Evaluation of k-epsilon models for simulating wind-induced mean airflow field and dispersion around a rectangular multi-storey building

D. Mu, N. Gao
School of Mechanical Engineering
Tongji University, Shanghai, 200092, China

SUMMARY
The environment within an isolated building is a great concern for inhabitants since people spend more than 80% of their time indoors. In this paper, we use computational fluid dynamics (CFD) method and wind tunnel tests to analyze the airflow field and pollutant transmission routes within a typical rectangular multi-storey building with openings. The Reynolds-averaged Navier-Stokes (RANS) approach was employed in the numerical calculation. The performance of three k-ε turbulence model were evaluated. It was found that the predicted surface pressure coefficient distributions by realizable k-ε model agree well with experiment data, while the prediction of pollutant concentration field was underestimated. The pollutant source released from the windward stagnation zone could spread horizontally and vertically, also disperse from windward to downward units. The source unit and the immediate upper and lower flats were quite vulnerable. The effects of altering the value of the turbulent Schmidt number (ScT) in the realizable k-ε model were also examined. It was showed that the concentration distribution results are sensitive to the value of ScT.

INTRODUCTION
The occupants in urban area have exceeded 50%, so a trend toward high-density development of multi-storey residential estate presents to adapt the housing problem. City dwellers spend more than 80% of their time indoors (Klepeis 2001), notably over half of this time is staying in their homes. All these lead to an increasing concern to ensure a better indoor environment quality related to thermal comfort, health and productivity issues. Proper ventilation design in buildings can create a comfortable and healthy indoor environment. Natural ventilation is an important strategy to replace indoor air with fresh outdoor air. To understand the wind-induced airflow patterns is helpful for predicting the airborne transmission route around buildings.

The indoor air quality could be the result of the joint effect of both inside and outside the building. Depending on a great number of ingredients include the airflow pattern, source location and the pollutant properties, etc. (Tominaga 2016), extensive research work on near-field pollutant dispersion by (semi-) empirical models, on-site/laboratory experiment and computational fluid dynamics (CFD) were carried out (Xia 2014). With regard to the numerical method, various turbulence models have been tested to develop understanding and proper modelling techniques for the flow and dispersion around buildings. Steady RANS (Reynolds-averaged Navier-Stokes) model is the most widely used approach for dispersion modelling in many applied studies refer to actual buildings. The most commonly used method to evaluate the CFD models is through the comparisons between CFD results and experimental data.

The airflow patterns within and around a cubic isolated body with and without openings have been thoroughly revealed (ASHRAE 2015, Jiang et al. 2003). However, the airflow patterns related to multi-storey buildings with openings are still lack of investigation. The purpose of present work is to explore the airflow patterns in and around a rectangular six-storey building with single-sided ventilation. And then investigate the pollutant concentration field within the building. We carried out a set of wind tunnel experiments to measure the mean pressure coefficients on building surfaces in the reduced scale building and the mean concentration distribution along specific façade. Then the numerical method was employed to reveal the visualized airflow patterns. The evaluation work on the performance of k-ε turbulence model was done with the assist of experiment results.

