Environmental Pollution 204 (2015) 280–288

Contents lists available at ScienceDirect

Environmental Pollution

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

Characteristics of air pollutant dispersion around a high-rise building

Y. Zhang [a, b], K.C.S. Kwok [a, *], X.-P. Liu [c], J.-L. Niu [d]

[a] Institute for Infrastructure Engineering, University of Western Sydney, Australia
[b] Medical Center, Tsinghua University, China
[c] School of Civil Engineering, Hefei University of Technology, China
[d] Department of Building Services Engineering, Hong Kong Polytechnic University, Hong Kong, China

ARTICLE INFO

Article history:
Received 6 February 2015
Received in revised form
28 April 2015
Accepted 1 May 2015
Available online 16 May 2015

Keywords:
Air pollutant
High-rise building
Windward emission
Leeward emission
CFD

ABSTRACT

A numerical wind tunnel model was proposed. The computed results of the pollutant diffusion around a typical Hong Kong high-rise building model (at a linear scale of 1:30), were found to show a similar trend to the outcomes of self-conducted experimental measurements that the pathways of pollutant migration for windward and leeward pollutant emission are different. For the case with windward pollutant emission at the 3rd floor within a re-entry, the pollutant migrated downwards due to the downwash created by the wind. In contrast, for the case with leeward pollution emission, dispersion is dominated by intense turbulent mixing in the near wake and characterized by the upward migration of the pollutant in the leeward re-entry. The simulated results of haze-fog (HF) studies confirm that the pathway of pollutant migration is dominated by wind–structure interaction and buoyancy effect only plays a minor role in the dispersion process.

1. Introduction

1.1. Motivations

The degradation of air quality in an urban environment becomes one of the most serious life quality and public health issues nowadays. Poor quality air promotes the spread of respiratory and other communicable diseases (Portney and Mullahy, 1990), and even results in serious cardiovascular diseases (Brook et al., 2004), particularly for the very young and very old, and people with chronic diseases and those with immune deficiency. The alarming progressive degradation of air quality worldwide is attributable to the growth of manufacturing and resource industry in East and South Asia to meet the burgeoning demand for goods and services by a growing affluent population, a lack of control of air pollution caused by fossil fuel combustion to meet the demand for energy and the inadequacy of air ventilation in urban area where a large proportion of the world population reside (Alejo et al., 2010; Lu et al., 2012). Evidently, building arrays and street canyons have an undesirable impact on the ground level wind flow, air ventilation within the urban fabric, as well as the dispersion of air pollution; however wind–structure interaction which governs the effects of buildings on atmospheric flow is often not properly considered when making urban development and urban planning decisions (Abd Razak et al., 2013).

Although dispersion of air pollutant and air ventilation in urban environment attracts great research interest, there remains a lack of understanding of the mechanism as it includes complicated phenomenon, for example pollutant transport in high turbulent flow, buoyancy flows, flow separation and reattachment around buildings, and vortex-induced flow around buildings. Among these elements, which one drives the spread of pollutant/respiratory ailment the most is still a controversial issue. For instance, after the outbreak of Severe Acute Respiratory Syndrome (SARS) in Hong Kong in 2002, many researches have been reported to address the mechanisms of disease spreading. Nevertheless, it remains unclear whether buoyancy effect or wind flow plays the more significant role (Niu and Tung, 2008; Yip et al., 2007; Zhou and Jiang, 2004; Li et al., 2005). Hence a reliable method is urgently needed for the assessment of air ventilation and air quality in urban environments.

1.2. Background

One of the most common approaches adopted for the study of air pollution and air ventilation is a wind tunnel model test, in which the measurements of airflow and pollutant transport around

---

* Corresponding author. University of Western Sydney, 2751 NSW, Australia.
E-mail addresses: lixuethermal@163.com (Y. Zhang), K.Kwok@uws.edu.au (K.C.S. Kwok).

http://dx.doi.org/10.1016/j.envpol.2015.05.004
0269-7491/

building models are performed in a physical wind tunnel. An alternative approach is a numerical wind tunnel method, which builds a computational model and performs computational fluid dynamics (CFD) to predict the air flow and distribution of pollutant concentrations. The advantages and disadvantages of both methods are outlined hereunder.

Physical wind tunnel test is a widely accepted method and considered the main source of information for wind flow and pollutant dispersion around buildings and in street canyons (Pavageau and Schatzmann, 1999; Meroney et al., 1996; Hajra et al., 2013; Liu et al., 2010). One of the advantages of wind tunnel test is that model testing in a wind tunnel produces a set of physical data. The physical data are reliable provided that the test processes and measurements are undertaken in accordance to established experimental methods. However, there are also obvious limitations of physical wind tunnel test: the building models must be scaled down to fit in the wind tunnel, which creates low resolution of the measurements; and the boundary effect of the wind tunnel cannot be fully eliminated from the test. Although the physical measurements obtained in a wind tunnel model test are reliable, a scaled model is not a true reproduction of the real world. The need to satisfy the governing scaling and similitude requirements, such as Reynolds Number and Froude Number scaling, remains a formidable challenge.

