Numerical simulations of gas dispersion in a residential area

Takenobu Michioka*, Ayumu Sato*
*Central Research Institute of Electric Power Industry (CRIEPI), Chiba, Japan

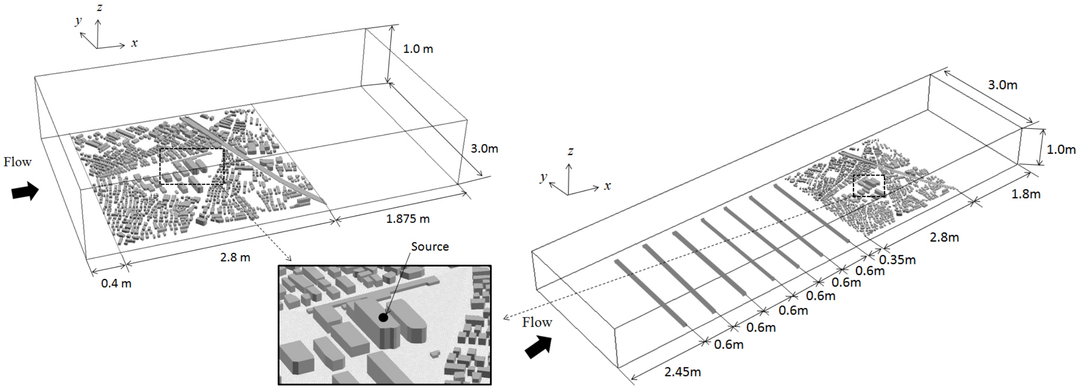
Abstract

The performance of a Reynolds-averaged Narvier-Stokes simulation (RANS) and a large-eddy simulation (LES) for gas dispersion in a residential area is investigated. RANS can capture the mean flow and the turbulent intensity, but can not reproduce gas dispersion unless a turbulent Schmidt number is properly chosen because gas dispersion is affected not only by the mean flow, but also by the turbulent scalar flux. On the other hand, LES can accurately represent the flow and concentration field.

Key words: Numerical simulations, gas dispersion, RANS, LES

1. INTRODUCTION

With the increasing availability of powerful supercomputers, numerical simulations have become a very attractive tool for simulating transport and dispersion of airborne materials in a residential area. Reynolds-averaged Navier-Stokes simulation (RANS) and large-eddy simulation (LES) have often been used for predicting velocity and concentration fields in a residential area. Flaherty et al. (2007) applied RANS to determine the dispersion of pollutant in Oklahoma City, where a field campaign was conducted (Joint Urban 2003). They showed that a single large building can have a substantial effect on the flow field in an urban area, although a large number of short buildings have a relatively small effect. They also found that the k-ε model underestimates both the near-surface wind speed and the turbulent kinetic energy. Hendricks et al. (2007) also performed RANS for Joint Urban 2003, and showed that the meteorological parameters input into the simulation are the most important factor for determining the urban flow and turbulence fields, which drive scalar dispersion. Hanna et al. (2006) applied five different CFD models (4 RANSs and 1 LES) to the same urban atmospheric boundary layer scenario in New York City, and indicated that these models produce similar wind flow patterns and provide results in good agreement with flow observations. Thus, the previous simulations on scalar dispersion in a residential area have only been compared with field observations. To assess the performance of a model in predicting scalar dispersion accurately, however, the results need to be compared with accurate wind tunnel data before being compared with field observations, which include a large margin of error. In this paper we focus on the performance of RANS and LES for predicting gas dispersion in a residential area by comparing the results of the simulations with data obtained from a wind tunnel.



(a) RANS (b) LES

Fig. 1 Schematic diagram of computational region.

2. NUMERICAL SIMULATIONS

LES and RANS were applied to simulate tracer gas dispersion released from the roof of the building in Komae Research Laboratory at CRIEPI (Central Research Institute of Electric Power Industry) in Japan, where wind tunnel experiments were conducted by Sato et al. (2007). Figure 1 shows a schematic diagram of the computational domains used in (a) RANS and (b) LES. The computational domains have dimensions of 5.075 x 1.0 x 1.0 m in RANS and 11.0 x 1.0 x 1.0 m in LES in the streamwise (x), spanwise (y), vertical (z) directions, respectively. The origin of the coordinate axes is set at the bottom of the computational domain 7.8 m and 1.8 m downstream of the entrance in LES and RANS, respectively, and at the spanwise center, which is at the same location in both simulations. The inlet conditions for the velocity and the turbulent kinetic energy in RANS were based on the results obtained in wind tunnel experiments over a flat plate without any obstacles. A laminar flow with a boundary height of 0.01 m was given as the inflow condition in LES, and to generate unsteady turbulent flows L-shaped roughness elements with height of 0.005 m were set at x = -5.35 m and 6 roughness elements with a height of 0.005 m were set from x = -4.75 to x = -1.75 at regular intervals of 0.6m. The tracer gas was released from the model stack, which was a rectangular pipe with 2.0 mm sides, at x=y= 0.0 m, z = 0.1215 m which is over the six-story building. The free stream velocity was 3.0 m/s and the vertical exhaust velocity of the gas was 2.2 m/s. The CFD code "FrontFlow/red" was employed (Unemura et al. 2004). The simulations are based on a finite volume method with an unstructured grid system to resolve the flow structure in a complex geometry. The computational grid system consisted of prismatic meshes in most of the area around the laboratory and tetrahedral meshes in other areas. The total grid points were about 135 million in RANS and about 151 million in LES. Slip boundary conditions were imposed on the velocity components on the upper and side walls. The wall function technique for velocity was applied to the ground and building surfaces. In LES, the convection term in the momentum equation was discretized by a second-order central scheme blended with a 5% first order upwind scheme, and the convection term in the mass conservation equation was discretized by a second order upwind scheme. The other terms were estimated using a second order central scheme. In RANS, the convection terms in both the momentum and mass conservation equations were discretized by a first order upwind scheme and the other terms were estimated using a second order central scheme. The standard Smagorinsky model was used for the sub-grid scale model in LES and the standard k-ε model was used for turbulence closure model in RANS.

