Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2011
(ICMERE2011) 22- 24 December 2011, Chittagong, Bangladesh

ICMERE2011-PI-147

COMPUTATIONAL AERODYNAMIC INVESTIGATIONS OVER THE CAR BODY

Apurba Das and M. A. Wazed

Department of mechanical Engineering
Chittagong University of Engineering & Technology (CUET)
Chittagong- 4349, Bangladesh
E-mail: apurba_cuet@yahoo.com , awazed@gmail.com

Abstract*-Computational Fluid Dynamics (CFD) emerges as an advanced investigative means in the fields of automobile, aeronautics and aerospace. It reduces the wind tunnel experiments. This paper outlines the process taken to optimize the geometry of a vehicle. Vertices and edges were imported into Gambit and a computational domain created. An unstructured triangular mesh was then applied. The Fluent was used to iterate toward a converged solution with the goal of obtaining a better flow around the car and lowering the coefficient of drag. By that we get the pressure field and velocity field around the vehicle, then acquire air drag coefficient (C_D) and drag force and compare it with the redesigned car geometry.*

Keywords: CFD; Drag Force; Aerodynamics; Car Body; Flow.

1. INTRODUCTION

The performance, handling, safety and comfort of an automobile are significantly affected by its aerodynamics properties. Defecting drag was the first major focus of automotive aerodynamics, beginning in 1960's. Low drag is important for fuel economy and low emissions. Other aspects of vehicle aerodynamics are no less important for quality of automobiles such as directional stability, wind noise, cooling of engine, ventilating and air conditional these all depend on flow field around and through vehicle. Nowadays, automotive designer rely on aerodynamics principle to create improvement in the power and handling of vehicle at high speeds. Passenger cars have become more shapely over the years as manufacturer discovered how streamlining can increase fuel efficiency, allowing a car to travel at the same speed using less horsepower. These designs reduce air resistance, or aerodynamics drag. Low drag coefficient make the vehicle enable to move easily through the surrounding viscous air with minimum of resistance. As an increasing of drag, the more power of car to do work than reducing the power train efficiency. In aerodynamic field there have two major studies need to be concerned where is study the airflow on the body and estimation of drag.

To understand the aerodynamics on the car model, flow visualization is the best technique as usual does by wind tunnel. But, in this project Computational Fluid Dynamics (CFD) analysis will be used as the technology of computer simulation to estimate the drag of car model after conventional technique due to economical factor[1].

With the increasing improvement of the automobile and freeway technique and the decrease of petrol resource, people ask for higher automotive tractive ability, lower fuel consumption and better handling. Automotive aerodynamics characteristics are very important to these capacities. At present, wind tunnel is a chief research means in our country, but Computational Fluid Dynamics, CFD, has developed very fast in some countries, with the appearance of high performance computer and accurate analytic method. As a modern method, CFD can not only shorten the automobile design period, but also greatly reduce the wind tunnel experiments. In this article, we study a certain car. In order to obtain the accurate result with less time and cost, we adopt simplified model of the car, which can efficiently stand out the main facets of the problem.

CFD is a computational technology that enables us to study things that flow. CFD not only predicts fluid flow behavior, but also the transfer of heat, mass, phase change, chemical reaction, mechanical movement and stress or deformation of related solid structures.

2. THEORY OF AERODYNAMICS

In this section, the fundamental of aerodynamics [2] is discussed to gain understanding in doing analysis of the project. The basics equation and terms in aerodynamics field or fundamental of fluid mechanics such as Bernoulli's Equation, lift and drag Forces are studied.

2.1 Bernoulli's equation

The basis of aerodynamics is the Bernoulli's equation which states that at any point in a streamlined flow if the local air stream is lower than that of the undisturbed flow, the loss is compensated by an increase in pressure [3]

$$P + \frac{V^2}{2} = constant \qquad (1)$$

From equation (1) shows the increasing of velocity will case the decrease in static pressure and vice versa.

2.2 Drag

The drag force[4] is a reactive force that tends to slow an object down as it falls through a medium. The drag coefficient is a value for a particular object that describes the ratio of the drag force to the factors that influence the drag force. The drag coefficient depends on the size, shape, and weight of the object but it is usually associated with the extent to which the object is streamlined. Generally, the larger the drag coefficient, the more a drag force it will produce while falling, and therefore, the slower it will fall.

