

CFD simulations of pollutant gas dispersion with different buoyancies around an isolated building

Yoshihide Tominaga and Lin Guo-Cheng

*Department of Architecture and Building Engineering, Niigata Institute of Technology,
Kashiwazaki, Niigata, Japan*

Abstract

CFD simulations are performed for flow and dispersion fields around an isolated cubic building model with tracer gases exhausted behind it. The tracer gases have three different buoyancies due to density differences with ambient air. Therefore, they behave as neutral, buoyant and dense gases. Two modeling approaches, steady RANS and LES, are adopted and compared. For the neutral gas case, all the RANS computations show very large concentrations near the floor, which did not appear in the experimental and LES results. The same tendency is observed for the buoyant gas case. These high concentrations are mainly caused by underestimation of turbulent diffusion behind the building. On the other hand, since the density gradient is negative (stable) in the dense gas case, the discrepancy in steady-RANS is not noticeable. The difference between the two modeling approaches for the dense gas case is relatively small compared with those for the neutral and buoyant gases.

Keywords: CFD, Dispersion, Buoyancy, Building, Turbulence model

1 INTRODUCTION

Prediction of contaminant dispersion around buildings is one of the most important subjects in the fields of wind engineering, air-conditioning engineering, etc. A number of investigations have been carried out for predicting turbulent diffusion around buildings using wind tunnel tests. However, there are several difficulties and limitations with wind tunnel tests in comprehending the very complicated turbulent diffusion process around buildings located in atmospheric boundary layers.

Numerical methods based on CFD (Computational Fluid Dynamics) for simulating flow and diffusion fields have developed rapidly as a new analysis tool for wind engineering. Much research has been conducted on pollutant dispersion around buildings using the CFD technique (e.g. Blocken et al., 2008; Tominaga and Stathopoulos, 2010). Most of these previous studies have treated tracer gases as neutrally buoyant (passive) scalar (e.g. Hanna et al., 2004). Even when using a lighter or heavier tracer gas than air in an experiment, buoyancy effect of pollutants on a dispersion field has not been explicit in many cases. However, negative or positive buoyancies in pollutants have a large influence on flow and dispersion fields (Gavelli et al., 2008; Meroney, 2010; Hu et al., 2011).

In the present study, CFD simulations are performed for flow and dispersion fields around an isolated cubic building model, where the tracer gases are exhausted from behind the building. The tracer gases are treated as neutral, buoyant or dense compared with ambient air. Various turbulence models including two modeling approaches, steady Reynolds-averaged Navier-Stokes (RANS) and Large-Eddy Simulation (LES), are used and compared. Effects of pollutant buoyancies on the prediction results by various models are investigated by comparison with wind tunnel experiment results.

2 FLOW AND DISPERSION FIELD

2.1 Target configuration

The target configuration was determined based on the experiment reported in Tomina-ga et al. (1992, 1997). Figure 1 illustrates the flow situation analyzed here. A cubic building model 0.2m high was located in the turbulent boundary layer. Here, x_3 is the vertical direc-tion. The Reynolds number based on $\langle u_b \rangle$ and H is 5.7×10^3 . It was confirmed in prelimi-nary experiments that Reynolds number effect is small. A square-shaped gas source with a side length of $0.025H$ was set at ground level in the recirculation region behind the cube. Exit gas speed $\langle w_s \rangle$ was $0.5\langle u_b \rangle$. The cases compared in this study are listed in Table 1. The densimetric Froude numbers Fr_d , defined by Eq. (1), for the buoyant and dense gas cases were determined so that their absolute values were identical.

$$Fr_d = \frac{g\Delta\rho H}{\rho_a \langle u_b \rangle^2} \quad (1)$$

where ρ_a is ambient air density; ρ_s is gas density; $\Delta\rho = \rho_s - \rho_a$.

