A REVIEW OF INVESTIGATING AIRFLOWS AROUND BUILDINGS

Qian Xia[1] and Jianlei Niu [2]
[1] Department of Building Services Engineering, Hong Kong Polytechnic University, Hong Kong
Email: 09901550r@polyu.edu.hk
[2] Department of Building Services Engineering, Hong Kong Polytechnic University, Hong Kong

ABSTRACT

With the awareness of sustainability in the built environment, implementing natural ventilation to provide cooling has become revitalized. But maintaining a healthy indoor environment, especially for those high-rise residential buildings, could be an issue. The purpose of this review is to systematically review investigation methods (On-site measurements, physical scale modeling (wind tunnel and water channel experiments) and different computational fluid dynamics turbulence models (DNS, RANS and LES) for these practical concerns and knowledge gaps with regard to air flow around building and to indentify effective investigation methods with the ultimate goal to find optimum ventilation for HRR buildings. Physical scale modeling can be more controllable for airflow and building configurations, but the accuracies of them are still arguable. On-site measurement is a somewhat dear approach. Regarding to its accuracy, it still could under-predict the roof concentration. CFD is a fast, cost effective, risk reduction and safe method for examining the issue regarding to airflow and pollutant dispersion around the building. But the accuracies of each different turbulence model are different. DNS can provide the most accurate results in finite volume calculations; but it is impractical for simulations due to large computational requirements. LES can over-predict the lateral pollutant concentration in the wake region of the building and requires somewhat larger computational demand and times. Although RANS is the most commonly used model for pollutant dispersion and infection control issues, it over-predicts surface concentrations downwind of the sources. The research also discovered that the vertical cross contaminant dispersion has not been widely studied, especially on the high-rise buildings, and the concentration fluctuation rates in the unsteady atmospheric environment has been missed.

KEYWORDS

Airflows around buildings; Pollutant dispersion; On-site measurement; Wind tunnel; Water tunnel; Computational fluid dynamics (CFD)

INTRODUCTION

With the ever increasing living standard worldwide, centralized mechanical ventilation usually in combination with air conditioning has become a common practice for providing thermal comfort and indoor air quality by taking in controlled fresh air and discharging controlled exhausted air out. With these mechanical designs, a high investment cost and intensive energy usage cannot be avoided in order to maintain the indoor temperature and air flow rate at constant values. Approximately more than 60% of the total energy has been used for heating, ventilation and air-conditioning systems (HVAC) in serving residential buildings in most European countries (Orme 2001). However, with the finite energy is available on this planet, especially for fossil fuels. Therefore, Using natural ventilation has become one of the practical approaches to reduce the energy consumption for HVAC of buildings, since a naturally ventilated building is only 60% of that of an air-conditioned one (Kato *et al.* 1992; Ohba *et al.* 2001). Providing sufficiently fresh air through openable window area and the depth of the space is the basic principle of a naturally ventilated room. In Asian countries, it is also common that split-type or window-type air conditioners or VRV units provide air-conditioning, and ventilation is obtained via open windows. But opening windows could be one of the most common routines of airborne transmission. Therefore, in HRR, both window exhaust from the apartment and rangehood exhaust from the kitchen can re-entry through open-windows to the adjacent apartments. Airborne spread concept was first introduced by wells (1934, 1955) and Riley and O' Grady (1961) to culminate in the well-known

Wells-Riley equation (1997) for evaluating the effects of ventilation, filtration and other physical processes on the transmission of airborne diseases. With respect to vertical cross-floor air contamination, second hand tobacco smoke from the lower floor and fish-cooking fumes from lower floors are two of most obvious examples. Consequently, airflows around building blocks influence indoor air quality, pollutant dispersion in the surroundings, and airborne transmission of infectious diseases. To address these practical concerns, researchers worldwide have adopted a number of techniques and methods in their investigations. In this paper, we reviewed these investigation methods in the three categories of on-site measurements, wind/water tunnel physically-scaled experiments and CFD (Computational fluid dynamics) numerical simulations. Also, the issues concerned are classified as cross contamination within a single building block and among building arrays.

ON-SITE MEASUREMENTS

On-site measurements have been classified into two categories. One uses full scale measurements; another one utilizes physical scale measurements. Detailed measurements of the spatial and temporal distribution of climatic variables are possible differences between the full and physical scale structure.

