A Streamlined Design of a High-Speed Coach for Fuel Savings and Reduction of Carbon Dioxide

Chul-Ho KIM 1)

1) Department of Automotive Engineering, College of Engineering, Seoul National University of Science & Technology, 172 Kongnung-dong, Nowon-gu, Seoul, 139-743, Korea (E-mail: hokim@snut.ac.kr)

ABSTRACT: Numerous aerodynamic designs of high-speed coaches have been made to reduce aerodynamic drag for lower fuel consumption and to keep the driving stability of the vehicle on a highway. However, the external body shape of a long-distance, high-speed coach manufactured around the world is in a rectangular shaped blunt body. With this conventional body configuration of the bus, it is not easy to have an optimum shape of the bus with the minimum aerodynamic drag. From the previous study, it was found that total aerodynamic drag on a running vehicle is comprised of around 70% in pressure drag by stagnation on the front-side of the vehicle and 30% in induced drag by vortex at the rear-side of it. In this study, a streamlined design concept was incorporated to the front-side of a long-distance model bus to see its effect on the reduction in aerodynamic pressure drag and a general type of RGV (rear guide vane) was applied to see its effect on the reduction in the induced drag at the rear flow field of the model bus. Computational fluid dynamics (CFD) method 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 27.4% of the total drag of the original bus (Model-0) was reduced on the model-3 that is equivalent to 17.3kW of an engine brake power at 120km/h in speed. The annual economical effect is the reductions in about 14,610 liters of fuel and 41.2 tons of CO₂ per year by each car at the assumed operating condition.

KEY WORDS: (Standardized) heat · fluid, aerodynamic performance/aerodynamic noise, computational fluid dynamics, algorithm/modeling [D1]

1. Introduction

Bus has been used as one of the major public transportation systems in society. Especially for a high-speed coach, it has been played very important role in a long-distance transportation system connecting city-to-city and state-to-state. 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 1,200 kW in brake power. Its fuel consumption is surprisingly high and the rate is about 2.5~3.5 km/liter at the cruising speed (100km/h) on a highway. From the prior study of automotive aerodynamics, it was known that over 70% of the brake power of a vehicle engine is consumed to overcome the aerodynamic drag generating on the front and rear side of the vehicle at 100km/h.

In this research, a streamlined design of a high-speed, long-distance coach was studied to understand its effect on fuel consumption and carbon dioxide reduction compared to the conventional shape of the coach. For this, one of the popular models of high-speed express bus in Korean was selected and modified its surface structure with the streamlined 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 and a rear-spoiler was installed to minimize the induced drag at the rear-side of the bus. Five models of the streamline designed vehicle were developed with the combination of the modified frontal area and rear spoiler shown in Figure 5 and compared their aerodynamic performances; drag coefficient (CD), to the original model's.

When a vehicle runs on a road at constant cruising speed, it has to overcome two types of external force to maintain its speed. These are the rolling resistance on a road and aerodynamic drag acting on the body. In the case of rolling resistance, it has an effect on the driving stability of the vehicle but the aerodynamic drag is directly related to the fuel consumption of the vehicle engine (1-3).

In this study, two different models of streamline-designed bus with two types of the rear-spoiler were developed and examined numerically to have their aerodynamic characteristics and compared the results to the original bus models(Model-0). The speed of the model vehicle was varied from 60 km/h to 120 km/h.

2. Aerodynamic Characteristics and Geometry of the Model Coach
2.1. Aerodynamic Characteristics of the Model Coach

There are two kinds of aerodynamic force acting 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 and driving stability for a small size sedan.

Figure 1 shows complicated airflow phenomenon around a running coach on a road. Aerodynamic drag generated on the vehicle is mainly form or pressure drag on the body. 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 energy source of the form drag. The second part of the total drag is seriously formed on 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.

The downward force on a running vehicle is due to the pressure difference between top and bottom side of a vehicle. It affects to engine power required by the coach 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 the three important physical forces that can be generated on a running vehicle. Therefore, a carefully consideration on these factors should be given for the optimum aerodynamic design of a high-speed coach.

2.2. Geometry of the Model Coach

One of the general models of high-speed coach operated in public transportation industry in Korea was incorporated for this study. The model bus (FX-212, 46seats) is manufactured in Daewoo Bus Corp. Korea. The dimension and configuration of the model bus is given in Figure 2. The wind shield angle (θ) is about 76 degree and height/width ratio is 1.406.

2.3. Streamlined Design of the Front-side of the Model Coach

Approximately 70% of the engine brake power is consumed to overcome aerodynamic drag generated on the frontal and rear-surface of a vehicle body. The most serious aerodynamic resistance on a vehicle that is about 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 the rear of the vehicle should be considered as well because it is also about 20% of the total engine power at 100km/h.

