International Journal of Heat and Mass Transfer 55 (2012) 7981–7990

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Vehicle queue effect on the characteristics of air flow, and exhaust scalar dispersion and distribution fields in the vehicle wake

J.F. Huang, T.L. Chan *

Department of Mechanical Engineering, The Hong Kong Polytechnic University, Kowloon, Hong Kong

ARTICLE INFO

Article history:
Received 8 February 2012
Received in revised form 12 August 2012
Accepted 15 August 2012
Available online 13 September 2012

Keywords:
Vehicle queue effect
Scalar distribution region
Vehicle wake
Wind tunnel
Cold- and hot-wire technique

ABSTRACT

The characteristics of air flow, and vehicular exhaust scalar (i.e., pollutant) dispersion and distribution fields in the near-wake region of a scale-model vehicle which was placed alone or behind the preceding one(s) in a closed-circuit wind tunnel facility were experimentally investigated for typical urban driving conditions. The wake structure behind a queue of studied vehicles is mainly dominated by the last one, while the preceding vehicle(s) will lead to a stronger downwash flow in the wake. For the vehicle with rear slant angle ($\alpha < 30°$) which has a pair of trailing vortices in the wake flow, the vehicular exhaust jet plume will be mainly trapped inside these two trailing vortices and fills an "m-shaped" scalar distribution region behind the vehicle. Half of the m-shaped region which is on the vehicular tailpipe exit side shares a larger portion of scalar distribution than the other half. This unbalanced scalar distribution is enhanced by the preceding vehicle(s). For the vehicle with rear slant angle ($\alpha > 30°$) which has a two-dimensional wake flow, the vehicular exhaust jet plume will be carried by such a wake flow to form an "n-shaped" scalar distribution region behind the vehicle with a peak scalar region at its center. The preceding vehicle(s) will further enlarge the n-shaped scalar distribution region and push the peak scalar region closer to the ground. It is clearly shown that the two- or three-dimensional flow behind the studied vehicle can provide different shapes of exhaust scalar dispersion and distribution fields in the vehicle wake.

1. Introduction

The major source of air pollution is the on-road vehicle exhaust emissions in most urban cities [1]. Due to their serious impacts on urban air quality and public health, many experiments and numerical models for the pollutant dispersion of on-road vehicles or the pollutant dispersion in urban street canyons/street transport environments have been performed for the monitoring and improvement of urban air quality in the past decades [2–5]. On the contrary, the pollution in close vicinity of on-road vehicles (e.g., the vehicular exhaust pollutant dispersion and concentration in a relatively short distance behind vehicle(s)) has received less attention. This kind of pollutant dispersion behavior not only has a direct impact on human health in urban microenvironments, particularly to those high peak exposures of ground vehicle drivers and passengers, bicyclists, motorcyclists, pedestrians and people passing, working or living nearby [5–18], but also constitutes a major fraction of the total pollutant dispersion [19].

Recently, Chang et al. [14] have shown that the flow fields and dilution rates in the near-wake region have more complex interactions between wind speed, vehicle shape, and tailpipe orientation and location than that farther downwind. Chan et al. [15] have investigated numerically the exhaust particle formation and evolution processes, and concentration field in the wake region of the studied ground vehicle for stationary and moving conditions in a typical urban road microenvironment. Carpentieri and Kumar [16] have also shown that the presence of two different groups of nanoparticles in the volume immediately adjacent to the back of the moving vehicle: the new particles freshly emitted from the tailpipe, and the relatively aged particles in the flow recirculation wake of the vehicle. Kumar et al. [17] have further reviewed the dynamics and dispersion modeling of nanoparticles at five spatial scales (i.e., vehicle wake, street canyon, neighborhood, city and road tunnel), together with highlighting associated challenges, research gaps and priorities. Liu et al. [18] have also comprehensively simulated the interaction effects of different vehicle speeds and exhaust tailpipe exit velocity and temperature conditions on the three-dimensional flow structure, exhaust particle dynamic behavior, formation and evolution processes, number and volume concentration, and nucleation rate fields in the near-wake region behind the studied ground vehicle in urban road microenvironment.

Urban traffic density has been increasing significantly with shortened interval distance between the consecutive vehicles (i.e., vehicle spacing) especially for the most dense urban

* Corresponding author. Address: Department of Mechanical Engineering, The Hong Kong Polytechnic University, Hung Hom, Kowloon, Hong Kong. Tel.: +852 2766 6656.

E-mail address: mmtlchan@inet.polyu.edu.hk (T.L. Chan).

0017-9310/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.08.031

Nomenclature

d	diameter of a scale-model vehicle exhaust tailpipe, mm
H	height of a scale-model vehicle, mm
Re_H	Reynolds number $Re_H = U_\infty H/\nu$
$\overline{T}^*$	normalized mean temperature excess, $\overline{T}^* = (T_i - T_a)/(T_j - T_a)$
$\overline{T}^*_{\max}$	peak normalized mean temperature excess, $\overline{T}^*_{\max} = \overline{T}_{\max}/T_j$
T_a	background air temperature inside the wind tunnel, K
T_i	measured average local temperature, K
T_j	tailpipe exit temperature, K
X	streamwise coordinate (i.e., axis along the roadway direction), mm
Y	transverse coordinate (i.e., axis in the cross-roadway direction), mm
Z	spanwise coordinate (i.e., axis perpendicular to the roadway direction), mm
U	component of local velocity in X direction, m/s
U_∞	free stream velocity in X direction, m/s
U_j	exit velocity of heated tailpipe air jet, m/s
$\overline{U}$	mean velocity of U, m/s
u'	root mean square velocity (rms) of U, m/s
V	component of local velocity in Y direction, m/s
$\overline{V}$	mean velocity of V, m/s
W	component of local velocity in Z direction, m/s
$\overline{W}$	mean velocity of W, m/s

