Paper ID: FM-229

Design optimization & Numerical analysis of different bus geometry to reduce air drag co-efficient and fuel consumption

Md. Yasin Ali*[1], Shabnoor Mahmud Joty[2], Dr. Mohammad Ilias Inam[3]

[1,2,3]Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh.

**E-mail: yasin079mekuet@gmail.com*

Abstract

The purpose of this study is to enrich fuel economy by reducing air drag. All over the world bus is used as a major medium of transportation. For long distance journey, people use buses. All the design of bus available today is a rectangular body having a blunt frontal shape that causes a huge air drag on the bus. From previous studies, it was found that about 60%-70% of engine power is used to overcome the drag force of presently running model of buses and rest is to overcome the frictional force between road and tire. The frictional force is important for balancing of a vehicle but drag force is for vein. In this research work, a modification of bus geometry is shown to reduce drag force coefficient as well as fuel consumption. A numerical analysis using Ansys (fluent) was incorporated in this work to show various aerodynamic property of currently running model of bus and modified model of bus. Velocity is considered 100km/h or 27.77m/s. From this study, an optimum frontal shape of bus geometry was suggested. It was found that air drug force reduced 15.36% of currently running model of bus using the modified model. As a result, a huge annual fuel saving, as well as carbon emission, is now possible. This design is more helpful for long distance coach bus such as Dhaka-Chittagong coach in Bangladesh that runs at a high speed but would be less helpful for local or town service that runs at very lower speed and short distance. Fuel economy is also very important for bus.
Bus run on underground oil, this oil is limited in world and also costly. Burning this fuel also emit huge carbon-di-oxide and pollute environment. Modifying the design of bus fuel consumption as well as carbon emission can be minimized to get a better fuel economy. This will also reduce the cost of the journey.

Keywords: fuel economy, air drug, blunt frontal shape, carbon emission, Ansys (fluent).

1. Introduction

Bus is a very popular transportation in all over the world. In Bangladesh it is the most popular public transport. People use bus to travel almost every corner of the country. For highway bus services, fuel is a very important thing. In general, the size of a diesel engine for a high-speed express coach is in the range of 12 to 16 liters in displacement volume and 800 to 1200 kW in brake power. Its fuel consumption rate is very high. It is almost 2.5 to 3.5 km/liter at the cruising speed (100 km/h) on a highway. From the previous study of automotive aerodynamics, it was shown that almost 70% or more brake power is wasted to overcome the aerodynamic drag that generate on the front and rear side of the vehicle at 100 km/h.

In this research paper, a design of a high speed express coach was studied to understand the aerodynamic drag and fuel consumption, compared to the conventional shape of the coach. For this, one of the most popular model of a high-speed express coach in Bangladesh was selected and modified its surface structure with the streamline design concept. For a streamlined design of the model bus, the frontal configuration of the model vehicle was modified to reduce the stagnation pressure drag. Five models of the streamline designed vehicle were developed with the combination of modified frontal area shown in Figure 5 and compared their aerodynamic performances; drag coefficient, to the original model's.

When a bus runs on a highway at cruising speed, it has to overcome two types of external forces to maintain its speed. These are the rolling resistance on the road and aerodynamic drag on the body. Rolling resistance has an effect on driving stability but aerodynamic drag is directly related to fuel consumption.

2. Aerodynamic characteristics and geometry of the model coach

2.1 Aerodynamic characteristics of the model coach

Aerodynamic drag is a very important parameter to be considered for a high-speed running vehicle. High drag force means more driving power is required to maintain the required speed.

Figure 1 shows complicated airflow phenomenon around a running coach. As shown in the figure, incoming air stream hits on the front side of the vehicle and the kinetic energy turns into stagnation pressure which is main source to form drag. The second part of the total drag is seriously formed on the rear–side of the body due to the vortex generated. Air stream is separated at the end of the roof and the flow turns into the circulating flow due to viscosity effect on the boundary layer. It contributes to increase the vortex intensity at the rear side and increase the induced drag of the vehicle.

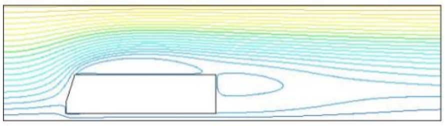


Figure 1. Stream line of original bus

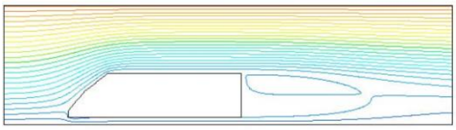


Figure 2. Stream line of modified bus

There is a downward force generated due to the pressure difference between top and bottom side of a vehicle. It affects to engine power required because the downward force is linearly proportional to the rolling resistance generated on the tires of the vehicle.

