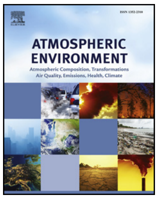
Atmospheric Environment 187 (2018) 117–130

Contents lists available at ScienceDirect

Atmospheric Environment

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



Numerical analysis of pollutant dispersion around elongated buildings: An embedded large eddy simulation approach

H. Foroutan[a,b,*], W. Tang[c], D.K. Heist[a], S.G. Perry[a], L.H. Brouwer[d], E.M. Monbureau[e]

[a] *Computational Exposure Division, National Exposure Research Laboratory, Office of Research and Development, U.S. Environmental Protection Agency, Research Triangle Park, NC, USA*
[b] *Department of Biomedical Engineering and Mechanics, Virginia Tech, Blacksburg, VA, USA*
[c] *Applied Research Associates Inc., Research Triangle Park, NC, USA*
[d] *Jacobs Technology Inc., Research Triangle Park, NC, USA*
[e] *Oak Ridge Institute for Science and Education, Oak Ridge, TN, USA*

ARTICLE INFO

Keywords:
Pollutant dispersion
Elongated building
Downwash
CFD
LES

ABSTRACT

High fidelity, scale-resolving numerical simulations of flow and pollutant dispersion around several elongated isolated buildings are presented in this paper. The embedded large eddy simulation (ELES) is used to model flow and concentration fields for six test cases with various source-building geometries. Specifically, the influence of building aspect ratio, wind direction, and source location is examined with these cases. Results obtained from the present ELES model are evaluated using available wind tunnel measurements, including those of streamwise and spanwise velocities, turbulent kinetic energy, and streamwise, lateral, and spanwise pollutant concentrations. Comparisons indicate that the ELES provides realistic representations of the flow and concentration fields observed in wind tunnel experiments, and captures several complex phenomena including the lateral shift and enhanced descent of the plume for rotated/elongated buildings. Furthermore, the ELES provides a means to study the advective and turbulent concentration fluxes, plume shapes, and geometry of vortical structures that is used to examine turbulent transport of pollutants around buildings. We investigate the enhancement of vertical and lateral plume spread as the building aspect ratio is increased. In addition, through the study of advective and turbulent concentration fluxes, we shed light on the physics behind higher ground-level concentrations observed for rotated buildings.

1. Introduction

The flow around a building in a turbulent atmospheric boundary layer includes several complex phenomena such as separation and reattachment, wake and recirculation, and horseshoe and arch vortices. The interaction of pollutants released near a building with these complex features often results in ground-level concentrations that are significantly larger than those occurring in the absence of the building. Elevated plumes brought quickly to the ground by these complex flows is referred to as building downwash. Many studies (e.g., Castro and Robins, 1977; Fackrell, 1984; Huber, 1989; Thompson, 1993; Snyder, 2005; Olesen et al., 2009) have shown that the magnitude and extent of the downwash effects depend on parameters including the building aspect ratio, the source height and location with respect to the building, and the meteorological conditions.

Wind tunnel and water channel measurements are common research methods to study the flow and dispersion around buildings. Smoke visualizations, velocity and turbulence measurements, and tracer concentration measurements have all demonstrated the complexity of the building-induced flow and the influence on nearby plume concentrations. For example, while examining simple rectangular buildings for a wide range of plume heights and locations, Thompson (1993) demonstrated that buildings with a wider dimension perpendicular to the incident flow have a down-washing effect that extends to higher plumes than that for buildings with otherwise similar dimensions. Snyder (2005) examined flow fields and Huber (1989) examined tracer concentration fields for releases near buildings at varying angles to the incident flow. They found a stronger downwash effect, higher ground-level concentrations, and strong lateral shifts in the plume when the building was at an oblique angle to the wind.