The drag force is related to these variables and the drag coefficient (C_D) by:

$$F_D = C_D \frac{1}{2} \rho A V^2 \qquad (2)$$

The drag force (F_D) is related to the density (ρ) of the medium in which the object is located, the planar area (A) perpendicular to the movement, and the velocity (V) of the object relative to the velocity of the medium. The value of the drag coefficient is quite variable and may vary with the relative velocity.

The Reynold's number **R** is a dimensionless quantity that is important in drag coefficient analyses. It is computed as:

$$R = \frac{\rho V D}{\mu} = \frac{\rho D}{\nu} \qquad (3)$$

Where ρ is the fluid density, μ is the dynamic viscosity, v is the kinematic viscosity, V is the velocity, and D is the length parameter such as the diameter of the object

2.3 Lift

The aerodynamic lift acts, by definition, into the direction normal to the direction of motion, thus in the case of a car it acts normally to the ground. It is accepted as a universal sign convention that the lift is positive when it acts upwards, away from the ground, and it is negative when it presses the vehicle to the ground. Positive lift reduces the tire load on the road and substantially lessens the grip. The Lift force is related to these variables and the lift coefficient (C_L) by

$$F_L = C_L \frac{1}{2} \rho A V^2 \qquad (4)$$

3. METHODOLOGY

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 (Version 2.2.3). In this program, geometry of the object is created, either from scratch or by importing a previously created model. After the modeling is done, a grid is applied. This grid, also called a mesh, represents the computational domain. Throughout this mesh is where the governing equations are solved numerically by the CFD software. After the grid is completed and boundary conditions specified, the mesh can be exported. Fluent (Version 6.2.16) is used to solve the exported problem. Fluent applies fluid flow physics in order to get solutions. The Fluent software performs discretization over the computational domain and allows the user to generate figures and plots as a means of visually comparing the results and data.

A redesigned car body shown in figure 2 was used to further analyze and compare the aerodynamic effects of a change in the body shape. This redesigned car body has the same vital dimensions i.e. length, width, average height, ground clearance and volume of the model car.

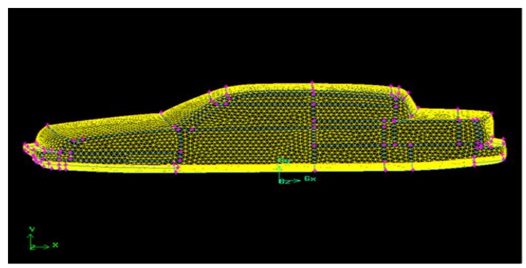

Fig 1: Original car geometry with meshed faces

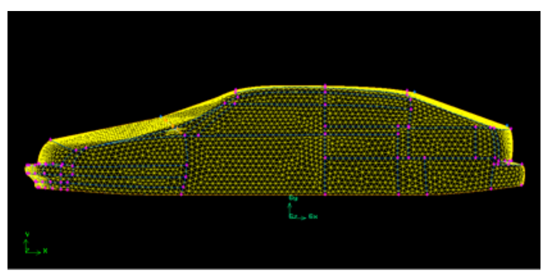

Fig 2: Redesign car geometry with meshed faces

4. RESULT AND DISCUSSION

CFD is applied to the automotive aerodynamics with Finite volume Method (FVM). By that we get the pressure field and velocity field around the vehicle, then acquire air drag coefficient (C_D) and compare it with the redesigned car geometry.

Figure 3 and 4 show static pressure contours for both case geometry with details of pressure distribution of car front. It is shown on figures that there is a higher pressure concentration on the car front in both cases.

From figure 3, it can be easily found that there are positive pressure areas at the front of the body, especially in the area before the radiator and the boundary area between hood and the front windshield. However negative pressure area is also found at the front end of the hood. Both the front and rear end of the roof are negative pressure areas. Particularly, air slows down when it is approaches the front of the car and results that more air molecules are accumulated into a smaller space. Once the air stagnates in front of the car, it seeks a lower pressure area, such as the sides, top and bottom of car. As the air flows over the car hood, pressure is decreasing, but when reaches the front windshield it briefly increasing. When the higher pressure air in front of the windshield travels over the windshield, it accelerates, causing the decreasing of pressure. This lower pressure literally produces a lift force on the car roof as the air passes over it.