METHODS
Wind tunnel experiment
The experiment was performed in the TJ-1 boundary layer wind tunnel in the State Key Laboratory of Civil Engineering for Disaster Prevention, Tongji University, China. It is a low speed open circuit one. The dimension of test section in this wind tunnel is 1.8m high, 1.8m wide and 12m long. The atmospheric boundary layer flow was generated by specific spires, grills and roughness elements, as shown in Fig. 1(a). The power law exponent of velocity profile was 0.22 (ASHRAE 2009). The turbulence intensity of the approaching wind flow was in a range of 10% to 20%, which were measured by the 3-D Cobra probe. The dimensionless mean velocity profile $U_{ref}$ defined as $U(h)/U_{ref}$ was presented to meet the similarity criteria of approaching boundary conditions, and the characteristic velocity $U_{ref}$ was set as the velocity at building height. The normalized velocity profile and measured turbulence intensity were presented in Fig. 1(b). The height was normalized by $h/H$. For a scaled modelling of airflow and plume dispersion in the wind tunnel study, a series of similarity requirements between prototype and scaled model should be examined carefully as reported in literature (Snyder 1972). Some of the similarity parameters can be neglected due to their poor relative importance when simulating pollutant transmission in and around buildings without thermal effect, while the Reynolds number must be paid attention to (Uehara et al. 2003). For the present tests, the mean flow velocity $U_{ref}$ measured at the building height, which was 0.59m in the scaled model, had a value of 2.89m/s. Hence, the building Reynolds number, $Re = U_{ref}Hv$, could be over 15,000 (Meroney 2004) and up to $1.15 \times 10^5$, assuring that the test results were independent of the Reynolds number.

In consideration of the blockage ratio and the capability to capture the airflow patterns and tracer gas dispersion behaviour, a 1:30 scaled hypothetical rectangular building.
model was structured. The building contains six floors and each floor contains a corridor with three units at each side. Each unit has a window on the exterior wall. All the units have the same dimensions, as shown in Fig. 2(a). Geometric similarity is satisfied and the blockage ratio is 5.46%.

![Image](https://example.com/image1)

**Figure 2(a). The building model**

![Image](https://example.com/image2)

**Figure 1. Photo of experiment configuration and approaching wind characteristics**

![Image](https://example.com/image3)

**Figure 2(b). Pressure taps for each floor**

For the analysis of transmission characteristics, the tracer gas sulphur hexafluoride (SF$_6$) was employed to simulate the pollutant. SF$_6$ was released at a constant flow rate of 15ml/s during the concentration tests. The release and detection of SF$_6$ were separately controlled by INNOVA 1303 and 1412i, manufactured by LumaSense Technologies, Inc. The source and sample locations were also shown in Fig. 2(a). The mean pressure coefficient distributions on the building façade were beneficial to analyse the flow pattern and the tracer gas transmission routes. Totally, 168 pressure taps with the diameter of 0.5mm were used to measure the building surface pressure, which means each floor has 28 test positions and the sequence numbers are shown in Fig. 2(b). The pressure taps were connected to three 64-channel electronic pressure scanners, which were positioned under the wind tunnel floor. The calibration work for each aforementioned instrument was carried out and repeated before and after the corresponding test process to ensure the stability of the equipment and reduce the systemic error.

**Configuration of simulation**

The CFD method was adopted by using a commercial program, Fluent 14.5. The computational domain size designed for CFD applications on the airflow and dispersion around building plays an important role on successful prediction. It should be large enough to generate correct flow around the building. However, if the domain is too large, the computing time will be significantly increased. Based on Hall’s (1997) work, the domain size was carefully designed and shown in Fig. 3, considering both efficiency and accuracy.

![Image](https://example.com/image4)

**Figure 3. The computational domain**

Three sets of structured hexahedral cells, respectively $2.6 \times 10^6$ grids, $5.0 \times 10^6$ grids, $6.8 \times 10^6$ grids were constructed in the computational domain. The grids in the near-building region were finer than those in the distant region, as shown in Fig. 4. The mesh independence test results by RNG $k$-$\varepsilon$ (Yakhot et al. 1992) model are shown in Fig. 5. It’s obvious that the coarsest mesh shows inconsistent results with the other two, especially in the upwind and building roof regions. Large differences show in the building wake region since the RANS simulations could not predict the flow field well in this region. Considering the accuracy and time consumption, the mesh system with $5.0 \times 10^6$ grids was used for the following calculations.

