Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 97 (2014) 1198 – 1207

Procedia Engineering

www.elsevier.com/locate/procedia

12th GLOBAL CONGRESS ON MANUFACTURING AND MANAGEMENT, GCMM 2014

Aerodynamic Study of Formula SAE Car

Sneh Hetawal[a,*], Mandar Gophane[b], Ajay B.K.[c], Yagnavalkya Mukkamala[d]

[a, b, c] *Second year M.Tech. (Automotive Engineering), School of Mechanical and Building Sciences, VIT University, Vellore, Tamilnadu, India.*
[d] *Professor, Thermal and Automotive Division, School of Mechanical and Building Sciences, VIT University, Vellore, Tamilnadu, India.*

Abstract

This paper describes the design and CFD analysis of a Formula SAE car. A numerical study of a rear engine SAE racecar is presented. The focus of the study is to investigate the aerodynamics characteristics of a SAE race car with front spoiler, without front spoiler and with firewall vents. Formula SAE is a college level student design competition where every year students of universities all over the world build and compete with open-wheel formula-style race [1]. Society of Automotive Engineers (SAE) INDIA racecar focuses on developing a simple, lightweight, easily operated open chassis vehicle. Compliance with SAE rules is compulsory and governs a significant portion of the objectives. The aerodynamics study of the SAE car is made to reduce the drag force. The study was performed using the CFD package. The main goal of this study is to enhance the stability of the vehicle and reduce the drag. With this the track performance will be increased also the resistance of air to the vehicle gets reduced. The CFD analysis is done on full scale model. The aerodynamic study is conducted in the ANSYS Fluent software to perform a turbulent stimulation (using k - ϵ model) of the air flow on the SAE car. The results are graphically shown with co – efficient of drag, velocity contour.

Selection and peer-review under responsibility of the Organizing Committee of GCMM 2014

Keywords: Drag reduction; CFD analysis; SAE car modification.

1. Introduction

Formula SAE will challenge the students to build a formula style race car. The Computational Fluid Dynamics (CFD) surely has played important role race cars in the last few decades. In order to get a good performance the vehicle has to aerodynamically efficient. The drag force is the unwanted thing which normally acts against the drive force of the car. The down force is useful to maintain the race car in ground.

* Corresponding author. Tel.: 9552751355
E-mail address: snehetawal@gmail.com

Selection and peer-review under responsibility of the Organizing Committee of GCMM 2014
doi:10.1016/j.proeng.2014.12.398

The capability of the aerodynamic engineer is to compromise between down force and drag. Initially the SAE car is analyzed and co – efficient of drag is found. In the second case the seats are slotted to reduce the drag. In the third case the spoiler is fitted to increase the stability of the vehicle.

1.1. Aerodynamic drag (C_d):

When the fluid flows over the surface, the surface will resists its motion. This is called drag. Aerodynamic drag is the sum of pressure drag and viscous drag. The pressure drag is the most dominant one of the both. The pressure drag is causes due the shear forces acting between the two layers of fluid.

$$\text{Aerodynamic drag } C_d = \text{Drag force}/0.5\rho v^2 A$$

Where ρ is the air density in kg/m^3, A is the effective frontal area in m^2, v is the velocity in m/s. The CFD is the effective tool which reduces times and is increasing importance. There are two different types of flow, laminar and turbulent. Laminar flow is smooth where the adjacent layers of fluid will undergo sliding between each other. The turbulent is the chaotic and random. It is normally unsteady, dissipative and 3 dimensional. Reynold's number decides whether the flow is laminar or turbulent. Over the surface there is a thin layer of air exists. The velocity of the air is reduces due the surface. This layer above the surface is called boundary layer. The length of the boundary layer increases as we move away from the surface. The boundary layer initially will be laminar and transforms into turbulent as it moves further over the larger distance. When there are sudden change in the surface this would lead to flow separation. Flow separation normally takes place when the upper layer of the fluid can no longer pull the lower layer of fluid within the boundary layer.