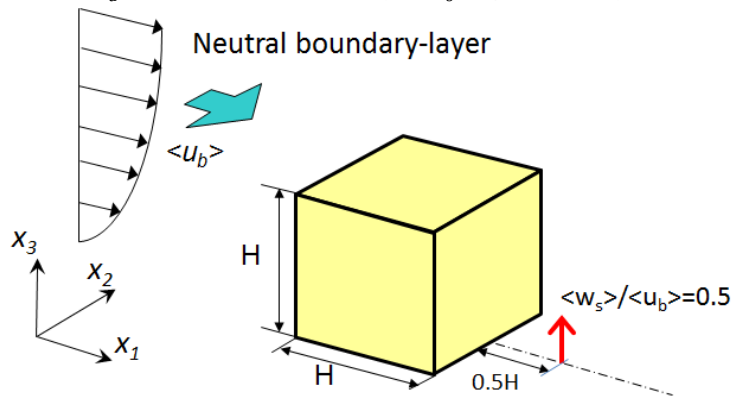


Figure 1. Model flowfield.

Table 1. Cases investigated.

Case	ρ_s/ρ_a	Fr_d	
1	1.0	0.0	neutral gas
2	0.3	-8.6	buoyant gas
3	1.7	+8.6	dense gas

2.2 Experimental setup

Experiments were carried out in the boundary layer wind tunnel at the Institute of Indus-trial Science, the University of Tokyo. The power law exponent of the vertical profile of the inflow velocity was 0.25. The streamwise turbulent intensity at building height H was ap-proximately 20%. The upwind mean velocity at building height H was 0.4 m/s. The neutral gas C_2H_4 was used as a tracer gas. In order to take into account negative and positive buoy-ancies shown in Table 1, a mixture of He and C_2H_4 and of SF_6 and C_2H_4 were used for the buoyant and dense gases, respectively. Concentration measurements were performed using a high-speed total Hydro-Carbon Analyzer (Technica; H-THCA-01). Wind velocity was measured by a split fibre probe (Dantec Dynamics; 55R55) and a CTA module, which can discern the three-dimensional components of velocity vector. Sampling frequency was set to 10Hz to obtain 4,096 data for each measuring point.

3 COMPUTATIONAL SETUP

3.1 Numerical methods

The commercial software ANSYS FLUENT 13.0 was used for both RANS and LES computations based on a control volume approach for solving flow and concentration equations (FLUENT Inc., 2010).

1) RANS: Three types of turbulence models were used and compared: the standard k - ε model (SKE), the RNG k - ε model (RNG) (Yakhot et al., 1992) and the realizable k - ε model (RLZ) (Shih et al., 1995). All model constants were set to be the default values in the software. All the transport equations were discretized using a second-order upwind scheme. The SIMPLE algorithm was used for pressure-velocity coupling. The generation of turbulent kinetic energy due to buoyancy was included in the k equation by default of the software. Although the buoyancy effect on ε was also included, it was confirmed that its effect on the results was negligible.

2) LES: Two types of sub-grid scale (SGS) eddy viscosity models were used: the standard Smagorinsky model (SS; Smagorinsky 1963) with the empirical constant $C_s=0.12$ (Tominaga 2008a) and the dynamic Smagorinsky SGS model (DS; Germano et al. 1991). The filtered equation was discretized using a bounded central-differencing scheme. Time integration was second-order implicit. The computations were conducted for 800 non-dimensional time units t^* ($=t \times \langle u_b \rangle / H$), which corresponds to 400 s in real time scale, to determine the time-averaged values. It was confirmed that the statistical results were almost repeatable over a longer averaging period.

3.2 Domain, computational grid and boundary conditions

The boundary conditions were set following basic guidelines (Tominaga et al., 2008b). The computational domain covered a volume of $21H(x_1) \times 13H(x_2) \times 6.5H(x_3)$. This domain was discretized into 1,112,200 hexahedral grids. These conditions were the same in both computations. Turbulence in the exhaust outlet flow was not considered.

1) RNG: The vertical distributions of $\langle u_1 \rangle$, k and ε at the inflow boundaries were based on the experiment. For the ground surface, the wall functions were modified for roughness, specified by an equivalent sand-grain roughness height k_s and a roughness constant C_r . k_s was defined using the function proposed by Blocken et al. (2007): $k_s = 9.793z_0/C_r$. The value of z_0 was estimated by the velocity profile obtained by the experiment.

