Effect of Streamlined Design of High-speed Coach on Fuel Economy and Emission

C.H. Kim*

*Seoul National University of Science and Technology, Seoul, Korea
Department of Automotive Engineering, profchkim@seoultech.ac.kr

ABSTRACT

The most important aerodynamic design concept of an automobile body shape is on fuel savings with the reduction of harmful emission, driving stability and aerodynamic noise and vibration. From the previous studies [1]~[3], it was found that fuel savings and emission reduction has more important meanings on the aerodynamic design of heavy-duty vehicles such as buses and trucks but the driving stability and aerodynamic noise problems for the small-size sadan. In this study, a streamlined design concept is applied to the front side of a long-distance high-speed model bus to see its effect on reducing the aerodynamic pressure drag and to a rear spoiler to understand its effect in reducing the induced drag at the rear flow field of the model bus. Computational fluid dynamics (CFD) simulation scheme was incorporated to analyze the variation of aerodynamic effect on the model buses with the change of body configuration. From the study, it was found that the total drag of the conventional high-speed model coach(FX212) was reduced by 27.4% at a speed of 120km/h by applying the streamline-designed body shape, which is equivalent to 17.3 kW in the engine brake power. The resulting fuel saving effects are a reduction in fuel consumption of about 17,045 liters and a reduction in CO_2 emissions of 48.1 tons per year by each bus at the given operating condition.

Keywords: aerodynamic drag, CO_2, fuel economy, road-load power, driving stability, Computational Fluid Dynamics (CFD)

1 INTRODUCTION

Long-distance, high-speed coaches have been played an important role in a inter-city and inter-state public transportation. In general, the size of a diesel engine for a high-speed express coach is in the range of 12~16 liters in engine displacement volume and 800~1,200 kW in engine brake power and its fuel consumption rate is surprisingly high and it is about 2.5~3.5 km/liter (about 30~40 liters/ 100km) at its cruising speed (100km/h) on a highway.

In the previous study of automotive aerodynamic design [3][4], it was known that over 70% of the engine brake power of a vehicle is consumed to overcome the aerodynamic drag formed at the front and rear side of the vehicle at 100km/h.

In this research, an effect of streamlined design of a high-speed, long-distance coach on fuel consumption and carbon dioxide reduction compared to the conventional shape of the coach was studied. For this, one of the popular high-speed express buses in Korean was selected and its body configuration was modified with a streamlined design concept. The frontal side of the vehicle was modified to reduce the stagnation pressure drag and a rear-spoiler was installed to minimize the induced drag at the rear-side of the bus. Five models of the streamline designed vehicles were developed and the aerodynamic performance of each model was compared to that of the original model.

2 AERODYNAMIC CHARACTERISTICS AND GEOMETRY OF THE MODEL COACH

2.1 Aerodynamic Characteristics of the Model Coach

Two directional aerodynamic forces act on a running vehicle; drag and lift (or down force) with no side-wind effect. In general, it is known that drag effects on the driving power required and lift on the driving stability of a vehicle. From the previous study, it was known that the major concept of aerodynamic design of a heavy-duty vehicle is on the reduction of fuel consumption [4] but driving stability for a small size sedan [2].

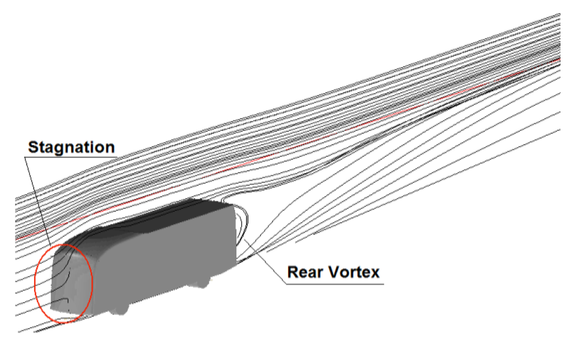

Figure 1: Airflow phenomenon on a high-speed coach.

Figure 1 shows the complicated airflow phenomenon around a running coach on a road. Aerodynamic drag generated on the vehicle is mainly the form or pressure drag on the body. As shown in the figure, the incoming air stream hits on the front-side of the vehicle and the kinetic energy turns into stagnation pressure which is main energy source of the form drag. The second part of the total drag is seriously formed at the rear-side of the body due to the vortex generated. As air stream passes through the roof surface of the vehicle, it 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 generated at the rear side of the body and increases the induced drag of the vehicle.

