

Numerical simulation of pressure and flow field in large group of low-rise buildings

Yong Chul Kim^a, Akihito Yoshida^a, Yukio Tamura^a,
Hirotoshi Kikuchi^b, Kazuki Hibi^b,

^a*Tokyo Polytechnic University, 1583 Iiyama Atsugi, Kanagawa, Japan*

^b*Shimizu Corporation, 3-4-17 Etchujima, Koto-ku, Tokyo, Japan*

ABSTRACT: The pressure and flow field around a target low-rise model immersed in a large group simulating various area densities were investigated using three-dimensional numerical RANS turbulence models. Wind pressure coefficients obtained from the numerical results were compared with those of wind pressure measurements, and the flow fields around the model were compared with those obtained from flow visualization. The results show that for the wind pressure coefficients for the isolated model, more reasonable results were obtained when the *rng k-ε* model was used, but there was little difference between turbulence models for the target model in a large group. For the flow field around the model, the RANS turbulence models were able to reproduce the overall flow pattern generally, even though some differences were found.

KEYWORDS: Group of low-rise buildings; Computational fluid dynamics; Wind pressure measurement; Flow visualization

1 INTRODUCTION

As low-rise buildings are normally built in large groups, the flow field around a low-rise building of interest is totally different from that around an isolated building. The significance of this different flow field should be focused on not only from structural but also from environmental viewpoints because the flow field determines wind pressure distributions and the degree of scalar dispersion. Flow field conditions are strongly dependent on building geometry and arrangement characteristics, more or less incident flow conditions, by interacting and overlapping, which governs the wind force/pressure characteristics as well as the nature of ventilation and dispersion of pollutants in urban street canyons.

Wind pressure measurements for a low-rise building in a large group have been performed by many researchers (Sun et al., 2008; Kim et al., 2011, 2012). Sun et al. (2008) investigated peak pressure coefficients through a number of wind tunnel tests, and reported that shielding effects generally depend on area density and relative height ratio between the target building and surrounding buildings, and although the shielding effects were significant in most cases, the positive peak wind pressure coefficients were increased by 100% in some cases. Most recently, Kim et al. (2011, 2012) measured fluctuating wind pressures applied to a cubical low-rise building immersed in various area densities and long upstream distances under suburban flow conditions. After a detailed examination of wind pressures and forces characteristics, methodologies of wind load estimation were suggested based on the interference factor for the structural frame and zoning interference factor for components and cladding.

The remarkable growth in computer technology has enabled increasing application of numerical simulations to a large group of low-rise buildings to the problems including wind loads,

flow conditions and pollutant dispersion. Much of the works have focused on investigating the flow and pollutant dispersion problems within urban street canyons using the RANS turbulence model. Chang and Meroney (2003) investigated the effect of surrounding buildings with different separation distances on flow distributions and transport from steady point sources of pollutants, or chemical and biological agents, in an idealized urban environment. Lien and Yee (2004) studied the neutrally-stratified flow within and over a large group of 3-dimensional cubical buildings using two turbulence models: the standard $k-\varepsilon$ model and the Kato-Launder $k-\varepsilon$ model. Santiago et al. (2007) examined the wind flow and pollutant dispersion inside and above a three-dimensional cubic array to describe and explain the flow structure and tracer dispersion using the standard $k-\varepsilon$ turbulence model.

Most recent numerical studies have discussed flow and pollutant dispersion, and although some previous studies have discussed the pressure and flow field for a target low-rise building in large group, the simulated area density was very limited, mostly an area density of 25%. The purpose of the present study was to investigate the pressure and flow field for a target low-rise building immersed in various area densities ranging from 6% to 44% by comparing numerical results with wind pressure measurements and flow visualizations using a DPIV system. For the numerical simulation, three-dimensional RANS turbulence models were implemented.

2 WIND TUNNEL TEST AND COMPUTATIONAL FLUID DYNAMICS

Wind tunnel tests were conducted at the Tokyo Polytechnic University in Japan. A 0.1m cubical building model was used for the target low-rise building, and the same sized cubes made of wood were used as dummy models. Area density, C_A , was defined as the ratio of the area covered by the building to the building lot area, and varied as 6%, 11%, 16%, 25%, and 44%. A schematic of the wind tunnel test is shown in Figure 1, and the experimental procedures were as follows.

