

Drag Force Reduction of a Bluff-Body with an Underbody Slant and Rear Flaps

Shinsuke Kowata, Jongsoo Ha, Shuya Yoshioka,
Takuma Kato and Yasuaki Kohama
Tohoku University

Copyright © 2008 SAE International

ABSTRACT

The drag reduction device of a bluff-body was developed by slanting the rear underbody as a hip-up shape and adding flaps to the rear-end. The experiments were conducted in an open-jet low turbulence wind tunnel while the bluff-body model was varied in both slant angles and rear flap configurations. Drag forces, surface pressures around the body, and the velocity distribution in wake were measured experimentally. The surface flow at the underbody was visualized by the oil-paint method. Force measurements showed that an underbody slant with rear flaps reduced the drag force. The most effective setting occurred when the underbody was slanted three degrees with the enclosure that had the upper, side and lower flaps. In spite of the negative pressure and the trailing vortex at the underbody, the underbody slant was useful to diminish the velocity defect in wake, which led to an increase in the rear-end pressure. Since the upward flow from the slanted underbody was blocked by the rear flaps, the vortex shedding into wake was repressed. In conclusion, in order to reduce the drag force of a bluff-body with the underbody slant, it is important to suppress the trailing vortex formation at the slanted underbody and the vortex shedding into wake as well as to raise the lowered pressure.

INTRODUCTION

Due to the recent rise in crude oil prices, concerns regarding the vehicle's fuel economy have been growing. Aerodynamic occupies no small portion for the vehicle's fuel consumption, lots of aerodynamic betterments have been doing to improve the fuel efficiency. One of the methods is changing the flow characteristics behind a bluff-body by attaching some subsidiaries such as flaps or vanes to the rear part. Because a bluff-body, a representative shape of a heavy truck or a bus, produces 25% of its total drag from the rear[1].

Wood et al.[2] achieved a combined fuel savings of 10% through real operational testings by adding three types of drag reduction devices to a tractor-trailer truck. Devices were installed to certain parts in which most of the truck's aerodynamic drag occurred

such as the tractor-trailer gap, the trailer base and the undercarriage. Later, Bayraktar et al.[3] optimized these devices by using a Design of Experiments (DOE) in combination with CFD. Simulations for each configuration and the identification of parameter sensitivities were demonstrated by DOE sequentially. Ortega et al.[4] reduced the wind-averaged drag of the tractor-trailer model experimentally by mounting three types of skirts and angled or curved base flaps to the model's underbody and base, respectively. Khalighi et al.[5] observed the drag reduction of the bluff-body model experimentally and computationally by the closure of the wake with horizontal and vertical plates that disturbed the shear layer behind the model. Cary III et al.[6] set the regression models and the response surfaces for the coefficient of drag in terms of two-panel wake boards' width and inset on the Ahmed body at yaw angle up to 12 degrees. Browand et al.[7] decreased the fuel consumption of the real standard semi-trailer through attachment flat flaps to the trailer base. Kim[8] investigated the three dimensional turbulent flow around the commercial bus model and the change in aerodynamic characteristics caused by the rear-spoiler.

The methods listed above did not modify the rear body shape but added certain devices to change the flow characteristics in wake. Song[9] suggested that the drag could be reduced by slanting the rear underbody, which shrunk the body in stream wise direction as a hip-up shape. Ruhrmann et al.[10] analyzed the flow characteristics of the bluff-body in ground effect by varying the slant angle and the ride height of the underbody from the moving ground. It was surmised that the force reduction phenomenon was caused by both vortex breakdown and flow separation. In this study, we examine experimentally the aerodynamic drag reduction of the bluff-body with a rear underbody slant and rear flaps. The effect of the drag reduction device is proved and the change of the flow characteristics around the body is analyzed through the wind tunnel experiments, which brought out oil flow visualization, the static surface pressure distribution at the rear part, and velocity distribution, power spectrum densities in wake.