In this study, two design concepts were introduced to a high-speed coach; a streamlined design of the front side of the vehicle to reduce profile or pressure drag and a rear-spoiler to minimize the turbulent kinetic energy formed at the rear of it.

Figure 3 shows an example of 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 in a new model. The wind shield angle (θ₁) and the roof angle (θ₂) set up 59 degree and 19 degree respectively.

3. Numerical Scheme and Its Conditions

In this study, FVM (Finite Volume 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-3D flow
- Turbulent flow
- Incompressible flow
- Steady state flow
The general-purpose CFD code, PHOENICS (ver.2008), was used for a numerical calculation of the turbulent incompressible flow field. 3-dimensional Navier-Stokes equations were solved with standard ($\kappa$-$\varepsilon$) turbulent model\(^6\). Since 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. 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\(^9\) has been employed for the velocity and pressure coupling in this application.

### 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,

1. **Continuity equation**
   \[
   \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} = 0 \quad (1)
   \]

2. **Momentum equation**
   \[
   \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} - \rho g_i \quad (2)
   \]

3. **Standard $\kappa$-$\varepsilon$ turbulent model**
   - Turbulent kinetic energy
     \[
     \frac{\partial (U_j \kappa)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \kappa}{\partial x_j} \right] + G - \varepsilon \quad (3)
     \]
   - Energy dissipation equation
     \[
     \frac{\partial (U_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_\mu G - C_{\varepsilon_2} \varepsilon) \quad (4)
     \]

   where 
   \[
   \begin{align*}
   -u_i u_j & = \nu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \\
   G & = -u_i u_j \frac{\partial U_j}{\partial x_i}, \\
   \nu & = C_\mu \frac{k^2}{\varepsilon}
   \end{align*}
   \]

   \[
   \begin{align*}
   C_\mu = 0.09, \\
   C_{\varepsilon_1} = 1.44, \\
   C_{\varepsilon_2} = 1.92, \\
   \sigma_k = 1.0, \\
   \sigma_{\varepsilon} = 1.0
   \end{align*}
   \]

### 3.2. Numerical Grid of Physical Model and Its Conditions

The CAD-to-CFD method\(^{10}\) in conjunction with orthogonal grids was incorporated for the numerical grid generation in the physical domain in this study. First, a 3-dimensional geometry of the model vehicle was modeled by Pro-Engineers, 3-dimensional CAD software and transferred the model into to the numerical domain to generate the numerical grid in the rectangular coordinate system.

![Fig. 4 A typical numerical grid of the model bus without a rear-spoiler (80x141x81).](image)

Figure 4 shows a typical numerical grid of the physical domain with the model bus without a rear-spoiler incorporated for this numerical study. 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.

### Boundary and initial conditions of the calculation:

1. **Velocity boundary condition** at the inlet of the control volume; \(U_{\text{in}} = 60\text{~to~}120 \text{~km/h}\)
2. **Constant pressure boundary condition** at the exit of the control volume
3. **No-slip condition** at the surface of the model bus
4. **Moving boundary condition** on the ground surface of road
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 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.

| 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              |

| Speed range of the model bus: \(U_{\text{in}} = 60, 80, 100, 120 \text{~km/h}\) |

Table 1 Specification of the model bus and its speed.

Figure 5 shows the various configurations of the model buses. In the case of Model-1 and 2, the roof air-conditioning unit covers the whole width of the model bus and the half of the width is covered in Model-3~Model-5 shown in Figure 6.
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\(^{(5)}\).

\[ \sum F_D = \sum P_{par} A_{par} \sin \theta \]  
(5)

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

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

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

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

\[ P_{sav} = (F_{D_{MODEL-0}} - F_{D_{MODEL-A}}) \times V_{bus} \]  
(8)

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

(3) Fuel saved by the reduction of the drag force

\[ m_{fuel} = Power_{saved} / (Q_{LHV} \times \rho_{fuel} \times \eta_{engine}) \]  
(9)

where $Q_{LHV}$ (\(=42.5\)MJ/kg) is the lower heating value of diesel fuel and $\rho_{fuel}$ (\(=860\)kg/m\(^3\)) is the fuel density at the ambient temperature and $\eta_{engine}$ (\(=35\%\)) is the thermal efficiency of a diesel engine\(^{(6)}\).

(4) Reduction of carbon dioxide (CO\(_2\))

\[ CO_2 \text{ emissions} = AL \times CL \times OF \times 44 / 12 \]  
(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)

(5) 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 environment by the reduction of fuel consumption and carbon dioxide (CO\(_2\)).

