*Proceedings of MARTEC 2010*
*The International Conference on Marine Technology*
*11-12 December 2010, BUET, Dhaka, Bangladesh*

# COMPUTATIONAL DRAG ANALYSIS OVER A CAR BODY

**M. M. Islam[1] and M. Mamun[2]**

[1]Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Dhaka, Bangladesh.
E-mail: engg_monir09@yahoo.com

[2]Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Dhaka, Bangladesh.
E-mail: mdmamun@me.buet.ac.bd

## ABSTRACT

*In the recent times, CFD simulations, with the advent of computer architectures with superfast processing capabilities are rapidly emerging as an attractive alternative to conventional wind tunnel tests which are either too restrictive or expensive, for aerodynamic styling of a car. In vehicle body development, reduction of drag is essential for improving fuel consumption thus protects the global environment and driving performance, and if an aerodynamically refined body is also aesthetically attractive, it will contribute much to increase the vehicle's appeal to potential customers. 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 CFD program Fluent was used to iterate toward a converged solution with the goal of obtaining a better flow around the car and drag force. The results are analyzed and only the drag force is compared with a recognized journal to validate the results. These practices were detailed in hopes that further research would use the ground work laid out in this paper to redesign existing vehicles in order to improve handling and increase fuel efficiency.*

**Key words**: CFD, car, drag, velocity vector, pressure contour, Gambit, Fluent.

## 1. INTRODUCTION

The rapidly increasing fuel prices and the regulation of green house gasses to control global warming have given tremendous pressure on the design engineers to enhance the current designs of the automobile using minimal changes in the shapes. To full fill the above requirements, design engineers have been using the concepts of aerodynamics to enhance the efficiency of automobiles [2, 4]. A lot of emphasize is laid on the aerodynamics in car design as an aerodynamically well designed car spends the least power in overcoming the drag exerted by air and hence exhibits higher performance- cruises faster and longer, that too on less fuel (Figure 1).

Apart from improved fuel economy, aerodynamically superior car offers better stability and handling at highway speeds and also minimization of harmful interactions with other vehicles on the roadway. In optimization of car aerodynamics, more precisely the reduction of associated drag coefficient (CD), which is mainly influenced by the exterior profile of car, has been one of the major issues of the automotive research centers all around the world. Average CD values have improved impressively over the time, from 0.7 for old boxy designs of car to merely 0.3 for the recent more streamlined ones [1].

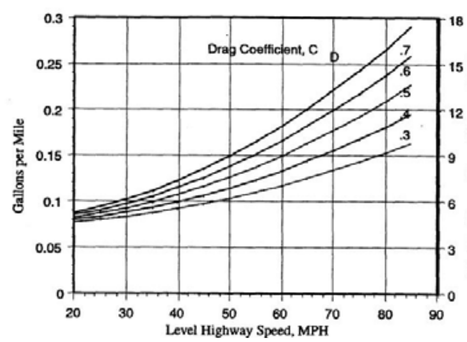


Figure 1. Fuel economy with reduction in drag coefficient (CD)

The Figure 2 shows the description of the fuel energy used in a modern vehicle at urban driving and highway driving. The shape of the vehicle uses about 3 % of fuel to overcome the resistance in urban driving, while it takes 11% of fuel for the highway driving. This considerable high value of fuel usage in highway driving attracts several design engineers to enhance the aerodynamics of the vehicle using minimal design changes [2].

The effect of drag on the moving vehicle is proportional to the square of velocity, so with increase in velocity (at approximately 50 km/h), aerodynamic drag becomes one of the most prominent factors contributing to the total drag experienced by the vehicle [5].

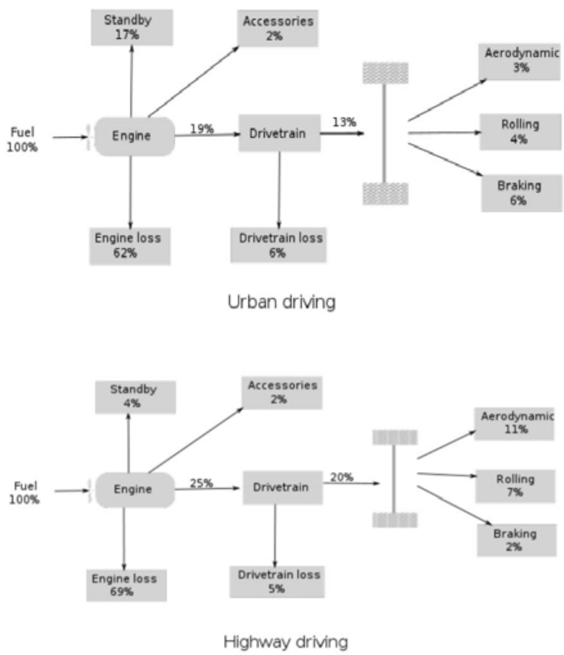


Figure 2. Typical Fuel Energy usage at urban and highway driving

Aerodynamic drag (AD), which compares the drag force, at any speed, with the force it would take to stop all the air in front of the car influences fuel consumption of a car, especially at higher speeds (Figure 3) and hence is considered a crucial factor in judging its performance. An aerodynamically well designed car spends the least power in overcoming the drag and hence yields higher performance - cruises faster and longer that too on less fuel [1].