The flow can be described by Navier-Stokes equation:

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

Where u is the velocity, t is the time, x is the position, p is the pressure and ρ is density in kg/m3 [2]. The same thing happens in the case of side view mirror. The flow impacts on the back side of the mirror, the flow get divided. As the flow moves further the flow separation takes place and large amount wake will be formed at the front part of the mirror. With the reduced scale the characteristic length gets reduced, hence the Reynold's no. also reduces. Further there should be geometrical similarity for the reduced scale model. Computational Fluid Dynamics is initial step before the wind tunnel validation is done. There are different methods to solve the flow. The most common of those are –

- Direct Numerical Simulation (DNS) – It solves the Navier-stokes equation numerically. It requires very fine grid size and requires very high computer memory. It can solve all turbulence scales.
- Reynold's Averaged Navier-Stokes Equation (RANS) – It gives the approximate time averaged solution to the Navier-Stokes equation. The velocity field is fluctuating and it cannot solve all turbulence scales.
- Large – eddy Simulation (LES): It solves the large eddies. The simulation is time dependent.

2. Methodology

The aerodynamic study of the formula SAE car is done in ANSYS Fluent software. The main aim of this aerodynamic study is to reduce the drag and increase the cornering stability of the SAE car. The basic model of SAE car was analyzed in Fluent and it was found that the value for lift and drag co – efficient were on higher side.

Appropriate design modification in the basic model were incorporated and analyzed separately. This design modification and its subsequent effects are discussed in the later part of this paper.

2.1. Modeling Of Car

The main aim of the aerodynamic study is to reduce the drag and increase the stability of SAE car. The reduction in drag will help in increasing the top speed of the SAE car. This is obtained by making the body aerodynamic and air flow should have lesser obstruction in its way. The stability of the SAE car is also very important in the aerodynamic study. The stability is obtained by providing wings or spoilers. In this paper the front wing is installed onto the SAE car to increase the stability. In this paper the aerodynamic study three different solid models are taken.

2.2. Design Modifications

- The first solid model is the basic SAE car model outlining the overall shape with actual dimensions;
- The second solid model is the SAE car model with cut out in the firewall to reduce drag
- The third solid model is the SAE car model with cut out in the firewall and a front wing. This will reduce drag and increase stability by increasing the down force.

The first model is the basic model of SAE car. All three cars were modelled in SOLIDWORKS 2013.

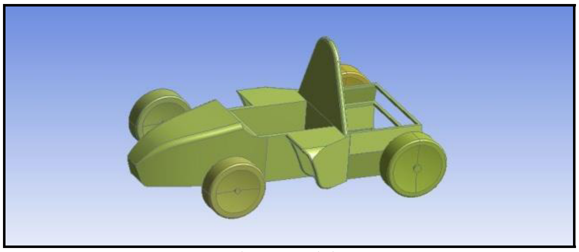

Fig.1. Model 1

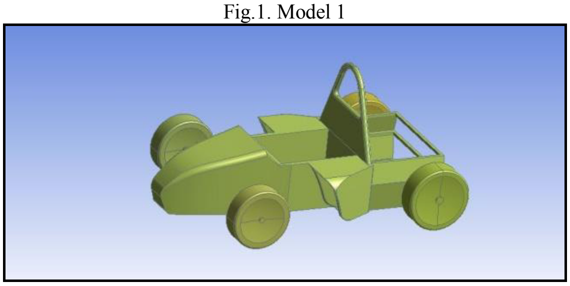

Fig.2. Model 2

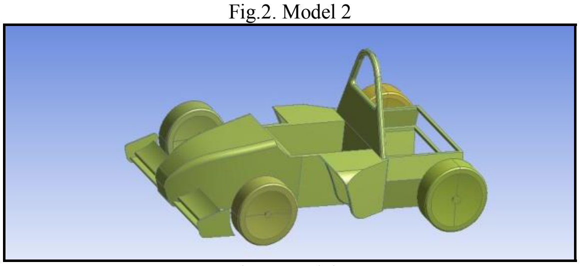

Fig.3. Model 3

2.3. Meshing For CFD Simulation

To reduce computational time, only half of the car model is used. This is possible because a vehicle has symmetry in vertical plane along its longitudinal axis.