An alternative approach is using computational technology to build a numerical wind tunnel. According to previous attempts (Li et al., 2006; Moonen et al., 2006; Endalew et al., 2009; Chavez et al., 2011; Blocken et al., 2012), a numerical wind tunnel approach has a number of advantages. The calculation domain of a numerical wind tunnel is adjustable and different boundary conditions can be applied so that the boundary effects on the simulations are minimized. If the computational power is large enough to accommodate a proper calculation domain, a numerical wind tunnel can test life-size objects, so that the scale effect can be minimized. However, there are limitations for existing numerical wind tunnels. Most notably the adopted numerical method affects the simulated results, i.e. different numerical scheme, different grid arrangement may make significantly different predictions.

It is noteworthy that the uncertainties of the simulation results are the main drawback of existing numerical wind tunnels. There are a number of studies using two-equation RANS models, including $k-\varepsilon$, RNG-$k-\varepsilon$, realizable $k-\varepsilon$ (Kim and Baik, 1999; Tsai and Chen, 2004; Jicha et al., 2002). However, those predictions were not fully reliable; in some circumstances, the predictions contradict the physical data, which suggests that either the general turbulent models adopted have common deficiencies, or the whole calculation process was not properly controlled, in terms of the convergency, time step and mathematical accuracy. Therefore, in Li's paper (Li et al., 2006), it was clearly indicated that a standardized quality assurance procedure is required. Furthermore, Blocken et al. (Bert Blocken, 2004) concluded that when using CFD, the best that one can do for the validation is to conduct the wind tunnel experiments oneself for the particular configuration.

1.3. Research objectives

The objectives of this research were to develop a numerical wind tunnel model with a self-developed solver on an open source CFD platform, to validate the model by self-conducted experimental measurement, and to discuss the air pollutant dispersion around high-rise buildings. The emphasis of this research was to investigate the different pathways of air pollutant dispersion around a high-rise building where the emission source is located at certain building heights in the windward, and leeward face of the building.

2. Numerical wind tunnel

Commercial software, such as Ansys-Fluent, Ansys-CFX (ANSYS Inc., Canonsburg, PA, USA), is widely used to create a numerical wind tunnel (Mo et al., 2013; Zheng et al., 2012; Huang et al., 2014). User-friendliness and robustness are the advantages of commercial software, but commercial software functions as a closed “black box” which does not allow the users to freely control the calculation process and hence is not convenient for the development of mathematical and physical models. Therefore, in-house/open-source CFD code is preferred for reliable and flexible simulations.

In our research, the numerical wind tunnel was built on an open-source CFD platform, OpenFOAM (www.openfoam.org), which was tested and validated in previous publications (Mack and Spruijt, 2013; Nagaosa, 2014). A self-developed solver was compiled for the simulation of air pollutant distribution. In this solver, PISO (Pressure-Implicit Splitting Operator (Seif et al., 2010)) algorithm, one of the extended versions of the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations (Barton, 1998)) algorithm was applied to solve the governing equations of momentum and air pollutant concentration. The general form of conservation equation is listed as below.

$$\frac{\partial}{\partial t}(\rho\varphi)+\frac{\partial}{\partial x_j}(\rho v_j\varphi)=\frac{\partial}{\partial x_j}\left(\Gamma_\varphi\frac{\partial\varphi}{\partial x_j}\right)+S_\varphi \qquad (1)$$

where φ is the generalized independent variables, and S_φ is the source item. Γ_φ is the diffusion coefficient of φ in turbulent flow. For fluid motion, Γ_φ is the dynamic viscosity of turbulence (μ_e). $S_\varphi=-\frac{\partial p}{\partial x_i}+\Delta\rho g_i+\frac{\partial}{\partial x_i}\left[\mu_e\frac{\partial u_j}{\partial x_i}\right]$. For pollutant dispersion, $\Gamma_\varphi=\frac{\mu_e}{\sigma}$, $\sigma\approx 1$ and $S_\varphi=0$.

The application of general turbulent models in the simulation of air pollutant dispersion have been compared extensively (Yuan et al., 2014). It is well known that LES and K–ω–SST models can give better predictions than others, but this is mostly meaningful in flow simulation. It has shown that the widely-used eddy-viscosity models in LES are not able to correctly predict time scales of turbulent mixing (He et al., 2002) and dispersions (Jin et al., 2010; Yang et al., 2008), which presents a new challenge for adopting LES in the study of pollutant dispersion. Often the improvement offered by these time-consuming models is not significant enough to justify the additional computational time. Furthermore, the measurements taken using conventional chemical probe are not accurate enough to necessitate model predictions of transient flow well in excess of the accuracy associated with physical measurements. Hence a standard $k-\varepsilon$ model was employed in the current numerical wind tunnel simulation. The model used in this paper is not dissimilar to that in commercial software, thus providing further advantages in using an in-house code with the self-developed eddy-viscosity models that are convenient to embed in the proposed numerical wind tunnel for this study and for future work.

Four millions tetrahedron/hexahedron computational cells were placed in the computational domain, with more than half of them were applied in the region close to the building model to bolster the spatial resolution around the building. Unsteady state calculations were performed. Auto-adapted time step was applied, which allows the variation of time step (10^{-5}s to 10^{-4}s) at iterations to ensure that the maximum Courant number during the calculation is less than 0.5, as per required by unsteady state solver (Patankar, 1980). The residuals of both momentum and mass conservation equations were setup as 10^{-6}, allowing sufficient numerical solution to capture the air pollutant with low concentration. The CFD results were time-averaged after the calculation reached a “stable state”, which is typically used to compare unsteady state simulation with steady state measurement (Zhang et al., 2011).