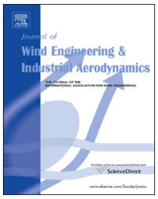
Journal of Wind Engineering & Industrial Aerodynamics 164 (2017) 138–151

Contents lists available at ScienceDirect

Journal of Wind Engineering
and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia



Salient drag reduction of a heavy vehicle using modified cab-roof fairings

Jeong Jae Kim[a], Sangseung Lee[b], Myeongkyun Kim[b], Donghyun You[b,*], Sang Joon Lee[a,*]

[a] *Advanced Fluid Engineering Research Center, Department of Mechanical Engineering, Pohang University of Science and Technology (POSTECH), 77 Cheongam-Ro, Nam-Gu, Pohang, Gyeongbuk 37673, South Korea*

[b] *Flow Physics and Engineering Laboratory, Department of Mechanical Engineering, Pohang University of Science and Technology (POSTECH), 77 Cheongam-Ro, Nam-Gu, Pohang, Gyeongbuk 37673, South Korea*

ARTICLE INFO

Keywords:
Heavy vehicle
Cab-roof fairing
Drag reduction
Forebody flow
Wind tunnel
Large-eddy simulation

ABSTRACT

Aerodynamic drag reduction is directly related to fuel consumption and CO_2 emission and is thus a main research interest in heavy vehicles. Approximately half of the total aerodynamic drag is attributed to the flow at the forebody of a vehicle and the gap between the tractor and trailer. Therefore, flow control devices that can reduce aerodynamic drag on the forebodies of heavy vehicles offer a considerably practical significance. Cab-roof fairing (CRF) is one of the most widely used drag reduction devices installed at the roofs of trucks or tractor-trailers. However, the drag-reducing effect and the three-dimensional flow characteristics around forebodies as a function of the external shape of CRFs have yet to be fully investigated. In this study, the drag reduction effects of typical and modified CRF models are quantitatively examined through wind tunnel tests and numerical simulation (coarse large eddy simulation (LES)). The wind tunnel experiment at a Reynolds number of $> 5.5\times 10^5$ is conducted for a scaled-down model of a 15-tonne truck. The modified CRF significantly changes the flow structure, leading to approximately 19% drag reduction. PIV flow field measurement was conducted to figure out the differences in flow characteristics around the forebody of the vehicle with and without CRFs.

Flow characteristics, including vortical structures, turbulent kinetic energy, mean pressure field, and mean velocity field around the forebody of the vehicle model (1/8 scale) with and without CRFs are numerically investigated using coarse LES for further understanding of the mechanism associated with drag reduction. The present results are expected to provide useful information for the design of new CRF models and the improvement of the aerodynamic performance of heavy vehicles, including trucks and tractor-trailers.

1. Introduction

Economical and eco-friendly vehicles have recently received significant attention along with the increased importance of fuel saving and the strengthening of environmental regulations. Therefore, aerodynamic drag reduction in heavy vehicles, such as trucks and tractor-trailers, is of great practical significance because it is directly related to energy saving. Sovran (1983) reported that a 3% reduction in fuel consumption can be obtained by reducing aerodynamic drag by 10%. Thus, numerous studies have attempted to reduce aerodynamic drag in heavy vehicles (Allan, 1981; Ahmed et al., 1984; Hucho and Sovran, 1993; Cooper, 2003; McCallen et al., 2004).

Various attempts have also been made to reduce forebody drag because approximately 45% of aerodynamic drag is induced by the front face of a tractor (25%) and by the gap (20%) between the tractor and trailer of a typical tractor-trailer when driven on a highway (Wood, 2006). Gilhaus (1981) experimentally investigated the effect of cab shape on forebody drag. Numerous forebody drag-reducing devices, such as forebody convex molding (Garry, 1981), front spoilers (Hyams et al., 2011), vertical fences (Allan, 1981), cab side extenders (Martini et al., 2011), and front spoilers (Choi et al., 2014), have been developed to control forebody flow in heavy vehicles. Cab-roof fairing (CRF) affixed to the cab of a truck or tractor is the most commonly used drag-reducing device to control forebody flow. CRF streamlines the step between the top part of a cab and the front part of a container. A CRF and a side extender attached on a 40 ft tractor-trailer model reduced drag by 12% at 65 mph (Cooper and Leuschen, 2005).

However, previous studies mostly dealt with the effect of drag-reducing devices without acquiring sufficient information on flow characteristics around the forebodies of heavy vehicles. Numerical and experimental studies were conducted using simplified vehicle models, such as the Ahmed model (Ahmed et al., 1984) and GM model (Han et al., 1996). The flow around a simplified tractor-trailer model (Östh and Krajnović, 2012) and a square-back vehicle (Verzicco et al., 2002) attached with drag-reducing devices were analyzed using large eddy simulation (LES). Detached eddy simulation was also conducted

---

* Corresponding authors.
E-mail addresses: dhyou@postech.ac.kr (D. You), sjlee@postech.ac.kr (S.J. Lee).

http://dx.doi.org/10.1016/j.jweia.2017.02.015
Received 12 March 2016; Received in revised form 28 October 2016; Accepted 15 February 2017
0167-6105/© 2017 Elsevier Ltd. All rights reserved.