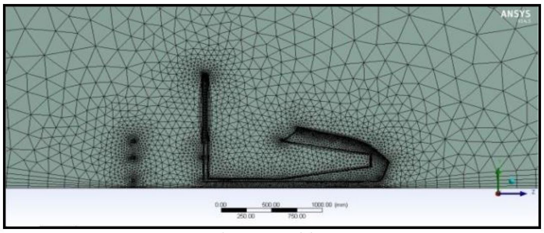

Fig.4. Meshing

Triangular meshing is done in order to obtain an unstructured grid. In order to get boundary layer effect, Program controlled inflation layer is used. Minimum element size was 120mm and number of grid elements is around 1119641.

2.4. Boundary Conditions For Simulations

Analysis has been done to simulate the car model in the wind tunnel. In order to achieve this, the domain around the body is considered as that of actual size of the wind tunnel [3]. Wall of the wind tunnel, the side, top faces of the domain are given boundary conditions of symmetry. The inlet velocity is given 25 m/s. Blue and red faces indicate velocity inlet and pressure outlet respectively. White represents wall whereas yellow represents symmetry conditions.

Boundary	Boundary Type	Values
Inlet	Velocity_inlet	25m/s
Outlet	Pressure_outlet	0 gauge pressure
Top	Symmetry	-
Side	Symmetry	-
Bottom	Road	-
Car body	-	-

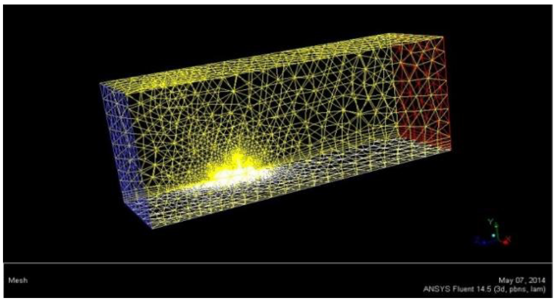

Fig.5. Boundary conditions

2.5. CFD Methodology

For analysis purpose, CFD solver ANSYS FLUENT is used for calculations. Transition SST 4 equation model is used. The courant number was set to 50 and the relaxation factors were taken as 0.25. For inlet condition the turbulence intensity was set to 1% and turbulent viscosity ratio as 10. For outlet, they were set to 5% and 10 respectively.

3. Results

CFD analysis of flow over the car is carried for speed of 25 m/s for all three models. Results are obtained the three models and graphs are plotted.