Greek symbols

α	rear slant angle of the scale-model vehicle, °
ρ	density of fluid, kg/m^3
ν	kinematic viscosity of fluid, m^2/s

areas which have considerably slower average vehicle speeds or encountered slow traffic under road congestion. Watkins and Vino [20] have recently discussed the application of modern traffic technologies (i.e., intelligent transport systems) for further reduction on the vehicle spacing. Here rises a question: how much of our knowledge gained from a single vehicle can still be applied to the case like a queue of vehicles? Up to now, very few research works concerning the interaction effect of flow structures, and scalar (i.e., pollutant) or aerosol particle dispersion and concentration fields from a queue of vehicles have been performed. Clifford et al. [21] identified that 75% of the total received pollution for a following vehicle comes from the preceding one. More recently, Kanda et al. [22,23] have found that for a queue of vehicles, the symmetric vehicle wake has more impact on the pollutant dispersion than the offset tailpipe position. It can be concluded that the wake flow of a queue of vehicles affects not only their aerodynamic properties but also the pollutant dispersion and concentration level behind the vehicles. A better understanding of the interactions between the flow structures, and scalar (i.e., pollutant) dispersion and distribution fields from a queue of vehicles becomes essential but has not been well documented.

Among a queue of vehicles, the air flow characteristics behind the vehicle are highly dependent on the arrangement of the incoming flow and the wake flow, as discussed comprehensively in our recent work of Huang et al. [24]. The aim of the present study is to investigate experimentally (i) the spatial evolution of air flow patterns, and scalar (i.e., pollutant) dispersion and distribution fields for different downstream distances within the near-wake region for the preceding and following vehicles which have pronounced effects on a single vehicle; and (ii) how the spatial evolution is affected by the presence of the preceding vehicle(s) for typical urban driving conditions with the introduction of an exhaust jet plume.

2. Experimental set-up

The present study was performed in a closed circuit wind tunnel with a test section of 0.6 × 0.6 × 2.4 m (width × height × length) and 0.5% background turbulence intensity for different incoming flow velocities, U_∞. All 1/22 scaled-down model vehicles have 90 × 81 × 180 mm (width × height × length) with a rear slant angle α = 25° or 60° which can be referred to our previous work of Chan et al. [12]. The two vehicle rear slant angles were chosen to cover the critical rear slant angle, 30° whereas two-dimensional wake flow (α > 30°) or three-dimensional wake flow (α < 30°) could be generated accordingly [11,12,24,25].

The vehicular exhaust tailpipe scalar (i.e., pollutant) distribution of on-road vehicle was simulated using a heated air jet plume at certain temperature T_j and velocity U_j through a small tailpipe exit (i.e., diameter d = 3 mm) for the scale-model vehicle. The tailpipe exit was located at the rear side of the vehicle (i.e. Y/H = 0.278 and Z/H = 0.222. A dedicated temperature control feedback system was placed inside the last model vehicle to maintain the tailpipe exit temperature. For the present study, the potential core temperature of the heated exhaust air jet and the background air temperature inside the wind tunnel were maintained constant [12,26]. The heated air jet exit velocity U_j of the model vehicle was then determined by the conservation of the mass flow rates from the vehicular exhaust tailpipe conditions of the real and scale-model vehicles for different vehicle speeds [12]. The detailed discussion and derived equations for analyzing the studied scale-model vehicles and simulating tailpipe exit conditions on wind tunnel similarity criteria used for the present study can be referred to Tables 1 and 2 in the experimental set-up section of our previous established work [12,24].

The x-axis is oriented in the direction of the main stream flow, U_∞, the y-axis is perpendicular both to the side wall of the vehicle and the main stream flow, and the z-axis is represented for the spanwise direction, as shown in Fig. 1. The studied vehicle was set on a designed flat plate (2.0 m in length × 0.59 m in width × 0.02 m in thickness) above the wind-tunnel floor in order to limit the boundary layer thickness [27]. The boundary layer thickness of this designed flat plate was reported to be around 0.1H by Huang et al. [24] and is relatively small if compared with the 1.5H height of the studied plane. A single vehicle or a queue of vehicles was/were placed in line on this horizontal flat plate to simulate the flow structure and scalar (i.e., pollutant) dispersion and distribution fields in the vehicle wake region for corresponding real world cases. Since the single vehicle cases of 30 and 50 km/h have shown similar results as described in Chan et al. [12], therefore only the results of vehicle speeds, 10 and 30 km/h will be presented and discussed. Their corresponding Reynolds numbers Re_H equals to 1.48 × 10^4 and 4.44 × 10^4, respectively. For a queue of vehicles, the “vehicle spacing” between the rear of the preceding vehicle and the front of following one was fixed to allow for two seconds of the vehicle movement (i.e., vehicle spacing = U_∞ × 2s). Hence, the corresponding vehicle spacing was 3.1H and 9.3H in respect to the vehicle speed of 10 and 30 km/h respectively, as shown in Fig. 1. The origin point of Cartesian coordinate system