2.2 Geometry of the Model Coach

The model bus, FX-212(46seats) has been manufactured at Daewoo Bus Corp. Korea since 2008. Dimension and configuration of the model bus is given in Figure 2. The wind shield angle (θ) is about 76degree and height/width ratio is 1.406.

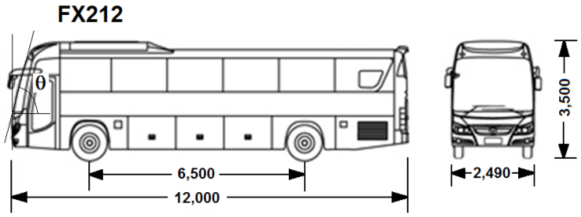

Figure 2: 2-dimensional view of the original model coach and its dimension; θ=76degree.

2.3 Streamlined Design of the Front-side of Model Coach

Figure 3 shows an example of the streamlined design of a high-speed coach with a rear-spoiler. Comparing to the wind shield angle (θ=76 degree) of the original model bus shown in figure 2, the angle is modified into two steps on a new model. The wind shield angle (θ_1) and the roof angle (θ_2) set up 59 degree and 19 degree respectively.

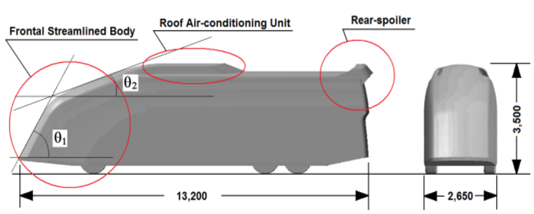

Figure 3: 2-D view of a streamline designed coach with a general shape of rear spoiler; θ_1=59 and θ_2=19 degree.

3 NUMERICAL SCHEME AND ITS CONDITIONS

FVM (Finite Volume Method) numerical scheme was employed to simulate flow phenomenon around a high-speed bus traveling on a level road at a constant speed without side wind. The airflow field of the control volume is reasonably assumed to be ;

- Quasi-3D flow
- Turbulent flow
- Incompressible flow
- Steady state flow

A general-purpose CFD code, PHOENICS (ver.2008) [6] was used for a numerical calculation of turbulent incompressible flow field. 3-D Navier-Stokes equations were solved with standard (κ-ε) turbulent model [5].

3.1 Governing Equations

The basic equations of fluid dynamics in the control volume are based on Navier-Stokes equations 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:

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j}(U_i U_j) = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \overline{u_i u_j}\right] - g_i \qquad (2)$$

Standard κ-ε turbulent model:

Turbulent kinetic energy equation :

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

Energy dissipation equation :

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

where $-\overline{u_i u_j} = \nu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}k\delta_{ij}$, $G = -\overline{u_i u_j}\frac{\partial U_i}{\partial x_j}$,

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}$$

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

3.2 Numerical Grid of Physical Model and Its Conditions

The CAD-to-CFD method [6] in conjunction with orthogonal grid was incorporated for the numerical grid generation in the physical domain in this study. First, a 3-dimensional model vehicle was modeled by Pro-Engineers, 3-D CAD software and transferred the model into to the numerical domain to generate the numerical grid in the rectangular coordinate system.

The optimum grid size of the 3-D model was decided to $(80 \times 141 \times 81)$ from the prior validation test of numerical grid.

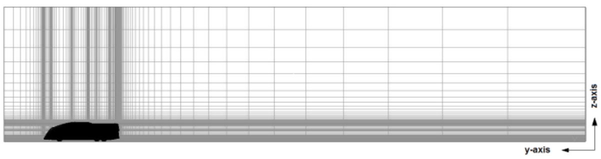

Figure 4 A typical numerical grid of the model bus without a rear-spoiler (80x141x81).

Boundary and initial conditions for the calculation:

- Velocity boundary condition at the inlet of the control volume; U_{car} = 60~120 Km/h
- Constant pressure boundary condition at the exit of the control volume
- No-slip condition at the surface of the model bus
- Moving boundary condition on the ground surface of road
- 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 and configuration of the rear-spoiler are two 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 bus.