Full scale measurement

Full scale measurement means testing flow and concentration around a single building, high-rise building or building array in prototype size and the under real atmospheric conditions. A number of researchers have investigated the pollutant dispersion with the full scaled on-site measurements. Mavroidis et al. (1999; 2000; 2007) released a dual source of ammonia and propane, normal and 45^0 to the mean wind direction upwind of a single cubic building, and deployed of a Flame Ionization Detector (FID) co-located with a Ultra-Violet Ion Collector (UVIC) to measure the concentration levels. The atmospheric stability conditions ranged from very stable to very unstable. The purpose of Mavroidis et al's (1999) study was to discover the potential influence of atmospheric stability conditions on dispersion behaviour of entrainment and detrainment in the wake region of an obstacle. It was found that the concentration in the recirculation region decayed exponentially and the decay duration (it is the time that the tracer gas is fully detrained at the wake) was longer in stable atmospheric conditions due to the lower wind speeds and higher concentrations in the wake region of the cube. Hence, it could be concluded that the rate of the gas detrainment from the wake region is mainly affected by the wind speed and much less by atmospheric stability. Mavroidis et al. (2007) summarized both the mean concentration and the concentration fluctuation statistics in the vicinity of an isolated building. Concentration fluctuation results showed that intermittency values increased as the source was displaced from the centerline. Meanwhile, it also implied the importance of concentration fluctuation studies, even at the wake region, because sudden peaks could occur when a source displaced further away from the centerline where gas entrainment was intermittent. The research also revealed that a dual source/receptor system technique was useful for pollutant dispersion investigation for providing data for statistical analysis and modeling. But investigations of the possibilities of this technique should be further studied. In terms of those densely built-up districts with buildings over 30 stories, the actual airflow around the buildings, particularly at lower floors, is much lower due to the obstructions around. On-site measurements had been reported by Niu and Tung (2008) on the virus-spread mechanisms of inter-flat or inter-zonal airflow through open-windows caused by buoyancy effects. For a multi-family building in sub-urban private residential area, with 3-storey and 15 residential units on each floor, a dual tracer gas of CO_2 and SF_6 were emitted simultaneously to examine the quantity of exhaust air coming out of the window of the lower floor that re-enters the open-window in the upper floor. CO_2 was for examining the ventilation rate. SF_6 was taken as a tracer of indoor pollutant originating from the lower floor and also used to quantify air change rate of the lower room. It was found that the air in the room upstairs contained up to 7% of the air directly from the room downstairs. The work proved that window flush with a flat façade can be a major route for the vertical spread of pathogen-containing aerosols in high-rise buildings.

A street canyon generally refers to a relatively narrow street between buildings that line up continuously along both sides. This unit is bounded by the ground surface at the bottom and the roof level at the top. It has a distinct climate where micro-scale meteorological processes dominate (Nakamura and Oke 1988) and the air ventilation and pollutant removal are solely through the roof level. Therefore, these unique micro-scale meteorological processes will affect the local air quality. The flow pattern inside street canyons depends on their geometry, especially, the building-height-to-street-width ratio (H/W, where H is the height of the building and W is the street width). For widely spaced buildings ($H/W<0.3$), the flow field between the buildings do not interact, which results in the isolated roughness flow (IRF) regime. At closer spacing ($0.3<H/W<0.7$) the wake behind the upwind building is disturbed by the recirculation created in front of the windward building, which is the wake interference flow (WIF) regime. At reduced spacing (H/W>0.7) the skimming flow (SF) regime is resulted in. IRF, WIF and SF regime have all been displayed in below.

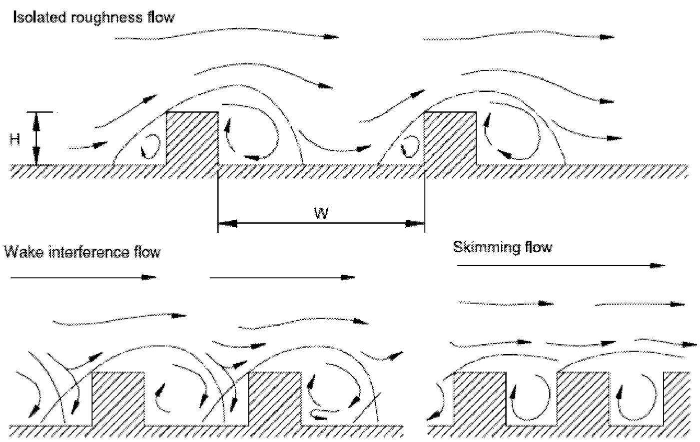

Figure 1 Three flow regimes with different building-height-to-street-width ratios H/W (Nakamura and Oke 1988).