EXPERIMENTAL SETUP

Model Description – Figure 1 shows the bluff-body model, 1/6th scale of the Ahmed body[11], with a length(L) of 174 mm, a height(H) of 48 mm, and a width(W) of 65 mm. All four front edges were rounded by a radius of 17 mm. Though the Ahmed body[11] was slanted at the rear upper body, the rear roof angle of the model in this study was set as 0 degree to design the rear shape of a bus or a trailer. The model was supported by four cylindrical struts with a diameter of 6 mm. Slanting was started at the line of $x=-37$ mm from the rear-end and its angle (θ) was classified 0, 3, 6, 9 and 12 degrees separately. Four types of rear flap configurations named as "Basic", "Upper Flap", "Upper+Side Flaps", and "Enclosure" were attached to the rear part together, as shown in Figure 2. All flaps were extended 20 mm into wake and rounded with a radius of 50 mm except for the lower flat flap of the Enclosure. At the Enclosure, rear flaps were attached up to slanting of 6 degrees because of the mechanical problem in mounting the lower flap.

Wind Tunnel Setup – The wind tunnel, an open-jet low turbulence type, experiments were carried out at the Institute of Fluid Science, Tohoku University. It had an octagonal test section with a subtense length of 293 mm. The model was positioned at 150 mm behind the front edge of the plate that was installed to simulate the ground with a length of 734 mm and a width of 490 mm, as shown in Figure 3. In order to prevent the separation at the upper surface of the plate, the plate's leading edge was formed in an elliptical shape, whose ellipse axis ratio was 1/8. The free stream velocity (U_∞) was 20 m/s, and the turbulence intensity was 0.8%. Reynolds number was $Re=2.2 \times 10^5$, which was based on the overall length of the model. The Reynolds number in this study was lower than that in the Ahmed body experiments[11] with 4.3×10^6 or that in real driving vehicles with $10^6 \sim 10^7$ order. However it is known that the Reynolds number between $1.0 \times 10^5 \sim 1.0 \times 10^6$ is enough to simulate the flow around a vehicle model in a wind tunnel[12][13]. The ground clearance of the model was adjusted as 8 mm, which was large enough to cover the boundary layer thickness of 1.8 mm and 2.4 mm at the forepart and the rear part of the model, respectively.

Measurements – Mean aerodynamic drag was measured by a load cell, which was joined to the model by four struts, as shown in Figure 3. The coefficient of drag force was averaged from data collection at a rate of 1,000 Hz for 100 seconds. The estimated uncertainty was less than 0.2%[14]. The oil, a mixture of liquid paraffin and titanium dioxide (TiO_2), was painted at the underbody to investigate the time-averaged surface flow. At the oil visualization, the free stream velocity was increased to 40 m/s so that the flow pattern could be seen more clearly. The qualitative flow characteristics, however, were not changed because the flow was turbulent[12][13]. This tendency was verified by drag measurements not shown in this paper. Static surface

pressures were measured from 6 static pressure holes on the slanted underbody and 10 holes on the rear-end, as shown in Figure 4. Static pressure measuring tubes were inserted to the inside of the model through two front struts. The particle image velocimetry (PIV) was used to analyze the flow characteristics at $y=0$ mm plane and $x=30$ mm plane in wake. Two hundred image pairs were taken with the spatial resolution of $1,008 \times 1,016$ pixels, and the time between images was 100 μs at $y=0$ mm plane and 40 μs at $x=30$ mm plane. Velocity vectors were computed by two frame cross-correlation with an area of 64×64 pixels. The power spectrum density (PSD) was used to observe the vortex shedding in wake with a hot-wire anemometer. The vortex shedding frequency was determined by a hot-wire probe of single normal type for 100 seconds with 1,000 Hz measurement frequency at one point ($x=50$, $y=0$, $z=56$ mm) in wake.

