

Investigations of Drag and Lift Forces over the Profiles of Car Using CFD

Edla Franklin¹ Pradeep Halder² Dr. P. Ravinder Reddy³

^{1,2,3}Department of Mechanical Engineering
^{1,2,3}CBIT, Hyderabad, Telangana state, India

Abstract— Aerodynamic characteristics of racing car are of significant interest in reducing racing accidents due to wind loading and save the fuel consumption. This work outlines the process taken to optimize the geometry of a vehicle. Vertices and edges of car were imported into GAMBIT and a computational domain is created. An unstructured triangular mesh was then applied. The goal is to obtain a better flow around the car model to lower the coefficient of drag force; the work is carried out in a ANSYS CFD FLUENT program towards a converged solution. These practices are helpful to redesign existing vehicles in order to improve handling and increase fuel efficiency. In the present work an attempt has been made by considering three models of car by varying speed of vehicle, the pressure coefficients and drag coefficients are obtained.

Key words: Aerodynamics, Drag force, Fuel efficiency, Lift force, Profile, Recirculation.

I. INTRODUCTION

The importance of aerodynamics to a Hybrid Electric Vehicle (HEV) is determination of drag estimation to know how much the car performance on the road against air resistance [1]. It can be used to improve the stability, reducing noise and fuel consumption. In view of the fact that many of car makers, formulate research and continue to develop the HEV models focused on higher propulsion efficiency in order to integrate the energy saving by reduce the rolling resistance of wheel and reduce the drag by aerodynamically losses [2].

The Proton Iswara Hatchback body was used by the group to modify the conventional power train to the hybrid power train. As drag is increased, the more power of the car is used to do work to push the air. This in turn reduces the power train efficiency [3]. Computational Fluid Dynamics (CFD) analysis will be used as the technology of computer simulation to estimate the drag because it is cheaper than the conventional technique. Therefore, the body of passenger car needs to be studied in term of aerodynamics losses. The car has been optimized by the manufacturer by having rounded body corners, raked windows and hatchback as found by [4]. However, the passenger car has to be large enough to accommodate people, the power train and support components, making it extremely difficult to realize an aerodynamically ideal body shape.

The present paper deals with three different profiles of passenger car which includes a comparison of profiles by the results obtained and optimum car profile is selected.

II. MODELING AND ANALYSIS OF CAR

As mentioned in the introduction, the first step of CFD is to design a computational model. This general process is done using a program called GAMBIT. In this program, geometry of the object is created by importing a previously created model from database. After the modeling is done, a grid is applied that represents the computational domain.

Throughout this mesh the CFD software solves the governing equations numerically. After the grid is completed and boundary conditions specified, the mesh can be exported. ANSYS Fluent (Version 6.1.22) is used to solve the exported problem.

The GAMBIT software package was used to generate an unstructured mesh around a body that could then be analyzed using Fluent. Before meshing could be applied to the geometry, it had to be cleaned up. This is due to the fact that when it is imported, geometry is considered to be “dirty.” To fix this problem, virtual geometry was laid over the top of the real geometry [5]. For this research a standard car was used (Fig.1) which was imported from the database.

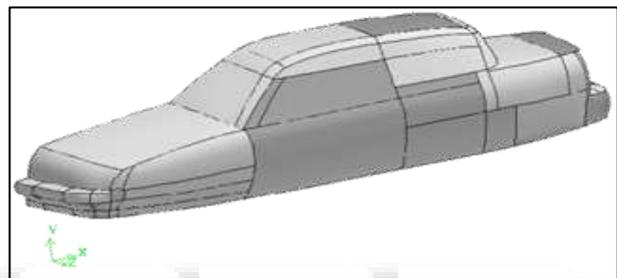


Fig.1: Imported geometry

GAMBIT was made to do some automatic clean, the model can still have some short edges and gaps between faces that can hinder meshing [5]. The Connect Edges Form was used to locate the shortest edge by selecting all the edges, using the Real and Virtual (tolerance) option and the default values for the tolerance and shortest edge entries. Before the edge could be deleted the default value for edge geometry had to be modified, the Connect Remove Short Edges value was changed to 1. This feature prevents the user from creating invalid geometry [5]. Then the removal was completed using the Virtual (forced) option. The next shortest edge was deleted using the same methods. Eliminating the short edges helps to ensure that the edge meshing intervals are of a reasonable size, so that distorted elements are not created [5]. To further clean the geometry, duplicate edges were connected the Real and Virtual (tolerance) option was chosen with the shortest edge tolerance set to 10% and T-junctions enabled. When the T-junctions utility is used, unconnected edges are split and reconnected [5]. An example of this can be found in Fig. 2. This process automatically connects all edges less than the 10% tolerance.

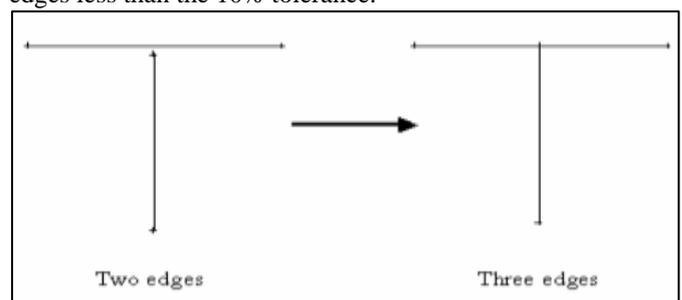
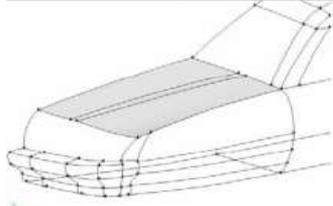
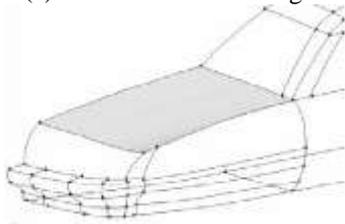