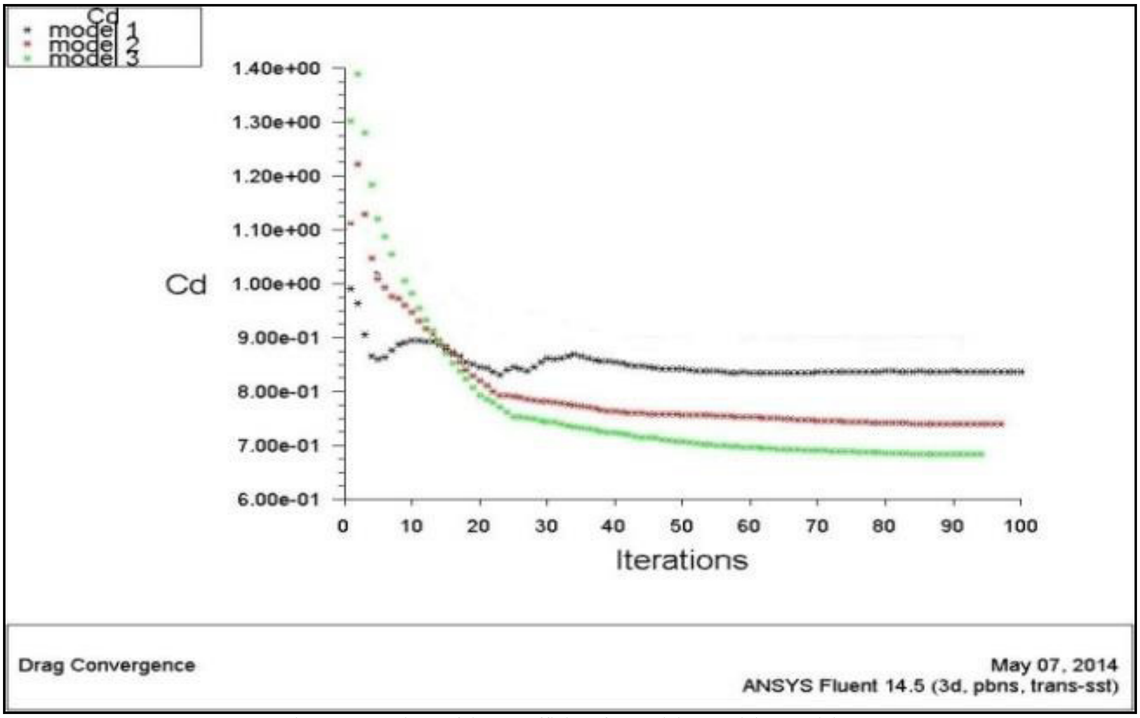

Fig.6. Comparison of drag coefficient for model 1, model 2, model 3.

Co-efficient of drag always depends on shape of the vehicle body. In this study, shape of the race car is modified by cutting out firewall and provided wing at front end. From the above graphs it can be observed that C_d for the modified car is lower, compared to the standard race car. C_d for the car with wing is found to 0.7 and race car with cutting out of fire wall have drag of 0.75, whereas standard race car have C_d of 0.85. Cutting out of fire wall helps to provide space for air flow. It also provides the attached flow for the streamline reducing the drag resistance.

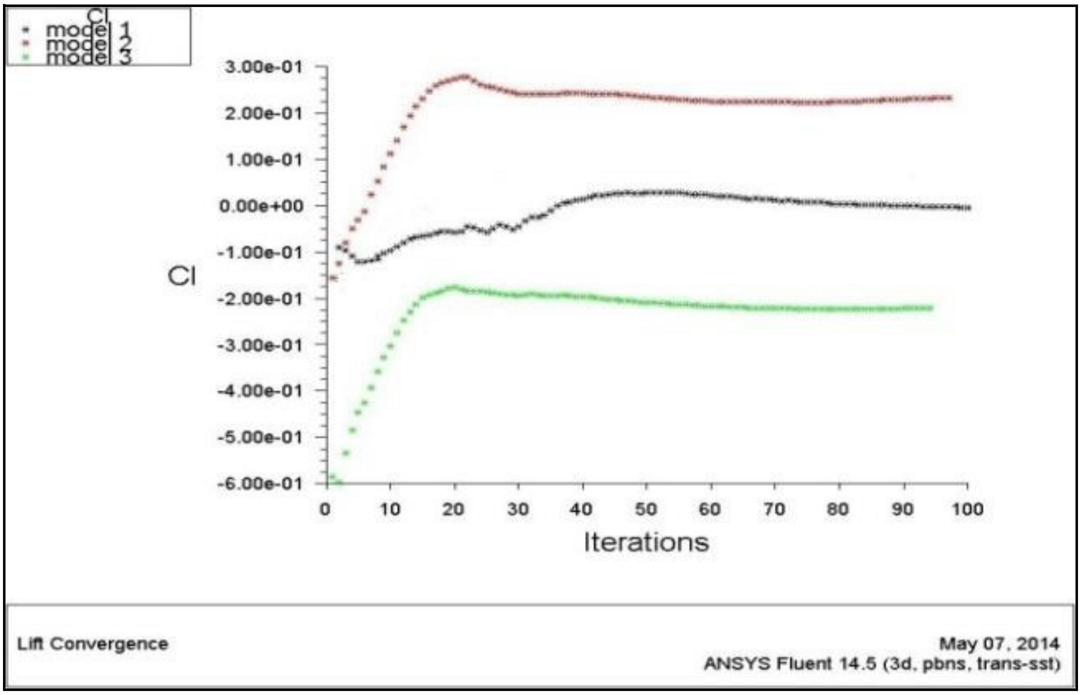

Fig.7. Comparison of lift coefficient for model 1, model 2, model 3.

