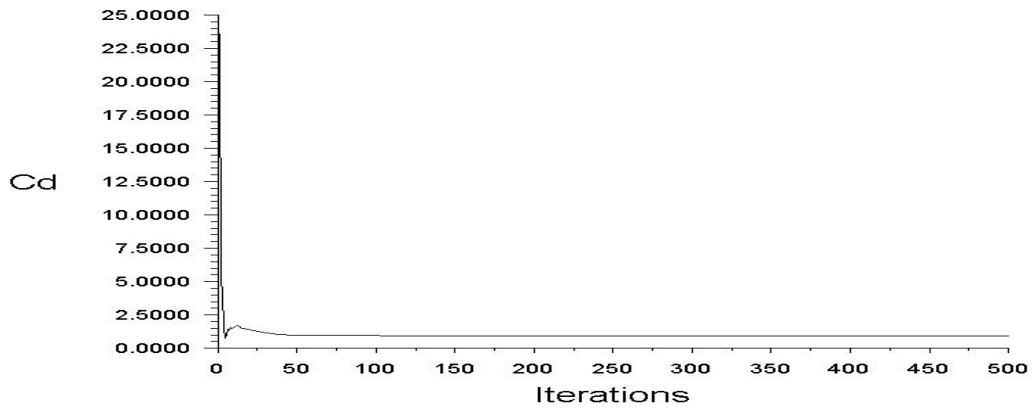
Coefficient of Drag

Fig 13: Cd for Base Model

Fig 14: Cd for Modified model 1

Fig 15: Cd for Modified model 2

Coefficient of Lift
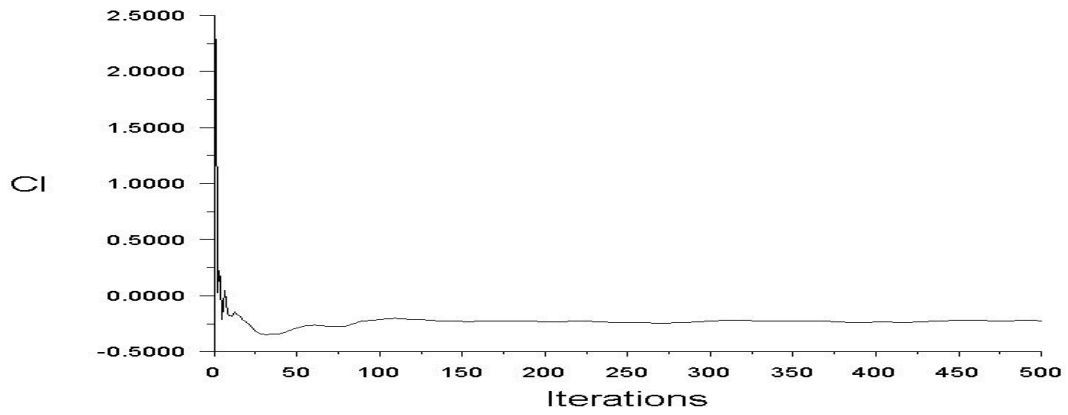
Fig 16: Cl for Base Model
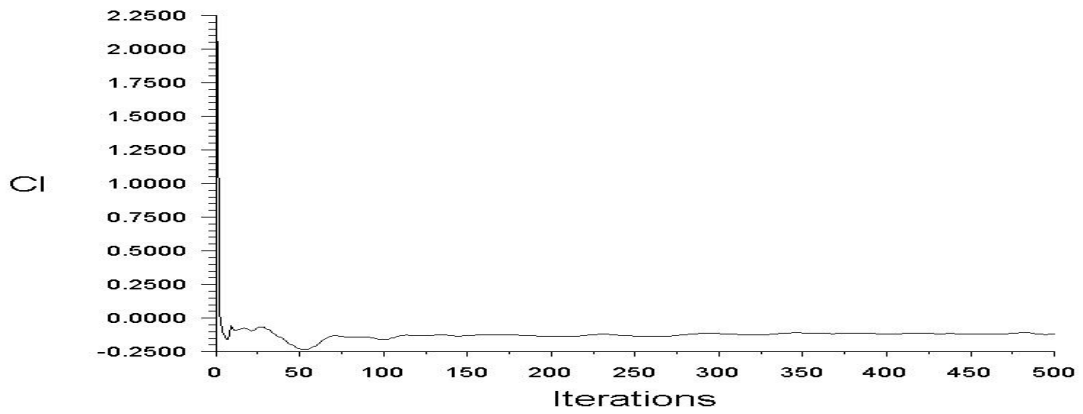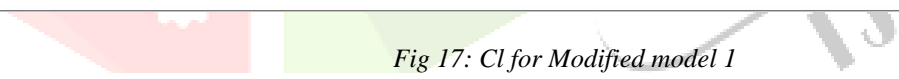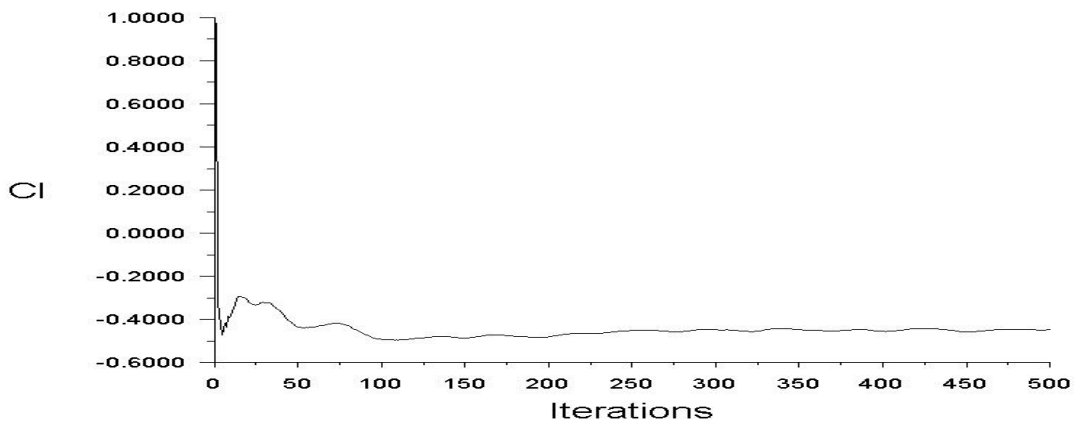
Fig 17: Cl for Modified model 1