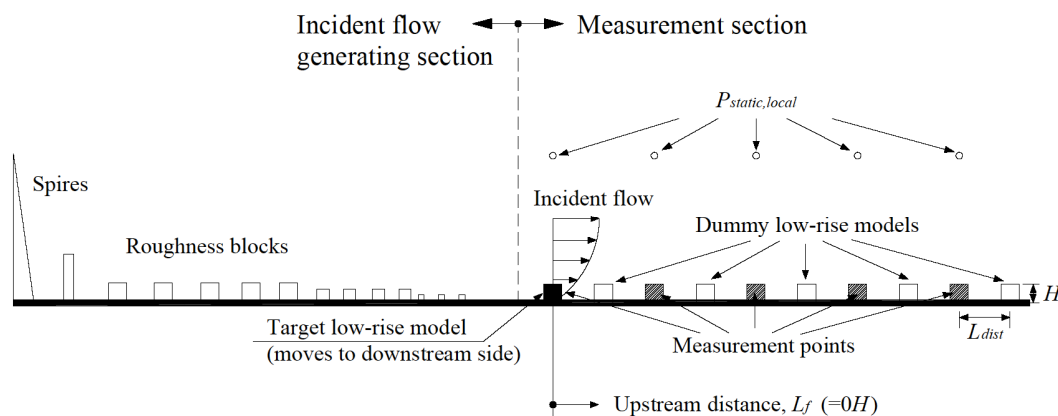


Figure 1. Schematic of wind tunnel test

As shown in Figure 1, half the wind tunnel was used as an incident flow generating section using spires, barriers, and three different-sized blocks. The incident flow was measured at $L_f=0H$. To clarify the effect of surrounding buildings, the wind pressure measurement of the isolated model was conducted at $L_f=0H$. After arranging the dummy low-rise models, the wind pressure measurements for the target model were conducted by moving it downstream at specified intervals determined by area density. Here, the term “target model” is used to indicate the model surrounded by dummy low-rise models in order to differentiate it from “isolated model”. In previous studies,

the dummy models were arranged on the upstream side while the target model was fixed at a position. But in the present study, the target model was moved to the downstream side, which makes the changes of wind pressures and flow conditions more understandable depending on various area densities and upstream distances. The number of measurement points was dependent on area density, and there were 9 measurement points for area densities of 11%, 16%, and 44%, and 6 measurement points for area densities of 6% and 25%. 125 pressure taps were installed, and all pressures were measured simultaneously using a multi-channel pressure measurement system, and the tubing effects were numerically compensated using the gain and phase shift characteristics of the pressure measurement system. As an incident flow, a turbulent boundary layer representing a suburban area with a power-law exponent of 0.2 ($U_H \approx 7.2 \text{ m/s}$, $I_{u,H} \approx 23\%$) was simulated as shown in Figure 2 (thick solid line). The Reynolds number based on U_{ref} and B was about 5×10^4 , and the wind direction was fixed at 0° , with the incident wind normal to the model surface.

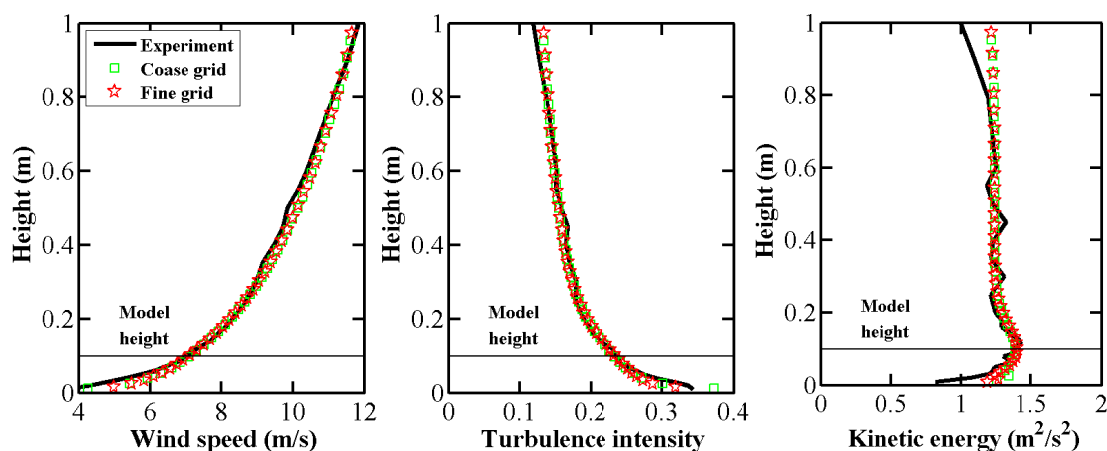


Figure 2. Comparison between incident flow conditions from wind tunnel test and simulation

Flow visualization using a Dynamic Particle Image Velocimetry (DPIV) system was carried out in a Boundary Layer Wind Tunnel at Shimizu Institute of Technology, Japan. The system consisted of a high-speed digital video camera, a double pulse ND: YAG laser, and a particle generator. Tracer particles were discharged from the particle generator, which was located at the end of the group of low-rise buildings, and then circulated in the wind tunnel. The particles were illuminated by a pulsed laser light sheet. The image size was adjusted to be large enough to adequately capture the flow field. A pixel resolution of 800×600 was used, and 2000 pictures were taken, resulting in sampling frequencies of 1000 frames per second. The measuring time was set to 2.95 seconds (5900 pictures).