Three turbulence model, standard $k$-$\varepsilon$ (Launder and Spalding 1974), RNG $k$-$\varepsilon$ and realizable $k$-$\varepsilon$ (Shih et al. 1995) models were employed to simulate the turbulent effect. The standard wall function was used with $30 < y^+ < 300$. The approaching wind profile obtained from the experiment was used as numerical inflow condition. In the simulation, the mass fraction of the pollutant is predicted through the solution of the advection-diffusion equation. The turbulent Schmidt number is known to have a large influence on the simulation of dispersion. $Sc=0.7$ was used for most of early CFD
studies for the turbulent mass diffusion. Blocken et al. (2008) tried several values of $Sc_t$ in the CFD simulation of dispersion from a rooftop vent on an isolated cubic building, and $Sc_t=0.3$ was chosen in the studies of dispersion around a low-rise rectangular building with rooftop structures. For a better comparison between experiment and simulation, four turbulent Schmidt number, 0.1, 0.3, 0.5, 0.7 were set in the present study.

For all the simulations, the SIMPLE algorithm was adopted to handle the velocity-pressure coupling and second order upwind scheme was used for spatial discretization. Convergence was assumed to be obtained when the scaled residuals reach $10^{-5}$ and the monitoring variables were steady.

Data analysis

The pressure coefficient distribution along the building façades could be established based on the measured pressure at the test points arranged along the exterior walls of the building. The pressure coefficient is defined as,

$$P_c = \left( \frac{p_s - p_{ref}}{0.5 \rho U_{ref}^2} \right)$$

where $p_s$ refers to the measured surface pressure, $p_{ref}$ is the reference static pressure examined by a Pitot-static probe at the building height.

For the analysis and comparisons of pollutant concentration distribution characteristics under different source positions, the measured tracer gas concentrations were normalized by the following equation,

$$K = \frac{100C}{C_s}$$

where $K$ represents the normalized concentration, $C$ refers to the measured tracer gas mass concentration of SF$_6$ and $C_s$ is the measured source mass concentration. Eq. (2) means the normalized concentration at the source is 100. The dimensionless index implies the relative differences of tracer gas concentrations between source and sample points.

RESULTS AND DISCUSSIONS

Pressure coefficient distribution

In this part, the condition on the windward façade was mainly paid attention to. The comparison between simulative mean pressure coefficients and the experiment data at three horizontal test lines were shown in Fig. 6, located at 1st, 3rd, 5th floor, respectively. According to Eq. (1), the pressure coefficient is a relative quantity. The positive and negative
values have a relation with the original measured value and the reference static pressure. Through the comparison between experimental data and three turbulent model simulation results in Fig. 6, it can be seen that the standard \( k-\varepsilon \) model obviously over-predicts the pressure coefficient values. The results obtained from realizable \( k-\varepsilon \) model and RNG \( k-\varepsilon \) model were close in Fig. 6(a) and Fig. 6(c), while the realizable \( k-\varepsilon \) model provided a relatively closer profile in Fig. 6(b). In general, the profile shape of pressure coefficient distributions along three test lines were similar. The measured data showed that with the height going up from the first floor to the fifth floor, the \( P_c \) values mainly increased.

![Image](https://via.placeholder.com/150)

**6(a) Horizontal lines at the first floor (1→8)**

![Image](https://via.placeholder.com/150)

**6(b) Horizontal lines at the third floor (1→8)**

![Image](https://via.placeholder.com/150)

**6(c) Horizontal lines at the fifth floor (1→8)**

*Figure 6. Pressure coefficient distributions at three horizontal lines on “U” façade*

To have a visualized understanding, the contour map of pressure coefficient from experiment data and realizable \( k-\varepsilon \) model were show in Fig. 7. The pressure coefficient distribution reflected a good symmetric tendency on the windward and leeward façades because of the symmetric geometry feature of the building. The highest \( P_c \) value was up to 0.8 around the window area of UM5th unit, forming the stagnation zone. The outflowing air from the stagnation area could spread to all around. Most \( P_c \) values were positive on the windward façade, while several negative ones showed at the lower floors near the edge. The qualitative \( P_c \) distribution reflected from experiment and realizable \( k-\varepsilon \) model agreed well at most area of “U” façade. Main discrepancies exist in the near-edge region, especially the top and bottom edge. Since the pressure taps were limited in the experiment, the interpolation of contour map has inaccuracy.