Dispersion models such as AERMOD (Cimorelli et al., 2005), OML (Olesen et al., 2007), and ADMS (Carruthers et al., 1994; Robins et al., 1997) have been developed on the basis of wind tunnel measurements such as those mentioned above. These operational models use a steady-

---

[*] Corresponding author. Department of Biomedical Engineering and Mechanics, Virginia Tech, Blacksburg, VA, USA.
E-mail address: foroutan@vt.edu (H. Foroutan).

https://doi.org/10.1016/j.atmosenv.2018.05.053
Received 6 September 2017; Received in revised form 23 May 2018; Accepted 26 May 2018
Available online 29 May 2018
1352-2310/ © 2018 Elsevier Ltd. All rights reserved.

state, Gaussian dispersion approach and serve as important tools for both air quality research and management. However, these models do not currently account for the asymmetry in the wake and the associated lateral and vertical shifts in the plume. Continuing to improve our understanding of the characteristics of the flow and pollutant dispersion near sources influenced by building wakes is crucial for the development of improved parametrizations in these models.

Recently, a wind tunnel study (Perry et al., 2016) was conducted at the United States Environmental Protection Agency (U.S. EPA) to specifically examine the influence of elongated rectangular buildings on the near-field dispersion of pollutants downwind of neutrally-buoyant pollutant releases. Four parameters were examined: building aspect ratio, source stack height, source location with respect to the building, and building angle with respect to the incident wind. These experimental scenarios both complement and enhance historical wind tunnel data related to building downwash. The results of Perry et al. (2016) suggest that wind direction has a strong influence on both the location and magnitude of maximum ground-level concentrations (GLCs). Specifically, the asymmetry of the building-induced wake for obtuse angles skews the lateral plume distribution and enhances the descent of plume material to the surface. Wind angle effects were particularly significant for sources located on the corner of a building. For the range of buildings studied, they also found that building wake influences on GLCs diminishes rapidly for stack heights above twice the building height. Finally, the wind tunnel results were used to demonstrate some inadequacies of the AERMOD model in simulating some of these wind-angle and stack location effects.

Despite being a critical basis for developing near-field pollutant dispersion algorithms, wind-tunnel studies can be somewhat constrained by the required time and expense, resulting in limited scenarios studied as well as limited spatial coverage of flow and concentration data. It is important to note that in these extremely complex flow situations, understanding the complete velocity, turbulence, and concentration fields around a building is important for adequate dispersion model development. The relative low cost and availability of a detailed and continuous set of data, make computational fluid dynamics (CFD) an important complementary tool for investigating near-field pollutant dispersion. This is evident from the increasing number of CFD simulation studies in this field in recent years (see Tominaga and Stathopoulos (2013) for a review). Most of these previous CFD studies have been carried out employing the steady Reynolds-averaged Navier-Stokes (RANS) equations with two-equation turbulence models (Li and Stathopoulos, 1997; Tominaga and Stathopoulos, 2009, 2017). Although the RANS simulations were generally able to predict the gross flow and concentration behavior, they fail to correctly describe the anisotropic and unsteady nature of the problem, which is manifested in the significant underestimation of the lateral plume spread (Tominaga and Stathopoulos, 2009). Therefore, scale-resolving methods including large eddy simulation (LES) and hybrid RANS/LES, which directly resolve (to some extent) the more influential, energy carrying eddies and models only the small (subgrid) scales, have been used and promising results were obtained (Tominaga and Stathopoulos, 2010; Gousseau et al., 2011; Jadidi et al., 2016a). Nevertheless, these studies are generally limited to cubical buildings perpendicular to the wind direction (Tominaga and Stathopoulos, 2010; Jadidi et al., 2016a), thus not addressing the complexities arising in the case of an elongated/rotated building.

The objective of the present paper is to employ a scale-resolving turbulence model, namely the embedded LES (ELES), to study near-field pollutant dispersion around elongated/rotated buildings. It includes the evaluation of the ELES performance against wind tunnel measurements, as well as the analysis of the computational results for various source-building geometries while emphasizing the aspects important for dispersion modeling.