As mentioned above, pressure drag, induced drag and lift or downward forces are three important physical forces that can be generated on a running vehicle. Therefore, a consideration on these factors should be given for the optimum aerodynamic design of high-speed coach.

2.2 Geometry of the model coach

One of the most popular high-speed express coach in Bangladesh was taken for this study. The model bus (HINO-RM2) is used.

Figure 3. Currently running bus (Hino RM2)

Figure 4. One of the Modified Bus (Model 5)

The dimensions and configuration of the bus is given in Figure 1. The wind shield angle (θ) is about 74 degree and height/weight ratio is 1.406. In the modified bus the angle of frontal wind shield have been divided into two parts. The first angle θ_1 was varied from 50 degree to 70 degree with a general interval of 5 degree and the second angle θ_2 was varied from 39 degree to 27 degree with a general interval of 3 degree. On the basis of result the model 5 was chosen having $\theta_1 = 65°$ and $\theta_2 = 30°$.

2.3 Streamlined designed of the front-side of the model coach

Approximately 70% of engine power losses due to aerodynamic drag. The most serious aerodynamic resistance that is 50% of the total road-load power is formed on the frontal side of the body due to the stagnation of air flow on the surface. The induced drag due to the vortex at rear is about 20% of the total engine power at 100 km/h.

In this study two design concepts were introduced to a high-speed coach. A streamlined design of the frontal side of the vehicle to reduce drag and a rear-spoiler to reduce turbulent kinetic energy formed at rear of it.

Figure 3 shows an example of streamlined design of a high-speed coach.

Comparing to the wind shield angle (θ=74 degree) of the original model bus shown in figure 3, the angle is modified into two steps in a new model. The wind shield angle (θ_1) and the roof angle (θ_2) set up 65 degree and 30 degree respectively.

3. Numerical Scheme and its conditions

In this study, FEM (Finite Element Method) scheme was employed to simulate flow phenomenon around a high-speed bus traveling on a road at a constant speed without the side wind effect. Therefore, the airflow field of the control volume is reasonably assumed to be;

- Quasi- 2D flow
- Turbulent flow
- Incompressible flow
- Steady state flow

Ansys fluent is used for the simulation of this study. 2-D Navier-Stockes equations were solved with standard (k-ϵ) model. The process was assumed as steady state and adiabatic process, the energy equation was not required to be solved in the numerical calculation. The turbulent no-slip condition near solid boundary was modeled by the logarithmic law. The time differencing has been fully implicit backward while advection terms are hybrid differenced. Conjugate gradient techniques for pressure corrections in transport equations has been incorporated and 'SIMPLE' algorithm has been employed for the velocity and pressure coupling in this application.

3.1 Governing Equations

The basic fluid dynamics in the control volume are based on Nervier-Stocks equation that are comprised of equations for conservation of mass and momentum and given as,

Continuity equation,

$$\frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial y_i} + \frac{\partial U_k}{\partial z_i} = 0 \qquad (1)$$

Momentum equation

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j}\left(U_i U_j\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[v\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \overline{u_\iota u_J}\right] - g_i \qquad (2)$$

Standard κ-є turbulent model

$$\frac{\partial}{\partial x_i}\left(U_j k\right) = \frac{\partial}{\partial x_i}\left[\left(v + \frac{v_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G - \varepsilon \qquad (3)$$

Energy dissipation equation

$$\frac{\partial}{\partial x_i}\left(U_j \varepsilon\right) = \frac{\partial}{\partial x_i}\left[\left(v + \frac{v_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{\varepsilon 1} G - C_{\varepsilon 2}\varepsilon\right) \qquad (4)$$

Where $-\overline{u_\iota u_J} = v_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}k\delta_{ij}$ and $G = -\overline{u_\iota u_J}\frac{\partial U_i}{\partial x_j}$, $v_t = C_\mu \frac{k^2}{\varepsilon}$

$\left(C_\mu = 0.09, C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.0\right)$

3.2 Numerical grid of physical model and its conditions

Ansys is used as the numerical computation platform for this study. The model is created in Ansys. A 2-dimensional geometry of the model vehicle was modeled.

Figure 4 shows a typical numerical grid of the physical domain with the model bus without a rear-spoiler. The optimum grid size was decided to (380x1400) from the prior validation test of numerical grid.

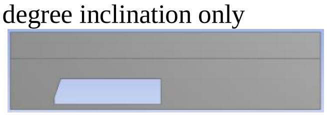
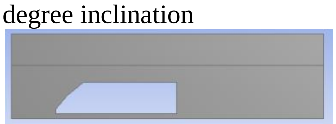
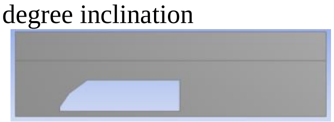
Boundary and initial conditions of the calculation are,

(1) Velocity boundary condition at the inlet of the control volume; $U_{car} = 100\ Km/h$
(2) Constant pressure boundary condition at the exit of the control volume
(3) No-slip condition at the surface of the model bus
(5) Potential flow conditions on the open surface of the control volume; east and west sides and top surface.