2) LES: A time-dependent inlet profile was generated by using the vortex method with a number of vortices $N_v=190$ (Gousseau et al., 2011). For the boundary condition at the solid walls (floor and building surfaces), a logarithmic-law was assumed.

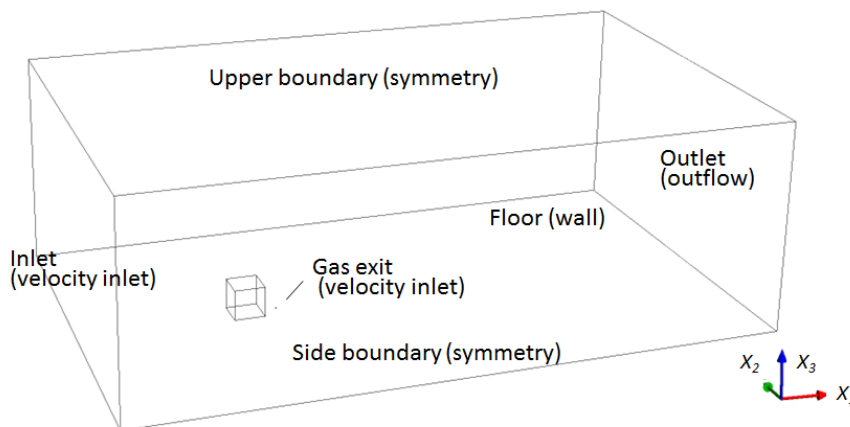


Figure 2. Computational domain.

4 RESULTS AND DISCUSSION

4.1 Neutral gas case

Figure 3 compares the contours of the time-averaged velocities of streamwise component ($\langle u_1 \rangle$) at the central section. Three computational results by the RANS approach show stronger reverse flow and longer reattachment length behind the building than those by two LES computations. In the RANS computations, the strongest recirculation flows are predicted by RNG and the weakest by SKE. The choice of SGS model in LES hardly influences the velocity predictions. The small recirculation region behind the building in the LES cases are mainly due to the fact that the periodic velocity fluctuation behind the building is well reproduced in comparison with RANS, as pointed out in previous studies (e.g. Tominaga et al. 2008a).