Fluent, included in the package of ANSYS 13.0, was used to carry out all numerical simulations based on the finite volume method. In the present study, well-known turbulent Reynolds Averaged Navier-Stokes (RANS) models, the standard $k-\varepsilon$ model and the $\text{rng } k-\varepsilon$ model, were used for the turbulent flow of an incompressible viscous fluid. The modified law-of-the-wall function was used in combination with RANS turbulence models. As input parameters for the modified law-of-the-wall function, K_S and C_S needed to be input to the solver, and K_S and C_S were determined using the incident flow from the wind tunnel test considering the relationship between K_S and z_0 for the fully rough regime (Blocken et al., 2007). The obtained z_0 for the incident flow was 4.76×10^{-4} , and using $C_S = 1$, K_S was set to 0.001 for the wind tunnel's floor roughness condition. Note that this K_S was smaller than y_p , which is the distance from the floor to the center point P of the wall adjacent to the cell. The boundary conditions of the two sides and the

top were modeled as slip walls, and the free outlet boundary condition was used. Inlet profiles for streamwise mean wind speed, turbulent kinetic energy, and turbulent dissipation rate were simulated using a user-defined function. Turbulent kinetic energy was obtained considering only the streamwise fluctuating component, and turbulent dissipation rate was obtained assuming local equilibrium of $G_k = \varepsilon$ (Tominaga et al., 2008). Lateral and vertical wind speeds were assumed to be zero. For the isolated model, two grid systems were employed: a coarse grid with y^+ larger than 61 and a fine grid with y^+ larger than 30. The simulated profiles at the model position in the empty three-dimensional computational domain are shown in Figure 2 for the fine and coarse grids, and show good agreement with incident flow from the wind tunnel test regardless of the y^+ values. Here, the reference wind speed U_{ref} means the mean wind speed at the model position. Other important computational conditions and the position of the target model are summarized in Table 1. For the isolated model, two grids were employed to examine the grid dependence of the results, and for the target model, the computational domain was reduced considering the symmetry condition. One example of a grid for the target model when $C_A=25\%$ is shown in Figure 3.

Table 1 Computational conditions

		$C_A=0\%$	$C_A=6\%$	$C_A=11\%$	$C_A=16\%$	$C_A=25\%$	$C_A=44\%$
Computational Domain ($X \times Z \times Y$, unit: H)		21×10×7	49×10×2	51×10×1.5	51×10×1.25	49×10×1	51×10×0.75
Target Model Position (L_b , unit: H)		0	8 and 24	6 and 24	5 and 25	8 and 24	6 and 24
Pressure-velocity Coupling		SIMPLE					
Spatial Discretization		QUICK					
Convergence Criteria		Continuity: 10^{-6} Others: 10^{-6}					
Grid Number ($X \times Z \times Y$)	Coarse Grid	60×33×39	293×19×41	336×16×41	352×14×41	395×12×41	416×9×41
	Fine Grid	117×47×52					
y^+	Coarse Grid	61< y^+ <263	57< y^+ <395	54< y^+ <398	62< y^+ <400	54< y^+ <401	27< y^+ <420
	Fine Grid	30< y^+ <357					
y_P (simulation scale, m)	Coarse Grid	0.00514	0.00629				
	Fine Grid	0.00262					
Grid Expansion Factor (y_i/y_{i+1})		Less than 1.2					

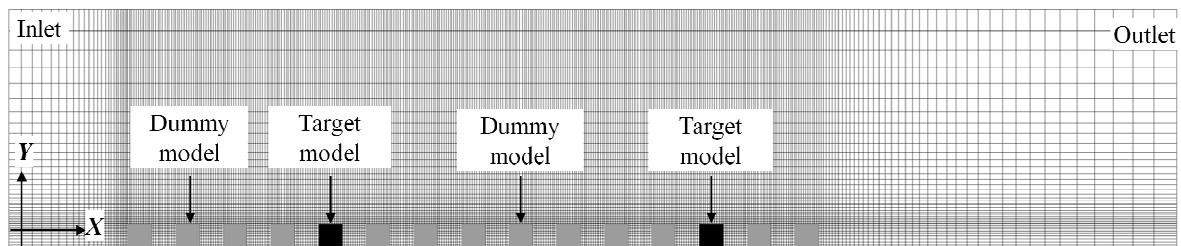


Figure 3. Grid for target model when $C_A=25\%$

3 RESULTS AND DISCUSSION

3.1 Results for isolated model

Mean wind pressure coefficients are defined as:

$$\bar{C}_p = \frac{\bar{P} - \bar{P}_{ref}}{0.5\rho U_{ref}^2} \quad (1)$$

where, $\bar{P}$ is the time-averaged surface pressure and $\bar{P}_{ref}$ is the time-averaged reference static pressure in the computational domain, which is at exactly the same position as the pitot tube in the wind tunnel test.