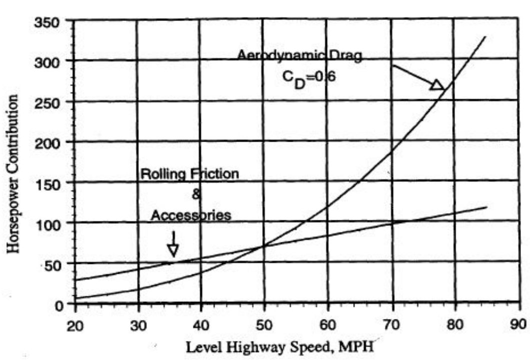


Figure 3. Increasing influence of drag co efficient (CD) at higher speeds

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 CFD program Fluent was used to iterate toward a converged solution with the goal of obtaining a better flow around the car and drag force. The results are analyzed and only the drag force is compared with a recognized journal to validate the results. These practices were detailed in hopes that further research would use the ground work laid out in this paper to redesign existing vehicles in order to improve handling and increase fuel efficiency.

## 2. PROCEDURE

The first step of CFD is to design a computational model. This general process is done using a program called Gambit (Version 2.3.16). In this program, geometry of the object is created in AutoCAD then it imported in Gambit. After the modeling is done, a grid is applied. This grid, also called a mesh, represents the computational domain. All the faces were meshed using a Tri element and an interval size of 0.003. This creates an unstructured grid of triangular mesh elements **(**Figure 4**).** Unstructured meshes are more applicable, especially to irregular faces. After the meshing was finished, the outer domain for the problem was created. 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. Then volume mesh was done using a Tet/hybrid element and TGrid type, the interval size was left at the default value of 1 (Figure 5).

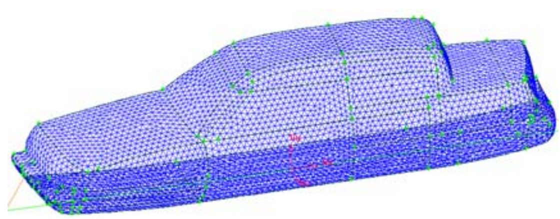

Figure 4. Meshed faces of the car

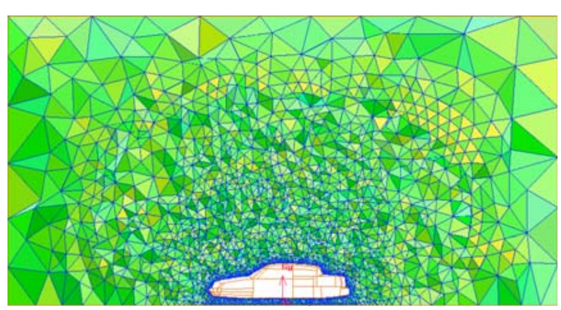

Figure 5. Side view of car and mesh

After the meshing was complete, the boundary types must be set. "Boundary-type specifications define the physical and operational characteristics of the model at those topological entries that represent model boundaries". All of the boundaries can be found in Table 1. Finally, the mesh was exported to Fluent for analysis.

Fluent (Version 6.2.16) is used to solve the exported problem. Fluent applies fluid flow physics in order to get solutions. The choice of the turbulent model depends on the required level of accuracy, available computational resources, and the required turnaround time. For the problem analyzed in this paper, standard k - ϵ turbulent model is selected. The k - ϵ model is one of the most common turbulent models. It is a semi - empirical, two equation model, that means, it includes two extra transport equations to represent the turbulent properties of the flow. The first transported variable is turbulent kinetic energy k. The second transported variable is the turbulent dissipation ϵ. It is the variable that determines the scale of the turbulence, whereas the first variable k determines the energy in the turbulence. The model transport equation for k is derived from the exact equation, while the model transport equation for ϵ was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. 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.

**Table 1. Boundary types**

| Boundary Conditions | | |
|---|---|---|
| **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 |

## 3. RESULT AND DISCUSSION

Figure 6 shows the pressure contour around the car, 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 produce a lift - force on the car roof as the air passes over it.

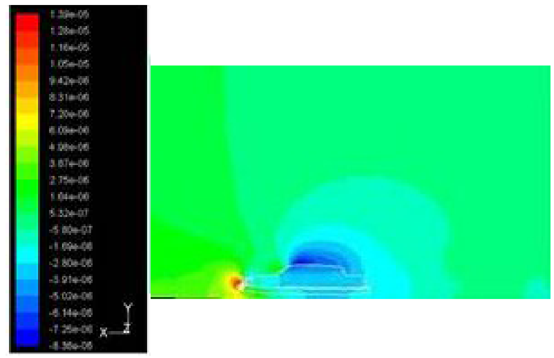

Figure 6. Pressure contour around the car body

Figure 7 shows that air velocity is decreasing as it is approaching the front of the car. Air velocity then increases away from the car front. 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.

