Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2015
(ICMERE2015) 26 – 29 November, 2015, Chittagong, Bangladesh

ICMERE2015-PI-126

DESIGN AND ANALYSIS OF THE AERODYNAMICS SHAPE AND SIZE OF HIGHWAY BUSES IN BANGLADESH

Md. Maksudur Rahman[1,*] and Md. Mamunur Roshid[2]

[1]Department of Piping & HVAC, Western Marine Shipyard Ltd.,
Chittagong, Bangladesh.
[2]Department of Mechanical Engineering, Chittagong University of Engineering and Technology,
Chittagong, Bangladesh.
[1,*] maksud09.me@gmail.com,[2] mamun_cuet2003@yahoo.com

Abstract- *The present highway buses of Bangladesh have poor aerodynamic exterior design which reduces the vehicle speed & at the same time increase the fuel consumption per km. We proposed a new design for highway buses using SOLIDWORKS (design software) in order to reduce the effect of drag force which will result in reduction of fuel consumption and increment in speed of the vehicle. Our designed bus has been numerically simulated by using ANSYS simulation software. The final result shows a considerable reduction of drag force which proves the validation of our work. I think this will be mostly applicable for all types of commercial vehicle in developing countries of the world.*

Keywords: Aerodynamics, Vehicle simulation, Solidworks, Fuel consumption, ANSYS

1. INTRODUCTION

The requirement for developing exterior shape of vehicle based on aerodynamics has gained impetus recently. The exterior styling and aerodynamically efficient design for reduction of engine load which reflects in the diminution of fuel consumption are two crying factors for successful implementation of vehicles in the competitive world. For heavy vehicles such as tractor-trailer combinations and buses, pressure drag is the dominant component due to the large surfaces facing main flow direction and due to the large wake resulting from the bluntness of the back end of such vehicles. By far the most efficient method of reducing body drag is to minimize flow separation by combining the rounding of the forward corners (sides and top) with the tapering of the body.

The discipline of aerodynamics deals with the motion of air around and through a body and the interactions associated with this relative motion between the air and the vehicle system. The aerodynamic properties of a road vehicle include effects on its performance, handling, safety and comfort.

The objectives of this project are designing an exterior shape of a vehicle based on aerodynamics, determining the drag co-efficient (c_d) of designed body using Computational fluid dynamic (CFD) called ANSYS [1] and comparing our simulated data with standard value.

With the recent increase in threat to the environment due to vehicular pollution it has become a demand of the hour to increase automobile efficiency. In order to achieve this, either change has to be brought about in engine functioning and supplementation of presently used fuel by eco-friendly fuels or by enhancing current automobile design. As far as engine optimization is concerned, we have reached a saturation point. Using eco-friendly fuels is an area still under development and it will take a few more years for it to be adopted worldwide. So, it can be easily concluded that enhancing automobile design is the easiest way to step-up vehicle efficiency.

1.1 Flows Around A Passenger Vehicle:

A vehicle that travels in a road is exposed to different kinds of resistance. The total resistance on moving vehicle with constant velocity is some of the rolling resistance (F_R), acceleration resistance (F_A) and the aerodynamic drag resistance (F_D)[2].These resistances must be overcome to propel the vehicle forward. To express the total resistance the equation of motion is used,

$$F_x = F_A + F_R + F_G + F_D = ma + fRmgcos\alpha + mgsin\alpha + (1/2)\,\rho CDAfU2[1] \quad (1)$$

The inclination resistance is independent of the velocity and since CFD calculations are performed on flat road, the inclination resistance can be neglected. During lower velocities the rolling resistance is the dominating resisting force. For velocities above 70km/h the aerodynamic drag will be the dominating resistance due to the fact that the drag is increasing with the square of the speed. Figure 1.1 shows the rolling resistance and the aerodynamic resistance as a function of the velocity, the rolling resistance slightly increase with the total resistance but can be assumed constant for velocities below 100 km/h. The aerodynamic drag force (F_D) can be divided into two components, form and fiction force. The form [pressure] drag acts in the direction normal to the surfaces, meanwhile the friction force is acting tangential to the surface. Due to the scale and shape of a passenger car the form drag is dominating and the vehicle can be assumed to be a bluff body. This means that a big wake and vortices will be present in the flow field. The total resistance force on a moving vehicle is shown below:

