

Aerodynamic Drag Reduction of Ahmed Model Using Synthetic Jet Array

Hoonil Park and Jun-Ho Cho
Seoul National University

Joonho Lee
Hyundai-Kia Motors

Dong-Ho Lee and Kyu-Hong Kim
Seoul National University

ABSTRACT

As speed of ground vehicle increases, there are increased concerns on the aerodynamic drag reduction of ground vehicle. Recently, synthetic jet is emerging as a promising active flow control technology for aerodynamic drag reduction. In this paper, we performed an experimental parametric study on synthetic jet for aerodynamic drag reduction of Ahmed model. Synthetic jet array is constructed by twelve synthetic jet actuators, and installed on two kinds of Ahmed model, of which slant angles are 25° and 35°. The jets are emanated between the roof and the rear slant surface. Jet angle, momentum coefficient, and driving frequency are changed to assess the effect of synthetic jet array on aerodynamic drag. To quantify the effect of synthetic jet, the aerodynamic drag and rear surface pressure are measured and analyzed. From the result, the effect of synthetic jet actuation on aerodynamic drag differs according to the slant angle of the body. The aerodynamic drag is reduced for 25° slanted body, but increased for 35° model. And jet angle, momentum coefficient, and driving frequency affect the quantity of change in aerodynamic drag.

CITATION: Park, H., Cho, J., Lee, J., Lee, D. et al., "Aerodynamic Drag Reduction of Ahmed Model Using Synthetic Jet Array," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(1):2013, doi:10.4271/2013-01-0095.

INTRODUCTION

Active Flow Control technology (AFC) is a promising area of vehicle aerodynamics. It introduces relatively small energy into the flow field in a vicinity of the object, and achieves much of aerodynamic enhancement by changing the flow pattern [1]. There are several methods on AFC: steady blowing, suction, micro jet, plasma actuator, synthetic jet, and so on. Among those methods, synthetic jet, so called a zero-net-mass-flux (ZNMF) jet, has advantages that it does not need additional air supply system and complex moving parts. With the advantages, there have been numerous researches to apply synthetic jet on aerodynamic applications, such as static and dynamic stall control of airfoils [2-3], jet vectoring [4], mixing enhancement of jet [5], and thermal mixing [6].

For literatures of AFC on ground bluff body, Beaudoin and Aider applied flaps at every rear edges of Ahmed body to reduce the drag and lift [7]. Muminovic *et al.* applied steady

blowing at the corners of roof end edge of Ahmed body, and designed the closed loop controllers [8]. Roumeas *et al.* performed numerical simulation of steady blowing on square back blunt body [9]. Bruneau *et al.* applied porous medium and pulsed jets on a square back blunt body [10]. Kang *et al.* suggested actively translating rear diffuser device to reduce aerodynamic drag by controlling under flow of car body [11]. McNallay *et al.* performed experiments using steady micro jets on Ahmed body [12]. Leclerc *et al.* performed a two-dimensional numerical study on aerodynamic drag reduction of Ahmed body using synthetic jet [13]. Brunn *et al.* attempt to reduce the aerodynamic drag by synthetic jet. For two-dimensional body, the drag is considerably reduced in a wide range of frequency [14]. And, for three-dimensional body, the spanwise vortex is remarkably weakened [15].

In this study, the synthetic jet is employed to reduce the aerodynamic drag by controlling the rear wake flow. As a wind tunnel model, the Ahmed reference model with 25° and 35° slant are chosen, showing different wake topologies

[16-17]. The synthetic jet is emanated from the roof end edge in a form of array, which consists of twelve synthetic jet actuators. Considered operating parameters are jet direction, momentum coefficient, and driving frequency. The change in aerodynamic drag and rear surface pressure are studied.

EXPERIMENTAL SETUP

Wind Tunnel Model and Test Model

The experiments are conducted in a closed-type subsonic wind tunnel at Seoul National University of South Korea. The dimensions of test section is $0.95 \times 1.34 \times 2.44$ ($w \times h \times l$, m), and turbulent intensity is less than 0.2%. All experiments are performed at 20 m/s of wind tunnel speed, corresponding Reynolds number based on model length is 4.29×10^5 .