Physical scale model on-site measurements

Some researchers use physical scale model under the real atmospheric environment conditions for studies. In this review, we call it physical scale model on-site measurements. It is one of the alternatives to the high expense full scale measurement, especially in the urban area with groups of obstacles. But this method may suffer from the inherent similarity problems, since the wind speed at the scaled-down model should be lower than that at the full scale model. Mavroidis *et al.* (2003) aimed for studying the plume entrainment behaviours from longitudinal, lateral and vertical source locations by using physical scale models for an isolated cube in the field. The experiment released a dual-gas with ammonia and propane 2.0H upwind of different cubes. The building with height H = 1.15m and diameter D = 1.15m, and a taller building with H = 2.3m at a nominal scale between 1/10 and 1/20. The results demonstrated that taller obstacles resulted in a reduction of ground level concentration when the tracer gas was released from upwind of the obstacles continuously.

More researches for physical scale model on-site measurements for building arrays have been conducted. Macdonald *et al.* (1997; 1998) released a tracer gas of pure-grade propylene (C_3H_6) as a plume dispersion upwind of 1/10 regular arrays of cubes in the field. In the experiment, the flow rate of the tracer gas was measured by a float-type variable area rotameter. UVIC detectors had been deployed to measure the propylene concentrations. Ultrasonic anemometers were placed at 1H height and 6 to 10H upwind of the arrays for measuring the mean and fluctuating velocity. Macdonald *et al.* (1998) illustrated that the ground concentrations were much higher than rooftop ones in the front rows of the array possibly due to the horseshoe vortex system around front row buildings. Higher lateral concentration profile could be found to the closer source. Macdonald's *et al.* (1997) study observed a greater initial dispersion plume was also found when releases were located inside the array than upwind of the array. Mavroidis *et al.* [(2001) also examined pollutant dispersion within building arrays by means of applying model obstacles with S/H = 1.5 (where S is the space between two consecutive array elements) at a scale between 1/10 and 1/20 in the field. A dual-gas with ammonia and propane had been released constantly 2.0H upwind of the obstacle at a height of 0.5H. The ammonia could be moved along three coordinate directions from the obstacle but the propane remained the same position. It suggested that mixing and dispersion were enhanced within the array.

WIND/WATER TUNNEL PHYSICAL SCALE MEASUREMENTS

To have steady weather conditions in on-site measurements is always difficult. A physical scale measurement in a simulated controlled environment can help us to overcome this problem. The physical scaled measurement shares similar principle as physical scale model on-site measurement, which means that by reducing the geometrical scale of a given domain and adjusting the reference parameters to reproduce measured data under the original full-scale conditions. Therefore, it has often been adopted as a complementary tool to numerical modeling and been proved especially useful in model development (Baker and Hargreaves 2001). Wind tunnel and water channel are two types of typical physical scale facilities in studying airflow around buildings.

