Atmospheric Environment 79 (2013) 716–730

Contents lists available at ScienceDirect

Atmospheric Environment

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

Review

CFD simulation of near-field pollutant dispersion in the urban environment: A review of current modeling techniques

Yoshihide Tominaga [a,*], Ted Stathopoulos [b]

[a] Department of Architecture and Building Engineering, Niigata Institute of Technology, Kashiwazaki, Niigata, Japan
[b] Centre for Building Studies, Department of Building, Civil and Envir. Engineering, Concordia University, Montreal, Quebec, Canada

HIGHLIGHTS

- CFD simulations of near-field dispersion in the urban environment are reviewed.
- Key features of near-field pollutant dispersion are identified and discussed.
- To understand inherent strengths and limitations of numerical models is important.
- Careful model evaluation while paying attention to their uncertainty is necessary.

ARTICLE INFO

Article history:
Received 26 November 2012
Received in revised form
5 July 2013
Accepted 15 July 2013

Keywords:
CFD simulation
Near-field
Pollutant dispersion
Urban environment
Modeling technique

ABSTRACT

Near-field pollutant dispersion in the urban environment involves the interaction of a plume and the flow field perturbed by building obstacles. In the past two decades, micro-scale Computational Fluid Dynamics (CFD) simulation of pollutant dispersion around buildings and in urban areas has been widely used, sometimes in lieu of wind tunnel testing. This paper reviews current modeling techniques in CFD simulation of near-field pollutant dispersion in urban environments and discusses the findings to give insight into future applications. Key features of near-field pollutant dispersion around buildings from previous studies, i.e., three-dimensionality of mean flow, unsteadiness of large-scale flow structure, and anisotropy of turbulent scalar fluxes, are identified and discussed. This review highlights that it is important to choose appropriate numerical models and boundary conditions by understanding their inherent strengths and limitations. Furthermore, the importance of model evaluation was emphasized. Because pollutant concentrations around buildings can vary by orders of magnitudes in time and space, the model evaluation should be performed carefully, while paying attention to their uncertainty. Although CFD has significant potential, it is important to understand the underlying theory and limitations of a model in order to appropriately investigate the dispersion phenomena in question.

1. Introduction

Air pollution near and around buildings is an important environmental problem. However, it is difficult to predict pollutant dispersion with certainty due to the complex interaction between atmospheric flow and flow around buildings. The pollutants that are brought into the atmosphere by various sources are dispersed (advected and diffused) over a wide range of horizontal length scales, which can be classified into near-field and far-field phenomena. The near-field pollutant dispersion involves the interaction of the plume and the flow field, which may be perturbed by building obstacles. Therefore, it is important to consider when assessing both outdoor and indoor air qualities, because it covers both pollutant concentrations in the surrounding streets and those on building surfaces. For this purpose, the dispersion region that is to be treated is the very short range (i.e. vicinity of the emitting building; within a few hundred meters of the source) rather than the entire (neighborhood) region of significant impact. Alternatively, in far field phenomena, the horizontal motion prevails over the vertical motion and the influence of individual buildings on a dispersion field becomes relatively small. This issue is discussed mainly in regards to public health regulations on air quality in urban environments.

Until recently, modeling studies on urban air quality were typically conducted by operational models derived from an integral

* Corresponding author.
E-mail addresses: tominaga@abe.niit.ac.jp (Y. Tominaga), statho@bcee.concordia.ca (T. Stathopoulos).

1352-2310/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.atmosenv.2013.07.028

nature of atmospheric dispersion (Stern, 1976; Pasquill and Smith, 1983). The operational models, which are mostly based on the Gaussian dispersion model, are often referred to as 'fast response models'. These models are frequently modified for various purposes and have been used for many comprehensive formal evaluations, because they are designed to enable many different cases to be calculated expeditiously (Hanna et al., 2001; Hall et al., 2002). Furthermore, they include many complicated dispersion processes; e.g. atmospheric stratification, buoyancy, chemistry, deposition, concentration fluctuations etc. These are no longer simple Gaussian plume models and are not constrained to simple, straight mean streamlines. For example, ADMS (UK-ADMS), the UK atmospheric dispersion modeling system (Carruthers et al., 1994, 1999; CERC, 2006), is one of the advanced operational models and can take into account a building effect based on a two-plume approach using wake averaged flow values to calculate plume spread (Robins et al., 1997). However, it cannot treat the detailed plume behavior affected by building obstacles explicitly due to the modeling derivation. Therefore, when these models are applied to near-field dispersion in the urban environment, it is important to understand the fundamental concepts used in these models, as suggested by Macdonald (2003).

A method that is more oriented to practical design related to near-field dispersion around a building is based on the so-called ASHRAE model (ASHRAE, 2007, 2011). This model was derived from the amassed results of wind tunnel experiments (Wilson, 1982; Wilson, 1983; Wilson and Lamb, 1994) and is also a semi-empirical model. It can be used for determining the appropriate stack height and air intake position for the case of an isolated building. This model is specialized for use in building design, but has limited applicability and less accuracy concerning building configuration details (Hajra et al., 2010, 2011).