Model No.	Specification of Fairing
Model-0	Original model (FX212)
Model-1	Streamlined Model Bus1 without spoiler
Model-2	Streamlined Model Bus1 with spoiler1
Model-3	Streamlined Model Bus2 without spoiler
Model-4	Streamlined Model Bus2 with spoiler1
Model-5	Streamlined Model Bus2 with spoiler2

4 AERODYNAMIC PERFORMANCE ANALYSIS OF THE MODEL COACH

For the analysis of aerodynamic performance of the model bus at its running condition, the static pressure distribution on the surface of the vehicle was analyzed. The drag and its coefficient (C_D) was calculated from the equations given below [7].

Drag force (F_D) and the coefficient (C_D):

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

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

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

where $A_{y\text{-}dir}$ is the projection area of the model bus on (x-z) plane; Model-0=7.233m^2, Model-1~ 5=7.698m^2.

Power saved on the streamlined bus model with respect to the original bus model(Model-0)

$$P_{sav} = (F_{D_{MODEL-0}} - F_{D_{MODEL-X}}) \times V_{bus} \quad (8)$$

where
P_{sav} is brake power saved (kW) and
$F_{D_{MODEL-0}}$ is total drag force (kN) of Model-0 and
$F_{D_{MODEL-X}}$ is total drag force (kN) of Model-1~Model-5 and
V_{bus} is velocity of the model bus (m/s).

Fuel saved by the reduction of the drag force :

$$\dot{m}_{fuel} = Power_{saved} /(Q_{LHV} \times \rho_{fuel} \times \eta_{engine}) \quad (9)$$

where Q_{LHV} (=42.5MJ/kg) is the lower heating value of diesel fuel and ρ_{fuel} (=860kg/m3) is the fuel density at the ambient temperature and η_{engine}(=35%) is the thermal efficiency of a diesel engine [8].

Reduction of carbon dioxide(CO_2) :

CO_2 emission produced from the combustion of diesel fuel can be estimated using the analytical equation given below [9]:

$$CO_{2\ emissions} = AL \times CL \times OF \times 44 / 12 \quad (10)$$

where
AL : amount of combusted fossil fuel, Gg
CL : carbon content of fossil fuel, 0.77ton/kℓ, (fraction)
OF : oxidation factor for fossil fuel, (fraction)

Annual operating condition of the model bus :

The operating condition of the original model bus (Model-0) is given below and the same condition is applied to the new designed model buses (Model-1~Model-5) to estimate annual effect on the fuel economy and the reduction of fuel consumption and carbon dioxide (CO_2).
(offered by an express bus company in Korea)

- 10-hour driving per day at its cruising speed
- 300 duty days per year

5 RESULTS AND DISCUSSION

Figure 5 shows the averaged C_D of each model bus. The C_D of the original bus (Model-0) is 0.457 and the lowest C_D is 0.332 on Model-3. The model bus (Model-5) with Spoiler2 shows lower C_D than the model buses with

Spoiler1 but still higher value of C_D than the Model-1 and Model-3 that has no rear-spoiler.

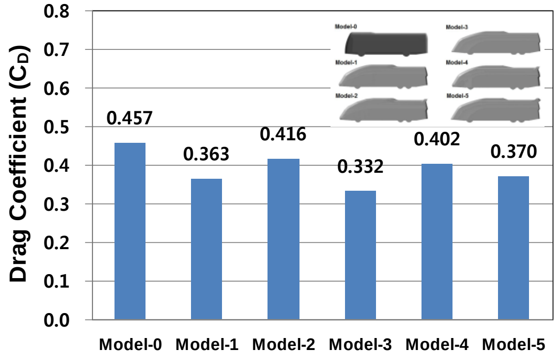

Figure 5: Comparison of the averaged drag coefficient (**C_D**) of the model vehicle.

Annual effect of the streamline-designed bus on fuel savings and CO_2 reduction is analyzed in Figure 6 and Figure 7. As shown in Figure 6, annual fuel savings of the streamlined model bus (Model-3) is 14.6kliters/yr at 120km/h compared to the original bus (Model-0). The annual reduction of CO_2 is estimated to be 41.2tons per year on Model-3 at 120km/h compared to the original bus (Model-0).