Figure 4 compares the mean wind pressure coefficients from the wind tunnel test with the simulation results from the coarse and fine grids. Differences are observed near the stagnation point on the windward surface and on the leading edges of the roof and side surfaces where large suction occurs due to the separated shear layer. On the leeward surface, the simulation results show slightly larger values than the experimental values (smaller absolute value). From the different turbulence models, clear differences are also observed near the stagnation point and leading edges. For the standard $k-\varepsilon$ model, the mean wind pressure coefficient near the stagnation point is larger than unity, which is theoretically impossible and which is also much larger than the experimental values. Contrary to the results of the standard $k-\varepsilon$ model, those of the $rng\ k-\varepsilon$ model show relatively good agreement with the experimental results on the windward surface. On the side surfaces, the variation trend of the $rng\ k-\varepsilon$ model is similar to that of the experimental value, but the values are much larger than the experimental results and the standard $k-\varepsilon$ model results. Focusing on the smallest mean wind pressure coefficient (larger absolute value), it seems that the standard $k-\varepsilon$ model results in more reasonable but slightly conservative values. There is little difference between the pressure coefficients on the leeward surface given by the two turbulence models, and it is interesting to note that almost the same results were obtained for the two different grids, although the fine grid gives slightly larger absolute values.

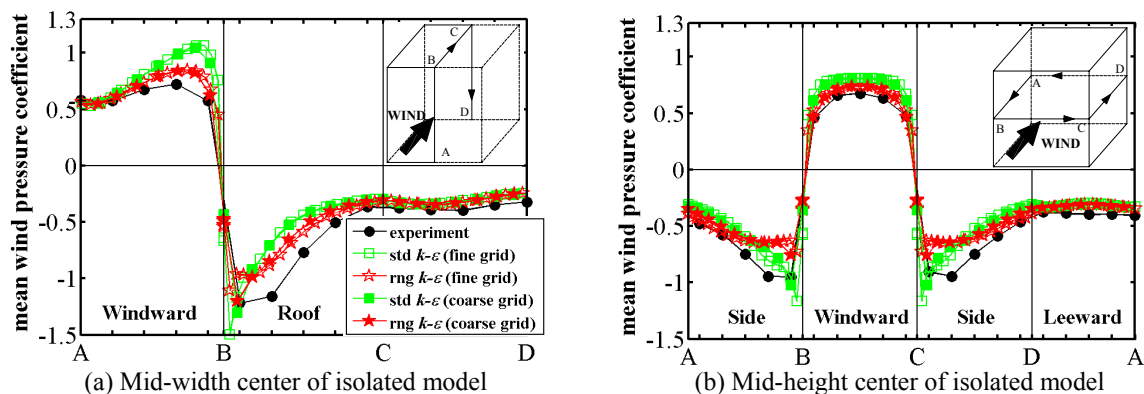


Figure 4. Comparison of mean C_p for isolated model

Wind pressure coefficients at mid-width center were compared quantitatively using three statistical parameters, normalized mean square error (NMSE, Eq. (2)), fractional bias (FB, Eq. (3)), and correlation coefficients (R), and the results are summarized in Table 2. For comparison, the numerical results corresponding to pressure tap positions were selected. NMSE indicates a value of normalized discrepancies between the computed and experimental values, and FB gives

information about overestimation (negative value of FB) or underestimation (positive value of FB).

$$NMSE = \frac{1/n \sum_{i=1}^n (E_i - C_i)^2}{1/n \sum_{i=1}^n (E_i C_i)} \quad (2); \quad FB = \frac{(E_i - C_i)}{0.5(E_i + C_i)} \quad (3)$$

As shown in Table 2, the NMSE of the rng $k-\varepsilon$ model is very small compared to those of the standard $k-\varepsilon$ model on the windward and roof surfaces, while that on the leeward surface is almost the same. As expected, the FB of the windward surface is negative, while those of the roof and leeward surfaces are positive, meaning that the numerical results are overestimated for the windward surface and underestimated for the roof and leeward surfaces. The FB of the rng $k-\varepsilon$ model is smaller than that of the standard $k-\varepsilon$ model, showing especially large differences on the windward surface. Like NMSE, FB shows little difference on the leeward surface. Similar observations can be made for correlation coefficient R . Considering the different grids, NMSE and FB obtained from the coarse grid are smaller than those obtained from the fine grid, implying that the numerical results from the coarse grid better correspond to the experimental results, but the differences between them are not significant. The statistical parameters at mid-height center show a similar trend to the mid-width center, generally showing numerical results from the coarse grid using the rng $k-\varepsilon$ model in better correspondence with the experimental values.