Negative lift is the down force which pushes the vehicle closer to the ground. Underside of the race car is responsible for creating the down force. In order to reduce the lift, floor panel height of the car should be reduced. Front wing helps to provide the stability. From the above graph, it can be observed that co-efficient of lift is reduced from 0.2 for the standard race car to –0.25 for the modified race car with front wing. It is because the floor panel height of the model 3 is affecting on lift coefficient, which is comparatively lower than the other two models. Reducing the lift ultimately assists to achieve vehicle stability.

3.1. Static Pressure Contours:

Standard race car model has flat firewall; hence more air flow impinges on frontal area which leads to rise in pressure. For model 2, cutting out upper portion of firewall provide space for air flow. This raises the velocity of air and lowers the pressure at that point. For the all models, more pressure is developed at the stagnation region, on front tires [5].

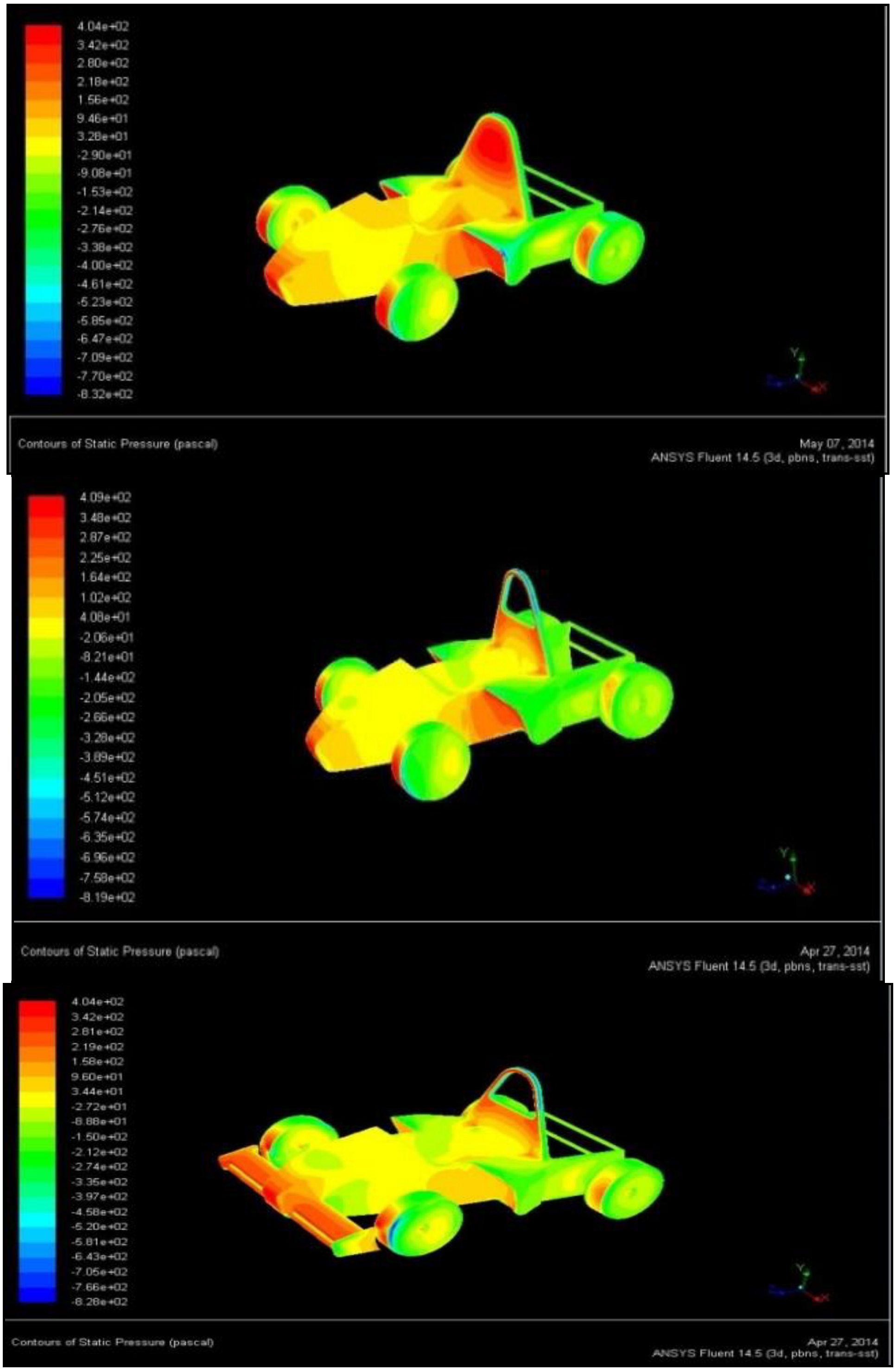

Fig.8. Comparison of static pressure contour for model 1, model 2, model 3.

As the aerodynamic shape provides more nozzle effect at frontal surface, pressure reduces tremendously towards the leading edge.

3.2. Total Pressure Contours Along Symmetry:

Rounded edges at the front surface accelerates the air flow, but that air flow is gets obstructed by firewall, inside the car or at driver space total pressur foud to be less for model 2 and found least for model 3. More wake region is observed for model 3 compred to other two.

