Nineteenth Australasian Wind Engineering
Society Workshop, April 4-6, 2018,
Torquay, Victoria

Effects of density and source location of pollutant particles on pollution dispersion around high-rise buildings

E. Keshavarzian[1], R. Jin[1], K. Dong[1], K.C.S. Kwok[2], M. Zhao[1], and Y. Zhang[3]

[1]*Centre for Infrastructure Engineering Western Sydney University, Sydney, Australia*
[2]*School of Civil Engineering, University of Sydney, Sydney, Australia*
[3]*School of Clinical Medicine, Tsinghua University, Beijing, China*

ABSTRACT

Using CFD simulations, the transmissions of aerosol particles emitted by pollutant sources to a nearby high-rise building is studied. The building is exposed to a realistic wind profile and the pollutant sources are located along different lines and at different distances from the building. Navier-Stokes equations for wind around the building are solved with a Standard k-ε model to simulate the turbulent flow. The Eulerian approach is also used to determine the concentration of aerosol particles with different density on the building at different heights. the total and regional deposition fractions are evaluated, and the impact of different parameters, such as location of pollutant source, pollutant density, shape of building, are investigated.

1. Introduction

Facilities like cooling towers and hot and cold water services can play a role of pollutant sources in urban environment. On one hand, their medium with constantly warm temperatures and a weak alkaline pH provide ideal conditions for colonies of bacteria to grow; on the other hand, their operations lead to the generation of massive aerosol particles which may carry the bacteria and spread them out with the plume. As a result, many outbreaks of different pathogens caused by the aerosol particles originating from the mentioned facilities in urban areas have been reported around the world. In many medical publications, it is reported that residents living in different buildings near a facility which is potential source of pollutants can be infected by pathogens, with definitive evidence that the bacteria were emitted from the source (Engelhart et al., 2008). The location of pollutant sources is therefore an important design consideration to control the transition of this infectious and potentially fatal disease.

As aerosol particles are mainly airborne, the dispersion of air pollution is generally controlled by meteorological factors such as ambient wind speed and wind direction. Although many researches show that building arrays and street canyons have a significant influence on wind flow, air ventilation and pollutant dispersion within urban area, it is mostly neglected in urban design and development to consider wind-structure interaction as a dominant feature in atmospheric flows (Razak et al., 2013). Building shape has a profound effect on the flow field as well as building arrangement. This factor can be more critical when pollutant dispersion is investigated around a specific building (Zhang et al., 2015). In particular, as the pollutants can be generated under different conditions and composed of both particulate matters and liquid droplets, the emitted aerosol particles may have different densities, which will affect the dispersion behavior of the particles around buildings. The pollutant density affect the pollutant pathway around building and the distance that pollutant is carried along with the wind flow.

In our research, the numerical wind tunnel has been validated and wind profiles (velocity and turbulent intensity profiles) has acceptable agreement with the ones from the wind tunnel test. Also, computed and experimental results of the dispersion of air pollution has similar trend. Using our model, the total and regional deposition fractions are evaluated, and the impact of different parameters, such as location of pollutant source, pollutant density, shape of building, are investigated. The findings can benefit urban design with guidelines for the relative location between potential pollutant sources and buildings.