Table 2 Comparison of normalized mean square error (NMSE), fractional bias (FB) and correlation coefficient (R)

		NMSE		FB		R for all points	
		std $k-\varepsilon$	rng $k-\varepsilon$	std $k-\varepsilon$	rng $k-\varepsilon$	std $k-\varepsilon$	rng $k-\varepsilon$
Fine Grid	Windward Surface	0.1	0.01	-0.23	-0.09	0.90	0.97
	Roof Surface	0.15	0.06	0.36	0.23		
	Leeward Surface	0.04	0.04	0.21	0.19		
Coarse Grid	Windward Surface	0.08	0.01	-0.21	-0.06	0.91	0.97
	Roof Surface	0.12	0.06	0.26	0.2		
	Leeward Surface	0.03	0.04	0.18	0.18		

Using the numerical results obtained from the rng $k-\varepsilon$ model with the coarse grid, prediction of peak pressure coefficients on the windward and roof surfaces were attempted, considering that the largest $\max C_p$ occurred on the windward surface ($\max C_p \approx 2.95$) and the smallest $\min C_p$ occurred on the roof surface ($\min C_p \approx -4.75$). The largest $\max C_p$ was obtained near the stagnation point, and the smallest $\min C_p$ was obtained at pressure tap near the edges. For the pressure taps shown as solid thick lines in Figure 5, the relationship between normalized turbulent kinetic energy (k/U_{ref}^2) and fluctuating wind pressure coefficients ($\text{rms} C_p$) were examined, and the results are shown in Figure 5. There is a clear linear relationship between them, implying that $\text{rms} C_p$ can be estimated by k/U_{ref}^2 . From these relationships, peak C_p values were obtained from various peak factors (g). The comparisons are shown in Figure 6, showing good agreement with the experimental data when $g_{\text{max}}=5$ for windward and $g_{\text{min}}=-6$ for roof surface.

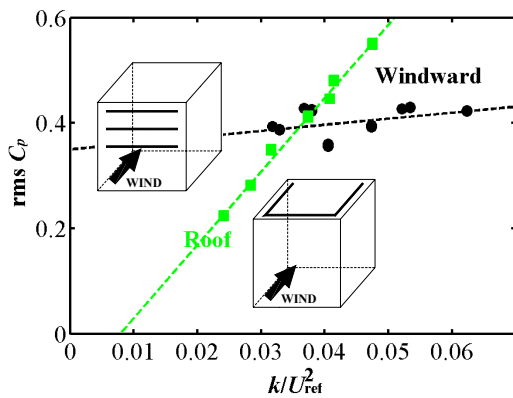


Figure 5. Relationship between k/U_{ref}^2 and $rms C_p$

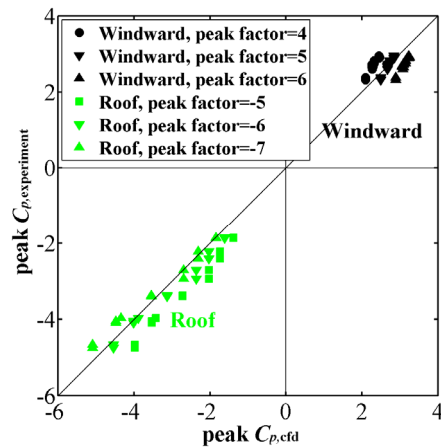


Figure 6. Comparison of peak C_p from numerical and experimental results

The flow conditions on the roof are shown in Figure 7 using contours of mean wind speed and streamlines for the coarse grid. The black rectangle indicates the isolated model, and the gray region in Figure 7(a) indicates a mask region where the particles cannot be captured by the high-speed digital video camera. From the DPIV result in Figure 7(a), it is not clearly ascertained but there is a reverse flow region on the roof. This region is well reproduced by the rng $k-\epsilon$ model (Fig. 7(c)), but not by the standard $k-\epsilon$ model (Fig. 7(b)).