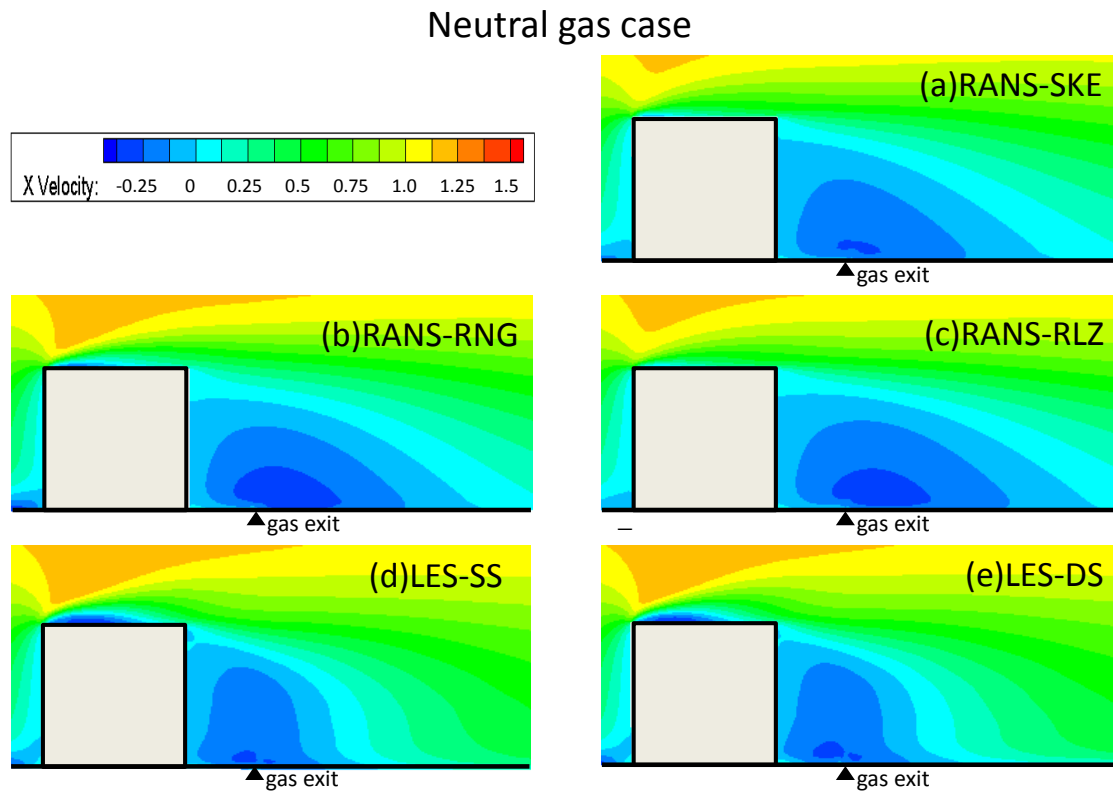


Figure 3. Contours of time-averaged streamwise velocity $\langle u_1 \rangle / \langle u_b \rangle$ at center section: (a) RANS-SKE; (b) RANS-RNG; (c) RANS-RLZ; (d) LES-SS; (e) LES-DS.

Figure 4 shows the time-averaged concentrations at the center section. There is no significant difference among the general distributions of all models. It should be noted that the contour obtained by the experiment may give different expression from those of CFD because the experimental measuring pitch was coarse. In all the results, concentrations from the source exit are advected by the recirculation flow behind the building, and a high concentration region appears windward of the exit. The two LES cases predict smaller spread of concentration leeward of the exit in comparison with RANS. This difference between LES and RANS is mainly caused by the fact that the turbulent diffusion in the wake region is underestimated and the convective transport is dominant in RANS, as previously mentioned.

Figure 5 compares the vertical distributions of concentration at the mid-point between the building and the exit. All RANS computations show extremely large concentrations near the floor, which differs from the experiment and LES. The concentrations obtained by the two LES cases are lower and are closer to that of the experiment than those by RANS.