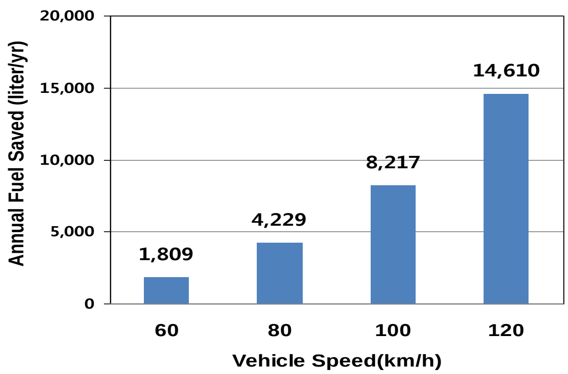

Figure 6: Annual fuel savings of Model-3 compared to the original bus (Model-0).

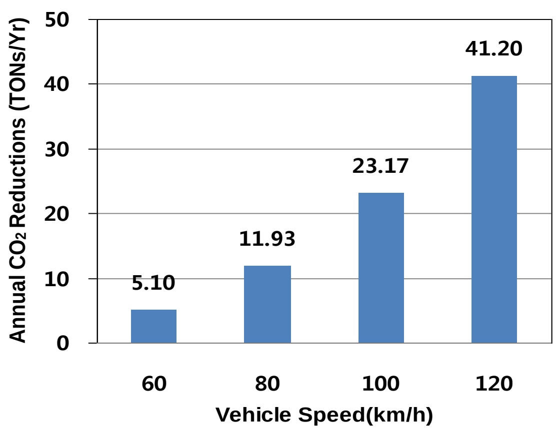

Figure 8: Annual CO_2 reduction of Model-3 compared to the original bus (Model-0).

6 CONCLUSION

Pressure drag on a running vehicle directly affects not only to fuel consumption but also to the generation of carbon dioxide gas from combustion of fuel. Therefore, an aerodynamic design of the long distance, high-speed coach is very important matter for fuel economy in transportation industry and for environment protection.

27.4% of the total drag(C_D) is reduced remarkably on the streamline designed coach (Model-3) compared to the original model (Model-0). In the case of Model-5 which is equipped with Spoiler2 at the rear-side of the coach, C_D is reduced to 19.2% but still shows higher drag coefficient relatively to the Model-3 with no spoiler.

The road-load power is expected to be decreased due to drag reduction on new designed model buses. The maximum engine power saved on Model-3 is about 9.73kW at 100km/h and is almost double (17.31kw) at 120km/h.

Due to the savings of the engine brake power with the drag reduction, the expected fuel savings and CO_2 reduction is about 14,610 liter/year and 41.2ton/year respectively at 120km/h with the Model-3 at the given operating condition.

ACKNOWLEDGEMENT- This research work is supported by the research fund of Seoul National University of Science & Technology, Seoul, Korea, 2009 and many thanks for the help.

REFERENCES

[1] R.H. Barnard, “Road Vehicle Aerodynamic Design,” Addison Wesley Longman Ltd., 1996.
[2] C.H. Kim, D.G. Yun and C.M. Lee, “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.
[3] C.H. Kim, C.B. Youn, “Aerodynamic Effect of Roof-Fairing System on a Heavy-duty Truck,” IJAT, Vol.6, No.3, p221-227, 2005.
[4] C.H. Kim, “An Effect of Roof-Fairing and Deflector System on the Reduction of Aerodynamic Drag of a Heavy-Duty Truck,” Trans of KSAE, Vol.14, No.2, p194-201, 2006.
[5] S.V. Patankar, “Numerical Heat Transfer and Fluid Flow,” Hemisphere Publishing Corp.’ 1980.
[6] CHAM, “Technical Report TR/326,” Concentration Heat And Momentum Limited, 2008.
[7] J.F. Douglas, “Fluid Mechanics,” Prentice Hall, p406~447, 2001.
[8] J.B. Haywood, “Internal Combustion Engine Fundamentals,” McGraw-Hill, 1989.
[9] Guendehou, S. : Draft 2006 IPCC Guidelines for National Greenhouse Gas Inventories, p5.10. (2006)