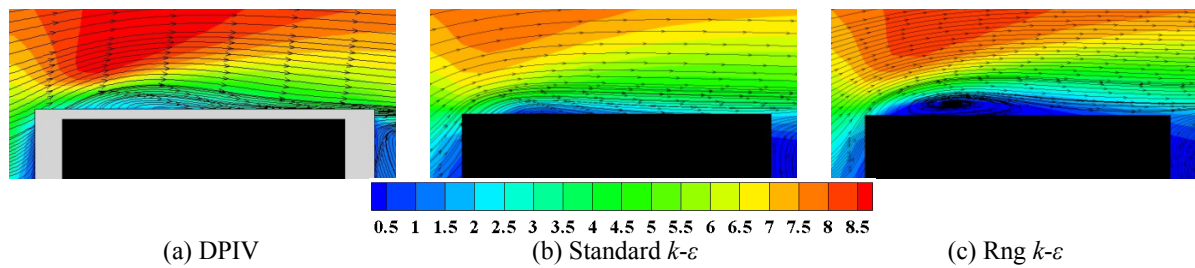


Figure 7. Contours of wind speed and streamlines on roof of isolated model

The flow fields measured and simulated at mid-height center are shown in Figure 8 for the coarse grid. The three black rectangles indicate half the isolated model adjusted to the same scale, making direct comparison possible. The flow field including the circulation in the wake seems to be simulated appropriately, but some differences were found. The circulation center (red solid circle) and wake center (red solid square) move to the downstream side for the simulation results, and the deviation is more for the rng $k-\epsilon$ model. There is reversed flow on the side surface for the rng $k-\epsilon$ model as on the roof, and these reverse flows are responsible for the larger mean wind pressure coefficient

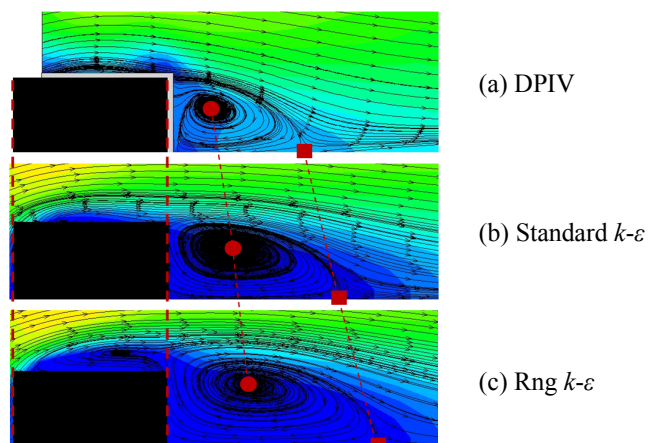


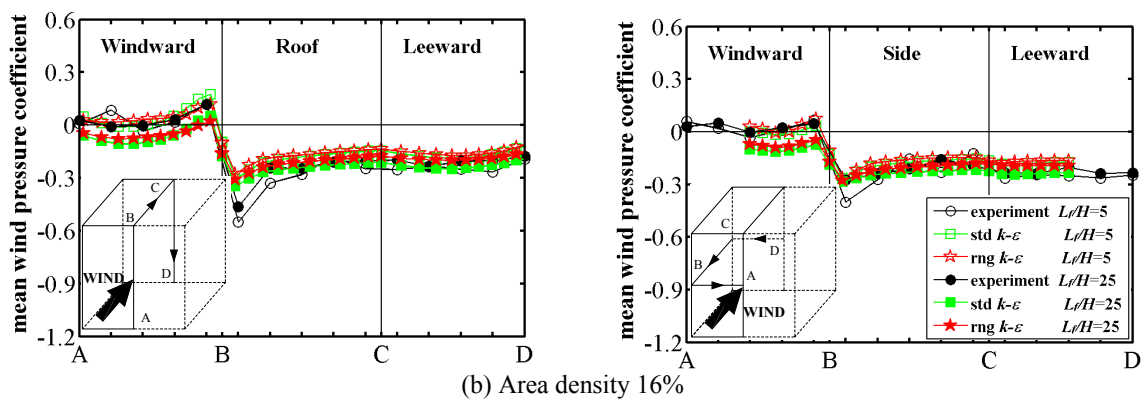
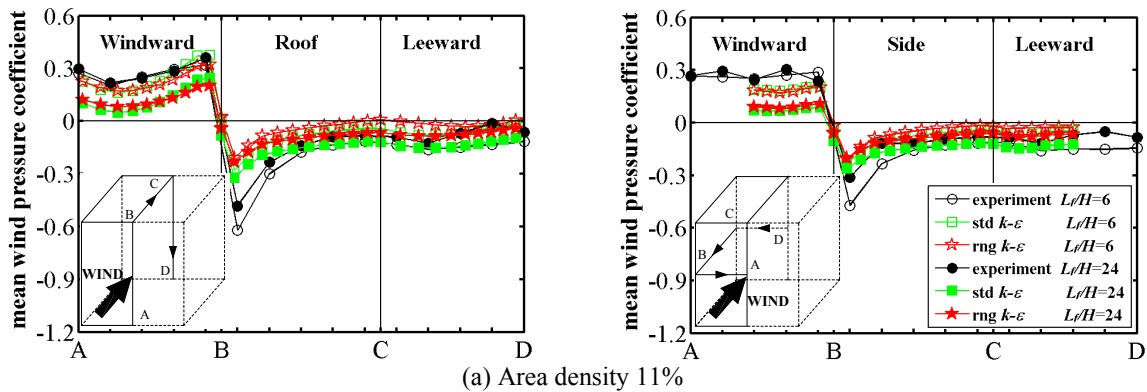
Figure 8. Comparison of flow field of isolated building for mid-height center

(smaller absolute value) on the leading edge on the roof and side surfaces, as shown in Figure 4. The results for the flow fields of the fine grid show a similar trend.