![Image](https://via.placeholder.com/150)

**7(a) Contour map from experiment**

**7(b) Contour map from realizable \( k-\varepsilon \) model**

*Figure 7. Pressure coefficient distributions at “U” façade*

Fig. 8(a) shows the airflow patterns on the windward side, i.e. “U” façade, which was obtained from realizable \( k-\varepsilon \) model. The airflow pattern contains an upwash region, a stagnation zone, a downwash region and an upwind vortex near the ground. The stagnation zone is at the fifth floor obviously. Fig. 8(b) gives the airflow patterns in horizontal section at the source location height. The approaching wind blowing to the windward side move laterally and then separate from the building edge. Some vortexes formed in each unit and the corridor resulting from the open windows.
Mean concentration profiles

The predicted normalized concentration profiles along the centre vertical line of UM column are compared with the experimental results, as shown in Fig. 9. The highest value is 100 at the source location, which was the centre of windows’ lower limb at UM5th unit. The numerical simulation illustrated the basic transmission tendency and features. The concentration levels along the test line decreased with the distance from the source location. All the CFD results under-predicted the concentration obtained by the wind tunnel tests. The differences of the concentration distribution results are sensitive to the value of $S_c$. The concentration was an intrusive tube with suction effect rather than a contact probe. The perturbation induced by the tube during the sample process could possibly cause small vortexes around the tube, resulting in overvalued concentration. In addition, the concentration field was inherently fluctuant during the experiments (Liu et al.), the concentration data showed in Fig. 9 was respectively a mean value averaged by several times of sample, while the RANS approach is built to represent the mean time values of the flow and scalar parameters, which was one of the reasons that deviations exist in the simulation and experiment. Besides, compared with velocity gradient, the concentration gradient is larger, which is not easy to get a precise distribution field using the time average method in simulation.

CONCLUSIONS

In the present study, both numerical and experimental methods were used to analyse the airflow and concentration field within a rectangular multi-storey building with open windows. The performance of three types of turbulence models were examined by comparing the pressure coefficient and concentration distribution on the windward side with experiment data. The steady flow patterns reflected by the RANS models could be a qualitative reference to help analysing the gaseous pollutant dispersion paths. The qualitative velocity and pollutant concentration distribution could be quite visualized given by $k$-$\varepsilon$ model. The predicted surface pressure coefficient distributions by realizable $k$-$\varepsilon$ model were consistent with experiment data, while the quantitative concentration distribution was underestimated both on upward and downward dispersion. It was also found that the concentration distribution results are sensitive to the value of $S_c$ in the realizable $k$-$\varepsilon$ model. One thing is for sure, the source unit and the immediate upper and lower ones could get high risks of infection. Further investigations on the wind-induced pollutant transmission and concentration field within experiments, the instrument used for the sample concentration was an intrusive tube with suction effect rather than a contact probe. The perturbation induced by the tube during the sample process could possibly cause small vortexes around the tube, resulting in overvalued concentration. In addition, the concentration field was inherently fluctuant during the experiments (Liu et al.), the concentration data showed in Fig. 9 was respectively a mean value averaged by several times of sample, while the RANS approach is built to represent the mean time values of the flow and scalar parameters, which was one of the reasons that deviations exist in the simulation and experiment. Besides, compared with velocity gradient, the concentration gradient is larger, which is not easy to get a precise distribution field using the time average method in simulation.
such an isolated multi-storey building with openings by more precise numerical method are worth of exploring.

Figure 10. Tracer gas mass fraction from realizable $k$-$\varepsilon$ model when $Sc=0.1$

ACKNOWLEDGEMENT

This research is funded by the Natural Science Foundation of China (No.51278348) and the Fundamental Research Funds for the Central Universities of China.

REFERENCES