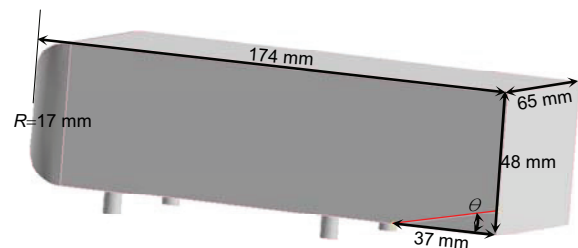


Figure 1. The Bluff-Body Model Geometry

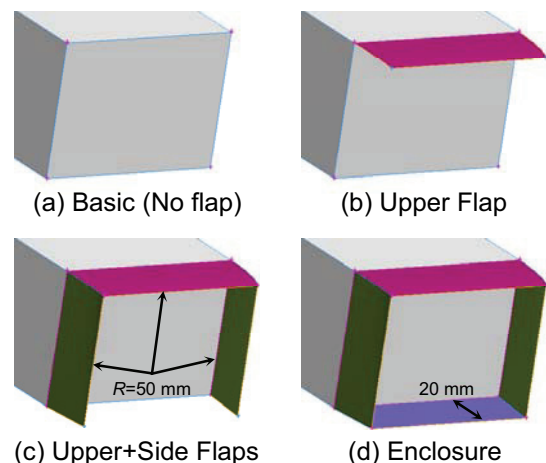


Figure 2. Four Types of Rear Flap Configurations

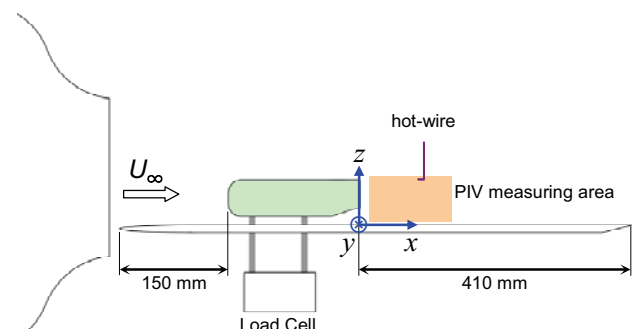


Figure 3. Wind Tunnel Experimental Setup

RESULTS

1) Drag Forces – Figure 5 shows the coefficients of drag force distribution at every slant angle with rear flap configurations. The drag was reduced as rear flaps were added at each slant angle. In regard to the rear underbody slant angle, the drag was reduced only at low slant angles. At the Basic, C_D had the minimum at slanting of 3 degrees, then rose as the body was slanted more. This tendency was in agreement with Song's result[9]. Similarly the Upper Flap and the Enclosure had the minimum C_D at 3 degrees while the Upper+Side Flaps at 6 degrees. The most effective setting was the Enclosure with slanting of 3 degrees, which reduced 27% of the drag of Basic with 0 degree. The reason why the drag force increased at high slant angles will be discussed later.

2) Flows at Slanted Underbody – As a result of the oil visualization, as shown in Figure 6, the flow was separated at the slanted underbody regardless of rear flap configurations. Red horizontal lines in the figures indicated the beginning of slanting. At slanting of 12 degrees, surface flows at the center were stagnated as soon as the body was slanted due to the separation. At the slanted underbody, the surface flow patterns around each side edge showed the outward flow, which was the consequence of the trailing vortex[15].

Slanting made the underbody flow accelerate so that the surface pressure at the slanted underbody was lowered, as shown in Figure 7, especially at upstream ($x=-31$ mm). This negative pressure decreased more as the slant angle increased. This lowered pressure enabled the outside flow of the body to come into the slanted underbody, which induced the trailing vortex. Besides this negative pressure was one of the factors for the drag force increment[16][17]. The pressure decline by slanting over 6 degrees was 10 times as much as the pressure recovery by adding rear flaps. This difference had an influence on the upward drag after 6 degrees, as shown in Figure 5.

Since the flowing cross section area under the body was enlarged at downstream ($x=-22$, -13 mm), the surface pressures on the slanted body approached to those on the non-slanted body. This surface pressure tendency was in good agreement with Ruhrmann et al.[10]. The surface pressure of $y=0$ mm plane became higher than that of $y=-20$ mm plane, since the latter was more close to the influential area of the evolving trailing vortex.