Table 1
Test case configurations and the number of computational cells for each case.

Case	Configurations (aspect ratio-wind direction-stack height-stack location)	LES zone cell count (× 10^6)	RANS zone cell count (× 10^6)
A	2-00-1.5-DM0	1.60	1.68
B	2-45-1.5-DM0	2.07	1.74
C	8-00-1.5-DM0	3.05	1.88
D	8-45-1.5-DM0	3.62	1.95
E	2-00-1.5-DC+	1.87	1.71
F	2-00-1.2-DM0	1.55	1.68

2. Description of test cases

The CFD simulations are designed to match six cases examined in the wind tunnel study of Perry et al. (2016). While Perry et al. (2016) examined a range of building aspect ratios, source heights and locations, and wind angles, a smaller subset of the cases available are the focus of the present study (Table 1). Four of the cases selected employed two building aspect ratios, $L:W:H = 1:2:1$ and 1:8:1 (where L = streamwise length, W = cross-stream width, and H = height = 150 mm), each oriented perpendicular (0°) and 45° to the approach flow. For these four cases, the source is located at the middle of the downwind face (DM0) of the building with a stack height of 1.5H (see Fig. 1). Two additional cases include a 1:2:1 building oriented perpendicular to the wind with a source located at the downwind corner of the building (DC+) with stack height of 1.5H, and at the downwind middle (DM0) with stack height of 1.2H. Table 1 summarizes the configurations of these six cases. Each case is defined by four variables: building aspect ratio, wind direction, stack height, and stack location. For example, case D (8-45-1.5-DM0) includes a 1:8:1 building rotated 45° with a stack height of 1.5H located at downwind middle (DM0).

A 1:150 scale was used for the wind tunnel study (Perry et al., 2016) which makes the full scale equivalent building height equal to 22.5 m. A tracer source consisting of a 7-mm diameter porous plastic ball was at the edge of the building at elevations of 1.5H and 1.2H. The approach boundary layer developed for this study was designed to simulate flow in a typical suburban area with the use of Irwin spires (Irwin, 1981) and a series of roughness tabs arrayed on the floor of the wind tunnel. The tabs (38 mm high, 76 mm wide, and 1 mm thick) were spaced 305 mm both laterally and longitudinally, with each row offset laterally by half the tab spacing to provide a staggered pattern to condition and maintain the boundary layer throughout the test section. The resulting boundary layer had a full-scale equivalent roughness length of 0.36 m, displacement height of 0 m, and a friction velocity of 0.25 m s^{-1}. The wind speed at building height (U_0) was maintained at 2.77 m s^{-1} in the tunnel. This wind speed along with a building height of 150 mm provides a sufficiently high Reynolds number (27,700) for Reynolds number independence. The wind tunnel results represent all wind speeds that result in neutral atmospheric conditions.

Velocity and turbulence profiles were measured in the study using Laser Doppler Velocimetry. Rosemount Model 400A hydrocarbon analyzers (flame ionization detectors) were used to process samples of tracer gas released from the source and measured throughout the tunnel. To compute the velocity and concentration results, signals were sampled at 20 Hz over 120 s intervals. Results of the wind tunnel data, as well as the CFD model, are presented in normalized concentration units $\chi = C/C_0$ with $C_0 = \frac{Q}{H^2 U_0}$, where C is the background-adjusted concentration (g m^{-3}), U_0 is the reference wind (m s^{-1}), and Q is the tracer gas emissions rate (g s^{-1}). To account for residual tracer concentrations in the wind tunnel lab, at the beginning and end of each measured profile the background was sampled and the measurements adjusted. In addition, at the beginning of each measurement day, the hydrocarbon analyzers were calibrated against a range of standardized

Download English Version:

https://daneshyari.com/en/article/8863608

Download Persian Version:

https://daneshyari.com/article/8863608

Daneshyari.com