Wind tunnel measurements

A wind tunnel is a device for directing airflow over an object for the analysis of the properties around it. Scaled building models may be used as objects in a wind tunnel. Three monitoring techniques should be involved in a wind tunnel experiment: flow visualization for revealing the possible flow patterns, tracer dispersion quantifying concentrations at receptor locations, and Laser Doppler Anemometry (LDA) for studying the flow in detail. Also, although pressure distribution may not be the target for air quality studies, it is an important parameter for estimating air-infiltration in envelopes and can also be used for the validation of numerical results. On the other hand, although wind tunnel might have the scaling difficulties, it can help us effectively approximate real atmospheric conditions in urban streets and investigate micro-scale pollutant dispersion around a building. To study the pollutant dispersion of an isolated building, tracer gas can be placed at different locations around a building in the wind tunnel. Ogawa *et al*. (1983a, 1983b); Saathoff *et al.* (1995); Yassin *et al.* (2005) studied the air flow patterns and concentration dispersion on and around a cube. The effect of increasing model scale to the mean concentrations of a source at the roof was also investigated (Saathoff *et al.* 1995). SF6 as tracer gas emitted from center of cube roof (Ogawa *et al*. 1983a, 1983b; Saathoff *et al.* 1995) and a 0.25m diameter stack with a height of 3m (Yassin *et al.* 2005). A two-colour Laser Doppler anemometer (LDA) was for measuring u and w of the flow (Ogawa *et al*. 1983a, 1983b). Hot-wire anemometers were used for measuring u, v and w of the flow (Ogawa *et al*. 1983a, 1983b; Saathoff *et al.* 1995). But Yassin *et al.* (2005) used an ultrasonic anemometer and a 3D LDA. Smooth upwind surface and $\theta = 0^0$ (wind direction) would generate reverse flow at the source position and high concentrations at the leading edge; but very rough surface and $\theta = 0^0$ could produce high rooftop concentrations (Ogawa *et al*. 1983a). The maximum concentration was found under stable conditions (Yassin *et al.* 2005). Rooftop pollutant concentration was overestimated by wind tunnel (Ogawa *et al*. 1983b). The changes in model scale did not vary much with leeward concentrations (Saathoff *et al.* 1995). Huber (1991, 1989) evaluated the concentrations downwind of a point source in the near wake of the model building. The buoyant ethane as tracer gas was emitted upwind and concentration (Huber 1991, 1989). Velocity and turbulence were obtained by x-array hot-wire probe with constant-temperature anemometer. Concentrations were measured by a FID for 2 minutes (Huber 1991, 1989). Four flow speeds and buildings sizes were considered (Huber 1991). Angles from -30^0 to 60^0 were to verify oblique orientation and W/H ratios from 2 to 22 to test building width effect. Concentrations were not significantly high near the building, whereas they were greater than 10H (where H is building height) at downstream distance (Huber 1991). The largest W/H=10 plume was the lateral plume from a point source near the center of the building (Huber 1989). In comparison, the vertical and horizontal cross contaminant around an isolated high-rise building has not been widely researched. The only one relevant paper has been found in the review is Liu *et al's* (2010) study. It adopted a 1:30 scale model of typical Hong Kong residential buildings for representing a 10-story in prototype into a 4m high x 5m wide x 41m long wind tunnel to examine the cross-contamination phenomenon of the re-entrance space with 0^0 and 90^0 wind directions. Tracer gas was continuously released by a flow meter at the constant flow rate as the exhausted room air at 3rd, 6th and 9th floor respectively. The measured points were taken in the re-entrance space of each floor. The research work presented that the pollutant could spread in both vertical direction and in the downward direction. Dispersion occurring in the horizontal direction had also been found.

A number of wind tunnel studies of pollutant dispersion phenomena in building arrays and street canyons were also carried out (Davidson *et al.* 1996; Macdonald *et al.* 1998; Yee *et al.* 2006). They studied flow field and plume dispersion around urban buildings by physical models. The W/H ratio for a fixed obstacle density had been particularly tested (Macdonald *et al.* 1998). Ethane (Davidson *et al.* 1996) and Methane (Macdonald *et al.* 1998) were released upwind of the building array. Yee *et al.* (2006) emitted pure ethylene upwind and within the building array. An x-array hot-wire, a pulsed-wire anemometer (Davidson *et al.* 1996), a type JBJ pulsed-wire anemometer mounted on an automated traverse gear (Macdonald *et al.* 1998) and crossed hot-wire anemometers coupled with a digital vane anemometer (Yee *et al.* 2006) were applied. A 1/20 scaled and a 1/200 scaled model (Davidson *et al.* 1996), a 1/50 and 1/100 scaled model (Macdonald *et al.* 1998; Yee *et al.* 2006) were all tested in different wind tunnels respectively. A number of Flame Ionization Detectors (FIDs) had been utilized for obtaining information for plume internal structure and mean concentration levels (Macdonald *et al.* 1998; Yee *et al.* 2006). But another additional FID used for measuring the background concentration of the tracer (Macdonald *et al.* 1998). The mean concentration of the plume was in a Gaussian form (Davidson *et al.* 1996; Macdonald *et al.* 1998). But concentration fluctuation within the plume was decreased by the small-scale and high-strength turbulence (Davidson *et al.* 1996). 2 to 3 times larger lateral and vertical spreads of plume measured from wind tunnel than those in the field and water channel (Yee *et al.* 2006). But the peak concentration was underestimated by wind tunnel.