2. CFD Model Setup

Building Geometry and Computational Model Setup

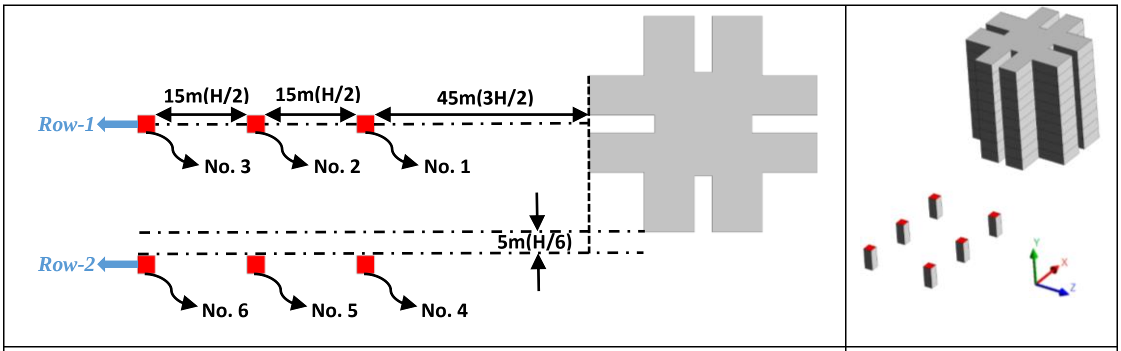
The physical model is based on a 1:30 scale model of a 10 storey high-rise building 30 m in height, which is a typical residential building in Hong Kong. The residents in one of this type of building were influenced heavily by severe acute respiratory syndrome (SARS) in 2002 and the spread of SARS virus in this building from an internal source has been studied in our previous work (Liu et al., 2010, Yu et al., 2017, Zhang et al., 2015). The dimensions of the building model and boundary conditions were extracted from a boundary layer wind tunnel test (Liu et al., 2010) and a numerical investigation followed the experiments (Zhang et al., 2015). The range of domain size used in previous studies was considered to design the flow domain, as shown in Figure 1. The cross section of the numerical wind tunnel domain was designed with the same dimension as the wind tunnel cross section which was 4m in height and 5m in width as recommended in the guidelines (Franke et al., 2011). The distance of 5H from inlet boundary to building is chosen, which is sufficient according to established guidelines (Franke et al., 2011). A distance of 15H is considered for the outlet from the building to allow for flow redevelopment behind the wake region. The blockage ratio of the constructed computational domain is about 2%, which is below the recommendation of 3% (Blocken, 2015). To understand behavior of plume originating from in the presence of the building, typical pollutant sources 2.4m×2.4m square-sectioned and 6m height are considered at six source locations upstream of the building. As shown in Figure 1, they were distributed in two columns and three rows. One of the columns was located in front of the building on the centerline and the other column 20m off the building centerline which was outside the building footprint. Pollutants were emitted from the top surfaces of the cooling towers which are coloured as red in the figure. Concerning the boundary condition, a constant flow rate of 58.5 ml/s of 10% pollutant concentration were allocated to the top surfaces of the pollutant sources. This flow rate was low enough to ensure that source momentum effects were not significant.

Based on the wind tunnel experimental data (Liu et al., 2010), the wind speed and turbulent intensity at the top of the building are about 3.27 m/s and 15%, respectively. Correspondingly, the following equation is prescribed at the domain inlet:

$$U(y) = \frac{u_*}{\kappa} \times ln(\frac{y + y_0}{y_0}) \tag{1}$$

u_* is friction velocity, κ is the von-Karman constant (0.42) and y_0 corresponds to aerodynamics roughness length of $\frac{0.02}{30}$ m of the wind profile in the wind tunnel which simulated according to an open terrain (Category 2) in the Australian Standards.

Other boundary conditions are considered as follows: slip wall condition at top boundary and lateral boundaries (zero normal velocity and zero normal gradients of all variables) and zero pressure outlet for outlet boundary.



(a) (b)

Figure 1: Building model and location of pollutant sources within the computational domain

Solution Method

A fully structured computational grid with approximately 3,700,000 cells was generated for the flow domain. The simulations were performed with CFD code ANSYS-Fluent 18.1. The 3D steady RANS equations are solved. The SIMPLE algorithm and QUICK discretization scheme are used for pressure-velocity coupling and convection terms, respectively.

Selection of proper turbulent for pollutant dispersion model has been found challenging in previous works (Yu et al., 2011, Yu et al., 2017). Although LES and K-ω-SST models are recognized as more precise models than others, these models are highly computationally expensive. In this research, the standard k-ε model has been shown to produce satisfactory results in agreement with the experimental data (Yu et al., 2017, Zhang et al., 2015). Hence, k-ε model is employed to model the turbulent flow in this study. The kinetic energy of turbulence was calculated from wind velocity, $U(y)$, and turbulent intensity profile, $k(y)$ (Eq. **Error! Reference source not found.**), and imposed to the inlet boundary condition as well as wind velocity and dissipation rate profile.