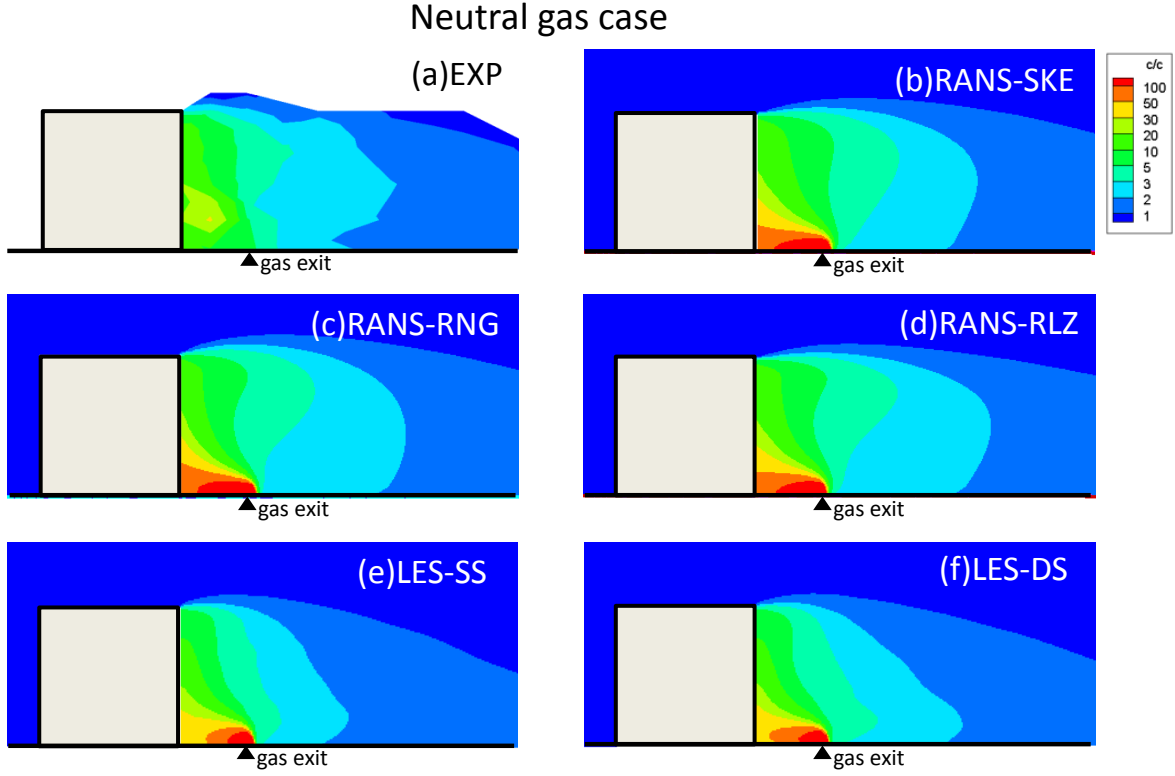


Figure 4. Contours of time-averaged concentration $\langle c \rangle / \langle c_0 \rangle$ (at center section): (a) experiment; (b) RANS-SKE; (c) RANS-RNG; (d) RANS-RLZ; (e) LES-SS; (f) LES-DS.

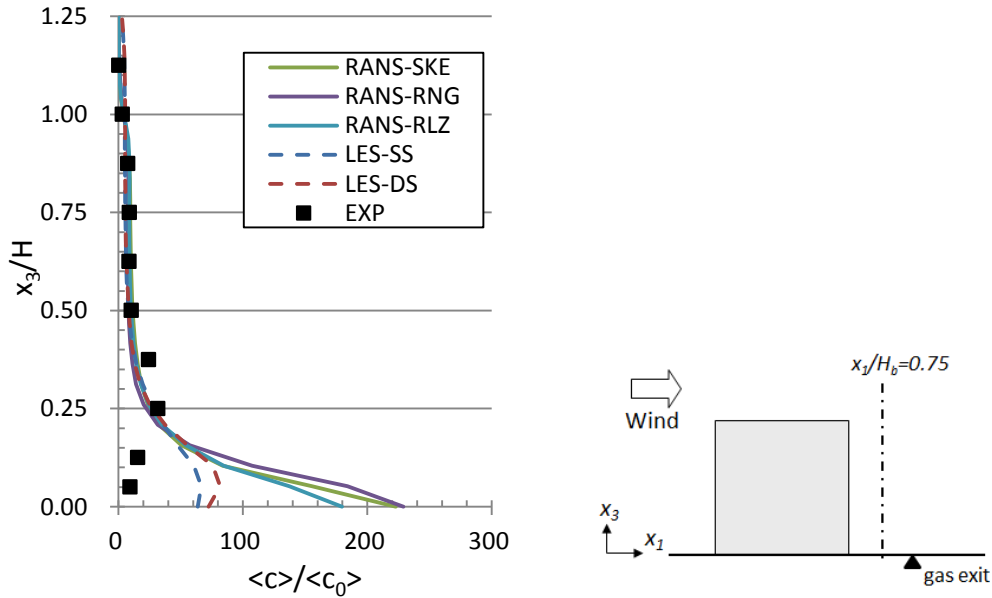


Figure 5. Vertical distributions of time-averaged concentration $\langle c \rangle / \langle c_0 \rangle$.

4.2 Buoyant and dense gas cases

Only the RANS-RLZ and LES-DS cases, which show the best results for the neutral gas case for each modeling approach, are compared. Time-averaged concentration in the buoyant gas case is illustrated in Figure 6. The experimental results show that the concentration for the buoyant gas case is very low windward of the exit in comparison with that for the

neutral gas case. The concentration at the lower part of the wake region is especially low. These results are clearly affected by the positive buoyancy of the tracer gas. Although both CFD results reproduce this tendency due to buoyancy, the predicted upward spreads of the concentrations are smaller than those of the experiment. The vertical distribution of the time-averaged concentration along the line between the exit and the building is also shown in Figure 6. LES-DS succeeds in reproducing the tendency of the concentration to decrease near the floor as in the experiment, while the concentration increases as the position approaches the floor in the RANS-RLZ case. However, the concentration is underestimated by both models in the upper part.