In the past two decades, micro-scale Computational Fluid Dynamics (CFD) simulation has been widely used as an emerging analysis method for pollutant dispersion around buildings and in urban areas, sometimes in lieu of wind tunnel testing. The CFD simulation method consists of solving the transport (advection and diffusion) equation of concentration based on the velocity field obtained from the Navier–Stokes equations. CFD can provide detailed information about the relevant flow and concentration variables throughout the calculation domain; however, it is more time-consuming than the two previously mentioned methods. Moreover, it is difficult to implement various dispersion processes such as atmospheric stratification, buoyancy, chemistry etc. to the model, whereas they are easily implemented to the operational models.

Recently, CFD has been studied extensively for the assessment of pollutant dispersion around buildings. However, these studies have been performed with different research purposes, configurations, boundary conditions, and modeling approaches. This makes it difficult to evaluation of the strengths and limitations of CFD for the evaluation of near-field pollutant dispersion in the urban environment. As already mentioned, the near-field dispersion around buildings is characterized by the complex interaction between the atmospheric flow and the flow around buildings. The phenomenon has both meteorological and building aerodynamic aspects, however the majority of previous research has been conducted within the frameworks of each aspect. Hence, a comprehensive review with cross-cutting aspects is required for CFD simulations of near-field pollutant dispersion in the urban area.

Furthermore, for evaluating the quality of CFD simulations, it is necessary to analyze its sensitivity and uncertainty appropriately. ERCOFTAC Best practice guidelines (Casey and Wintergerste, 2000) gives the best practical advice for achieving high-quality industrial CFD simulations and provides relevant information on the most important issues relevant to the credibility, especially with regard to the most common sources of errors and uncertainties in CFD. Recently, several best practice guidelines have been proposed as verification and validation process of CFD for urban wind environment applications, mainly intended for the prediction of pedestrian level winds (Franke et al., 2004, 2007; Tominaga et al., 2008b). Although these practical guidelines are quite effective in the pollutant dispersion problem, there are additional recommendations specific to the dispersion problem, i.e. requirements of modeling for a contaminant transport equation and sensitivity to wind and other climatic conditions. Robins et al. (2000) investigated uncertainty in CFD predictions of building affected dispersion through the Evaluation of Modeling Uncertainty (EMU) project which involved a group of four organizations undertaking CFD simulations for a series of realistic near-field dispersion test cases (Hall, 1977). The study identified some important qualitative guidelines for good modeling practice to indicate where attention should be focused. In the context of rapid increase of CFD applications to near-field pollutant dispersion around buildings in recent years, it is critical to examine past studies on this area.

This paper reviews the current modeling techniques in CFD simulation of near-field pollutant dispersion in the urban environment and discusses the findings to give insights into future directions of practical applications. In Section 2, previous studies of near-field pollutant dispersion around buildings using CFD are overviewed. The configurations used in previous studies are categorized into four typical cases: an isolated building, a single street canyon, building arrays, and building complexes. Section 3 identifies key features of near-field pollutant dispersion around buildings from previous studies and discusses their relevance in CFD modeling. The importance of proper choice of turbulence models and boundary conditions are emphasized in Sections 4 and 5. Finally, model evaluation methods are discussed, and future directions of practical applications are suggested.

2. Overview of previous CFD studies in near-field pollutant dispersion around buildings

2.1. Dispersion around an isolated building

In order to investigate the basic structure of pollutant dispersion around a building, many research studies examining dispersion around a single obstacle have been conducted by field measurements and wind tunnel experiments (Robins and Castro, 1977b; Huber and Snyder, 1982; Ogawa et al., 1983; Li and Meroney, 1983a,b; Mavroidis et al., 2003). Flow and contamination patterns around a rectangular building with a rooftop vent are illustrated schematically in Fig. 1 (ASHRAE, 2011). The effluent from the vent is entrained into the zone of recirculating flow behind the downwind face and is brought back up to the roof. The near-field pollutant dispersion is characterized by the interaction between atmospheric flow and flow around a building as expressed in this figure.

Since the 1990s, many numerical studies using CFD have been conducted to investigate the applicability of these models for pollutant dispersion around a single building by the Reynolds-Averaged Navier–Stokes (RANS) approach (Murakami et al., 1990; Zhang et al., 1993; Delaunay et al., 1997; Cowan et al., 1997; Selvam, 1997; Li and Stathopoulos, 1997, 1998; Leitl et al., 1997; Meroney et al., 1999) and by Large Eddy Simulation (LES) (Tominaga et al., 1997; Sada and Sato, 2002). The calculations utilized in these studies were conducted with relatively coarse meshes in comparison to present work because of restrictions of computational resources at that time. Moreover, good practice guidelines of CFD for predicting flow fields around buildings have not been established at the time. Therefore, validation of the models had