Fig. 2: T-junction

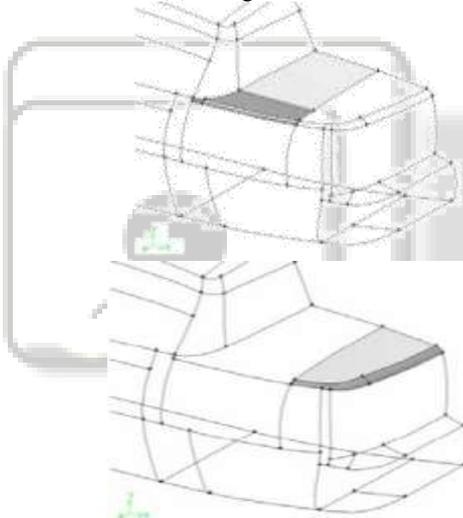
To further ensure appropriate meshing, some faces on the car were merged resulting in a less complicated geometry. The three long faces on the hood of the car, the four faces on the trunk closest to the window, and the three near the back to be merged were selected. This was done with the Merge Faces (virtual) Form and the Virtual (forced) option, the faces that were merged can be observed in Fig.3



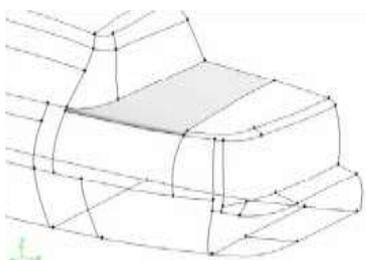
(a) Hood faces to be merged



(b) Merged faces on hood



(a) Trunk faces to be merged



(b) Merged faces on trunk

Fig. 3: Merging faces

The car faces were ready for meshing at this point. All the faces were meshed using a Tri element and an interval size of 0.03. This creates an unstructured grid of triangular mesh elements (Fig. 4). A real brick (shaped volume) was created around the car with dimensions of 10m x 5m x 5m (width*depth*height or x*y*z) with the coordinate system centered within the brick, then it was moved 2.5 meters in both the y and z directions in order to obtain proper positioning. This positioning is displayed in Fig.5. Then the volume just created was deleted, leaving behind the lower geometry (i.e. faces, edges, and vertices).



Fig. 4: Meshed faces of the car



Fig. 5: Car within domain

Next, some lines were created on the symmetry plane face near the front and back bumpers of the car in order to prepare for later face creation and to help to concentrate the mesh beneath the car. The line across the bottom of the symmetry plane was split twice, both times with a point as a real connected edge, in front of the car with a U-Value of 0.64 and behind the bumper with a U-Value of 0.57. "If you split a real or virtual edge using a point as the split tool, you must specify a U Value parameter that identifies the location of the point on the edge. The parameter value U represents a fraction of total edge length" [6]. Virtual straight edges were used, without a host, to connect the points to the vehicle bumpers. Faces were created above and below the car with the symmetry plane as the host and a tolerance of 0.001. The new plane is shown in Fig.6.



Fig. 6: Faces created around car, to serve as the symmetry plane

Face	Boundary Type	Zone Name
In front of the car	Velocity Inlet	Inlet
Side opposite the car	Symmetry	Side
Above the car	Symmetry	Top
Below the car	Wall	Ground
Two faces created at the Car center plane	Symmetry	Symmetry Plane
Behind the car	Pressure Outlet	Outlet
Faces of the car	Wall	Car

Table1: Boundary Types

A. First Alteration of The Geometry

In order to have means of comparison, the initial geometry of the car was altered in a way that would hopefully render a better vehicle. The first change was to essentially turn the car into a hatchback. A line was created along the symmetry plane that extended from the roof of the car to the back of the trunk. This edge was then used to create one large face from a wire frame, over the rear window and trunk. The old faces were then deleted and the new one was used to create the volume (Fig.7). Then all of the same steps were repeated as before for the initial geometry



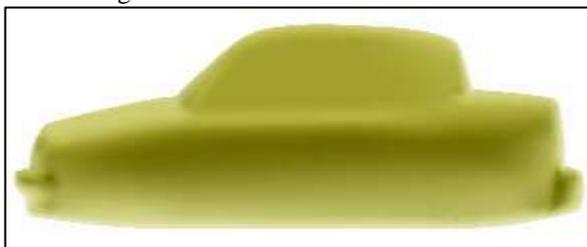
Fig. 7: First alteration to the model

B. Second Alteration of the Geometry

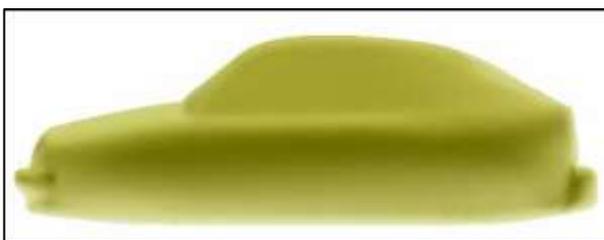
For the same reasons discussed in the previous section, the second version of the car was altered to create yet another model. This time the hatchback was left as a hatchback from the first alteration. This time the area of interest was the front of the car. The intent of this alteration was to decrease the angle between the hood and the windshield for a smoother flow. This was accomplished by copying the edge between them. The copy was placed away from the original by a magnitude of 0.04m in the negative x direction and the positive y direction; essentially up and to the left 4cm. Then an edge was created on the symmetry plane joining the front of the hood, the vertex of the new edge, and the roof of the car. The three forms of the car discussed in these past sections can be viewed and contrasted in Fig.8.



Fig. 8: Second alteration to the model



(a) The original geometry



(b) After the first alteration



(c). after the second alteration

Fig. 9: The three versions of the car

C. Fluent: Analysis of the Models

With the mesh completed and exported, Fluent was used to attain a solution. For the purposes of this research, the flow was assumed to be steady state, laminar, and incompressible, external flow. Cases were set up for each of the three models previously created for Reynolds's numbers of 100 and 1,000, yielding six cases. An attempt was made to use a Reynolds's number of 5e6 using turbulence, k-epsilon model to observe the practical case for first and second alterations.

D. Initial Geometry, with $Re=100$

First the mesh was read in and checked. Once the grid check was passed the size was reported to get an idea of the memory required for calculation. All the settings are set to default. The numerical method chosen for this problem was the segregated solver; which solves the governing equations sequentially. They are segregated from one another, hence the name.