The contours of static pressure shown in Figure 4 demonstrate that the redesigned car geometry has a reduced pressure head in the front end. This pressure head greatly contributes to the down force generated in the front half of the car therefore provides for stability. As the air flows beyond the front grille of the car, the static pressure reduces with an increase in the gradient of the bonnet. This graduated change in the pressure contours reduces the concentrated pressure heads along the upper surface of the redesigned car geometry.

It is also shown that the redesigned car geometry has smaller amount of pressure in front of the windshield of the car. Because of larger angle between hood and front windshield of redesigned car geometry it is achieved that pressure contour over hood is almost constant, which is not achieved in first case of car geometry.

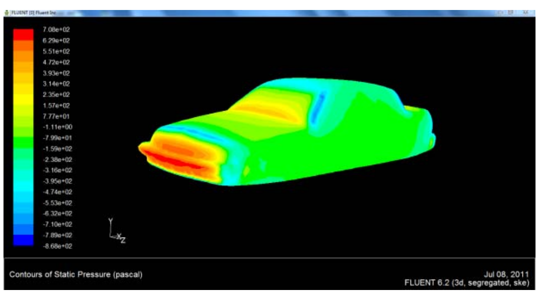

Fig 3: Pressure contours over actual car geometry

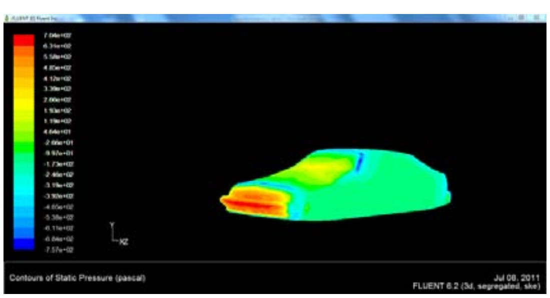

Fig 4: Pressure contours over redesigned car geometry

Figure 5 shows the velocity contours of actual car geometry, and figure 6 shows velocity contours of redesigned car geometry. Figures show that air velocity is decreasing as it is approaching the front of the car. Air velocity then increase away from the car front. It is obvious from figures that in case two velocity magnitude increases with a higher gradient, which means that air resistance is smaller, and that is the result of redesigned geometry in sense of increasing angle between hood and front windshield.

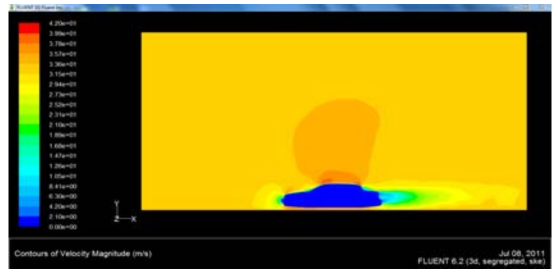

Fig 5: Velocity contours over the front end of actual car geometry

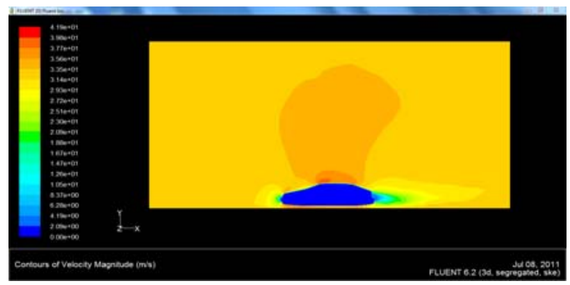

Fig 6: Velocity contours over the redesigned car geometry

In the velocity vector figure 7 and 8, it can be seen that there are small separation sectors in the areas above the radiator, near the bumper, and above the front and rear windshields and some eddy flow is found in the wake of the body. In the area near to the symmetric section, air flow is parallel to ground while it is apart from the roof, and slightly deflects to both sides. Air between floor and the ground goes upwards while leaving from the body. The main direction of the wake is upward. In the areas near to the sides of the body, air from the bottom is whirling and a large eddy is found. Air from the roof of either side whirls softly, it forms rather small eddies, which are not obvious.