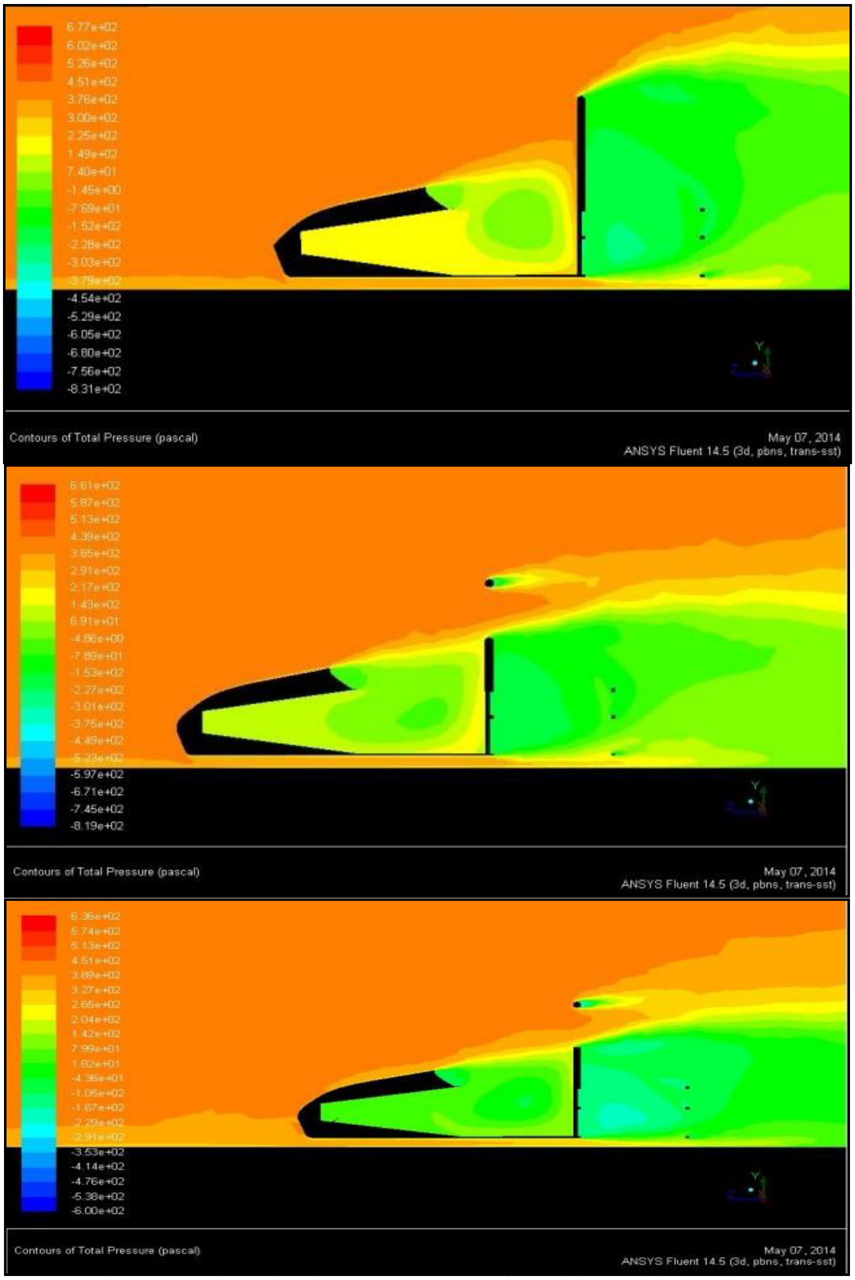

Fig.9. Comparison of total pressure at symmetry for model 1, model 2, model 3.

3.3. Velocity Contours:

More velocity of air is observed for model 2 near the cutting out section of firewall. Also more velocity is observed for model 3 below the stagnation point, air gets accelerate near that point.

