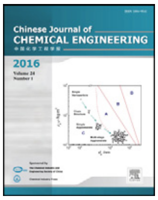
Chinese Journal of Chemical Engineering 24 (2016) 71–78

Contents lists available at ScienceDirect

Chinese Journal of Chemical Engineering

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



Fluid Dynamics and Transport Phenomena

A new approach to quantifying vehicle induced turbulence for complex traffic scenarios

Yesul Kim [1], Li Huang [1], Sunling Gong [1,2], Charles Q. Jia [1,*]

[1] Department of Chemical Engineering and Applied Chemistry, University of Toronto, 200 College Street, Toronto, ON M5S 3E5 Canada
[2] Air Quality Research Division, Environment Canada, Toronto, ON M3H 5T4 Canada

ARTICLE INFO

Article history:
Received 4 December 2014
Received in revised form 6 July 2015
Accepted 6 November 2015
Available online 2 December 2015

Keywords:
Computational fluid dynamics
Turbulent kinetic energy
Vehicle-induced turbulence
Road-induced turbulence
Mixed traffic

ABSTRACT

Traffic-related pollutants adversely affect air quality, especially in regions near major roadways. The vehicle-induced turbulence (VIT) is a significant factor that controls the initial dilution, dispersion, and ultimately the chemical and physical fate of pollutants by altering the conditions in the microenvironment. This study used a computational fluid dynamics (CFD) software FLUENT to model the vehicle-induced turbulence (VIT) generated on roadways, with a focus on impact of vehicle-vehicle interactions, traffic density and vehicle composition on turbulent kinetic energy (TKE). We show, for the first time, that the overall TKE from multiple vehicles traveling in series can be estimated by superimposing the TKE of each vehicle, without considering the distance between them while the distance is greater than one vehicle length. This finding is particularly significant since it enables a new approach to VIT simulations where the overall TKE is calculated as a function of number of vehicles. We found that the interactions between vehicles traveling next to each other in adjacent lanes are insignificant, regardless the directions of the traffic flow. Consequently, simulations of different traffic scenarios can be substantially simplified by treating two-way traffic as one-way traffic, with less than 5% difference in the overall volume-averaged TKE. We also developed equations that allow the estimation of the overall volume-averaged TKE as a function of the number and the type of vehicles.

1. Introduction

Various computational fluid dynamics (CFD) studies have modeled typical highway conditions using realistic vehicle shapes and compositions and investigated the turbulent kinetic energy (TKE) generated on roadways and its effect on pollutant dispersion [1–3]. One difficulty in reproducing realistic roadway conditions using CFD models is simulating two-way traffic. Hu *et al.* [4] developed a solution procedure, adopting a sliding-mesh approach in which the mesh points are updated as the vehicles move at a specified velocity in opposite directions. They were able to investigate the transient behavior of the airflow in between the two vehicles as well as the pressure distributions. However, such rigorous approaches are computationally expensive, and their transient nature made it not suitable for highway-scale TKE models. Other studies have suggested different approaches with some simplifying assumptions. One study made an assumption that if the vehicle flow is continuous enough for the TKE to stay constant over time, then a segment of highway can be used as a representative section of the overall highway [1] and vehicles were set to moving walls with specified velocities. Similar approach has been used by other studies [3].

These studies have built only one set of vehicles for each highway study, claiming that traffic volumes change little between seasons [3] and the traffic composition does not vary much [1]. Their works are therefore limited to the specific conditions under which their simulations were set up for, and the results cannot be extended to other roadway conditions. Although the traffic volume may change little between seasons, the hour-to-hour variations were proven to be more significant as shown in field measurements [5,6]. Also, these studies were not able to capture the different vehicle-vehicle interactions and its effect on TKE generation, or its decay characteristics, although these factors could be important in determining the fate and transport of air pollutants. One study showed how the pollutant dispersion and turbulent mixing are impacted by building arrays and packing density in street canyon environments [7]. Impacts of similar significance may be expected from different arrays and densities of vehicles on roadways. Since it is important that the results are taken and applied beyond the simulation domain, the current study aims to provide insights into different factors that may affect TKE on roadways and to develop parameterizations that can be applied in future studies.

2. Methodology

The commercial Computational Fluid Dynamics (CFD) package, FLUENT, was used in this study. FLUENT is a multi-purpose fluid dynamics software package, which has been widely used in complex airflow and pollutant dispersion applications in various environments. All of

---

[*] Corresponding author.
E-mail address: cq.jia@utoronto.ca (C.Q. Jia).

http://dx.doi.org/10.1016/j.cjche.2015.11.025

the governing equations are discretized using the finite volume method and are solved by using the SIMPLE (Semi-Implicit Method for Pressure-Linked Equation) algorithm in FLUENT, which uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field [8].

2.1. Quantification of vehicle induced turbulence (VIT)

The turbulent kinetic energy (TKE) of the air flow was used to quantify the vehicle induced turbulence (VIT). TKE is defined as the sum of the kinetic energy of the velocity fluctuations. The velocity u may be expressed as:

$$u = \bar{u} + u' \tag{1}$$

where $\bar{u}$ is the time-averaged mean velocity, and u' is the fluctuating part of the velocity that differs from the average value.

Then the TKE per unit mass of the flow can be expressed as:

$$\text{TKE} = k = \frac{1}{2}\left(u'^2 + v'^2 + w'^2\right) \tag{2}$$

where u, v, and w are the fluctuating velocity components in x, y, and z directions.