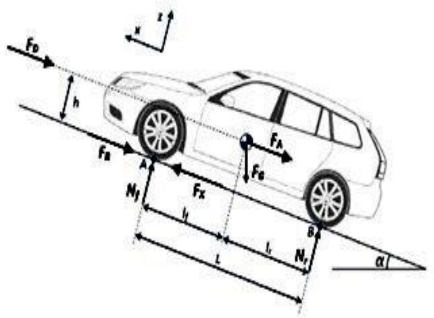


Fig.1.1: Resistance on a passenger car [2].

1.2 Equations of Motion:

For steady laminar flow of a viscous, incompressible, Newtonian fluid without free-surface effects, the equations of motion are the continuity equation [2]

$$\overline{\nabla}.\overline{V}=0 \qquad (2)$$

And the Navier–Stokes equation [2]:

$$(\overline{V}.\overline{\nabla})\overline{V} = -\frac{1}{\rho}\overline{\nabla}p' + v\nabla^2\overline{V} \qquad (3)$$

Eq. (2) is a conservation equation, while Eq. (3) is a transport equation that represents transport of linear momentum throughout the computational domain. In Eq. (3) and (4), $\overline{V}$ is the velocity of the fluid, ρ is its density, and v is its kinematic viscosity ($v=\frac{\mu}{\rho}$).

The lack of free-surface effects enables to use the modified pressure p', thereby eliminating the gravity term from Eq. (3). It is noted that Eq. (2) is a scalar equation, while Eq. (3) is a vector equation. Equations (2) and (3) apply only to incompressible flows in which we also assume that both ρ and v are constants. Thus, for three-dimensional flow in Cartesian coordinates, there are four coupled differential equations for four unknowns, *v, w,* and p'(Table 1.1).

If the flow were compressible, Eq. (2) and (3) would need to be modified appropriately. Liquid flows can almost always be treated as incompressible, and for many gas flows, the gas is at a low enough Mach number that it behaves as a nearly incompressible fluid.

The equations of motion to be solved by CFD for the case of steady, incompressible, laminar flow of a Newtonian fluid with constant properties and without free-surface effects. A Cartesian coordinate system is used. There are four equations and four unknowns, *v, w,* and p'.

Table 1.2: Equations of motion [3]

Continuity:	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$
x-momentum:	$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+\mathrm{w}\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p'}{\partial x}+v(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2})$
y-momentum:	$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+\mathrm{w}\frac{\partial v}{\partial z}=-\frac{1}{\rho}\frac{\partial p'}{\partial x}+v(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2})$
z-momentum:	$u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+\mathrm{w}\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p'}{\partial x}+v(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2})$

1.3 Drag Force:

A fluid may exert forces and moments on a body in and about various directions. The force a flowing fluid exerts on a body in the flow direction is called drag force.

$$F_d \propto V^2;$$
$$F_d = (1/2)C_D\rho Av^2[4] \qquad (4)$$

Where (F_d) is drag force, V is speed of the vehicle, A is size & shape of the vehicle, ρ is Air density, c_d is drag co-efficient.

"The lower value of (c_d), the higher the final speed, results in lower fuel consumption" [5].

2. METHODOLOGY

2.1 Geometry:

At first we have designed geometry of bus in SOLIDWORKS [6]. Then we have calculated the drag co-efficient by Computational fluid dynamics software, called ANSYS. Finally, we have compared the values with known values of drag co-efficient of present vehicles.