Accordingly the underbody slant made the separation and flow acceleration, which caused the lowered surface pressure at the slanted underbody and the trailing vortex formation. It is important to lessen the pressure decline and to prevent the trailing vortex formation.

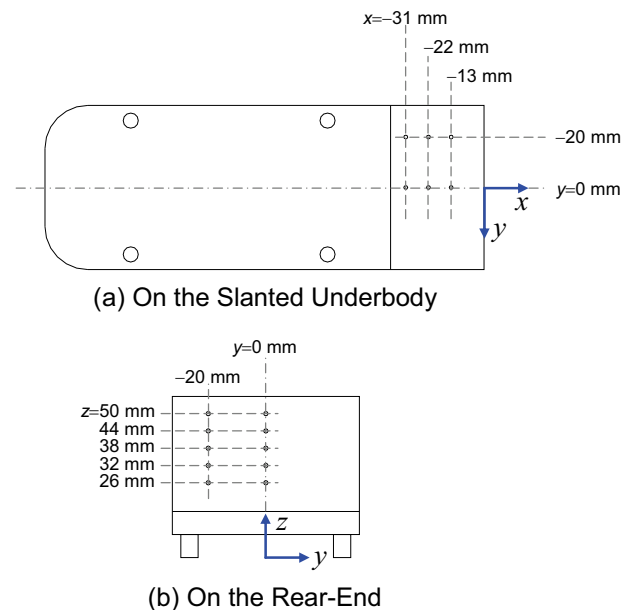


Figure 4. Layout of Static Surface Pressure Holes

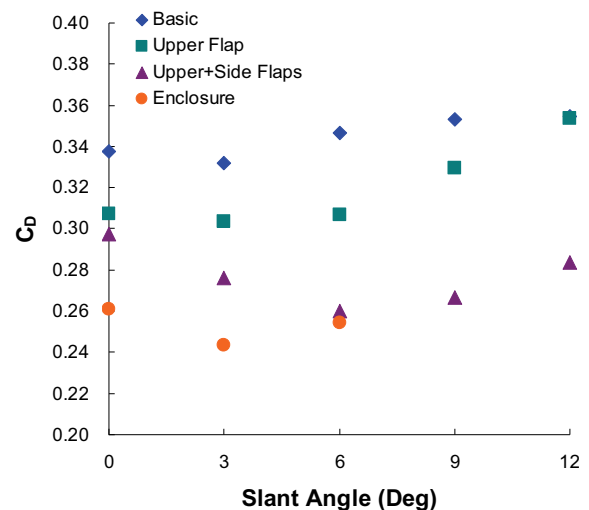


Figure 5. Coefficients of Drag Force Distribution

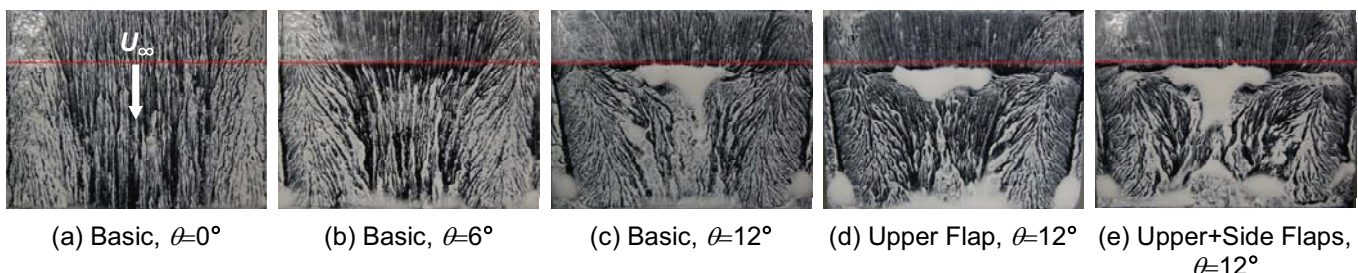


Figure 6. Oil Flow Visualization at the Slanted Underbody