Fig 18: Cl for Modified model 2

Coefficient of drag and coefficient of lift both are depends on shape of the vehicle body. In this paper, modification of the race car is done by fire wall cut and side rod design for reducing the drag and providing the front spoiler for increasing the downforce and reducing lift up to some values. In the above graphs it is showed that Cd for the modified model is lower, compared to the base model. Cd for the base model is 0.98 and that is for modified model is 0.71; which is quite better than the previous model. The fire wall cutting helps to provide the space for air to flow through it. And by providing spoiler the Coefficient of lift also changing from -0.10 to -0.42 which gives a good grip and stability to the race car.

REFERENCES

[1] Manan Desai, S.A Chaniwala, H.J Nagarserth "Experimental and Computational Aerodynamic Investigations of a Car" describes" in [2008].

[2] Mustafa Cakir "CFD Study on Aerodynamic Effects of a rear wing/spoiler on a passenger vehicle" in [2012]..

[3] Chainani. A, Perera " CFD Investigation of Airflow on a Model Radio Control Race Car''
in [2008].

[4] Bhagirathzala, Dr.Pravin P. Rathod, Prof. SorathiyaArvind S "Aerodynamic performance assessment of sedan and hatchback car by experimental method and simulation by computational fluid dynamics" in Journal of Engineering Research and Studies E-ISSN0976-7916, in [2012]

[5] . Sahil Gupta and Kishal Saxena "Aerodynamics Analysis of a Formula SAE car" in [2010]

[6] S. Wordley and J. Saunders, "Aerodynamics for Formula SAE: Initial design and performance prediction," SAE Paper Number 2006-01-0806, 2005

[7] Sneh Hetawala, Mandar Gophaneb, Ajay B.K.c, Yagnavalkya Mukkamalad, "Aerodynamic Study of Formula SAE Car" 12th Global congress on manufacturing and management, GCMM 2014

[8] R. K. Petkar, S. G. Kolgiri ,S. S. Ragit(2014): "Study of Front-Body of Formula-One Car for Aerodynamics using CFD" International Journal of Application or Innovation in Engineering & Management (IJAIEM),Volume 3, Issue 3

[9] SAE, Formula Student Germany Rules 2017, December 2016

[10] Wolf-Heinrich Hucho, "Aerodynamics of road Vehicles" Society of Automotive Engineers, Inc., 1997