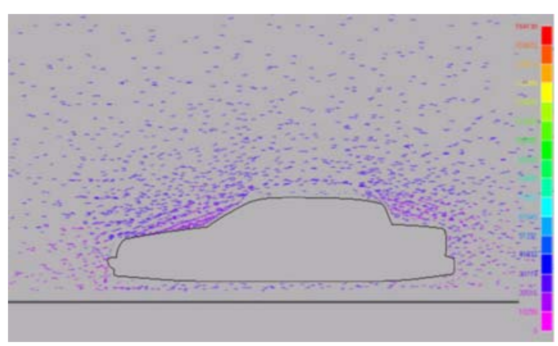

Figure 7. Velocity vector around the car body

Simulation is done for different air velocity and the value of drag force is obtained for each velocity. 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 M. Desai [1] experiments result. He did the experiments in two methods on a geometrically similar, reduced scale (1:15), clay model, differing from actual car (my model) only in size and simulating dynamically similar flow situations. He placed the model in a wind tunnel test section blows air at different velocities. He measures the variation in static pressure at different nodes by traversing Pitot tube in horizontal and vertical direction, thereby covering the entire flow field on upstream as well as downstream sides of the car and calculates in first method. In second method, he used pressure tapings along the centerline of the car over its profile.

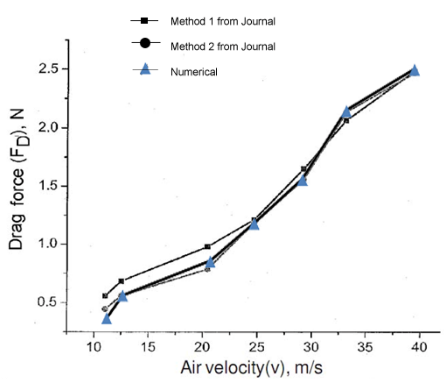


Figure 8. Variation of drag force with air velocity (Computational /Experimental from journal)

Figure 8 shows the comparison of drag force between the numerical result and the two experimental results. It appears a good agreement among the results; however in lower speed range the numerical drag force is a little bit lower than the experimental results. But in higher speed range a good agreement displays. Results differ due to the size of the model, especially in lower velocity. Size, however, plays a major factor in the aerodynamics of the vehicle. For same velocity, Reynolds number will be less if the size of the car is small. The boundary layer thickness and turbulence transition as well as the flow separation points are dependent on the Reynolds number, 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 high Reynolds number and complex shape [3]. So for the vastly different Reynolds numbers, Pressure separation drag is the dominant source of drag on these model cars and therefore the numerical value is some lower than experimental value. But it is able to overcome in higher speed.

## 4. CONCLUSION

This paper presents the process taken to optimize the geometry of a vehicle. The geometry was imported into Gambit, a computational domain created and unstructured triangular mesh was then applied. The CFD program Fluent was used to obtain the flow characteristics around the car. The results are analyzed and only the drag force is compared with a recognized journal to validate the results. Comparison of the numerically derived results with that of the experiments obtaining from well recognized journal shows a good correlation. However further investigations are suggested in order to reduce the differences in the results at different conditions. These practices were detailed in hopes that further research would use the ground work laid out in this paper to redesign existing vehicles in order to improve handling and increase fuel efficiency. It is also recommended that the methods used in this paper can be applied to other vehicles of interest and used to create better vehicles. Applying this study to different geometry of vehicles to reduce the drag optimum geometry can be designated, which is the ultimate objective.

## REFERENCES

[1] M. Desai, S A Channiwala and H J Nagarsheth, "A comparative assessment of two experimental methods for aerodynamic performance evaluation of a car," *Journal of scientific & Industrial research,* Vol. 67, pp 518 – 522 (July 2008)

[2] P. N. Krishnani, "CFD study of drag reduction of a generic sport utility vehicle," *Master's thesis,* Mechanical engineering department, California State University, Sacramento (fall 2009).

[3] R. T. Kell, "Aerodynamic Analysis of F1 in Schools Challenge Car Models," *Initial Thesis Report, ACME, UNSW@ADFA,* Canberra, Australia (2009).

[4] Sugiono, W. M. Hong, O. Ilias, "Developing the database for vehicle body shape design with the less of aerodynamic resistance and vibration" *ECCM 2010 , IV European Conference on Computational Mechanics,* Palais des Congrès, Paris, France (May 16-21, 2010).

[5] S. N. Singh, L. Rai, A. Bhatnagar, "Effect of moving surface on the aerodynamic drag of road vehicles" *Proceeding of IMechE.* Vol. 219 Part D: J. Automobile Engineering, pp 127- 134 (August 2004).