$$\varepsilon(y) = \frac{{u_*}^3}{\kappa(y + y_0)} \tag{2}$$

$$k(y) = \frac{3}{2}(U(y) \times I(y)) \tag{3}$$

An Eulerian approach is utilized to compute pollutant dispersion and particle transport in the domain of fluid. This approach treats the pollutant as a continuum field and assumes that the effects of pollutant particle inertia are negligible. Therefore, Eulerian method can be used for particles with such a low inertia. Large particle counts and low deposition rates can be easily simulated by this method. In this approach, the turbulent and steady form of the mass transport equation governing the convective-diffusive motion of pollutants can be written as(Fluent, 2015):

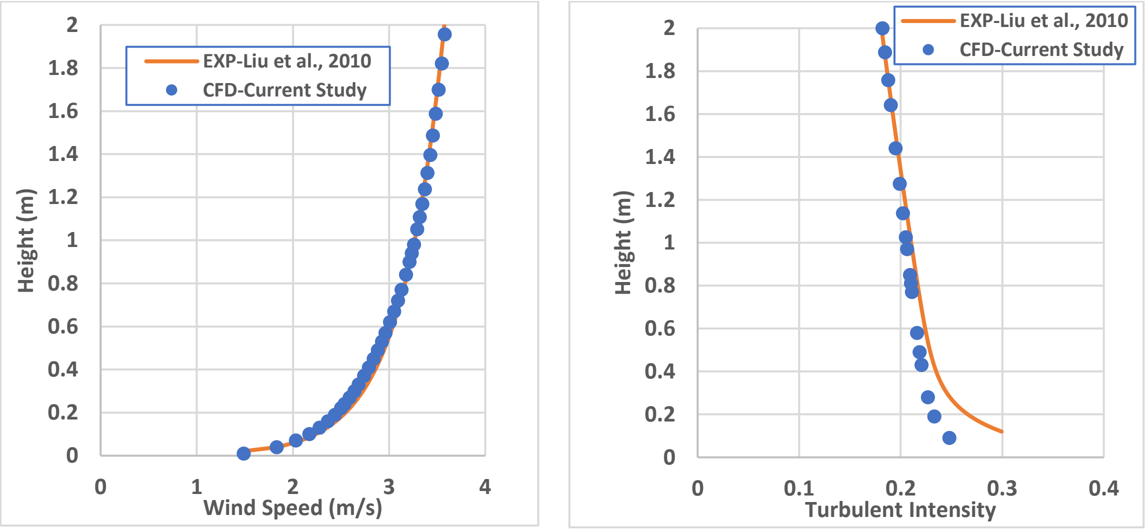
$$\frac{\partial}{\partial x_j}(u_j C) = \frac{\partial}{\partial x_j}\left((\mu + \mu_t)\frac{\partial C}{\partial x_j}\right) \tag{4}$$

where C is the pollutant concentration, μ is fluid viscosity and is μ_t turbulent viscosity.

In order to assess the contribution of the effects of pollutant source location and the pollutant density on the pollutant distribution compared with wind-structure interaction, emission of pollutant with different density was studied in this work. As the density of pollutants can be uncertain, two different molecular weight, heavier and lighter than air, was chosen for pollutant in this study. Firstly, the molecular weight was assumed to be 44, which is about 150% of the molecular weight of air and secondly, the pollutant molecular weight was 14 which is 50% lighter than air.

3. Results and Discussion

The wind profiles (wind velocity and turbulent intensity) utilized at the inlet boundary condition in the numerical wind tunnel were measured at the place of the building in the empty wind tunnel experimentally. The atmospheric boundary layer should be homogenous horizontally in the upstream and downstream of flow (Blocken et al., 2007) which means that the flow characteristics of inlet and incident flow should have accepted agreement. This challenging part to validate flow field in numerical wind tunnel and experimental one can be established through controlling wall functions and roughness parameters properly. Utilizing a UDF to set a value outside of that interval to control wall roughness height and choosing proper mesh size near to the ground, a compromise finally reached and incident wind profiles at the location of the building in an empty domain which was determined in CFD were consistent with the measured values in the wind tunnel test.