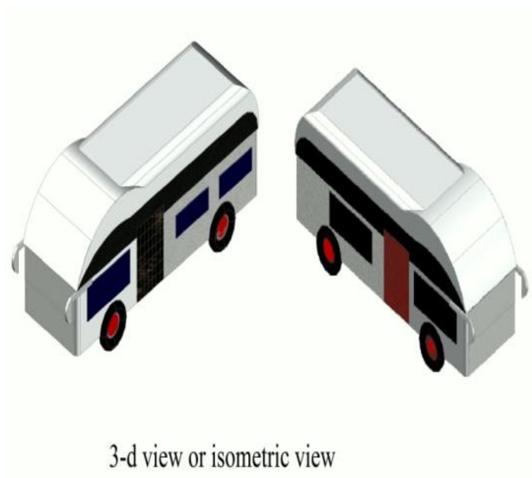


Fig.2.1 (a): Isometric view of primary design

The proposed design is simplified due to final simulation.

Simplified proposed design with proper dimension is shown below:

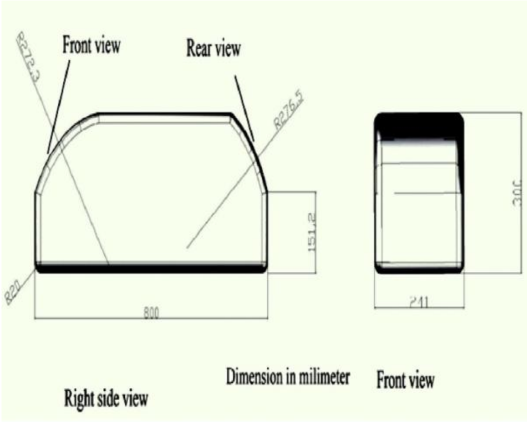


Fig.2.1 (b): Front and right side view with proper dimension

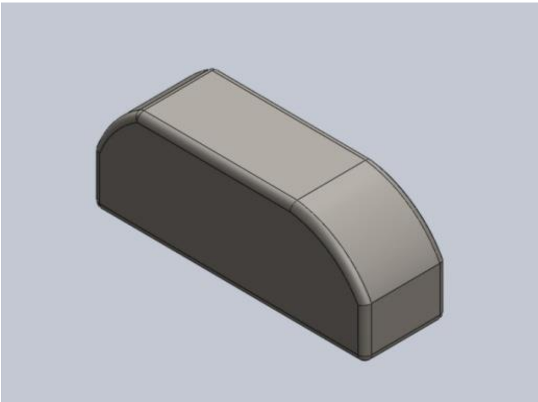


Fig.2.1(c): Simplified Isometric view

For geometry selection, a vehicle body is properly drawn by the engineering drawing tool SOLIDWORKS. To make the geometry acceptable a primary simulation is done in SOLIDWORKS.

2.2 Model Reference:

Weight is 4098.77 N, mass is 418.242 kg, Volume is 0.0531438 m^3, Density is 7870 kg/m^3.

2.3 Properties:

The material of the object is AISI 1020 steel, Cold rolled, Model type is Linear Elastic Isotropic, Yield strength is 350 N/mm^2 and tensile strength is 420 N/mm^2.

2.4 Study Result:

This simulation is consists of stress, displacement and deformation. The simulation procedure is described below sequentially.