In order to keep things simpler the model was assumed to be laminar flow instead of turbulent for $Re=100$ and 1000. The operating pressure was set to be 1 atmosphere. The material used was air. Due to the fact that the flow is incompressible, the density is considered to be constant. The default values for density ($\rho=1.225\text{kg/m}^3$) and viscosity ($\mu=1.7894 \times 10^{-5}\text{kg/ms}$) were used.

To complete the definition of this condition, the free stream velocity was calculated which was done by rearranging the equation for the Reynolds's number: $Re = \rho * v * l / \mu$

Where Re is Reynolds's number, ρ is density, v is velocity, l is length, and μ is viscosity. The length refers to the span of the car from front bumper to back bumper, a measurement of 2.55m. This dimension, along with others can be found in Fig.10. Through this equation the velocity was found and set to be $5.72 \times 10^{-4}\text{m/s}$, with magnitude and direction normal to the boundary. The velocity inlet BC must be accompanied by at least one outlet of some kind [7]. This was set as a pressure outlet with the default settings. This was the chosen outlet condition because it is suitable for either compressible or incompressible flows. It is where the gauge pressure is set, which is physically the static pressure present in the environment into which the flow is exhausted [8].

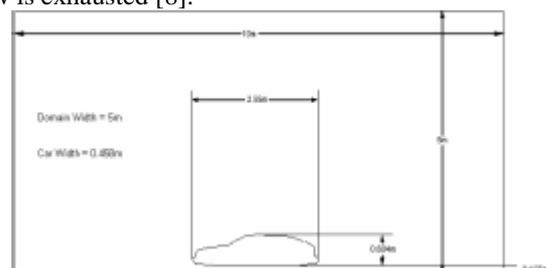


Fig. 10: Dimensions of the car and outer domain

Now that the problem was defined, Fluent had to be set to solve the fluid flow problem. The first step in doing this was to set the solution controls, all of which were left at the default values (pressure=0.3, density=1, body forces=1, momentum=0.7) except for the momentum option for the discretization. This was changed to “2nd order upwind” for increased accuracy.

1) Lift Forces are calculated using:

- $F_L = \frac{1}{2} \rho V^2 A C_L \dots \dots (1)$
- F_L = Aerodynamic Lift Force, kN
- ρ = air density, Kg/m³
- V = velocity, m/sec
- A = Frontal Area, m²
- C_L = Lift Coefficient

2) Drag Force is calculated as:

- $F_D = \frac{1}{2} \rho V^2 A C_D \dots \dots (2)$
- F_D = Aerodynamic Drag Force, kN
- ρ = air density, Kg/m³
- V = velocity, m/sec
- A = Frontal Area, m²
- C_D = Drag Coefficient

III. RESULTS AND DISCUSSIONS

CASE 1- In this case geometry of the car is not altered

A. Velocity of Flow used for simulation is 11.1m/s

The Fig.12 shows the pressure distribution along the symmetry plane of the car. It is observed that the maximum pressure is 75.5 Pa in front of the vehicle at bottom of the windshield and in front of hood because of sharp edges. In Fig.13 shows the velocity (14.2 m/s) distribution on the surface of car it is maximum on the top surface. Velocity Vectors are plotted in the Fig.14 it predicts the flow around the car and the vector plots gives a better indication of air flow direction, the contours gave a better presentation of the overall magnitude at various stages throughout the flow. Heavy recirculation is observed behind the car and also in front of the car it is due to the influence of the Car geometry. Path lines give the best visualization of the flow around the car. Hence it is plotted in Fig .15.

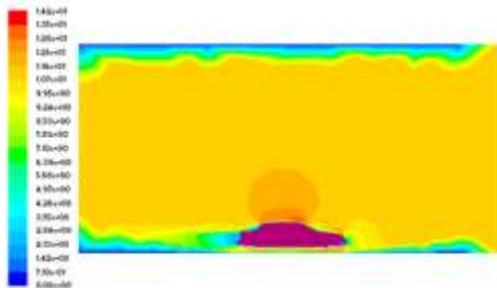


Fig. 12: Pressure contours at symmetry plane

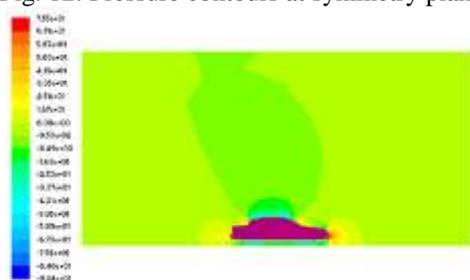


Fig. 13: Velocity contours at symmetry plane

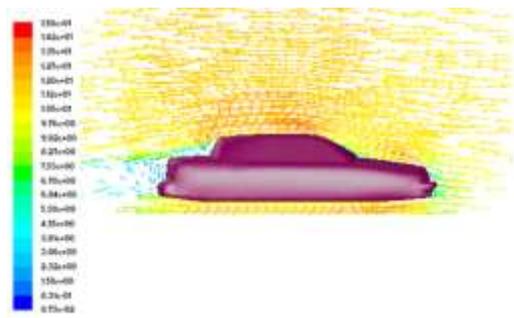


Fig. 14: Velocity vectors over a car profile

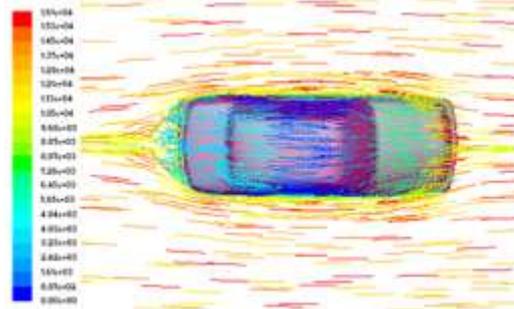


Fig. 15: Path lines

B. Velocity of Flow used for Simulation is 16.6 m/s

The Fig.16 shows the pressure distribution along the symmetry plane of the car. It can be seen that the maximum pressure is 168 Pa is at in front of the car. In Fig.17 shows the velocity (21.3m/s) distribution is maximum at top surface of the car, behind the car the velocity has fallen down to negative velocity it means the direction of velocity vectors have reversed. This can be observed in velocity vectors plots which are shown in the Fig.18 on the symmetry plane. Recirculation has increased than the previous case of velocity (11.1 m/s) since in this case the velocity is increased from 11.1 m/s to 16.6 m/s. Path lines give the best visualization of the flow around the body. Hence it is plotted in Fig .19.