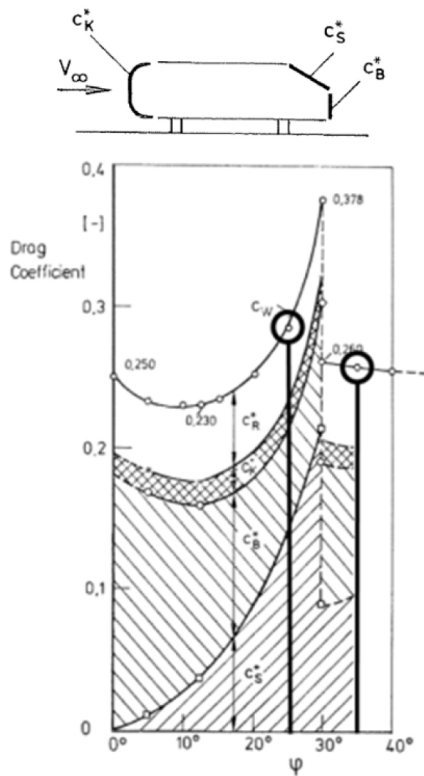


Figure 1. Variation of drag with base slant angle taken from Ahmed et al.[16].

The aerodynamic drag of Ahmed body is highly dependent on rear slant angle (Figure 1), and it is significantly related to the rear wake flow pattern. There is a critical slant angle, which is 30°. Below the critical angle, the aerodynamic drag increases as the streamwise vortices grow, which are created from slant side edges. The wake flow is fully three-dimensional. And then, for higher angle than the critical angle, the streamwise vortices burst and the roof end edge separation governs the wake region, and the aerodynamic drag experiences dropdown [16-17]. For the flow pattern varies according to the slant angle, two Ahmed

bodies which have different slant angle, 25° and 35°, are chosen for this study.

For the purpose of this study, Ahmed body is equipped with twelve synthetic jet actuators. The actuators are installed to emanate the synthetic jet at the roof end edge. Orifices of the synthetic jet actuators are placed in the roof surface without spacing with roof end edge. The model is scaled to 3.4:1, considering equipment of synthetic jet actuator array. Figure 2 shows the 35° slanted Ahmed body used in present wind tunnel tests. The model is produced by two parts (front and rear sections) to change the rear slant angle.

The aerodynamic drag is instrumented by one-axis load cell, and located at center of bottom surface. Theoretically, aerodynamic drag for this model may be measured within 0.2% accuracy. Rear surface pressure is instrumented at 32 pressure ports. The pressure ports are distributed on the half surface, assuming symmetric flow.

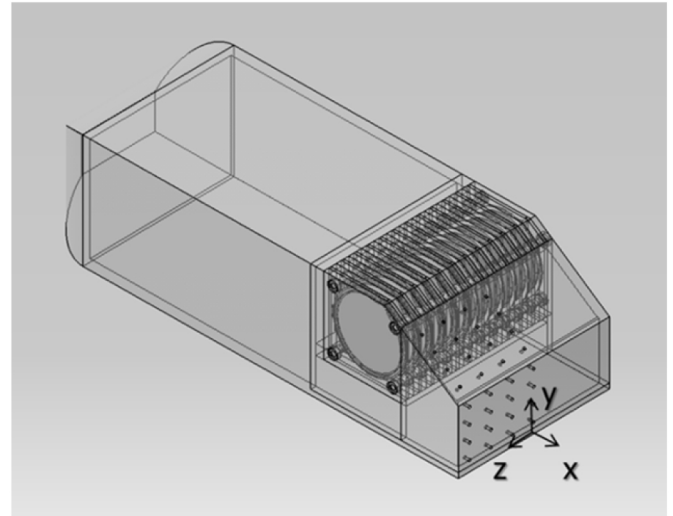


Figure 2. Ahmed body equipped with synthetic jet actuators.

The aerodynamic drag and rear surface pressure are non-dimensionalized by drag coefficient and pressure coefficient, respectively. Drag coefficient is defined as

$$C_D = \frac{\text{Drag}}{1/2 \rho_0 U_0^2 A_0} \quad (1)$$

, where *Drag* is the force acting on the model in free stream direction, ρ_0 is free stream density, U_0 is free stream velocity, and A_0 is frontal area of the model. The pressure coefficient is defined as

$$C_p = \frac{P - P_0}{1/2 \rho_0 U_0^2} \quad (2)$$