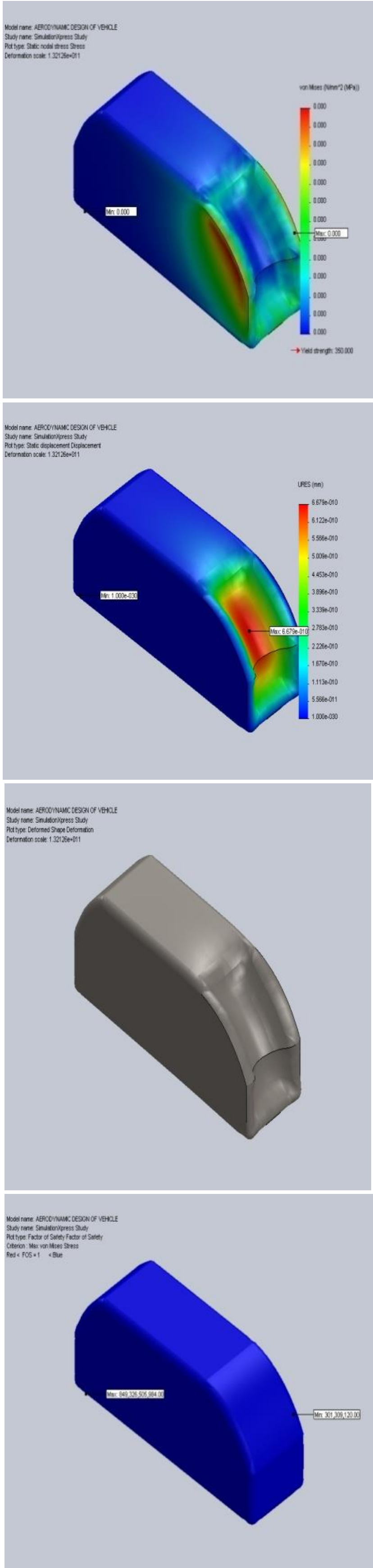


Fig.2.4: Study Result

The result is comparatively accurate for the further simulation. The geometry is now ready for the final simulation by ANSYS. In ANSYS the simulation setup is done as mentioned below.

2.5 Mesh:

The second step is mesh. It is also called computational domain. The computational domain is designed to lead to a free flow with neglect able blockage which essentially means a box that consists of an inlet, an outlet, two sides, a roof and a ground surface. The surface mesh was created on the geometry of the vehicle. Mesh parameters has been maintained as below:

Physics preference	CFD
Relevance Center	Coarse
Span Angle Center	Fine
Curvature Normal Angle	56.0 °
Min Size	1 mm
Max Face Size	250.0 mm
First aspect ratio	5

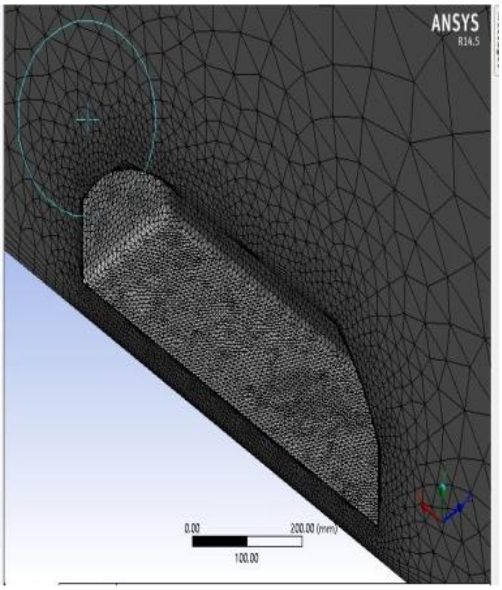


Fig.2.5 (a): The close view (Mesh)

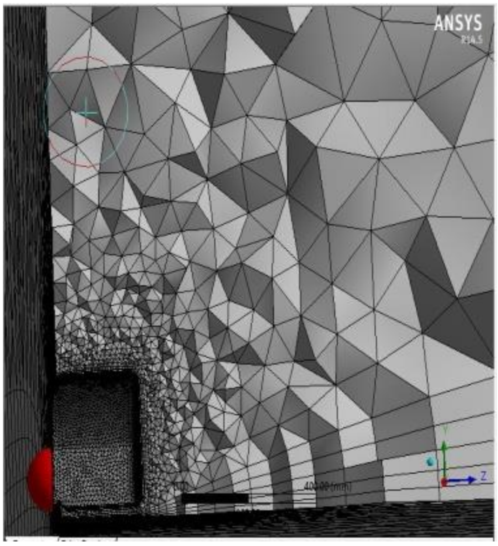


Fig.2.5 (b): The section plane (Mesh)

2.6 General Task:

Here Pressure-Based solver type is selected as it enables the pressure-based Navier-Stokes solution algorithm (the default). For this work absolute velocity formulation is used to enable the use of the absolute velocity formulation. Steady response is selected to specify that a steady flow is being solved.

2.7 Viscous Model:

As the bus average speed is around 40-60 m/s (around 200 km/h), the Reynolds number dealing with the front wing are from 106 to 3x106 (depending on the device picked on the front wing). So the flow would definitely be a turbulent one at the front wing surfaces. Thus a 'k-ε' model has been selected for analysis. k-epsilon specifies turbulent flow to be calculated using one of three models. Here realizable k-epsilon is selected.