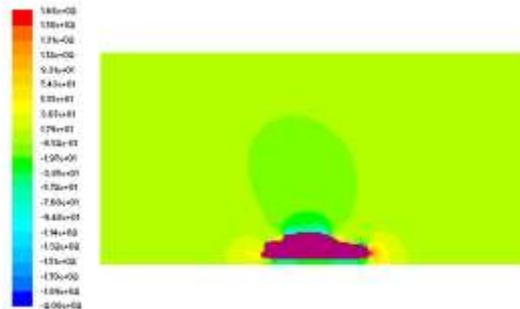


Fig. 16: Pressure contour at symmetry plane

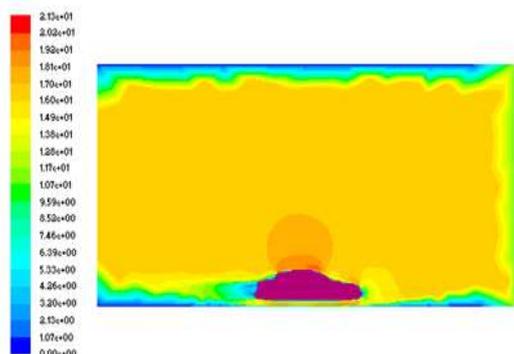


Fig. 17: Velocity contour at symmetry plane

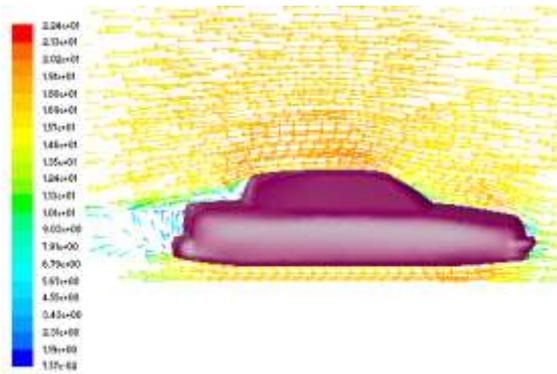


Fig. 18: Velocity vector over a car profile

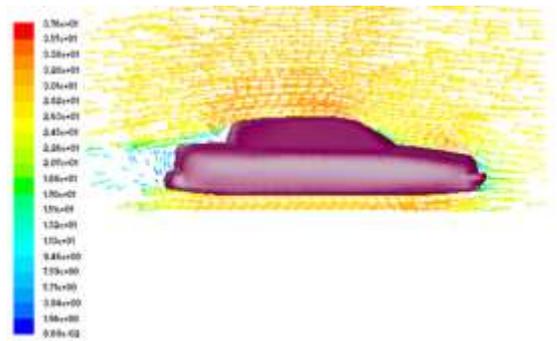


Fig. 22: Velocity vector over a car profile

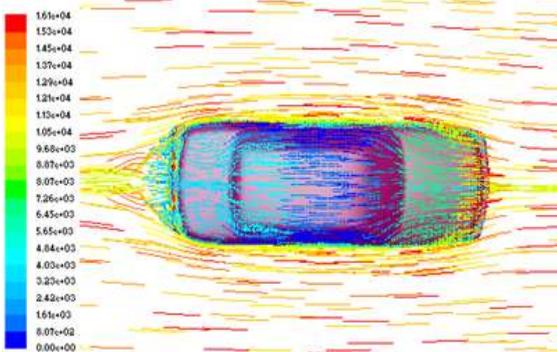


Fig. 19: Path lines

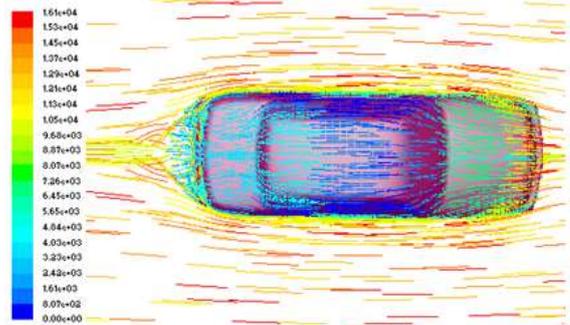


Fig.23: Path lines

CASE 2: In this case geometry of the car at rear hood is modified.

C. Velocity of Flow used for Simulation is 27.7 m/s

The Fig.20 shows the Pressure distribution along the symmetry plane of the car. It can be seen that the maximum pressure is 468 Pa is at in front and top surface of the car. In Fig.21 shows the velocity (35.6 m/s) distribution is maximum at top surface of the car. Velocity Vectors are plotted in the Fig.22 on the symmetry plane. Recirculation has further increasing; path lines give the best visualization of the flow around the body. Hence it is plotted in Fig.23.

D. Velocity of Flow used for Simulation is 11.1m/s

The Fig.24 shows the pressure distribution along the symmetry plane of the car. It can be seen that the maximum pressure is 76.3 Pa is at in front and the windshield bottom side of the car it is due to the sharp edges. In Fig.25 shows the velocity (13.8m/s) distribution is maximum velocity on the top surface of car. The velocity vectors are plotted in the Fig.26 on the symmetry plane. Recirculation is observed behind the car and also in front of the car. Path lines are plotted in Fig.27.