Since the instantaneous values of TKE can vary dramatically, a mean TKE value is often calculated to represent the overall flow.

$$\text{TKE}_{\text{avg}} = \overline{k} = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right) \tag{3}$$

Although it is not possible to exactly predict the random and irregular details of turbulent flow, various models have been developed to provide “closure” to the equations governing the average flow. The standard k–ε turbulence model is one of the most widely used and validated CFD turbulence model [1–3,9–11]. It offers a good compromise between result accuracy and computational cost in the absence of swirling flow [8]. The assumptions used in this model are that the flow is fully turbulent and the effects of molecular viscosity are negligible.

2.2. Simulation setup

2.2.1. Simulation domain and mesh setup

In order to model a realistic roadway condition, three different types of vehicles were used in this study (Fig. 1): a passenger vehicle, a sport utility vehicle (SUV), and a truck, which were modeled in real-shape rather than block-shape, since the block-shaped vehicles are estimated to produce 25% more turbulence than real-shaped vehicles [12]. Vehicle dimensions are given in Table 1. These vehicles were set to travel along the x-axis, while the y-axis is the width of the domain going into the screen, and the z-axis is the height of the domain.

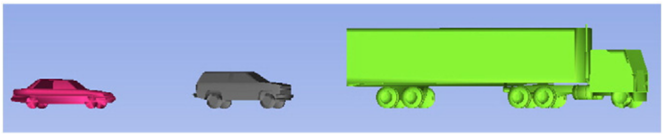


Fig. 1. Shapes of three different types of vehicles: a passenger vehicle, an SUV and a truck.

Table 1
Vehicle dimensions

Vehicle type	Length (m) in x direction	Width (m) in y direction	Height (m) in z direction
Passenger vehicle	4.5	1.8	1.5
SUV	4.5	2	1.85
Truck	15	2.5	4

The domain size in this study was fixed at 100 m × 20 m × 20 m in x, y, z directions, respectively. This was to compare the volume-averaged TKE when there are multiple vehicle interactions, changes in traffic densities and changes in traffic compositions. The computational domain was meshed using ICEM-CFD — a software widely used for generating meshes. To maintain a proper balance between results accuracy and computational expense, variable mesh size was used. Since the gradients in the model variables are more steep right around the vehicles and in the vehicle wake region, finer mesh density was used around the vehicles, and the mesh size was set to grow at a certain ratio as it moves away from the vehicles. Up to 6 million cells were created in each mesh setup. Fig. 2 shows a typical mesh setup where the fine mesh density is shown right around the vehicle; note that there are volume meshes that fill the whole domain but only the surface meshes are shown here and also that some surfaces are left invisible for the ease of view. More specifically, the smallest grid size was 1 cm around the vehicle tailpipes. The expansion ratios were 1.2 and 1.3 for inner and outer density regions as shown in Fig. 2. The mesh size near the ground shares the same parameters as described above. In addition, sensitivity tests were conducted to ensure that two density regions were adequate to obtain relatively constant volume-averaged TKE around a single passenger vehicle. The orders of numerical schemes used in this study are standard for pressure and second order upwind for momentum, TKE and turbulence dissipation rate. The convergence criteria are 10^{-3} for continuity, velocity components, TKE, turbulence dissipation rate and 10^{-6} for energy.

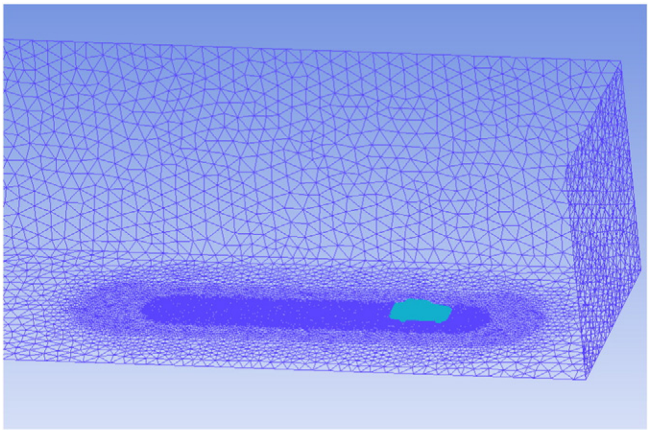


Fig. 2. Mesh density setup around a single passenger vehicle.

2.2.2. Boundary conditions

Movement of the vehicles in a steady-state flow was simulated by modeling the vehicles as moving walls with specified translational velocities in x-direction. For the vehicle surface, an equivalent roughness height of 0.0015 m was used [3]; while the ground was set as a stationary wall with a surface roughness of 0.01 m. The road is not raised and there is no barrier or obstacle to air flow, other than the surface roughness. Non-slip boundary conditions and a specified surface temperature (300 K) were applied to the vehicle and the ground surfaces.

As shown in Fig. 3, the symmetric boundary conditions were applied to the two side faces and the top face; which means that there is zero gradient in the variables normal to these surfaces, and that there is no flux of all quantities across these surfaces. The front side was modeled as a velocity inlet with zero velocity; while the back side was modeled as an outflow with zero normal first derivatives of all quantities, which means that there is bulk flow only and no diffusive flux for all flow variables in the direction normal to the plane. When external wind was introduced, the side from which the wind blows from was also treated as a velocity inlet, with specified wind velocity components. For the “velocity inlet” boundary in Fig. 3, the actual velocity was set to