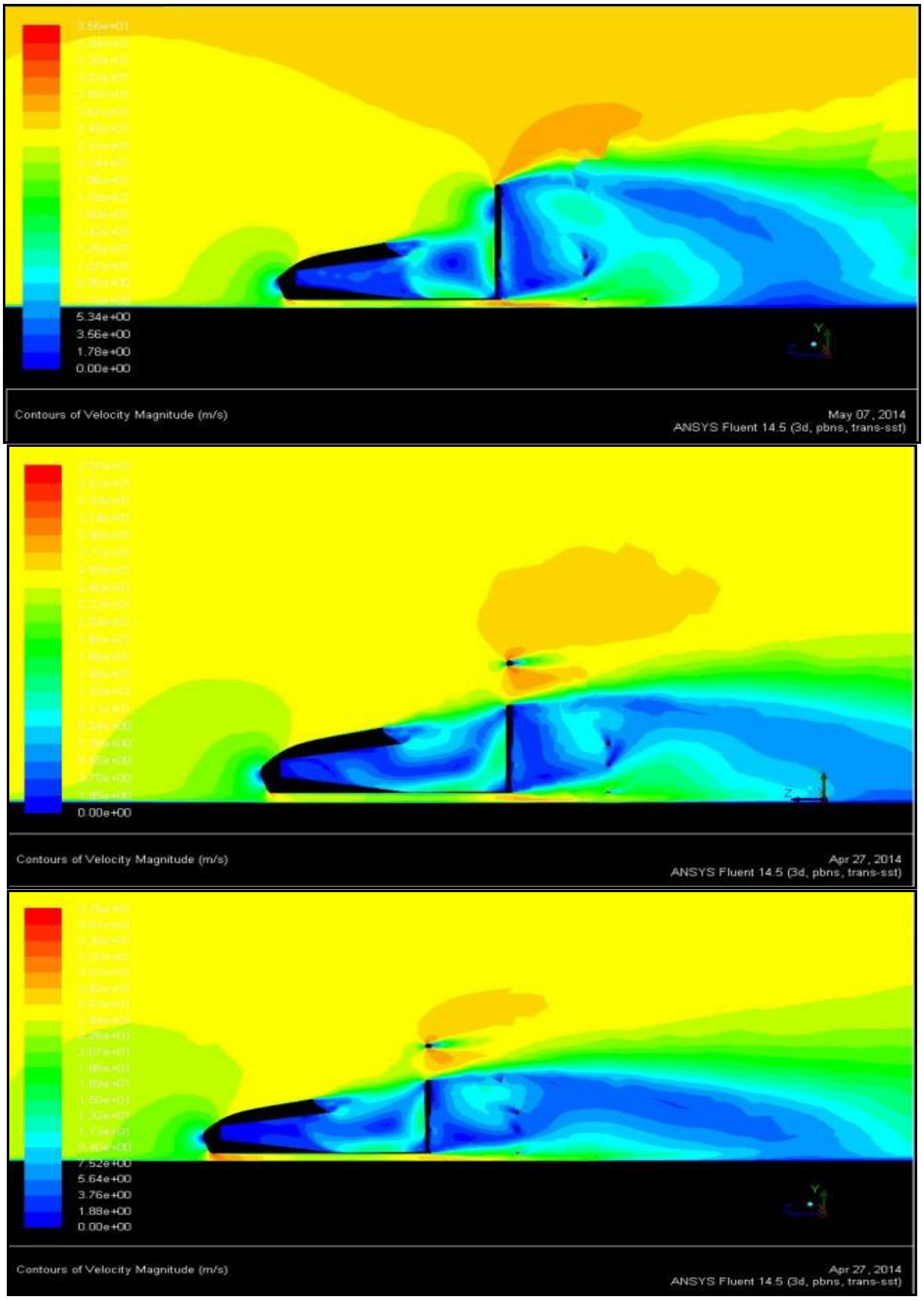

Fig.10. Comparison of velocity contour at symmetry for model 1, model 2, model 3.

4. Conclusion

To increase the aerodynamic performance of race car, an attempt is made to modify the design of a Formula SAE car. Comparative study is done on three car models by carrying out CFD simulations. Cutting out the section of firewall and providing wing at front end. Drag co-efficient is found to get reduced from 0.85 for the standard race car to 0.70 for the modified car with front wing, whereas negative lift is increased from 0.2 for standard race car to -0.25 for the model 3. The pressure at firewall found to be reduced for the modified cars due to providing space to flow the air through cut out section, where flow remains attached and helps to decrease the drag. Thus overall pressure near the driver head region is reduced from 340Pa to 80 Pa. for the modified car with front wing. Velocity of air is found to be increase below the stagnation point of car from 26m/sec to 32m/sec for model 3. Whereas at the rear end more wake region is found for standard race car. Model 3 having wing at the front end and having cut section at firewall shows less drag and lift, shows better aerodynamics characteristics than other two models.