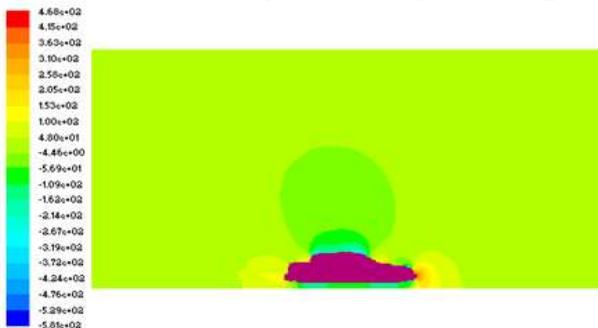


Fig. 20: Pressure contours at symmetry plane



Fig. 24: Pressure contours at symmetry plane

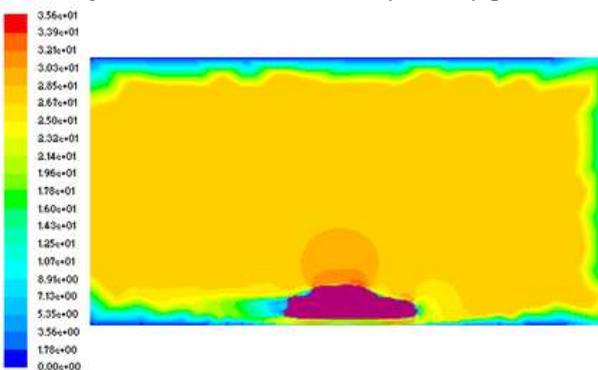


Fig. 21: Velocity contours at symmetry plane

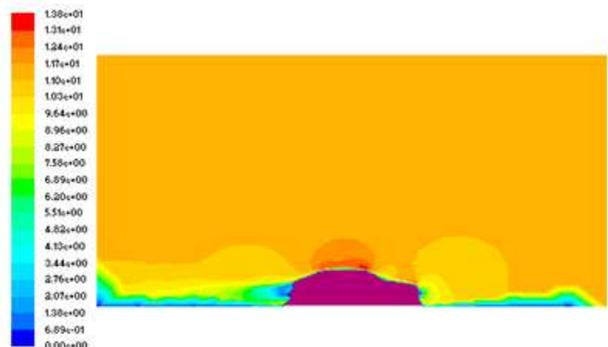


Fig. 25: Velocity contours at symmetry plane

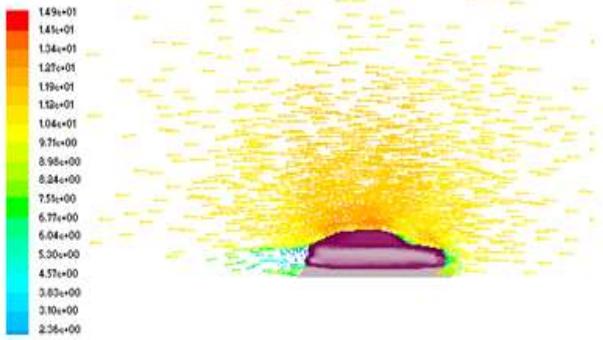


Fig. 26: Velocity vector over a car profile

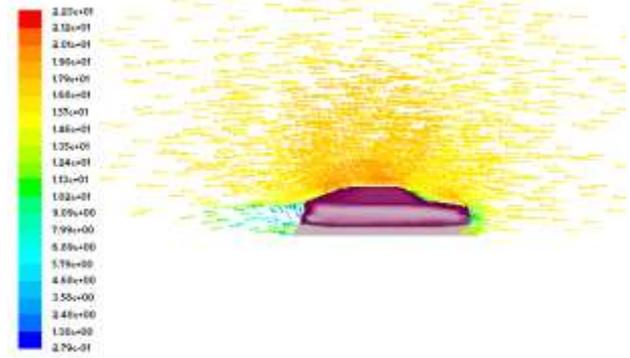


Fig. 30: Velocity vector over a car profile

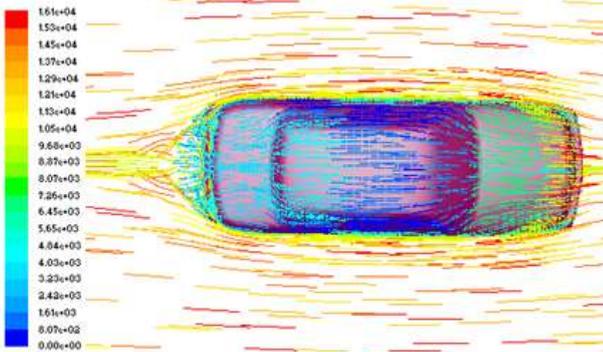


Fig. 27: Path lines

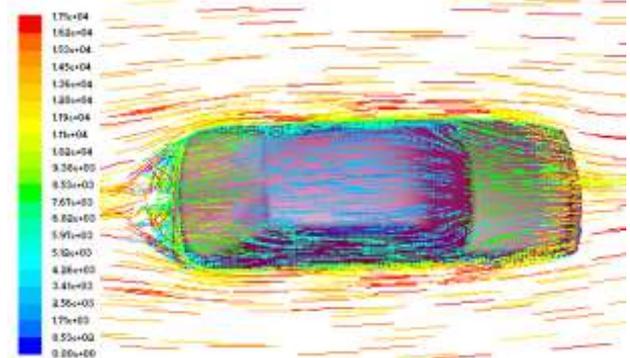


Fig. 31: Path lines

E. Velocity of Flow used for simulation is 16.6 m/s

The Fig.28 shows the Pressure distribution along the symmetry plane of the car. It can be seen that the maximum pressure is 170 Pa. In Fig.29 velocity distribution is shown in Fig.. The maximum velocity reached is 20.7m/s. Velocity vectors are plotted in the Fig.30 on the symmetry plane. Recirculation is observed behind the car and also in front of the car.

The path lines over the car at a location of $y = 0.4$ m is shown in Fig.31

F. Velocity of Flow used for simulation is 27.7 m/s

The Fig.32 shows the pressure distribution along the symmetry plane of the car. It can be seen that the maximum pressure is 472 Pa. In Fig.33 velocity distribution is shown the maximum velocity reached is 34.6m/s. Velocity vectors are plotted in the Fig.34 on the symmetry plane. Recirculation is observed behind the car and also in front of the car. Path lines are plotted in Fig. 35