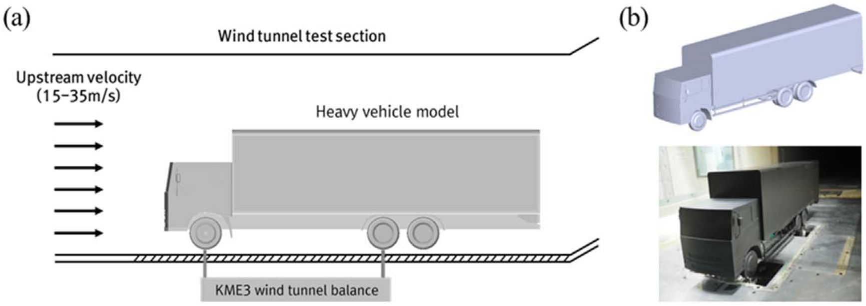


Fig. 1. Schematic of (a) the wind tunnel test section and drag measurement system and (b) the 15-tonne truck model.

for an Ahmed reference car model (Kapadia et al., 2004).

On the other hand, different turbulent intensities of the flows used in the wind-tunnel and on-road tests cause discrepancies in the aerodynamic results. Cooper and Campbell (1981) reported that the drag is usually increased due to the existence of turbulent flows for both wind-tunnel and on-road tests. Watkins et al. (1995) investigated the effect of the low turbulence flow in wind tunnel tests and turbulent flow in full-scale on-road measurements on the drag reduction effect of add-on devices. However, the effect of turbulent intensity on the drag reduction of add-on devices has not been fully investigated.

The use of a real vehicle model to study the influence of CRF is highly recommended because the flow characteristics around the forebody of a simplified vehicle model is considerably different from those of real vehicle models. In addition, conventional CRFs are required to be further optimized to maximize their drag reduction effect. With the advancements in manufacturing technologies, such as 3D printers, various shapes of drag-reducing devices can be easily made at a low cost. In the present study, three types of CRF shapes are fabricated, and their drag reduction effects in a scaled-down model (1/8) of a heavy vehicle are quantitatively evaluated using an experimental approach and coarse LES. The results would provide a new design concept for CRFs to improve the aerodynamic performance of trucks and tractor-trailers.

2. Experimental setup and method

2.1. Wind tunnel and drag measurement

The wind tunnel tests are performed in the POSTECH subsonic wind tunnel (closed-return type) with a test section of 4.3 m in length, 1.5 m in height, and 1.8 m in width. The wind speed (U) in the wind tunnel ranges from 5 m/s to 75 m/s. The freestream U is measured using a Pitot tube connected to a micro-manometer (FCO510, Furness Controls). Turbulence intensity and flow uniformity are lower than 0.2% and 0.25%, respectively. In the present study, the wind-tunnel test with low turbulent intensity is conducted for comparing the drag reduction effects of additive devices because turbulent intensity in the on-road tests is not easy to control, although there is discrepancy in turbulent intensities of the flows used for wind-tunnel and on-road tests.

The aerodynamic drag and moments exerted on the test vehicle model are measured using a seven-component external balance made by the German Aerospace Center (DLR). The measurement accuracy for these orthogonal coordinates is approximately 0.2% of the full scale. The drag force acting along the wind direction (X axis) is determined by statistically averaging the data acquired from five independent measurements. A vehicle model is affixed to the seven-component balance by connecting the fairs of its front and rear wheels to the four contact posts of the balance through the bottom surface of the wind tunnel test section (Fig. 1a). All wheels of the vehicle model are fixed. This vehicle model of a 15-tonne truck is designed on the basis of a prototype of the commercial truck Hyundai Trago Xcient, which measures 3.3 m high, 2.5 m wide, and 12 m long. The vehicle model is scaled down to a scale of 1/8 to achieve a reasonable ratio of the blockage of the model to the cross-sectional area of the wind tunnel test section (Fig. 1b). The effects of the model blockage on the measured pressure distribution and drag coefficient (C_d) are negligible when the blockage ratio is lower than 6% (West and Apelt, 1982). In this experiment, the actual blockage ratio of the 15-truck model is 4.7%. The drag coefficient is determined by using the following equation:

$$C_d = \frac{2F_d}{\rho U^2 A}$$

where F_d is the drag force measured along the streamwise direction, ρ is the fluid density, U is the wind speed, and A is the frontal cross-sectional area of the vehicle model.

2.2. PIV measurement

A schematic diagram of the experimental setup for PIV measurements of the velocity fields of the flow around the vehicle model is

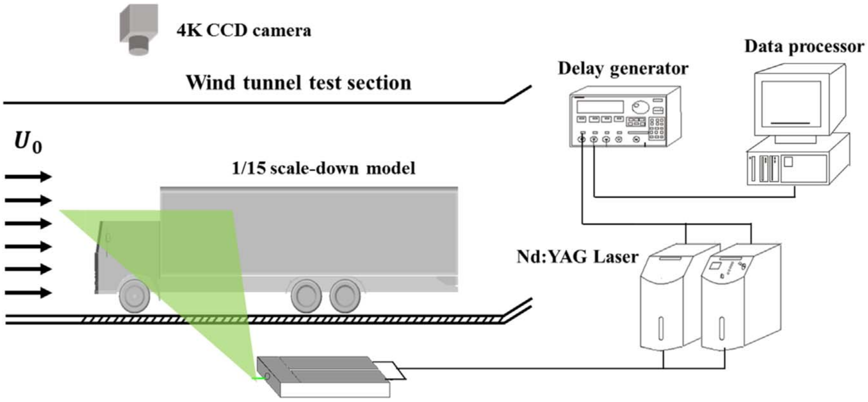


Fig. 2. Schematic diagram of the experimental setup for PIV measurement. The PIV system consists of a 120 mJ Nd:YAG laser, 4 K CCD camera, a delay generator, and a data processor.

Download English Version:

https://daneshyari.com/en/article/4924896

Download Persian Version:

https://daneshyari.com/article/4924896

Daneshyari.com