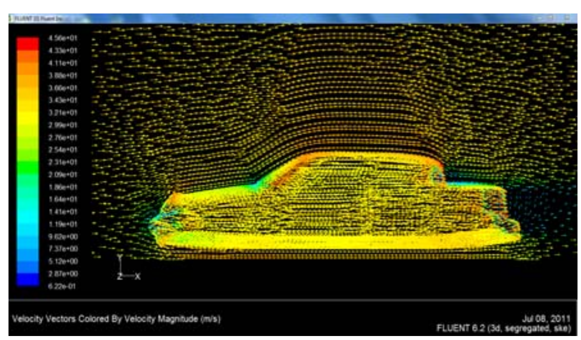

Fig 7: Velocity vectors over the actual car geometry

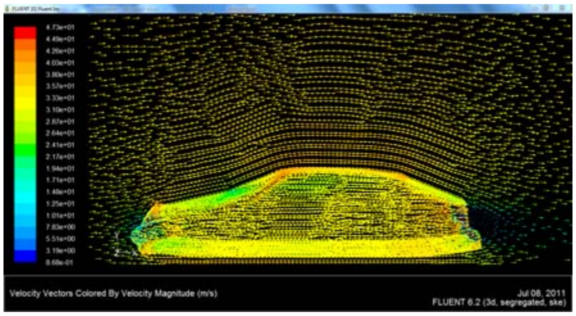

Fig 8: Velocity vectors over the redesigned car geometry

The Figure 7 demonstrates that the velocity of the air reduces as it reaches the front of the car. This velocity then increases due to a positive slope above the front end of the car. The airflow velocity over the roof reduces significantly and remains constant as it flows towards the rear of the car. This airflow trend is attributed in part to the rear airfoil which does not trip the flow or decrease its velocity. The reshaped front grille on the redesigned model car from figure 8 acts like a flow tripper delaying the formation of a potentially stagnant boundary layer. This reduces the viscous forces caused by the boundary layer of the car therefore contributing to a reduction in the overall drag force.

The resolution of the boundary layer on the body is directly responsible for the size of the wake. Based on the velocity vectors in figure 9 and 10, the air flows produce two counter rotating vortices behind the car body which is typical for separating flows behind the car body. The combinations of large vortex and separation cause a large effect on the drag force.

As shown in Figure 9 and Figure 10 the wake i.e. region of air recirculation is at the back of the car. In this region both models experience high turbulence and reverse velocities. However redesigned the back reduces the boundary layer stagnation and greatly changes the flow pattern behind the car. This causes a reduction in the wake region which should contribute to a decrease in the drag force.

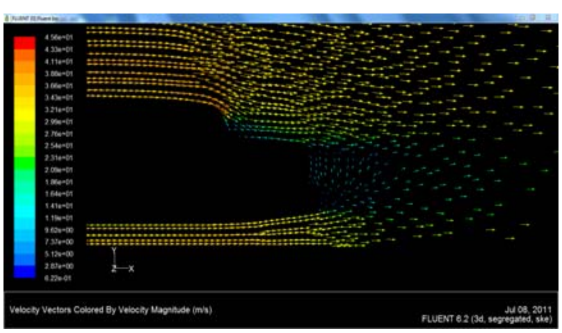

Fig 9: Velocity vectors of rear end of the actual car geometry

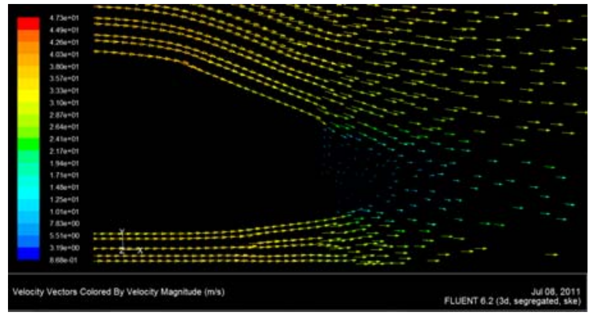

Fig 10: Velocity vectors over the rear end of redesigned car geometry

4.1 Drag Force Calculation: The Drag force value for a moving vehicle is given by the following expression Drag Force

$$F_D = C_D \frac{1}{2} \rho A V^2$$

Table 1: Variation of drag force, drag coefficient with velocity of actual car geometry