Fig. 28: Pressure contours at symmetry plane

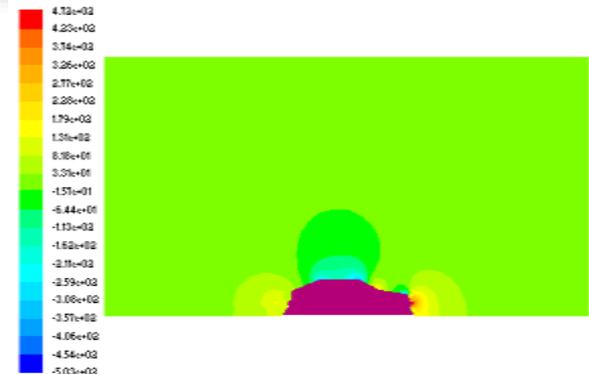


Fig. 32: Pressure contours at symmetry plane

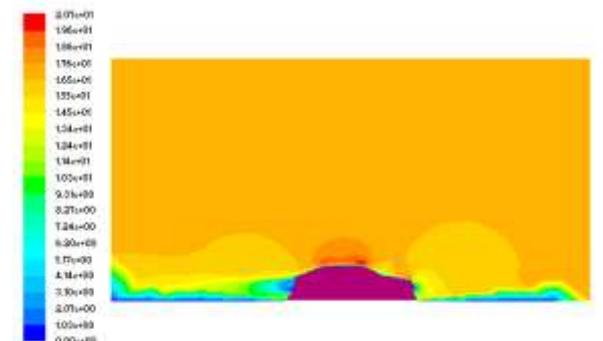


Fig.29: Velocity contours at symmetry plane

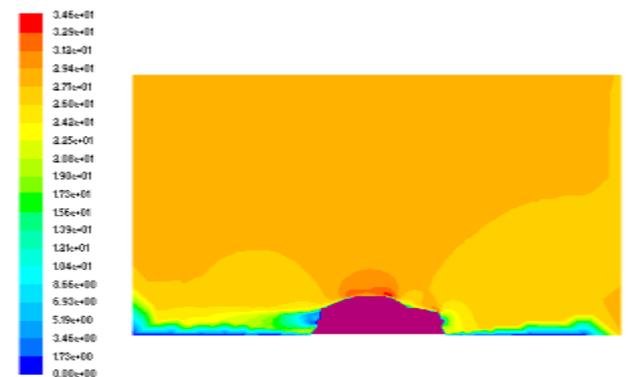


Fig. 33: Velocity contours at symmetry plane

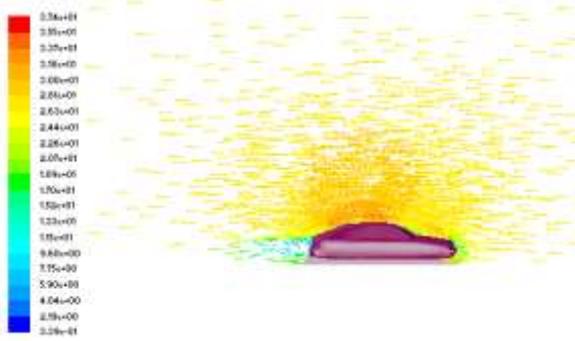


Fig. 34: Velocity vector over a car profile

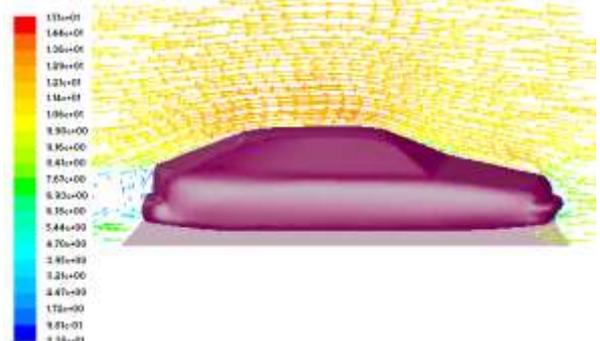


Fig. 38: Velocity vector over a car profile

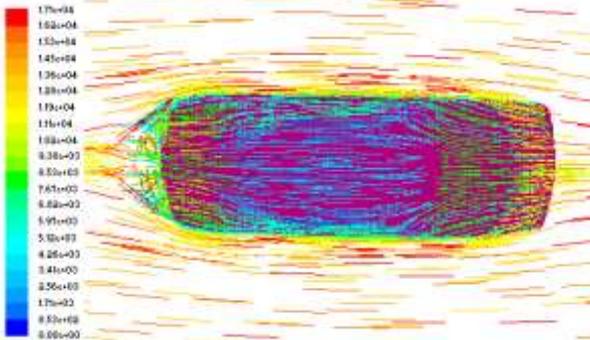


Fig. 35: Path lines

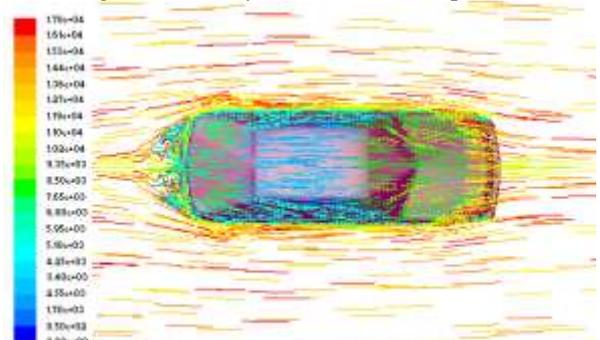


Fig. 39: Path lines

CASE 3: In this case geometry of the car is modified at two locations i.e. in front and at the rear hood of the car.

G. Velocity of Flow used for simulation is 11.1m/s

The Fig.36 shows the pressure distribution along the symmetry plane it is observed a maximum pressure of 76.1 Pa. In Fig.37 velocity distribution is shown the maximum velocity achieved is 13.7 m/s. Velocity vectors are plotted in the Fig.38 on the symmetry plane. Recirculation is observed behind the car and also in front of the car. The path lines are plotted in Fig.39

H. Velocity of Flow used for simulation is 16.6 m/s

The Fig.40 shows the pressure distribution along the symmetry plane of the car. It can be seen that the maximum pressure is 170 Pa. In Fig.41 velocity distribution is shown, the maximum velocity achieved is 20.6 m/s. Velocity vectors are plotted in the Fig.42 on the symmetry plane. Recirculation is observed behind the car and also in front of the car; path lines give the best visualization of the flow around the body. Hence it is plotted in Fig.43.