Summarizing the above results, it was found that there is clear dependence of wind pressure on the turbulence models, but little dependence on grid, meaning that the numerical simulations for the grids with $(30\sim61)<y^+$ result in almost identical results. Thus, the simulation of the target model surrounded by a large group of dummy models was conducted for the grid with $(30\sim62)<y^+<(\text{roughly } 400)$.

3.2 Results for target model

Figure 9 shows the variation of mean wind pressure coefficients derived from wind tunnel tests (solid and open circles), the standard $k-\varepsilon$ model (solid and open squares), and the rng $k-\varepsilon$ model (solid and open stars) at mid-width and mid-height center at $L_f/H=8$ and 24 for various area densities. Note that as only half of the target model was used in the simulation, the results from half the windward and leeward surfaces were used for the mid-height center. For all area densities, the variation trends of the numerical results along the two centers are similar to the experimental results, and correspondence becomes better at higher area density. For example, the mean wind pressure coefficients on the windward surface decrease with area density, and at higher area density the coefficients at all surfaces become similar. However, large differences were found near the separation region on the roof and side surfaces as for the isolated model, showing larger values than the experimental results (smaller absolute value). But the differences in the turbulence models are generally small for all surfaces at different upstream distances, showing similar degrees of correspondence to the experimental results.



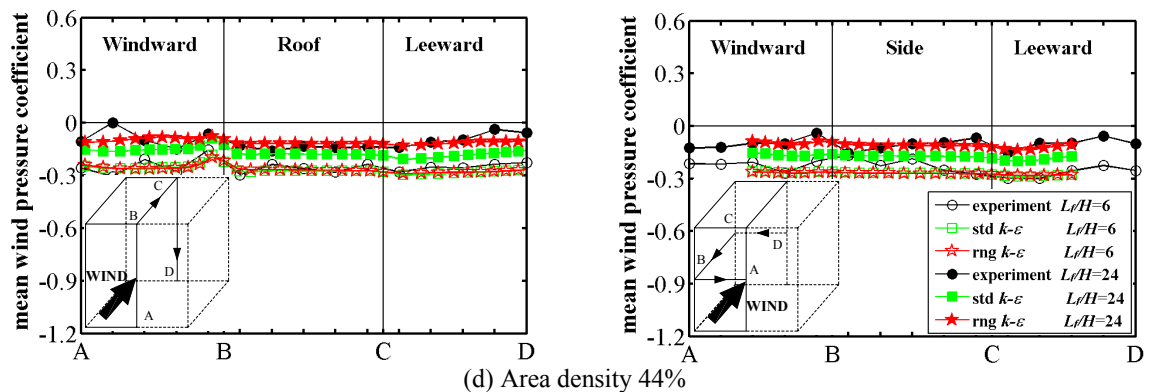
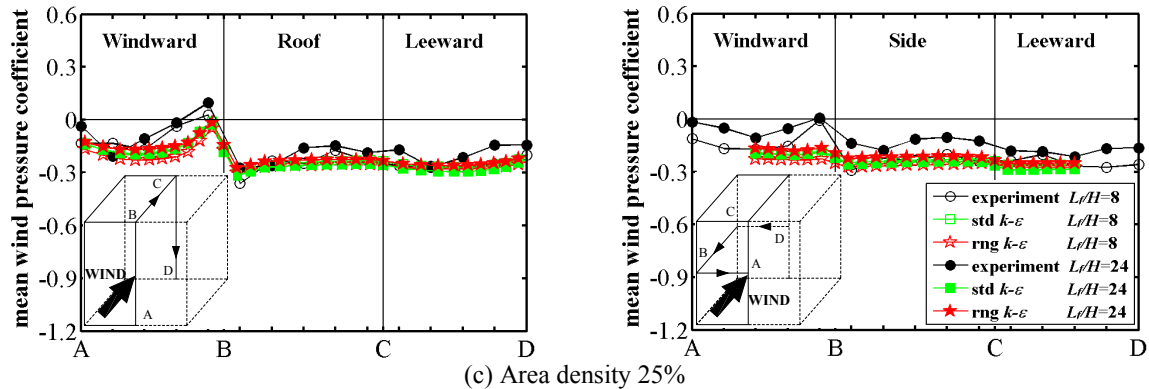