(a) (b)

Figure 2:(a) Mean velocity and (b) Turbulent intensity as incident profiles at the target location in the empty domain

To determine the impact of distance and location of pollutant sources on the pollutant distribution around the building, in each simulation pollutants are emitted only from one source of pollutant. As it was mentioned in the previous section, pollutant with two different molecular weight was released from pollutant sources to evaluate the effect of pollutant density around the building. Figure 3 shows the average pollutant mole fraction of on different façades (windward, sides and leeward) between different levels of the building. This figure shows a consistent result and reveal that the pollutant sources closer to the building cause a higher pollutant concentration in windward, side and leeward faces. This is consistent with the natural mixing and dilution processes as pollutants are transported along the streamwise direction. Hence, closer sources of pollutants experience less dilution and mixing to create a higher deposition rate on the building. Additionally, it can be seen that in contrast to the leeward face, in the windward face, lower levels have higher pollutant concentration. This trend is similar for all the locations of pollutant source, which is consistent with the flow field

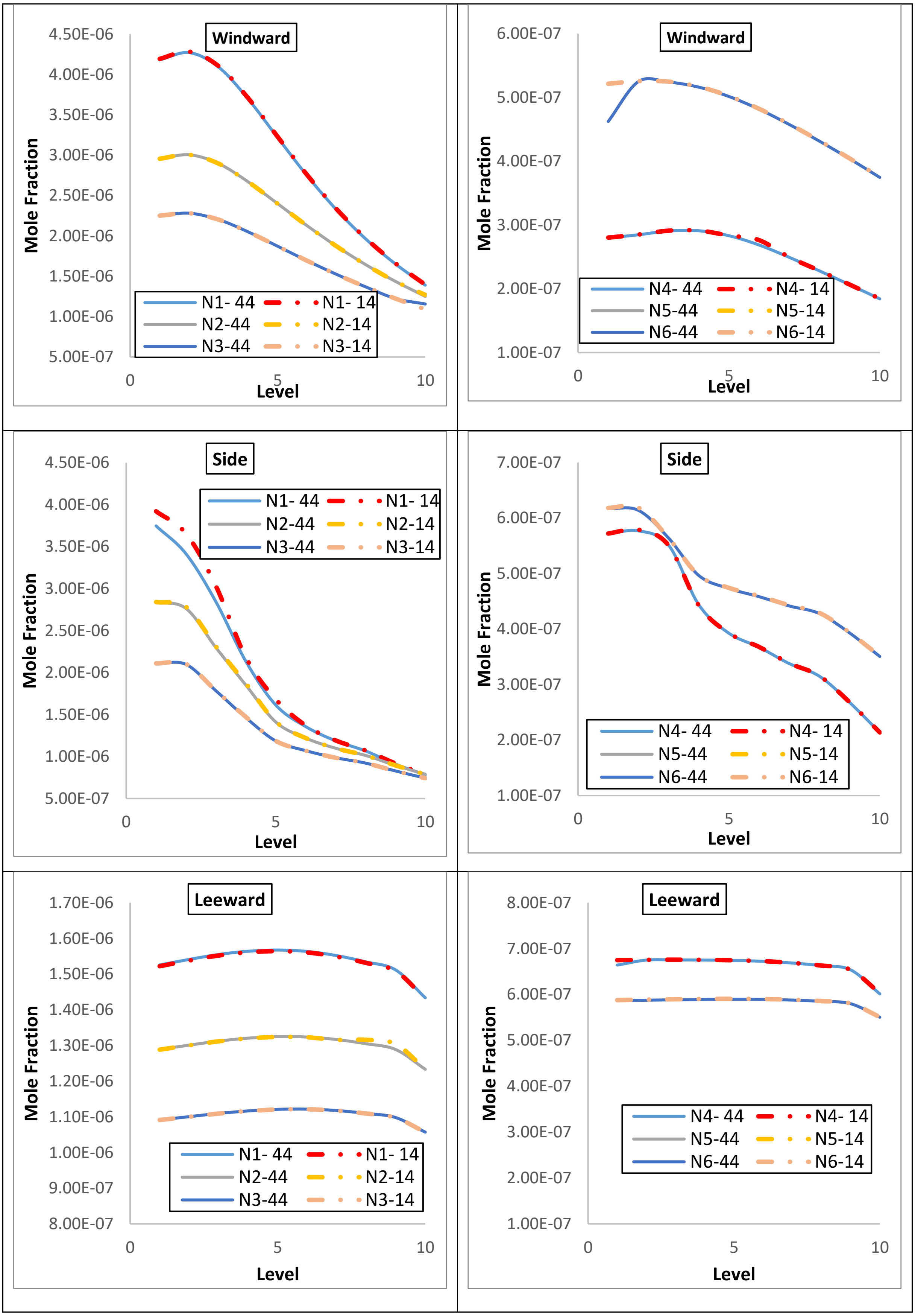


Figure 3: Average mole fraction pollutant on different facades emitted from sources (N1-N6) with two different molecular weight (44 and 14)