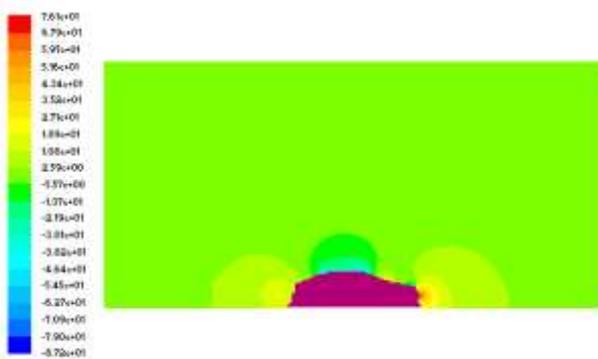


Fig. 36: Pressure contours at symmetry plane



Fig. 40: Pressure contours at symmetry plane

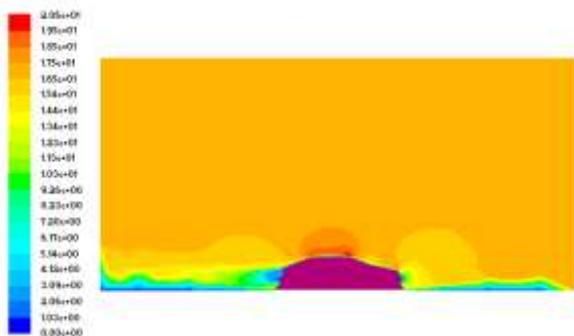


Fig. 37: Velocity contours at symmetry plane

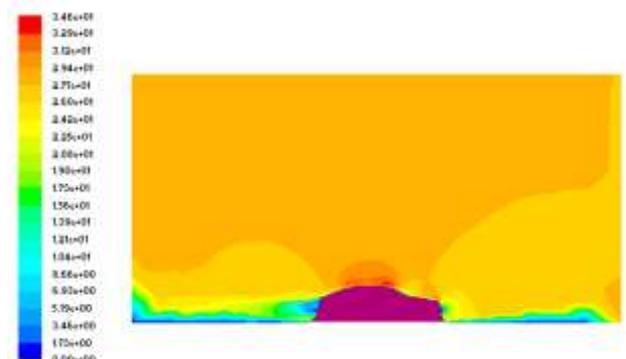


Fig. 41: Velocity contours at symmetry plane

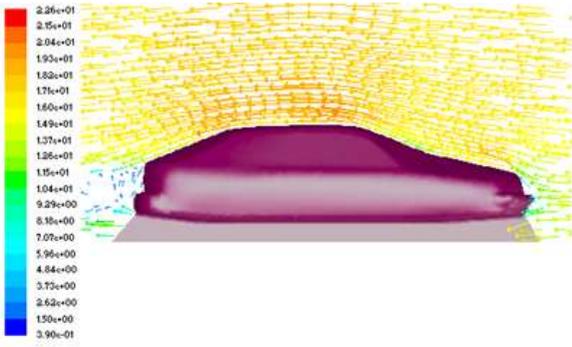


Fig. 42: Velocity vector over a car profile

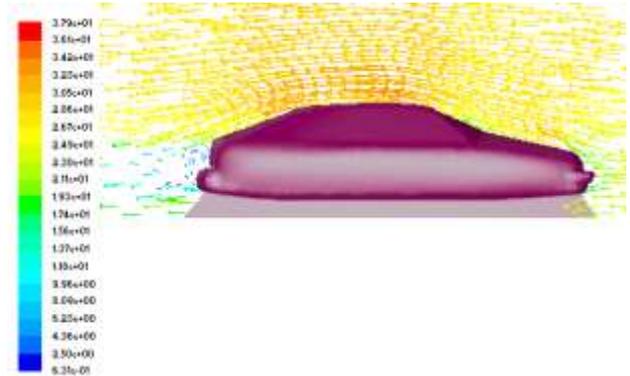


Fig. 46: Velocity vector over a car profile

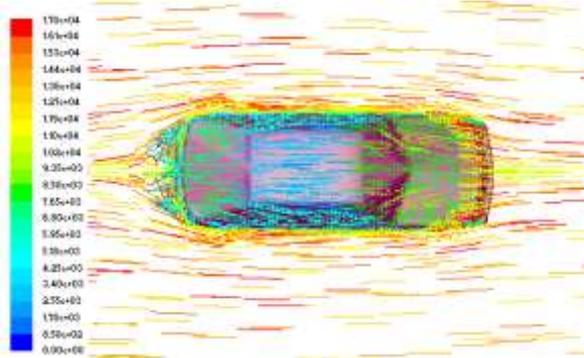


Fig.43: Path lines

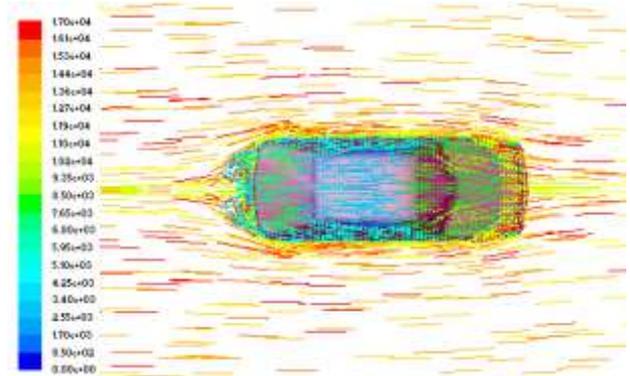


Fig. 47: Path lines

I. Velocity of Flow used for Simulation is 27.7 m/s

The Fig.44 it is observed that the maximum pressure of 474 Pa at the front end of hood and at the bottom of the windshield intensity of pressure decreased. In Fig.45 velocity distribution is shown, the maximum velocity achieved is 34.4 m/s. Velocity vectors are plotted in the Fig.46. Recirculation is observed behind the car and also in front of the car; blue color indicates the negative velocity due to recirculation. Path lines are plotted in Fig.47