Figure 9. Comparison of mean C_p for various area densities (Mid-width center (left) and mid-height center (right))

3.3 Flow over large group of low-rise buildings

The variations of mean wind speed and turbulence intensity are shown in Figure 10. In simulation and experiment, the profiles were measured without the target model, and in the wind tunnel tests the profiles were measured from 0.1m (model height) to 0.4m using hot wire. The mean wind speed profiles from the wind tunnel tests and simulations agree well with each other, showing little differences depending on the turbulence models. But in the turbulence intensity profiles, the turbulence intensity for the standard $k-\epsilon$ model is slightly higher than that for the rng $k-\epsilon$ model when the height is less than 0.4, and is more consistent with the wind tunnel test results. The profiles from the standard $k-\epsilon$ model seem to be more similar to the wind tunnel test results. For other area densities, similar results with a similar degree of correspondence to the experimental results were obtained.

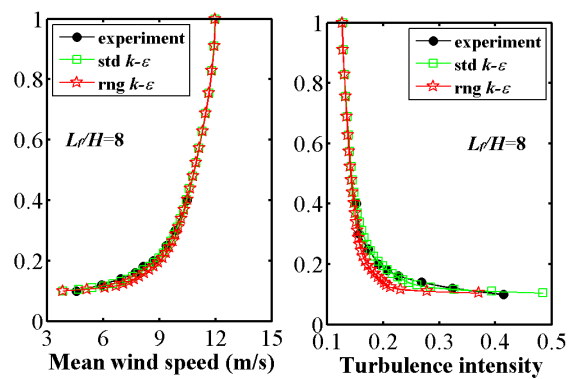


Figure 10. Comparisons of mean wind speed and turbulence intensity profiles at $C_A=25\%$

4 CONCLUSIONS

Mean wind pressure coefficients of the isolated model obtained from the $\text{rng } k\text{-}\varepsilon$ turbulence model match well with those from the pressure measurement. However, the differences in turbulence models were small for the target model in a large group, and the correspondence to experimental data was generally good, still showing some discrepancy near a large suction area. Using a limited number of pressure taps on the windward and roof surfaces, the prediction of peak C_p was attempted, and the results agreed with the experimental results, once the peak factors were given appropriately. For the flow field, the RANS turbulence models were able to reproduce the overall flow pattern around the model generally, but some differences were found i.e., the $\text{rng } k\text{-}\varepsilon$ model provided reasonable simulation near the regions where large suction occurs, and in other many regions the standard $k\text{-}\varepsilon$ model gave more similar results to experiment.

5 ACKNOWLEDGEMENTS

This study was funded by the Ministry of Education, Culture, Sports, Science and Technology, Japan, through the Global Center of Excellence Program, 2008-2012. The authors gratefully acknowledge their support.

6 REFERENCES

- 1 Blocken, B., Carmeliet, J., Stathopoulos, T., CFD evaluation of wind speed conditions in passages between parallel buildings-effect of wall-function roughness modifications for the atmospheric boundary layer flow, *J. Wind Eng. Ind. Aerod.*, 95 (2007) 941-962.
- 2 Chang, C.H., Meroney, R.N., Concentration and flow distributions in urban street canyons - wind tunnel and computational data, *J. Wind Eng. Ind. Aerod.*, 91 (2003) 1141-1154.
- 3 Kim, Y.C., Yoshida, A., Tamura, Y., Characteristics of surface wind pressures on low-rise building immersed in large group of surroundings, *Eng. Struct.*, Vol. 35 (2011) 18-28.
- 4 Kim, Y.C., Yoshida, A., Tamura, Y., Influence of surrounding buildings on wind loads acting on low-rise building, *J. Struct. Eng.*, ASCE, (To be published) (2012)
- 5 Lien, F.S., and Yee, E., Numerical modeling of the turbulent flow developing within and over a 3-D building array, Part I: A high-resolution Reynolds-Averaged Navier-Stokes approach, *Boundary-Layer Meteorol.*, 112 (2004) 427-466.
- 6 Santiago, J.L., Martilli, A., Martin, F., CFD simulation of airflow over a regular array of cube. Part I: Three-dimensional simulation of the flow and validation with wind-tunnel measurements, *Boundary-Layer Meteorol.*, 122 (2007) 609-634.
- 7 Sun, Y., Tamura, Y., Quan, Y., Matsui, M., The interference effect of surrounding roughness on wind pressures of rectangular prism, 6th Int. Colloquium Bluff Bodies Aerody. Appl., 2008, pp. 1-10.
- 8 Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirasawa, T., AIJ guidelines for practical applications of CFD pedestrian wind environment around buildings, *J. Wind Eng. Ind. Aerod.* 96 (2008) 1749-1761.