Velocity(V)(m/s)	Drag Force(F_D)N	Drag Coefficient(C_D)
25	150.49	0.40173228
33.33	266.51	0.40026862
40	361.48	0.399289262
45	484.13	0.39888537
50	596.71	0.39823019

Table 2: Variation of drag force, drag coefficient and with velocity of redesigned car geometry

Velocity(V)(m/s)	Drag Force(F_D)N	Drag Coefficient(C_D)
25	142.33484	0.38005132
33.33	251.55	0.36970461
40	342.66	0.37703
45	456.744	0.3764087
50	562.53	0.3755093

Simulation is done for different air velocity and the value of drag force is obtained for each velocity Shown in table 1 and table 2. Variation of drag force with air velocities is very much in confirmation with theoretical understanding. For validation of the result, only variation of drag force with air velocities are compared with another redesigned car model.

From equation of drag force, Aerodynamic drag increases with the square of speed of car. Figure 11 shows the comparison of drag force between actual car geometry and redesigned car geometry results. It appears a good agreement among the results. Results differ due to the shape of the model, especially with velocity. Shape, however, plays a major factor in the aerodynamics of the vehicle. For same velocity, Drag force is less than actual car geometry of redesigned car geometry. The boundary layer thickness and turbulence transition as well as the flow separation points are dependent on the shape of the car, therefore determining the most dominant source of drag. The boundary layer creates viscous drag through shear stress onto the surface, and separation creates pressure drag. On a full scale car, the pressure drag is stronger than viscous due to the complex shape of car.

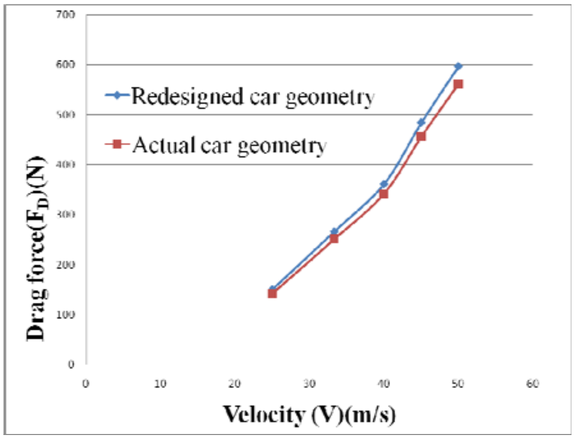

Fig. 11: Comparison of drag force between actual car geometry and redesigned car geometry

5. CONCLUSION

From the above discussions and result, we can conclude into the following points:

1. The vehicle's shape has the most direct influence in the final value of drag force.
2. Aerodynamic drag increases with the speed of car.
3. Drag force of redesigned car geometry is less than actual car geometry.
4. Applying this study to different geometry of vehicles body to reduce the drag, optimum geometry can be found

6. REFERENCES

[1] Mahazis, M.B, Design And Analysis Front Hood Fairing For Cars, in Mechanical Engineering. November 2009, Universiti Malaysia Pahang.

[2] Salleh, M.K.A.B., Simulation and analysis drag and lift coefficient between sedan and hatchback car, in Mechanical Engineering. November 2009, Universiti Malaysia Pahang.

[3] Chainani. A, P.N., CFD Investigation of Airflow on a Model Radio Control Race Car, in Proceedings of the World Congress on Engineering 2008.

[4] Kucner, L.K. and R.H. McCuen, Drag Coefficient. july,2004.

7. NOMENCLATURE

Symbol	Meaning	Unit
F_D	Drag force	(N)
F_L	Lift force	(N)
P	Pressure	(Pa)
A	Area	(m^2)
C_D	Drag coefficient	Dimensio-nless
C_L	Lift coefficient	Dimensio-nless
R	Reynold's number	Dimensio-nless
ρ	Density	(Kg/m^3)
μ	Viscosity	(kg/ ms)