The time-averaged concentration for the dense gas case is shown in Figure 7. The experimental results show very high concentrations in the lower part of the wake region. An extraordinarily high concentration region occurred near the floor windward of the source exit. This tendency is well reproduced in both CFD results. The vertical distribution of the time-averaged concentration is also indicated in Figure 7. The difference between the two models is small, and they both reproduce the tendency for the concentration to increase rapidly as the position approaches the floor. The difference between the prediction accuracies for the buoyant and dense gases can be explained as follows: For the buoyant gas, the flow field is more diffusive than for the neutral gas, because the density gradient is positive (unstable) in most of the wake region. Therefore, the previously-mentioned underestimation of turbulent diffusion behind the building in RANS acts more significantly than for the neutral gas. On the other hand, since the flow field is stable due to negative buoyancy and turbulent diffusion is suppressed for the dense gas, no such discrepancy occurs for RANS. Therefore, the difference between the two models is relatively small compared with the neutral and buoyant gas cases.

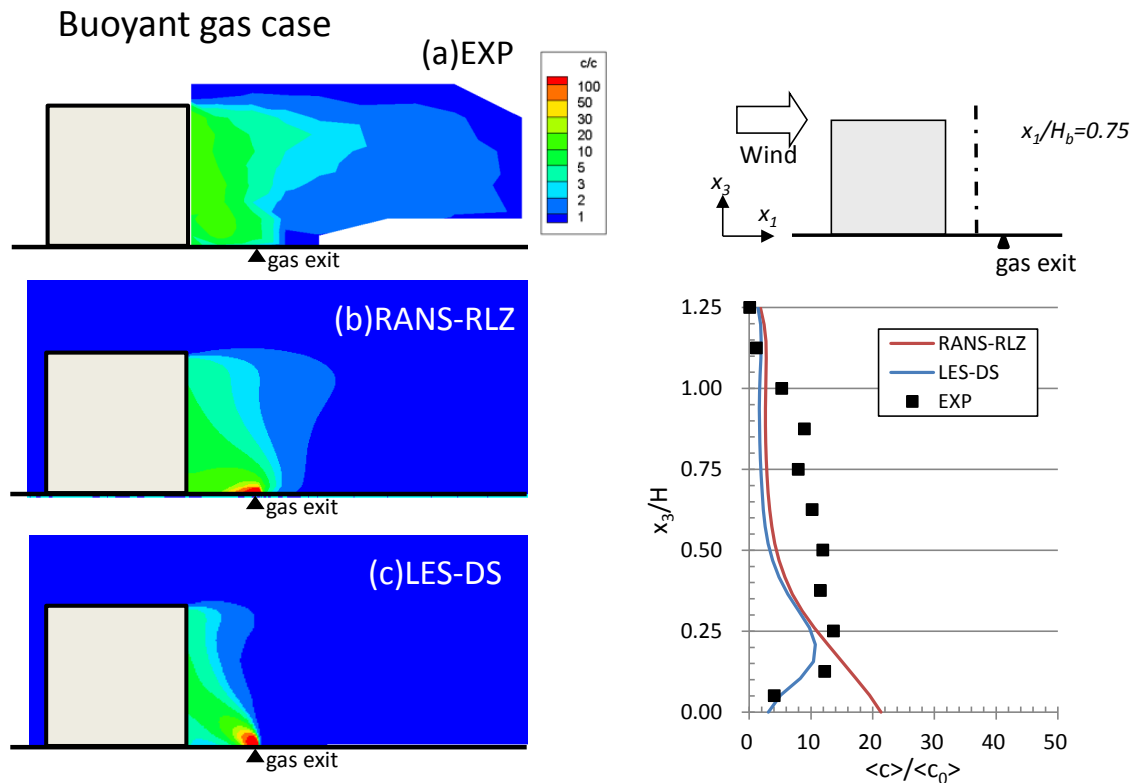


Figure 6 Time-averaged concentration $\langle c \rangle / \langle c_0 \rangle$ with buoyant gas. Left: Contour at center section obtained from (a) Experiment; (b) RANS-RLZ; and (c) LES-DS. Right: Comparison of vertical distributions at indicated line behind a building.