Acknowledgements

We would sincerely like to thank to VIT University for providing us the facilities and support to carry out our experimentations. Also we thank to Mr. Yagnavalkya Mukkamala for providing us the opportunity to work under his guidance and for being the supporting author for the paper.

References

[1] David Rising, Jason Kane, Nick Vernon, Joseph Adkins, Craig Hoff and Janet Brelin – Fornari, "Analysis of a Frontal Impact of a Formula SAE Vehicle, 2006, SAE journal.

[2]Michael R. Wilson, Robert G. Dominy and Adam Straker, "The Aerodynamic Characteristics of a Race Car Wing Operating in a Wake", 2008-01-0658.

[3] R.H. Barnard, Road Vehicle Aerodynamics, Third edition.

[4] W.H. Hucho, "Aerodynamics of Road Vehicles"

[5] Inchul Kim, Hualei Chen and Roger C. Shulze, "A Rear Spoiler of a New Type that Reduces the Aerodynamic Forces on a Mini-Van", 2006-01-1631.

[6] Angel Huminic and Gabriela Huminic, "On the Aerodynamics of the Racing Cars", 2008-01-0099.

[7]Wael A. Mokhtar and Jonathan Lane, "Racecar Front Wing Aerodynamics", 2008-01-2988.

[8] Douglas Gogel and Hiroshi Sakurai, "The Effects of End Plates on Downforce in Yaw", 2006-01-3647.