Water tunnel measurements

Water and wind tunnel share the same principles and considerations. The major potential advantage of water tunnel over wind tunnel is its capability of simulating buoyancy effects. Salt bath modeling is a technique that uses water infused with salt solutions of various concentrations to act the same way as thermal buoyancy in a full scale structure. But dimensional analysis is still needed to maintain consistency between the model and prototype and providing a similar atmospheric boundary layer in water as in full scale experiments is still a challenge. Cheah *et al.* (1993); Bara *et al.* (1992) and Yee *et al.* (2006) studied the plume dispersion and contaminant profiles around the building by a released source located at the ground level. A 1/2000 scaled model was tested in a recirculating water channel (Cheah *et al.* 1993). Bara *et al.* (1992) used saline tracer technique from a ground level source in a water tunnel simulation of ground level plume to be examined in a stably rough surface atmospheric boundary layer. On the other hand, sodium fluorescein dye tracer was emitted as a ground-level source in the flow. The concentration was measured simultaneously at multi-points by the laser-induced fluorescence (LIF) technique of a 1/205 scale model under neutrally stratified atmospheric boundary layer in a water channel with a working section of 0.9m high x 1.5m wide x 10m long (Yee *et al.* 2006). The mean velocity was measured by a pitot tube (Cheah *et al.* 1993), a fibre-optic Laser Doppler velocimeter (LDV) powered by an argon-ion laser (Bara *et al.* 1992) and a Laser-Doppler anemometer in backscatter mode with a 16mV He-Ne laser (Yee *et al.* 2006). Turbulent velocity (Cheah *et al.* 1993) was also tested by single sensor and X-array, cylindrical fibre film probes with a two channel DISA 55M Series anemometer. A similar flow speed and a higher Reynolds number could be obtained in water channel. Therefore, mean and fluctuating concentrations could be accurately measured by a conductivity probe and flow can be visualized easily (Cheah *et al.* 1993). Yee *et al.* (2006) also agreed that the lateral and vertical plume and decay of concentration fluctuation intensity with increasing downwind distance could be reproduced well by water channel; the maximum mean concentration, concentration fluctuation intensity and its decay could be more accurately measured by a water channel than others. But it overestimated fraction of non-zero concentration periods near the edges of the plume (Bara *et al.* 1992).

NUMERICAL EXPERIMENTS/SIMULATIONS

CFD technique has started to be used since 1960s to analyze the issues regarding to fluid flow, heat transfer and associated phenomena (e.g. chemical reactions). More and more popular numerical simulation turns into a major tool for designers of fluid flow and thermal systems, which is attributed to the availability of faster and larger computers enabling us to solve complex fluid flow and thermal problems. In this review, we focus on its application in airflow and pollutant dispersion around buildings.

Direct Numerical Simulation (DNS)

DNS solves Navier-Stokes equations directly without the use of any turbulence models. It captures the mean flow and all of the relevant scales of turbulent motions, so extreme fine grids are required in consideration of the small eddies of an external flow. It appears that some essential flow features like the vortex behind a building can be best captured by DNS models. However, it was not amenable to use DNS in engineering applications due to the prohibitive memory and computational requirements. Consequently, data from reliable DNS are not usually available (Aberdi and Omidyeganeh, 2008).

Reynold-averaged Navier-Stokes numerical simulations (RANS) and Large eddy simulation (LES)

RANS are the oldest approach to turbulence modeling, which considers an integral approach for the whole turbulence spectrum so that turbulence modeling assumptions are required for the closures. But this approach does not require as large computing resources as DNS, reasonable results can still be provided. Thus, they have been the mainstay of engineering flow calculations over the last three decades (Versteeg and Malalasekera 2007). The attention of RANS models is focused on the mean flow and the effects of turbulence on mean flow properties. The standard $k-\varepsilon$ models, the RNG $k-\varepsilon$ models and the Realizable $k-\varepsilon$ model are three main types of RANS models. The most popular model is the standard $k-\varepsilon$ model developed by Launder and Spalding (1974). But Tominage and Stathopoulos (2009) pointed that the standard $k-\varepsilon$ model provides insufficient results for the concentration field around a cubic building due to its poor reproductivity of the basic flow structure, such as the