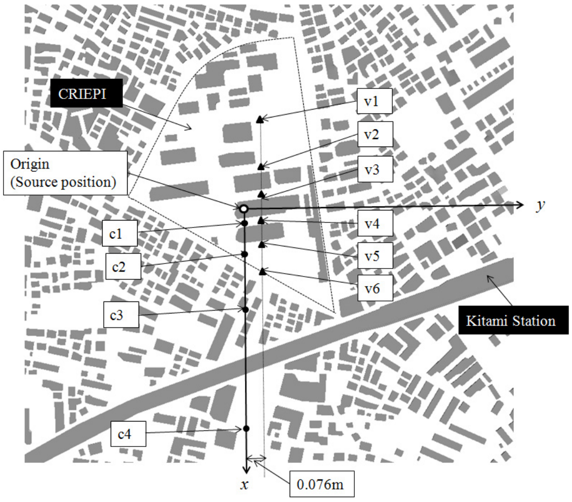


Fig. 2 The locations where the obtained velocity and concentration were observed with the wind tunnel.

3. RESULTS

Figure 2 shows the locations where the obtained velocity and concentration were compared with the wind tunnel data. The velocities were compared at six points with y = 0.076 m (v1-v6), and the concentrations were compared at four points with y = 0.0 m (c1-c4), where the velocity and the concentration were measured in the wind tunnel.

Figure 3 shows the vertical distributions of time-averaged streamwise velocities and the rms values of streamwise velocity fluctuations at six points (v1-v6) shown in Fig. 2. The squared value of streamwise velocity fluctuation in RANS was estimated using,

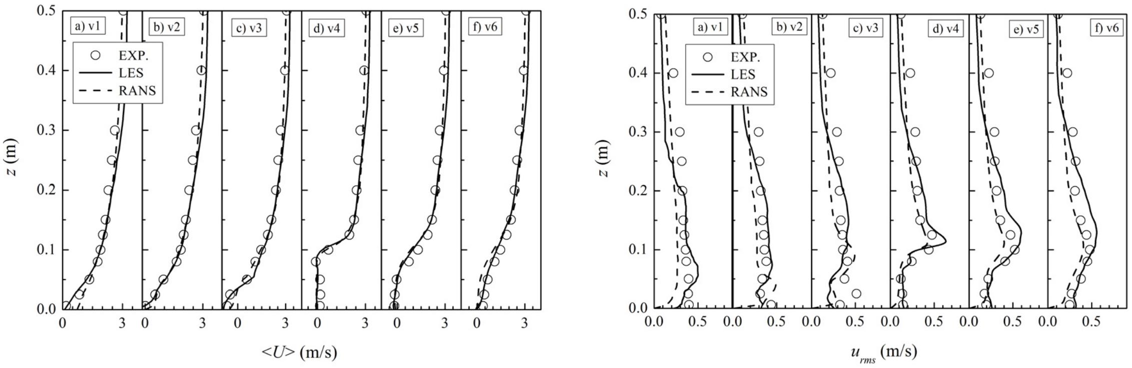
$$< u^2 >= \frac{2}{3}k - 2\gamma_T \frac{\partial \langle U \rangle}{\partial x}, \qquad (1)$$

where k is the turbulent kinetic energy, ν_T is the turbulent kinetic viscosity, and < > denotes time-averaged values. The time-averaged velocities and velocity fluctuations obtained by both RANS and LES are in relatively good agreement with those obtained from the wind tunnel experiment, and both simulations capture the recirculation between the buildings at point v3. At points v5 and v6 behind the tall building, however, the streamwise velocities are underestimated in RANS. In addition, RANS generally underestimates the turbulent energy behind a single building, as pointed by Tominaga et al. (2008), because RANS can not represent vortex shedding induced by the building, which has a strong effect on the turbulent energy. The present results obtained by RANS are, however, in good agreement with the wind tunnel experiments. This is because no marked vortex shedding behind the buildings has been observed in the present complex residential area and the intensities of streamwise velocity fluctuation are almost identical to those of horizontal turbulent fluctuations, which is assumed in the standard k-ε model used in the RANS.