patterns around the building. Briefly, near the windward face, velocity vectors mostly show downward inclination associated with downwash, thus causing a higher pollutant concentration at the lower levels. In contrast, in the leeward face region, the velocity vectors reflect a more turbulent flow regime with a tendency towards upward inclination, particularly at close to the leeward face, which drives pollutants to higher building levels. As shown in this figure, the pollutant concentration at in different levels had a same trend, regardless of the density of pollutant, heavier than air (with molecular weight of 44) or lighter than air (with molecular weight of 14). The results reinforced our finding that pollutant dispersion in the near wake is dominated by wind-structure interaction, with buoyancy effect associated with pollutant density, within the range tested, playing only a minor role.

4. Conclusions

In the paper, particle pollutant dispersion emitted from sources outside of a high-rise building has been studied using a CFD model. Effect of pollutant density and effect of distance and location of the sources have been investigated through the simulations. Evidently, the pollutant concentration distribution on different faces of the building revealed that convective mechanism as a consequence of wind-structure interaction dominates the pollutant dispersion processes. Regarding flow field aspect, the recirculation formed behind the building can change the particles' path and cause particle impingement on the leeward side of the building. Also, downwash in front of the building on the windward face cause higher pollutant concentration at the lower levels of the building. Along the leeward face, the pollutants drifted upward also because of wind-structure interaction.

Although the pollutant density was adjusted to be heavier and lighter than air in this research, the results show that the pathway of pollutant for both windward and leeward remain unchanged. Evidently air pollutant dispersion around the building model is dominated by wind-structure interaction and buoyancy effect associated with the pollutant specific weight within the range tested only plays a minor role in the dispersion process.

References

BLOCKEN, B. 2015. Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. *Building and Environment,* 91**,** 219-245.

BLOCKEN, B., CARMELIET, J. & STATHOPOULOS, T. 2007. CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow. *Journal of Wind Engineering and Industrial Aerodynamics,* 95**,** 941-962.

ENGELHART, S., PLEISCHL, S., LÜCK, C., MARKLEIN, G., FISCHNALLER, E., MARTIN, S., SIMON, A. & EXNER, M. 2008. Hospital-acquired legionellosis originating from a cooling tower during a period of thermal inversion. *International Journal of Hygiene and Environmental Health,* 211**,** 235-240.

FLUENT, A. 2015. Theory Guide and User's Guide. *Ansys Inc, USA*.

FRANKE, J., HELLSTEN, A., SCHLUNZEN, K. H. & CARISSIMO, B. 2011. The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: a summary. *International Journal of Environment and Pollution,* 44**,** 419-427.

LIU, X. P., NIU, J. L., KWOK, K. C. S., WANG, J. H. & LI, B. Z. 2010. Investigation of indoor air pollutant dispersion and cross-contamination around a typical high-rise residential building: Wind tunnel tests. *Building and Environment,* 45**,** 1769-1778.

RAZAK, A. A., HAGISHIMA, A., IKEGAYA, N. & TANIMOTO, J. 2013. Analysis of airflow over building arrays for assessment of urban wind environment. *Building and Environment,* 59**,** 56-65.

YU, X., ZHU, B., YIN, Y., YANG, J., LI, Y. & BU, X. 2011. A comparative analysis of aerosol properties in dust and haze-fog days in a Chinese urban region. *Atmospheric Research,* 99**,** 241-247.

YU, Y., KWOK, K. C. S., LIU, X. P. & ZHANG, Y. 2017. Air pollutant dispersion around high-rise buildings under different angles of wind incidence. *Journal of Wind Engineering and Industrial Aerodynamics,* 167**,** 51-61.

ZHANG, Y., KWOK, K. C. S., LIU, X. P. & NIU, J. L. 2015. Characteristics of air pollutant dispersion around a high-rise building. *Environmental Pollution,* 204**,** 280-288.