reverse flow on the roof. The best tested turbulence models was RNG $k-\varepsilon$ model. Mahjoub *et al.* (2003) also provided that, among the three first order $k-\varepsilon$ models, only the RNG $k-\varepsilon$ models could provide good results in the exit region and in the trailing zone of a jet, although they rendered identical results in the upstream and far downstream regions of a jet. Hence, the RNG $k-\varepsilon$ models had been widely used recently (Yakhot *et al.* 1992). However, recent works indicated that these models seem to be very promising in describing complex processes (Canepa 2004); while Lakehal and Rodi (1997) highlighted difficulties in downwash simulation by $k-\varepsilon$ models.
The premise of LES is to simulate the large-scale turbulent motions and approximate the small-scale motions through modeling. The effects on the resolved flow (mean flow plus large eddies) due to the smallest, unresolved eddies are included by means of a so-called sub-grid scale model. Because small-scale turbulence is modeled, LES requires considerably less computational resources and time than DNS. With the increasing numerical capability of modern computer, LES is more and more widely used in predicting airflow. The key to successfully predicting airflow by the LES is to accurately represent the unresolved sub-grid-scale (SGS) motion.
RANS modeling might be inappropriate for some cases due to the averaging procedure used in these models. Johnson and Hunter (1998) found that $k-\varepsilon$ model predicted stronger concentration gradients in the wake region than those in wind tunnel experiments. Consequently, $k-\varepsilon$ model underestimates turbulent diffusion and turbulence intensities in the wake. Gao *et al.* (2008) revealed that RNG $k-\varepsilon$ models were not able to examine the turbulent fluctuations and instantaneous air exchanges through the opening, especially in wind-driven single-side natural ventilation. Meroney *et al.* (1999) also concluded that over-predicted surface concentrations downwind of the sources emitted in the vicinity of bluff bodies could be obtained from $k-\varepsilon$ models (since RANS $k-\varepsilon$ models cannot produce the intermittent nature of bluff-body flow). Tominaga *et al.* (2008) found that the $k-\varepsilon$ model could not reproduce the reverse flow on the roof, which was corrected by revised $k-\varepsilon$ models. But modified $k-\varepsilon$ models overestimated the reattachment length behind the building, which could be improved by LES. Also a good agreement was found between LES simulation and the wind tunnel measurement in respect of mean concentration, fluctuation intensity, peak concentration values, the windward length of a cavity region behind the building, the measured mean velocity and turbulent intensity (Stathopoulos and Li 1997). The major advantages of LES are its capability of handling the unsteadiness and intermittency of the flow as well as providing detailed information on the turbulence structure. But without inflow turbulence, the LES could produce too much vortex shedding behind the building, which would overestimate the pollutant fluctuation in the lateral direction in the wake region. However, owing to large demand in computational resources for the LES technique, it used to be regarded mainly as a research tool rather than a practical way to engineering applications (Versteeg and Malalasekera 2007). Consequently, to study the problems with a complex geometry or a large-scale site, LES has difficulty due to limitations of available memory and computing speed at present. With the fast development of computer capacity and sophisticated numerical models, comprehensive numerical investigation of complex buildings structures and urban layouts will not be far away. So the LES modeling would be a realistic choice in the near future.

CONCLUSION

In this review, it is found that the airflow, pollutant dispersion and infection spread risks in an isolated building and building arrays have been studied by means of field measurements, physical scale measurements and numerical simulations. Physical scale modeling can be more controllable in terms of stable airflow boundary conditions. But its accuracy is compromised due to due to its limitations in simulating the real atmospheric boundary layer in terms of the turbulence characteristics. The physical scale model on-site measurement, having similar problems as wind tunnel tests, can over-predict the pollutant concentrations along the roof center and at leeward façade.
Theoretically speaking, DNS can provide the most accurate results in finite volume calculations because of the consideration of small eddies, but due to its large computational requirements, it is impractical to use for airflow problems around buildings. The LES can solve the unsteadiness and intermittency of flow separation, but it over-predicts the lateral pollutant concentration in the wake region of a building. The RANS models, the most commonly used, currently have obvious deficiencies in predicting the separation at a building roof and the wake flow at the leeward of a building.
In terms of the practical concerns to be resolved, in spite of researches up to the present moment have revealed many phenomena about an isolated building or groups of buildings, but they still emphasize on horizontal contaminant dispersion and the effects to the horizontal neighborhood, and much less research is focused on the vertical contaminant dispersion, especially in high-rise buildings. Moreover, strong upward air motion dominated by

buoyancy effects by solar radiation under low wind condition has not been widely studied. Hence, combined buoyancy effects and wind forces should be considered. And this phenomenon would be difficult, if not impossible, to be investigated in wind tunnel tests. With the increasing computing power, LES models are perceived to play more roles in studying this practical problem.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the financial support by a Research Studentship from The Hong Kong Polytechnic University.