3.3 Major design parameters and operating range

Frontal shape of bus is important design points to improve the aerodynamic performance of the model bus. Table 1 shows the major design parameters of the model bus and its running condition for the numerical study.

Table 1. Specification of the model and its speed.

Model No.	Specification of Fairing
Model-1	Original model (HINO RM2) with 74 degree inclination only 
Model-2	Streamlined Model Bus with 50&39 degree inclination 
Model-3	Streamlined Model Bus with 55&36 degree inclination 

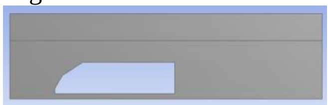
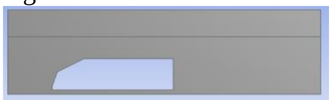
Model-4	Streamlined Model Bus with 60&33 degree inclination 
Model-5	Streamlined Model Bus with 65&30 degree inclination 
Model-6	Streamlined Model Bus with70&27 degree inclination 
Speed range of the model bus: U_{car} = 100(km/h)	

Figure 5. shows the various configurations of the model buses.

4. Aerodynamic performance analysis of the model coach

For the analysis of aerodynamic performance of the model bus on its running condition, the static pressure distribution on the surface of the vehicle was analyzed. The drag and its coefficient (C_D) were calculated from the equations given below.

4.1 Drag force (F_d) and the coefficient (C_d)

$$\sum F_D = \sum P_{par} A_{par} \sin\theta \qquad (5)$$

$$\sum F_D = C_D \frac{1}{2} \rho_{air} \sum A_{y=dir} {V_{bus}}^2 \qquad (6)$$

$$C_D = \frac{2 \sum F_D}{\rho_{air} A_{y=dir} V_{bus}} \qquad (7)$$

where $A_{y\text{-}air}$ is the projection area of the model bus on (x-z) plane; Model-1=0.1mm^2, Model-2=0.1mm^2.

4.2 Power saving

$$P_{sav} = \left(F_{D_{model-0}} - F_{D_{model-x}}\right) \times V_{bus} \qquad (8)$$

Where, P_{sav} is the brake power saved (kW), $F_{D_{model-0}}$ is the total drag force (kN) of model-0, $F_{D_{model-x}}$ is total drag force (kN) of model-1 to 5 and V_{bus} is the velocity of the model bus (m/s)

4.3 Fuel saved by the reduction of drag force

$$\dot{m_{fuel}} = \frac{Power_{saved}}{Q_{LHV} \times \rho_{fuel} \times \eta_{engine}} \qquad (9)$$

5. Result and discussion

In this numerical study, a streamlined body of high-speed coach was examined to see its effect on the driving economy. The assumption is that the model bus runs straightforward at a constant speed with no side-wind. Graph was taken for total pressure both on inclined line one and inclined line two. These two graph are given below.

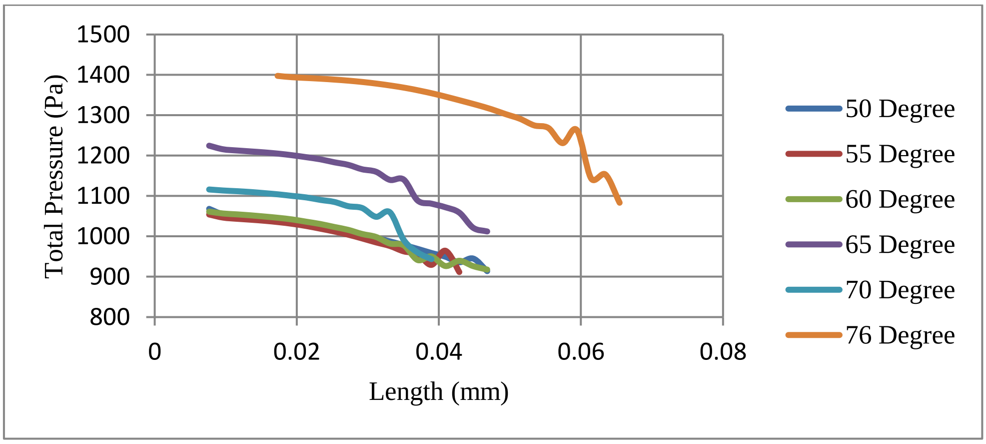


Figure 6. Total pressure along inclined line one.