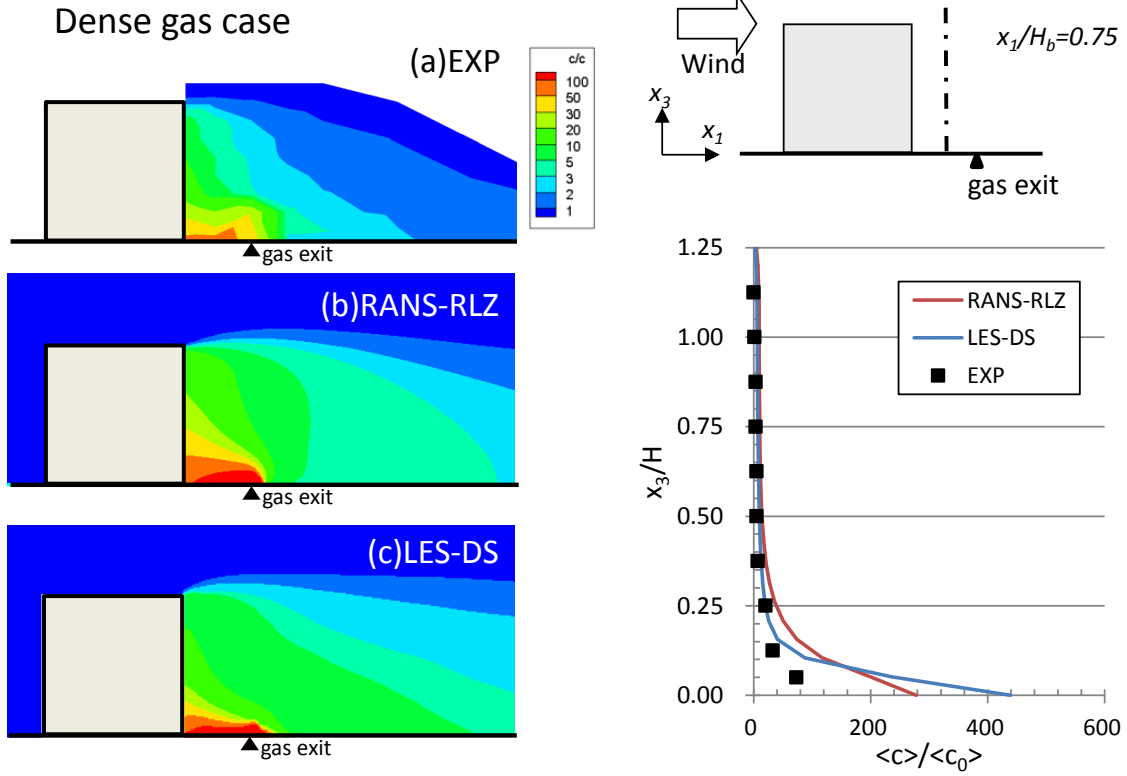


Figure 7. Time-averaged concentration $\langle c \rangle / \langle c_0 \rangle$ with dense gas. Left: Contour at center section obtained from (a) Experiment; (b) RANS-RLZ; and (c) LES-DS. Right: Comparison of vertical distributions at indicated line behind a building.

5 CONCLUSIONS

CFD simulations were performed for flow and dispersion fields around a cubic building model, in which the tracer gases were exhausted behind the building. The tracer gases were of three different densities, behaving as neutral, buoyant or dense gases. Two modeling approaches, steady RANS and LES, were used and compared. For the neutral gas, all RANS computations showed very large concentrations near the floor, which did not appear in the experimental and LES results. The same tendency was observed in the results for the buoyant gas. These high concentrations appearing in steady-RANS computations were mainly caused by underestimation of turbulent diffusion behind the building. On the other hand, since the density gradient was negative (stable) and turbulent diffusion was suppressed in the dense gas case, there was no noticeable discrepancy for steady-RANS. Therefore, the differences between the two models were relatively small in comparison with those for the neutral and buoyant gases. Detailed explanation on the influence of buoyancy effects on prediction accuracy will be presented in the final presentation.