REFERENCES

Aberdi, J. and Omidyeganeh, M. (2008). "Modeling of pollution dispersion around a cubic obstacle", BBAA VI International Colloquium on Bluff Bodies Aerodynamics & Applications, Milano, Italy.

Baker, C.J. and Hargreaves, D.M. (2001). "Wind tunnel evaluation of a vehicle pollution dispersion model", Journal of Wind Engineering and Industrial Aerodynamics, 89, 187-200.

Bara, B.M. Wilson, D.J. and Zelt, B.W. (1992). "Concentration fluctuation profiles from a water channel simulation of a ground-level release", Atmospheric Environment, Part A. General Topics 26(6), 1053-1062.

Canepa, E. (2004) "An overview about the study of downwash effects on dispersion of airborne pollutants", Environmental Modelling & Software, 19, 1077-1087.

Cheah, S.C. Cleaver, J.W. and Millward, A. (1983) "Water channel simulation of the atmospheric boundary layer", Atmospheric Environment, 17(8), 1439-1448.

Davidson, M.J. Mylne, K.R. Jones, C.D. Phillips, J.C. Perkins, R.J. Fung, J.C.H. and Hunt, J.C.R. (1995). "Plume dispersion through groups of obstacles — A field investigation." Atmospheric Environment, 29(22), 3245-3256.

Gao, N.P. Niu, J.L. Perino, M. and Heiselberg, P. (2008). "The airborne transmission of infection between flats in high-rise residential buildings. Tracer gas simulation", Building and Environment, 43, 1805-1817.

Johnson, G.T. and Hunter, L.J. (1998). "Urban wind flows: wind tunnel and numerical simulations-a preliminary comparison", Environmental Modelling and Software, 13(3-4), 279-286.

Kato, S. Murakami, S. Mochida, A. Akabayashi, S. and Tominaga, Y. (1992). "Velocity-pressure field of cross ventilation with open windows analysed by wind tunnel and numerical simulation", Journal of Wind Engineering and Industrial Aerodynamics, 41–44(1-3), 2575 – 2586.

Lakehal, D. and Rodi, W. (1997). "Calculation of the flow past a surface-mounted cube with two-layer turbulence models", Journal of Wind Engineering and Industrial Aerodynamics, 67-68, 65-78.

Launder, B.E. and Spalding, D.B. (1974) "The numerical computation of turbulent flows", Computer Method in Applied Mechanics and Energy, 3, 269–289.

Li, Y. and Stathopoulos, T. (1997). "Numerical evaluation of wind-induced dispersion of pollutants around a building", Journal of Wind Engineering and Industrial Aerodynamics, 67&68, 757-766.

Liu, X.P. Niu, J.L. Kwok, K.C.S. Wang, J.H. and 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(8), 1-10.

Macdonald, R.W. Griffiths, R.F. and Cheah, S.C. (1997). "Field experiments of dispersion through regular arrays of cubic structures", Atmospheric Environment, 31(6), 783-795.

Macdonald, R.W. Griffiths, R.F. and Hall, D.J. (1998). "A comparison of results from scaled field and wind tunnel of dispersion in arrays of obstacles", Atmospheric Environment, 32(22), 3845-3862.

Mavroidis, I. and Griffiths, R.F.L. (2001) "Local characteristics of atmospheric dispersion within building arrays", Atmospheric Environment, 35, 2941-2954.

Mahjoub, S.N. Mhiri, H. EL, G.S. Le Palec, G. and Bournot, P. (2003). "Three-dimensional numerical calculations of a jet in an external cross flow. application of dispersion of pollutants", Journal of Heat Transfer. Transactions of the ASME, 125, 510-522.

Mavroidis, I. and Griffiths, R.F. (1999). "Jones CD, Biltoft CA. Experimental investigation of the residence of contaminants in the wake of an obstacle under different stability conditions", Atmospheric Environment, 33, 939-949.

Mavroidis, I. Griffiths, R.F. and Hall, D.J.(2000). "Investigation of building-influenced atmospheric dispersion using a dual source technique", Environmental Monitoring and Assessment, 65, 239-247.