Annual operating condition of the model bus, (offered by an express bus company in Korea)

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

5. Results and Discussion

In this numerical study, a streamlined body of high-speed coach was examined to see its effect on the driving economy and environmental effects especially on the reduction of carbon dioxide. The initial assumption of this study is that the model bus runs straightforward at a constant speed with no side-wind.

Figure 7 shows variation of the aerodynamic drag coefficient of each bus model with the change of vehicle speed.
The drag coefficient does not change much at each model with the vehicle velocity. The models without the rear spoiler (Model-1, Model-3) show lower drag coefficient than the models with the spoiler. Model-5 with new type of spoiler (Spoiler2) has lower $C_D$ than the models (Model-2, Model-4) with the conventional rear-spoiler (Spoiler1) but still has higher $C_D$ than the models with no spoiler. The lowest $C_D$ is obtained by Model-3 which has the streamlined design in front with no rear spoiler.

The averaged $C_D$ of each model bus is compared in Fig. 8. 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 withSpoiler1 but still higher value of $C_D$ than the Model-1 and Model-3 that has no rear-spoiler.

As shown in Figure 9, the model with no rear-spoiler (Model-1, Model-3) shows over 20% reduction in $C_D$ compared to the original model. Compared to Model-1, Model-3 that the roof air-conditioning unit covers the half length in width direction of the model bus shows about 7% more reduction in $C_D$. Model-5 with Spoiler2, has about 7% more reduction compared to Model-4 with Spoiler1. It means that Spoiler2 has better effect on the reduction of the induced drag at the rear of the bus than Spoiler1.

Now, the best model (Model-3) is analyzed comparing to the original model (Model-0) to understand its effect on fuel economy and environmental protection from the automobile emissions.

The total $C_D$ is reduced about 27.4% in Model-3 compared to the original model (Model-0) as shown in Fig. 10.
Model-3 saves 17.31 kW of the engine power at 120 km/h comparing to the original bus. In this operating condition, the fuel saving is expected to be 4.87 liters/hr as shown in Fig. 12.

Now, the annual effect of the streamline-designed bus on fuel savings and CO\textsubscript{2} reduction is analyzed in Fig. 13 and Fig. 14. As shown in Fig. 13, annual fuel savings of the streamlined model bus (Model-3) is 14.6 liters/yr at 120 km/h compared to the original bus (Model-0). The annual reduction of CO\textsubscript{2} is estimated to be 41.2 tons per year on Model-3 at 120 km/h compared to the original bus (Model-0).

From the numerical study, it was found that a streamlined design of a high-speed, long-distance bus is very important to reduce aerodynamic drag of the bus. 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 to study for fuel economy in transportation industry and for environment protection.

(1) 27.4\% of the total drag(C\textsubscript{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\textsubscript{D} is reduced to 19.2\% but still shows higher drag coefficient relatively to the Model-3 with no spoiler.
(2) The road-load power is expected to be decreased due to the drag reduction on the 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.

(3) Due to the savings of the engine power with the drag reduction, the fuel consumption is decreased and CO₂ emission is reduced as well. The annual effect on the reduction of fuel and CO₂ is about 14,610 liter/year and 41.2ton/year respectively at 120km/h with the Model-3 at the given operating condition.

(4) Model-5 with the Spoiler2 has less effect on the total drag reduction compared to Model-3, therefore a further study is needed to have an optimum design of the rear spoiler of the bus model.

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) Barnard, R.H : Road Vehicle Aerodynamic Design, Addison Wesley Longman Ltd. (1996)
(2) 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)
(3) Kim, C.H., Youn, C.B. : Aerodynamic Effect of Roof-Fairing System on a Heavy-duty Truck, IJAT, Vol.6, No.3, p221-227. (2005)
(4) Kim, C.H. : 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) Patankar, S.V. : Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp. (1980)
(6) CHAM Technical Report TR/326 : Concentration Heat And Momentum Limited. (2008)
(7) Douglas, J.F. : Fluid Mechanics, Prentice Hall, p406~447. (2001)
(8) Haywood, J.B. : Internal Combustion Engine Fundamentals, McGraw-Hill. (1989)
(9) Guendehou, S. : Draft 2006 IPCC Guidelines for National Greenhouse Gas Inventories, p5.10. (2006)
(10) Kim H. C., Ha J. H. and Kim C. H. : A Numerical Study on the Aerodynamic Effect of a Rear-side Guide Vane on Recreational Vehicle, Proceedings of KSAE Autumn Conference, Vol.2, p894-898. (2007)