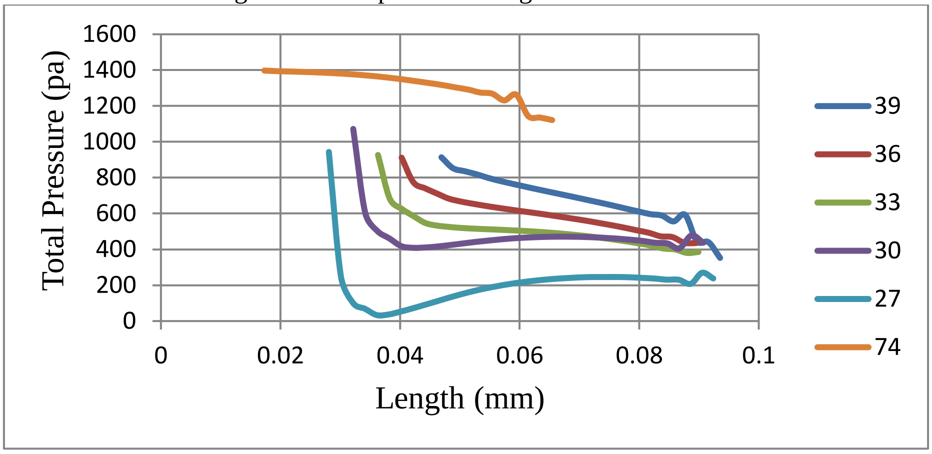


Figure 6. Total pressure along inclined line two.

From this two graph it is clear that total pressure is minimum at angle 55 degree for inclined line one or lower part of frontal shape and at an angle 27 degree with horizontal for inclined line two. Where the pressure is minimum, applying force will also be minimum. So making a bus having a combination of 55 degree first inclination and 27 degree second inclination is the most economic frontal shape of bus.
Again a third graph is shown calculating the total drag force on the inclined wind shield of modified bus geometry.

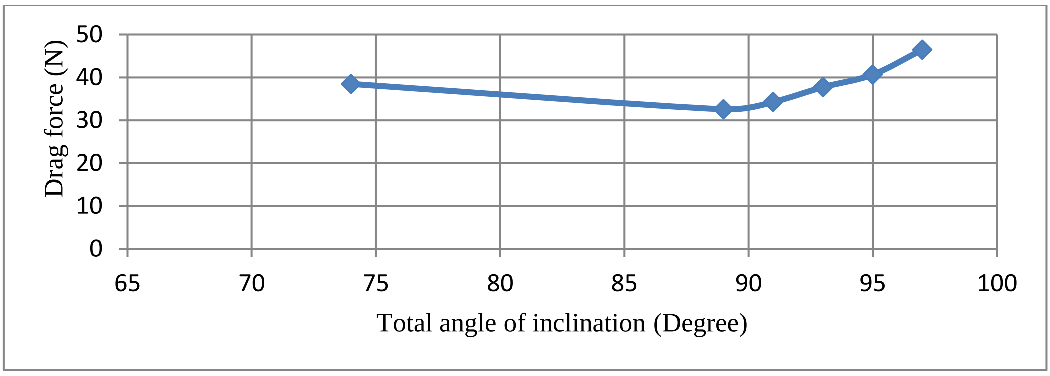


Figure 6. Total angle Vs Drag force for various model of bus

From the graph it is shown that drag force is minimum for total angle of inclination 87 (summation of angle in two phase). So taking this angle is more economic.

6. Conclusion

From this numerical study, it was found that an optimum aerodynamic shape is very important for a high-speed express coach. Drag force on frontal wind shield has been reduced 15.36% using a bus of frontal shape of two different angle in combination. These angles are 55 and 27 degree. Aerodynamic drag reduction also reduces fuel consumption and environment pollution. Therefore, it is important design an optimum aerodynamic shape for high-speed coaches for better fuel economy.

List of nomenclature:

Name of the symbol	Symbol
Car velocity	U_{car}
Total Drag force	F_D
Pressure	P_{Par}
Area	A_{par}
Mass of Fuel saved	M_{fuel}
fuel required	Q
Density of fuel	P_{fuel}
Engine efficiency	$^{n}{}_{engine}$

7. References

[1] Kim, C.H., Yun. D.G. and Lee C.M. : A Numerical Study on the Aerodynamic Effects of a Rear-Side Spoiler on the Driving Stability of a Passenger Car, Journal of SNUT, Vol.47. (1998)
[2] Patankar, S.V. : Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp. (1980)
[3] Douglas, J.F. : Fluid Mechanics, Prentice Hall, p406~447. (2001)
[4] Barnard, R.H : Road Vehicle Aerodynamic Design, Addison Wesley Longman Ltd. (1996)
[5] CHAM Technical Report TR/326 : Concentration Heat And Momentum Limited. (2008)