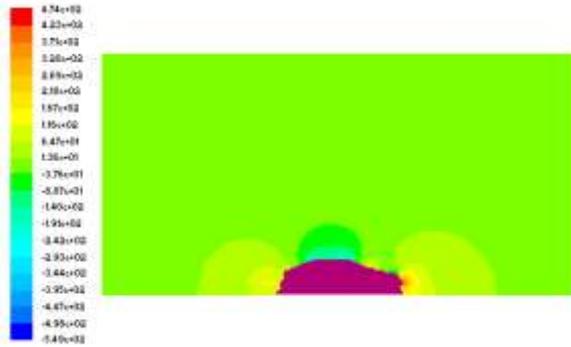


Fig. 44: Pressure contours at symmetry plane

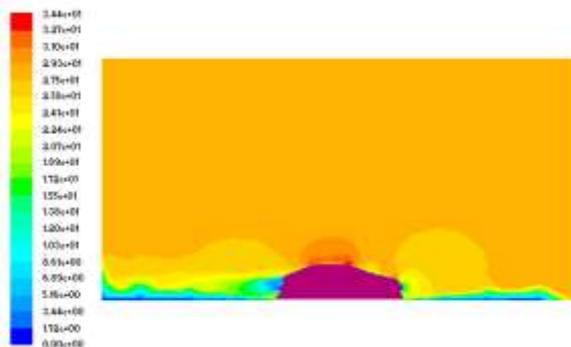


Fig. 45: Velocity contours at symmetry plane

Velocity, m/s	Non alteration		First alteration		Second alteration	
	Lift force, kN	Drag force, kN	Lift force, kN	Drag force, kN	Lift force, kN	Drag force, kN
10.0	0.381	1.114	0.45	1.02	0.546	1.045
11.1	0.477	1.368	0.56	1.25	0.680	1.280
16.6	1.120	3.021	3.88	1.53	1.580	2.802
22.2	2.053	5.354	2.35	4.83	2.904	4.945
27.7	3.265	8.133	3.72	7.54	4.593	7.625
33.3	4.960	11.90	5.37	10.9	6.704	10.91

Table 2: Variation of lift and drag force without geometry alterations and with geometry alterations

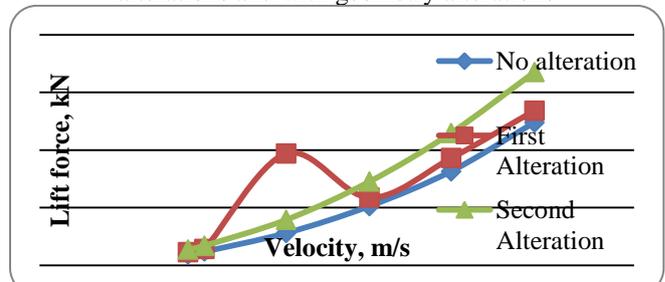


Fig. 48: Variation of lift forces with increase in speed for 3 cases

The Fig.48 shows the variation of lift forces with respect to velocity. For a non altered geometry and second alteration the lift forces are increasing gradually as velocity increases. For first alteration there is a sudden increase after

11.1m/s speed and drop after 16.6 m/s because of car profile after 22.2m/s velocity there is a gradual increase in lift force as the speed increases.

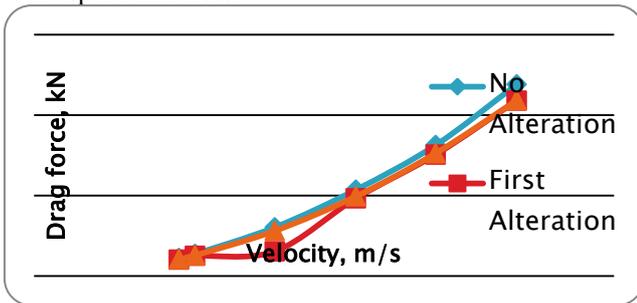


Fig.49: Variation of drag forces with respect to vehicle speed 3 cases

The Fig.49 shows the variation of drag forces with respect to vehicle speed for 3 cases. For Non alteration of car the drag force increases gradually as the vehicle velocity increases. For first alteration the drag force is increasing for first two speeds(10 and 11.1m/s) and then at 16.6m/s speed drag force is reduced and then gradual increasing as vehicle speed increases.

IV. CONCLUSIONS

Based on this study of the aerodynamic flow around a car, First alteration of the car stood best among the three profiles of car based on results obtained in terms of Drag it is about 10.90kN and Lift force is 5.37kN. To estimate the drag coefficient and flow visualization is achieved successfully by the use of CFD simulations. Aerodynamics drag for car body of profiles are successfully simulated with ANSYS Fluent solver, the analysis shows aerodynamics drag in term of drag forces or drag coefficient proportionally increased to the square of velocity for car body. The backflow became less prominent as the alterations were made and the flow progressed more smoothly over the front of the third model than over the others. Altering the geometry of a car can have a great effect on the flow of the fluid around it. Compared to the three geometries, the first altered geometry has the better performance because the velocity vectors are traveling very smoothly along the entire geometry of the car. By this the recirculation observed are very less when compared to the other two variants and the second altered geometry is better than the original geometry. The velocity is greatly affected at the symmetry plane, but not so much on the plane of the outer domain.

REFERENCES

- [1] Wolf-Heinrich Hucho: Aerodynamic of Road Vehicle; Fourth Edition; Society of Automotive Engineers, Inc. 1998.
- [2] Heinz Heisler: Advanced Vehicle Technology: Second Edition; Elsevier Butterworth Heinemann. 2002.
- [3] Rosli Abu Bakar, DevarajanRamasamy, Fazli Ismail, Design and Development of Hybrid Electric Vehicle Rear Diffuser, Science, Technology & Social Sciences 2008 (STSS), Malaysia.
- [4] Luca Iaccarino. Cranfield University Formula 1 Team: An Aerodynamics Study of the Cockpit. School of Engineering.Cranfield University. August 2003.

- [5] Gambit 2.1 Documentation Modeling Guide. Fluent, Inc., Ann Arbor, MI.
- [6] Fluent 6.1 Documentation User Guide. Fluent, Inc., Ann Arbor, MI.
- [7] Amir Shidique, Simulation and Analysis of Hybrid Electric Vehicle (HEV) by Addition of a Front Spoiler, p39, Thesis, University Malaysia Pahang, 2007.
- [8] Mark Coombs and Spencer Drayton, Proton Service and Repair Manual, Haynes Ptd. Ltd, P Ref 1, 2003, USA.