Experimental Thermal and Fluid Science 82 (2017) 58–74

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

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

Flow around an articulated lorry model

Kin Hing Lo *, Konstantinos Kontis

Division of Aerospace Sciences, School of Engineering, University of Glasgow, University Avenue, Glasgow G12 8QQ, United Kingdom

ARTICLE INFO

Article history:
Received 7 June 2016
Received in revised form 18 September 2016
Accepted 3 November 2016
Available online 4 November 2016

Keywords:
Articulated vehicles
Bluff bodies
Road vehicle aerodynamics
Square back vehicles
Wake flow
Wind tunnel tests

ABSTRACT

An experimental study has been conducted to investigate both the time-averaged and instantaneous flow pattern over a scale articulated vehicle model for understanding the flow physics of tractor-trailer vehicles. Fully turbulent flow was used in the study and smoke visualisation, surface oil flow visualisation and two-component particle image velocimetry were employed for flow diagnostics. Results obtained from the time-averaged and instantaneous flow fields show different flow pattern in the wake region downstream of the rear end of the trailer model. In the time-averaged flow field, a single counter-clockwise rotating vortex is presented in the wake region due to the coil-up of the lower shear layer. The instantaneous flow pattern shows that two wake vortices are presented in the wake region downstream of the trailer model. Moreover, the interactions between the wake vortex and the upper shear layer lead to the formation of the streamwise vortices within the shear layer. These streamwise vortices grow and propagate downstream which lead to the occurrence of vortex shedding in the upper shear layer downstream of the trailer model.

1. Introduction

Heavy Goods Vehicles (HGVs) play an important role in daily domestic goods transportation within the United Kingdom. According to the information shown in the report "Transport Statistics Great Britain 2015" [1], published by the Department of Transport of the United Kingdom Government, approximately 73.5% of domestic freight was transported by HGVs in 2014. In addition, about 15.5 million tonnes of greenhouse gas emissions came from HGVs. Due to the considerably poor aerodynamics efficiency of most HGVs, significant amount of fuel is consumed by HGVs to overcome the aerodynamics drag acting on the vehicles during high-speed operation. Altaf et al. [2] concluded that as much as 65% of fuel is consumed to overcome the aerodynamic drag encountered by buses and HGVs in long-haul journeys. Similarly, Bradley [3] indicated that the aerodynamic drag contributes approximately 21% of energy loss when a 36-tonne heavy goods vehicle is travelling at 105 km/h. Hsu and Davis [4] deduced that an annual fuel cost saving of US$ 10,000 could be achieved if the aerodynamic drag acting on a heavy vehicle is reduced by 40%. Similarly, Bradley [3] also anticipated that a 20% aerodynamic drag reduction on a heavy goods vehicle could lead to 4% of fuel saving during high-speed operation.

Aerodynamics of road vehicles is an active research topic nowadays due to the implementation of stringent regulations in many countries governing noise and exhaust gas emissions. In order to reduce noise and exhaust gas emissions, on top of improving engine technology the aerodynamic efficiency of road vehicles also requires to be improved. Aerodynamic efficiency of small vehicles such as cars and vans have been significantly improved in the last few decades [5]. In addition, the flow pattern over small vehicles has been extensively studied using both simplified models and actual vehicles [5,6]. One of the most common simplified models for investigating the flow physics of small vehicles is the Ahmed body originally proposed and used by Ahmed et al. [7]. Since then, many researchers investigated the flow pattern and drag characteristics over the Ahmed body with various slant angles to study the flow physics of simplified fast-back small vehicles [7–13]. The general time-averaged flow pattern over the wake region of the Ahmed body was concluded by Choi et al. [14]. Basically, the flow pattern is composed of a recirculating bubble on the slant surface and a wake region downstream of the vertical base of the Ahmed body. In addition, a pair of counter-rotating horseshoe vortices emanates from the side edges of the slant surface [14,15]. It was found that minimum drag occurs when the slant angle is 12.5° [14]. At this slant angle, the flow remains attached so that no recirculating bubble is formed on the slant surface of the Ahmed body.

Although the flow characteristics over the Ahmed body have been extensively investigated, the data collected could not be used

* Corresponding author.
E-mail address: kinhing.lo@glasgow.ac.uk (K.H. Lo).

http://dx.doi.org/10.1016/j.expthermflusci.2016.11.003
0894-1777/

to describe the general flow pattern over HGVs. This is because most HGVs have square-back rather than fast-back designs. As a result, another simplified model known as the General Motor (GM) model was used to represent square-back vehicles such as HGVs and buses [16]. The time-averaged flow pattern over the GM model was summarised in [14,16–19]. In general, the location at which flow separation occurs in the GM model is fixed at the sharp rear end. The occurrence of flow separation leads to the formation of a large and three-dimensional recirculating bubble downstream of the vehicle base. In addition, a pair of spanwise, counter-rotating vortices is formed downstream of the base of the GM model. The presence of these vortical structures significantly reduces the base pressure and increases the drag encountered [2,14].