The non-equilibrium wall function employs the two-layer concept in computing the budget of turbulence kinetic energy at the wall-adjacent cells, which is needed to solve the k- equation at the wall-neighboring cells. The wall-neighboring cells are assumed to consist of a viscous sub layer and a fully turbulent layer.

2.8 Materials:

Materials are considered as air.

2.9 Boundary Condition:

Boundary conditions are aerodynamic body, interior air, pressure outlet, symmetry, symmetry side, symmetry top, under and velocity inlet.

Table 2.9: Boundary conditions

Zone	Name	Type
1.	Aerodynamic body	Stationary wall, no slip.
2.	Interior air	Interior
3.	Pressure outlet	Pressure outlet
4.	Symmetry	Symmetry
5.	Symmetry side	Symmetry
6.	Symmetry top	Symmetry
7.	Under side	Stationary wall, no slip
8.	Velocity inlet	Velocity inlet.

2.10 Velocity inlet:

The Velocity Inlet dialog box sets the boundary conditions for a velocity inlet zone. Velocity magnitude is 40 m/s or 144 km/h and flow direction is defined as opposite of X-axis of the geometry. Turbulence intensity is 5% and Turbulence Viscosity Ratio is 10.

2.11 Reference values task:

Area is .072045 m^2, density is 1.224999 kg/m^3, Length is 1m, temperature is 288.16 k, Viscosity is $1.7894e^{-05}$, velocity is 40 m/s.

2.12 Data analysis:

Different relations are found from the simulation. They are scaled residuals, Cd-1 Convergence history, C_l-1 Convergence history, C_M-1 Convergence history, Convergence history of velocity magnitude**.**

The results are shown one by one in graphically.

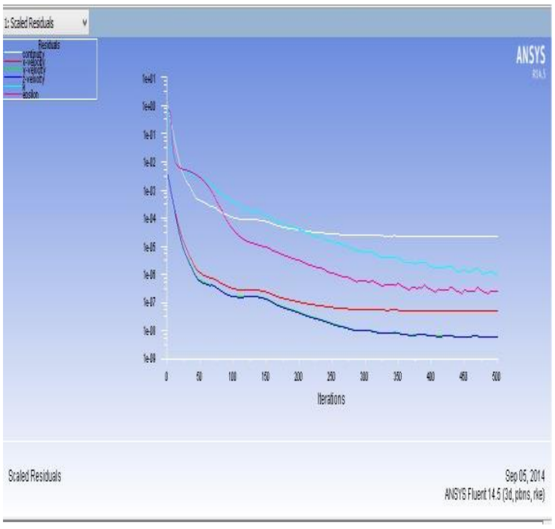


Fig.2.12 (a): Scaled residuals

(b) C_d-1 Convergence history:

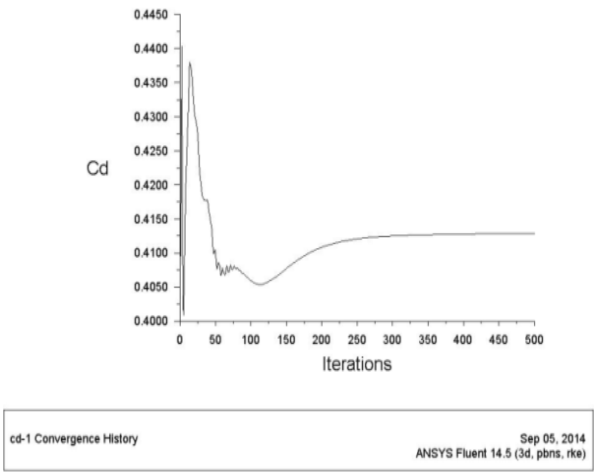


Fig.2.12 (b): C_d-1 Convergence history

(c) C_l-1 Convergence history**:**

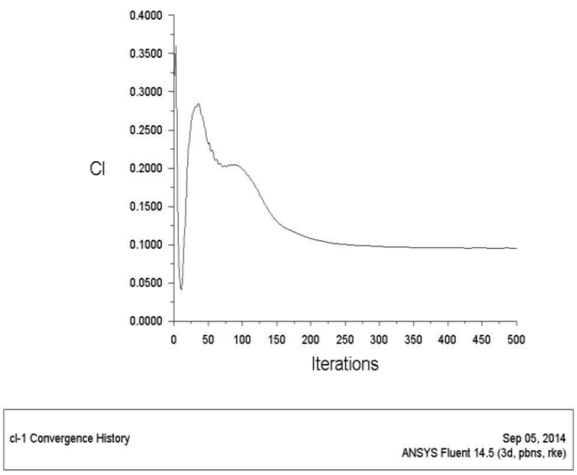


Fig.2.12(c): C_l-1 Convergence history

(d) C_M-1 Convergence history

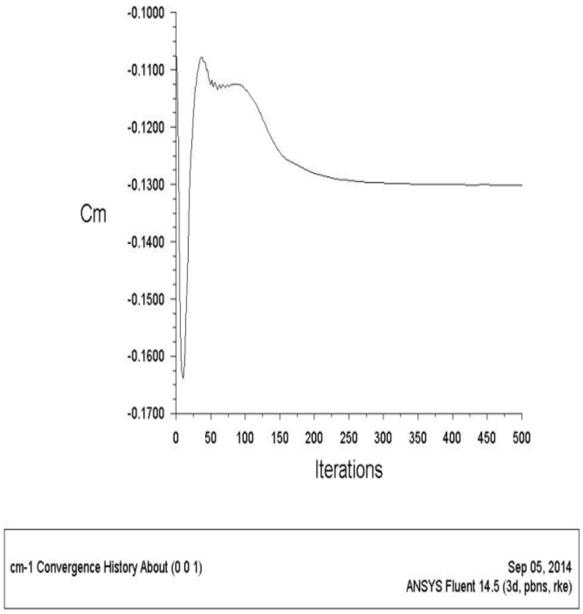


Fig.2.12 (d): C_M-1 Convergence history

(e) Contours of velocity magnitude**:**

Contours of velocity indicate how flow has been developed.

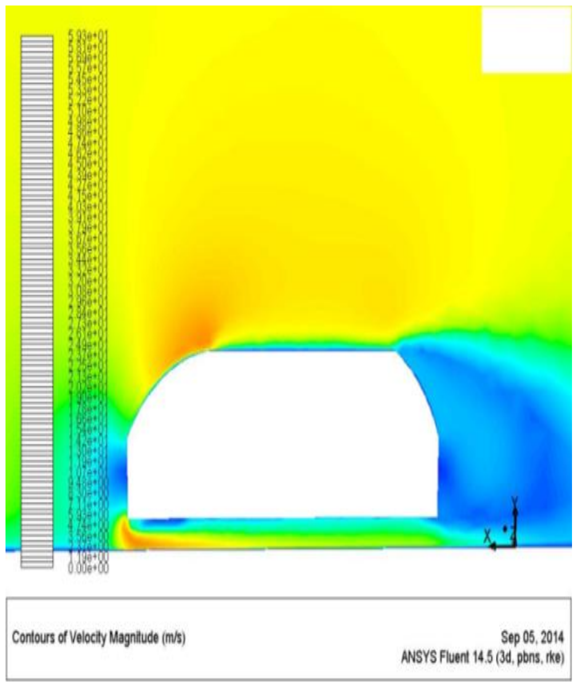


Fig.2.12 (e): Contours of velocity magnitude

(f) Velocity path line:

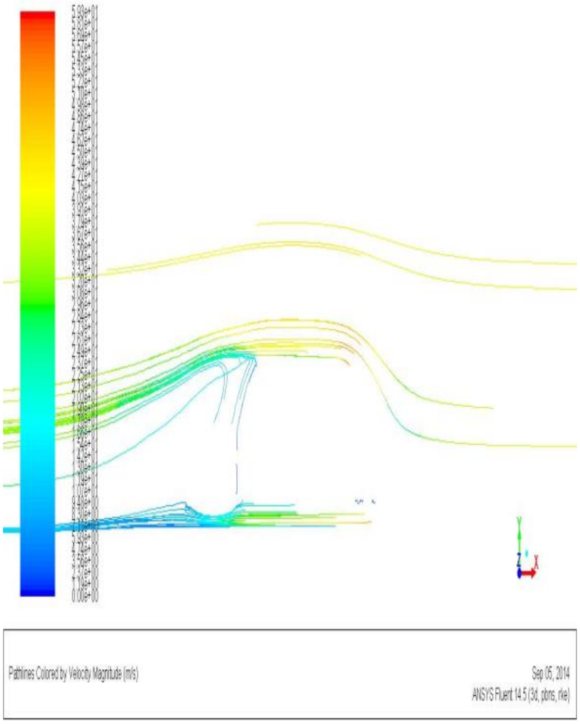


Fig.2.12 (f): Velocity path line

(g) Velocity path line in max view:

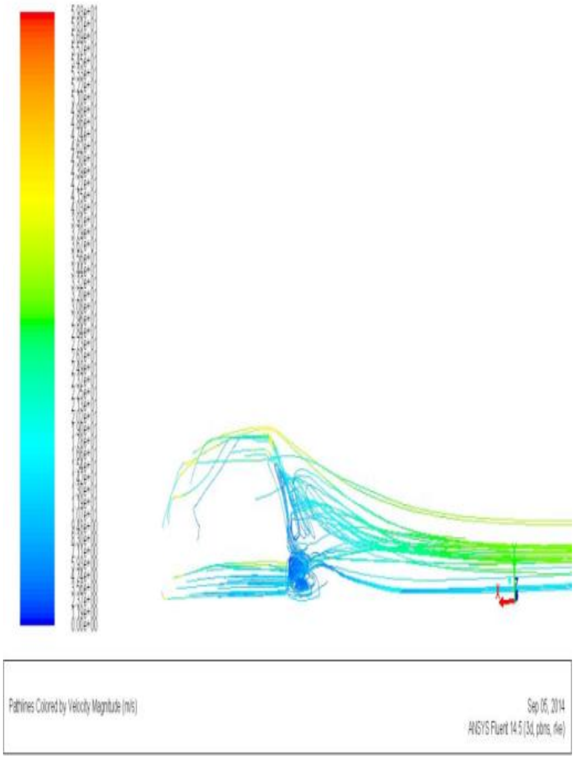
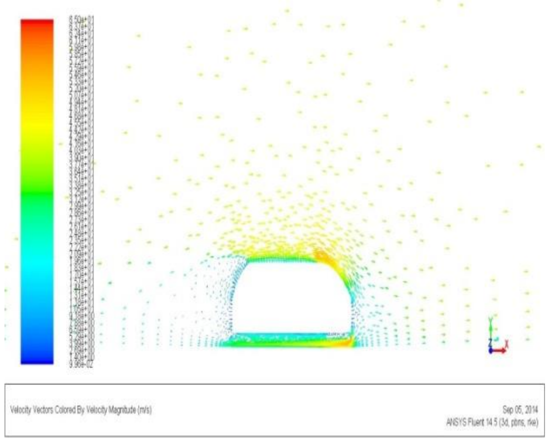


Fig.2.12 (g): Velocity path line in max view

(h) Velocity vectors:



Air flow in negative x-axis with a velocity of 40 m/s.

Fig.2.12 (h): Velocity vectors

(i) Contours of static pressure:

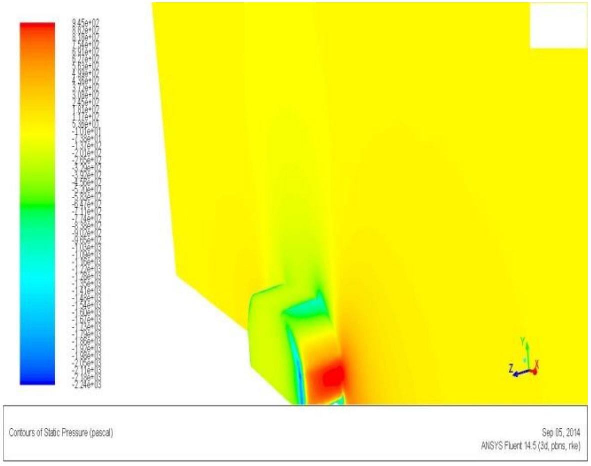


Fig.2.12 (i): Contours of static pressure

3 DATA COMPARISON

Comparison between standard and calculated values is given below:

Table 3(a): Standard value

Standard value		
No	**Vehicle name**	**Drag coefficiet(C_d)**
1	Lotus Seven	0.65 to 0.75
2	Hummer H2	0.57
3	Range Rover Classic	0.45
4	Honda Insight Hybrid	0.32
5	Koenigsegg Agera (R)	0.33 to 0.37

Table 3(b): Calculated value

Calculated value		
No	**Iteration period**	**Drag coefficient (C_d)**
1	1-50	0.42
2	50-100	0.41
3	100-150	0.29
4	150-200	0.308
5	200-250	0.309

4. RESULTS

From Table 3(a), we see that the average modern automobile achieves a drag coefficient of C_d=0.32 to 0.8. And our computed values from Table 3(b) fluctuate from .29 to .45 with respect to velocity. It is assured that the drag coefficient of a vehicle is affected by the shape of body of the vehicle.

Our simulation result concludes:

- Drag coefficient is convergence.
- Lift coefficient is convergence.
- Scaled residuals are convergence.
- Drag coefficient of C_d is varied from 0.29 to 0.42 which is quite similar of the present drag coefficient of the vehicles in the modern world.

5. CONCLUSION

The results from the study indicate that optimization of vehicle body is a viable option for the considerable reduction of fuel consumption and also for the improvement of comfort characteristics. Contrary to the usage of eco-friendly fuel for vehicle in the presence of rising price of itself where the aerodynamic design has become essential to maintain the proper vehicle shape and also for the reduction of the drag co-efficient. Our project estimated results might be acceptable as the values are tensed to convergence and varied within an optimum region. Since lower value of drag co-efficient reduces the possibility of wakes and separation of fluid flow around the vehicle body, our computed values as well as our design may be implemented practically to develop the exterior shape of the vehicle.

In addition the present intercity buses of Bangladesh does not maintained proper aerodynamic design, that is why we have to use the standard values of drag coefficient of foreign vehicles. So we think if the result of our project can be calculated practically in wind tunnel, then it will be more efficient for the further development.

Since the drag force co-efficient is convergence from the simulation, so it will result in speedier and less fuel consumable eco-friendly vehicle of present times.

6. REFERENCES

1. ANSYS ® Workbench ™ 2.0 Framework, Version 14.5.0, Getting Started Guide", ANSYS, Inc., April, 2012.
2. Aerodynamic analysis of drug reduction devices on the underbody for SAAB 9-3 by using CFD.. Chalmers University of technology, Sweden. And Jesper Marklund. Minimize vortex drag of a passenger car 2010.
3. Fluid mechanics, Fundamentals and applications, Second edition, By YUNUS A. CENGEL, JOHN M. CIMBALA, Page no: 854 to 855.
4. The Pioneers: Aviation and Airmodelling". Retrieved 26 July 2009. "Sir George Cayley, is sometimes called the 'Father of Aviation'. carrying glider."
5. Fluid mechanics, Fundamentals and applications, Second edition, By YUNUS A. CENGEL, JOHN M. CIMBALA, Page no: 586 to 597.
6. Http://www.Solidworks.com (15-08/2014)

7. NOMENCLATURE

Symbol	**Meaning**	**Unit**
C_d	Drag Coefficient	Dimensionless
μ	Viscosity	(kg/m.s)
A	Area	(m^2)
ρ	Density	(Kg/m^3)
υ	Kinematic viscosity	(m^2/s)
V	Speed	(m/s)
F_d	Drag force	(N)
g	Gravitational acceleration	(m/s^2)
C_l	Lift Coefficient	Dimensionless