Figure 4 shows the vertical distribution of time-averaged concentration. The concentration is normalized by the free stream velocity of 3.0 m/s, U_0, and the source strength, Q. The concentration estimated by LES is in good agreement with the wind tunnel data, but that estimated by RANS is dependent on the turbulent Schmidt number, S_{ct}, which is a selective parameter appearing in the model of turbulent scalar fluxes in RANS. The optimum values of S_{ct} in various turbulent flows can be selected from a wide range from 0.2 to 1.3 (Tominaga & Stathopoulos, 2007). When choosing a value of S_{ct} from 0.9 to 1.3, RANS accurately predict the mean concentration. However, Tominaga & Stathopoulos (2007) showed that RANS using S_{ct} = 0.3 gives better good agreement with the concentration distribution around a single building. This value is different from the values of S_{ct} = 0.9-1.3 used in the present simulations, and RANS using S_{ct} = 0.3 give the worse results as shown in Fig. 4(b). Thus, RANS can be used to predict the concentration only when an optimum empirical value of S_{ct} is chosen.

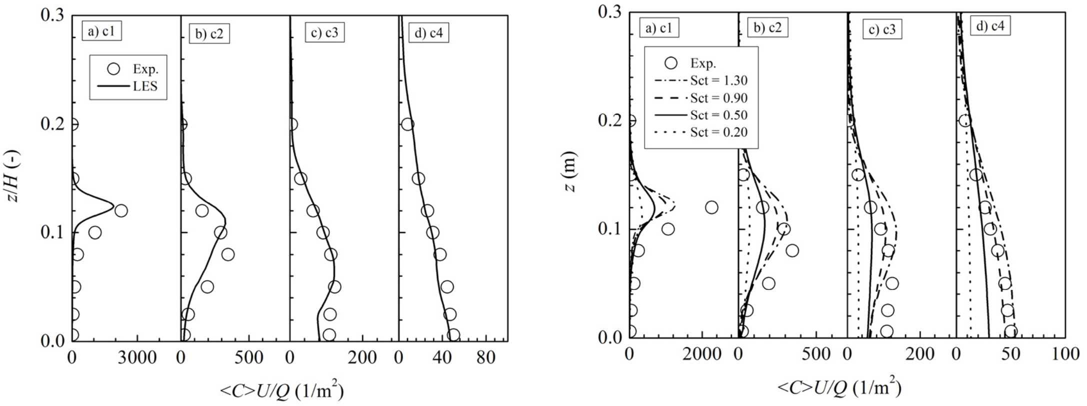
4. CONCLUSIONS

We performed a Reynolds-averaged Narvier-Stokes simulation (RANS) and a large-eddy simulation (LES) to simulate gas dispersion in a residential area. To evaluate the performance of these simulations, the results from the simulations were compared with those obtained by wind tunnel experiments, which were conducted for gas dispersion from a point source over the roof of a tall building in a complex residential area. The results showed that RANS can capture the mean flow, but can not reproduce gas dispersion because it is affected not only by the mean flow, but also by the turbulent scalar flux. The turbulent Schmidt number, which strongly affects the turbulent scalar flux, is the selective parameter appearing in the model of turbulent scalar flux, and RANS can be used to predict the concentration only when an optimum empirical value of S_{ct} is chosen. On the other hand, LES can accurately represent the flow and concentration field.



(b) (b)

Fig. 3 Vertical distribution of (a) time-averaged streamwise velocity and (b) rms of the streamwise velocity fluctuations.



(a) LES (b) RANS

Fig. 4 Vertical distribution of time-averaged concentration.(a) LES, (b) RANS.

References

Tominaga, Y., Stathopoulos, T., 2007. Turbulent Schmidt numbers for CFD analysis with various types of flowfield, *Atmospheric Environment*, 41, 8091-8099.

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

Hanna, S. R., Brown, M. J. Camelli, F. E., Chan, S. T., Coirier, W. J., Hansen, O. R., Huber, A. H., Kim, S., Reynolds, M., 2006. Detailed simulations of atmospheric flow and dispersion in downtown Manhattan, an application of five computational fluid dynamics models, *Bulletin of American meteorological society*, 1713-1726.

Flaherty, J. E., Stock, D., Lamb, B. 2007. Computational fluid dynamic simulations of plume dispersion in urban Oklahoma city, Journal of Applied Meteorology and Climatology, 46, 2110-2126.

Hendricks, E. A., Diehl, S. R., Burrows, D. A. Keith, R., 2007. Evaluation of a fast-running urban 2003 field data, *Journal of Applied Meteorology and Climatology*, 46, 2165-2179.

Sato, A., Ichikawa, Y., Michioka, T., 2008. Experimental and numerical simulation of pollutant dispersion in a high density residential area. (1) Field tracer experiments and wind tunnel experiments, *Proceedings of the 48th Annual Meeting of Japan Society for Atmospheric Environment*, 1J1324 (in Japanese).

Unemura, T., Zhang, H. Taniguchi, N., 2004. Development of the Fluid Dynamic Simulation Software “FrontFlow/Red”, *Monthly Journal of the Institute of Industrial Science, University of Tokyo*, 56, 40-43 (in Japanese).