NOMENCLATURE

- c : concentration
- $\langle c_0 \rangle$: standard gas concentration ($=q/(\langle u_b \rangle H^2)$)
- $\langle f \rangle$: ensemble average value of f
- Fr_d : densimetric Froude Number ($=g\Delta\rho H/\rho_a \langle u_b \rangle^2$)
- g : gravitational acceleration
- H : building height
- k : turbulent kinetic energy
- q : gas emission rate

t^* : non-dimensional time unit ($=t\langle u_b \rangle/H$)
 u_i : three components of wind velocity
 u_b : inflow velocity at building height H
 w_s : exit velocity
 x_i : three components of spatial coordinates ($i=1,2,3$: streamwise, lateral, vertical)
 ρ_a : ambient air density
 ρ_s : gas density

REFERENCES

- Blocken B, Stathopoulos T, Carmeliet J. 2007. CFD simulation of the atmospheric boundary layer: wall function problems. *Atmospheric Environment* 41, 238-252.
- Blocken, B., Stathopoulos, T., Saathoff, P. and Wang, X., 2008. Numerical evaluation of pollutant dispersion in the built environment: comparisons between models and experiments. *Journal of Wind Engineering and Industrial Aerodynamics* 96, 1817-1831.
- ANSYS Inc., 2010. ANSYS FLUENT 13.0 Users guide. FLUENT Inc., Lebanon.
- Gavelli, F., Bullister, E. and Kytomaa, H., 2008. Application of CFD (Fluent) to LNG spills into geometrically complex environments. *Journal of Hazardous Materials* 159, 158-168.
- Germano, M., Piomelli, U., Moin, P., Cabot, W.H., 1991. A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A* 3, 1760-1765.
- Gousseau, P., Blocken, B., & van Heijst, G. J. F., 2011. CFD simulation of pollutant dispersion around isolated buildings: on the role of convective and turbulent mass fluxes in the prediction accuracy. *Journal of Hazardous Materials* 194, 422-434.
- Hanna, S., R., Hansen, O., R., Dharmavaram, S., 2004. FLACS CFD air quality model performance evaluation with Kit Fox, MUST, Prairie Grass, and EMU observations. *Atmospheric Environment* 38, 4675-4687.
- Hu, L.H., Xu, Y., Zhu, W., Wu, L., Tang F, Lu K.H. 2011. Large eddy simulation of pollutant gas dispersion with buoyancy ejected from building into an urban street canyon. *Journal of Hazardous Materials* 192, 940-948.
- Meroney, R. N., 2010. CFD prediction of dense gas clouds spreading in a mock urban environment. The Fifth International Symposium on Computational Wind Engineering (CWE2010), Chapel Hill, North Carolina, USA, May 23-27, 2010.
- Smagorinsky, J., 1963. General circulation experiments with the primitive equations I. The basic experiment. *Monthly Weather Review* 91, 99-164.
- Shih, T.H., Liou, W. W., Shabbir, A., Yang, Z. and Zhu, J., 1995. A new k- ϵ eddy viscosity model for high Reynolds number turbulent flows. *Computers and Fluids* 24, 227-238.
- Tominaga, Y., Murakami, S., Mochida, A., Shibuya, A., Noguchi, Y., 1992. Wind tunnel tests on turbulent diffusion and concentration fluctuation of buoyant gas near building. *Proceedings of 12th National Symposium on Wind Engineering*, 119-124. (in Japanese)
- Tominaga, Y., Murakami, S., Mochida, A., 1997. CFD Prediction of Gaseous Diffusion around a Cubic Model using a Dynamic Mixed SGS Model based on Composite Grid Technique. *Journal of Wind Engineering and Industrial Aerodynamics* 67&68, 827-841.
- Tominaga, Y., Mochida, A., Murakami, S., Sawaki, S., 2008a. Comparison of various revised k- ϵ 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.
- Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirasawa, T., 2008b. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics* 96, 1749-1761.
- Tominaga, Y., Stathopoulos, T., 2010. Numerical simulation of dispersion around an isolated cubic building: Model evaluation of RANS and LES. *Building and Environment* 45, 2231-2239.
- Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B., Speziale, C.G., 1992. Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A* 4, 1510-1520.