Mavroidis, I. Griffiths, R.F. and Hall, D.J.(2003). "Field and wind tunnel investigations of plume dispersion around single surface obstacles", Atmospheric Environment, 37(21), 2903-2918.

Mavroidis, I. Andronopoulus, S. Bartzis, J.G. and Griffiths, R.F. (2007). "Atmospheric dispersion in the presence of a three-dimensional cubical obstacle: Modelling of mean concentration and concentration fluctuations", Atmospheric Environment, 41(3), 2740-2756.

Meroney, R.N. Leith, B.M. Rafailidis, S. and Schatzmann, M. (1999). "Wind-tunnel and numerical modeling of flow and dispersion about several building shapes", Journal of Wind Engineering and Industrial Aerodynamics, 81, 333-345.

Nakamura, Y. and Oke, T.R. (1988). "Wind temperature and stability conditions in an east-west oriented urban canyon", Atmospheric Environment, 22, 2691-2700.

Niu, J.L. and Tung, T.C.W. (2008). "On-site quantification of re-entry ration of ventilation exhausts in multi-family residential buildings and implications", Indoor Air, 18, 12-26.

Ohba, M. Irie, K. and Kurabuchi, T. (2001). "Study on airflow characteristics inside and outside a cross-ventilation model, and ventilation flow rates using wind tunnel experiments", Journal of Wind Engineering and Industrial Aerodynamics, 89(14—15), 1513—1524.

Orme, M. (2001). "Estimates of the energy impact of ventilation and associated financial expenditures" Energy and Buildings, 33, 199-205.

Ogawa, Y. Oikawa, S. and Uehara, K. (1983). "Field and wind tunnel study of the flow and diffusion around a model cube—I. Flow measurements." Atmospheric Environment, 17(6),1145-1159.

Ogawa, Y. Oikawa, S. and Uehara, K. (1983). "Field and wind tunnel study of the flow and diffusion around a model cube--II. Nearfield and cube surface flow and concentration patterns." Atmospheric Environment, 17(6), 1161-1171.

Riley, R.L. and O'Grady, F. (1961). "Airborne infection – transmission and control", New York, The MacMillan Company.

Saathoff, P.J, Stathpoulos, T. and Dobrescu M. (1995). "Effects of model scale in estimating pollutant dispersion near buildings." Journal of Wind Engineering and Industrial Aerodynamics, 54&55, 549-559.

Stathopoulos, T. and Li, Y. (1997). "Numerical evaluation of wind-induced dispersion of pollutants around a building", Journal of Wind Engineering and Industrial Aerodynamics, 67&68, 757-766.

Tominaga, Y. Mochida, A. Murakami, S. and Sawaki, A. (2008). "Comparison of various revised $k-\varepsilon$ models and LES applied to flow around a high-rise building model with 1:1:2 shape place within the surface boundary layer", Journal of Wind Engineering and Industrial Aerodynamics, 96, 389-411.

Tominaga, Y. and Stathopoulos, T. (2009). "Numerical simulation of dispersion around an isolated cubic building: Comparison of various types of $k-\varepsilon$ models", Atmospheric Environment 43, 3200-3210.

Versteeg, H.K. and Malalasekera, W. (2007). "An introduction to computational fluid dynamics—the finite volume method (Second Edition)", Pearson Education Ltd, Harlow, England.

Wells, W.F. (1934). "On air0borne infection study: droplets and droplet nuclei", Am. J. Hyg, 20, 619-627.

Wells, W.F. (1995). "Airborne contagion and air hygiene: an ecological study of droplet infection", Cambridge, MA, Harvard University Press.

Yakhot, V. Orzag, S.A. Thangam, S. Gatski, T.B. and Speziale, C.G. (1992). "Development of turbulence models for shear flows by a double expansion technique", Physics of Fluids A, 4(7), 1510–1520.

Yee, E. Gailis, R.M. Hill, A. Hilderman, T. and Kiel, D. (2006). "Comparison of wind-tunnel and water-channel simulation of plume dispersion through a large array of obstacles with a scaled field experiment", Boundary-Layer Meteorol, 121, 389-432.

Yu, I.T.S. Li, Y.G. Wong, T.W. Tam, W. Chan. A.T. Lee, W.H.J. Leung, D.Y.C. and Ho, Y. (2004). "Evidence of airborne transmission of the server acute respiratory syndrome virus", Journal of Medicine, 350,1731-1739.