Although the GM model could be used to represent some square-back heavy vehicles, this model could not be used to describe the flow characteristics over articulated lorries or tractor-trailers. Due to this reason, two simplified models known as the Ground Transportation System (GTS) [20–22] and the Generic Conventional Model (GCM) [23,24] were proposed and used to investigate the flow characteristics over articulated lorries. The GTS model represents the Cab-Over-Engine (COE) arrangement of the tractor unit of articulated lorries which is common in Europe. The flow characteristics over an articulated lorry with the COE arrangement is the subject matter of the present study. In contrast, the GCM model represents the conventional tractor unit arrangement which is commonly found in articulated lorries in the United States and Australia. In general, the original GTS model is the most simplified model to investigate the aerodynamic behaviour of articulated lorries. However, the lack of geometric features such as the gap and wheels in the GTS model suggests that the flow pattern over the model is unrealistic. These shortfalls were partially rectified in the modified GTS model [25] through the addition of a gap region and wheels to the model. However, it is still considered as an over-simplified model to be used in studying the aerodynamic characteristics of articulate lorries. In contrast, the GCM and later the modified GCM model [14] are more realistic due to the presence of the gap, wheels and other geometric features in the model.

Nevertheless, previous studies using the GTS and GCM models concluded that four main drag sources could be found in articulated lorries. These four regions are the front stagnation region, the gap flow region, the underbody flow and the large wake region that present downstream of the base of the square-back trailer [26–28]. Various devices and flow control strategies have been proposed in attempt to achieve drag reduction in these areas [14]. Since most of the previous studies mentioned in [14] concerned with the effects of various flow control devices in achieving drag reduction on articulated lorries, the flow physics of these vehicles is less well studied particularly for those articulated lorries with the COE tractor unit arrangement. In general, the time-averaged flow pattern over the modified GCM and GTS models under no crosswind condition could be summarised as follows. The flow stagnates at the front surface and separates at the rear end of the tractor model. Part of the separated flow enters the gap between the tractor base and the front face of the trailer. This results in the formation of a pair of counter-rotating spanwise vortices in the gap region. Along the trailer model, the flow remains attached until it reaches the rear end of the trailer at which massive flow separation appears. As a result, a large three-dimensional wake region is formed downstream of the base of the trailer model [14].

Recently, Mugnaini [29] investigated numerically the flow physics of a generic articulated lorry model with the conventional tractor unit arrangement. However, the results provided in [29] could only be treated qualitatively as the accuracy of the numerical procedures used is unclear. Similarly, Malviya et al. [30] investigated numerically the flow pattern over a generic articulated lorry model with the COE tractor unit arrangement. The result shown in [30] generally agreed with the conclusion drawn by Choi et al. [14]. However, due to the trailer model used in [30] was taller than the tractor model, an additional large recirculation bubble was formed immediately downstream of the front edge on the roof of the trailer model. Again, no information in the numerical scheme accuracy was provided in [30] which means that the data shown could only be understood qualitatively. Buil and Herrer [27] investigated numerically the flow characteristics over a COE typed articulated lorry with a circular-shaped trailer. The general flow pattern over the vehicle remains similar to that shown in an articulated lorry with a box-shaped square-back trailer. Once again, the accuracy of the numerical results shown in [27] is unknown. Altaf et al. [2] investigated numerically the flow pattern over a simplified generic square-back HGV model. Since only velocity and pressure contours were reported; no quantitative data about the characteristics of the wake vortex could be obtained from the study.

The lack of quantitative data in the flow physics of articulated lorries causing two significant problems in studying the aerodynamic properties of articulated lorries. Firstly, as already mentioned that the data presented in [27,29,30] could only be understood qualitatively due to lack of quantitative data available for validating the accuracy of numerical schemes. In addition, the lack of quantitative data also hindered the development of effective flow control devices that could be legally implemented in actual articulated lorries. The primary objective of the present experimental study is to provide some quantitative data in the flow characteristics over the wake region of a 1:20 scale generic articulated lorry model with the COE tractor unit arrangement. The data collected could be used for the purpose of numerical scheme validation and to improve our understanding in the flow physics over actual articulated vehicles with the COE tractor unit arrangement. The flow pattern over the model was visualised using surface oil flow visualisation, smoke visualisation and two-component Particle Image Velocimetry (PIV) techniques.

2. Experimental setup

2.1. Generic articulate lorry model

A 1:20 scale generic articulated lorry model was used in the present experimental study and its schematic is shown in Fig. 1. This model was designed based on some actual articulated lorries that commonly be found in the United Kingdom. Similar scale articulated lorry models also employed by Taubert and Wygnanski [31] and Ortega et al. [32]. It is generally agreed that the flow features appear over the model with this scale are comparable to those shown in the actual articulated lorries [14,31,32]. A normalised gap length ($G/\sqrt{A}$) of 0.1 is maintained between the tractor and the trailer to simulate the vehicle configuration during high-speed operation. It should be noted that G and A are the gap length between the tractor and trailer and the model frontal area, respectively.

2.2. The de Haviland wind tunnel

The present experimental study was conducted using the de Haviland wind tunnel of the University of Glasgow. The wind tunnel has a closed-loop design with a settling chamber to test section contraction ratio of 5:1. The dimensions of the wind tunnel test section are 4.0 m × 2.7 m × 2.1 m (length × width × height). Optical access is achieved through two glass-made side windows and two ceiling mounted Perspex windows. In order to ensure fully turbulent flow was used in the experiments, the freestream velocity