Wind and Structures, Vol. 13, No. 1 (2010) 21-36
DOI: http://dx.doi.org/10.12989/was.2010.13.1.021

Computational assessment of blockage and wind simulator proximity effects for a new full-scale testing facility

Girma T. Bitsuamlak*, Agerneh Dagnew and Arindam Gan Chowdhury

Laboratory for Wind Engineering Research, International Hurricane Research Center, Department of Civil and Environmental Engineering, Florida International University, Miami, Florida 33174, USA.

(Received March 23, 2009, Accepted August 27, 2009)

Abstract. A new full scale testing apparatus generically named the Wall of Wind (WoW) has been built by the researchers at the International Hurricane Research Center (IHRC) at Florida International University (FIU). WoW is capable of testing single story building models subjected up to category 3 hurricane wind speeds. Depending on the relative model and WoW wind field sizes, testing may entail blockage issues. In addition, the proximity of the test building to the wind simulator may also affect the aerodynamic data. This study focuses on the Computational Fluid Dynamics (CFD) assessment of the effects on the quality of the aerodynamic data of (i) blockage due to model buildings of various sizes and (ii) wind simulator proximity for various distances between the wind simulator and the test building. The test buildings were assumed to have simple parallelepiped shapes. The computer simulations were performed under both finite WoW wind-field conditions and in an extended Atmospheric Boundary Layer (ABL) wind flow. Mean pressure coefficients for the roof and the windward and leeward walls served as measures of the blockage and wind simulator proximity effects. The study uses the commercial software FLUENT with Reynolds Averaged Navier Stokes equations and a Renormalization Group (RNG) k-ε turbulence model. The results indicated that for larger size test specimens (i.e. for cases where the height of test specimen is larger than one third of the wind field height) blockage correction may become necessary. The test specimen should also be placed at a distance greater than twice the height of the test specimen from the fans to reduce proximity effect.

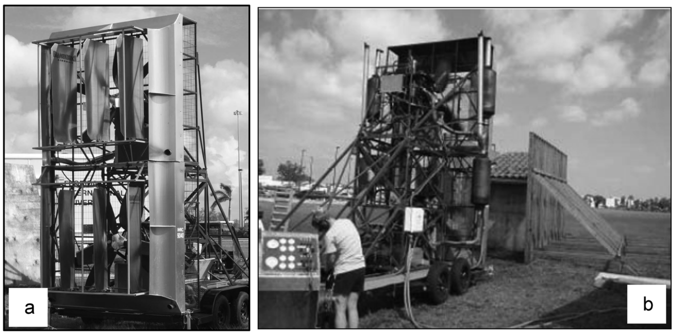
Keywords: full scale testing; blockage; wind simulator proximity; CFD; pressure coefficient; turbulence.

1. Introduction

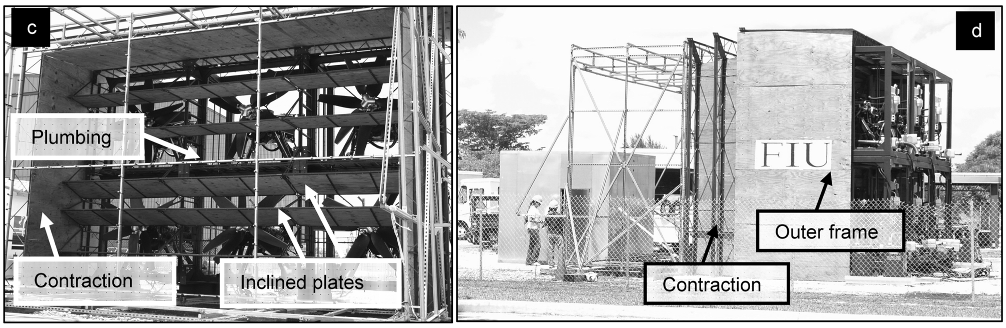
Central to FIU's research is the development in stages of full-scale testing facilities of the type generically called Wall of Wind (WoW), capable of producing hurricane level winds, in conjunction with wind-driven rain and wind-borne debris. The WoW, capable as it is of testing to failure entire structures at full scale, is an effective way of acquiring the experimental knowledge needed to mitigate hurricane damage in real buildings, and of powerfully demonstrating the damage wreaked by hurricanes on buildings as well as the dramatic loss reductions inherent in

* Corresponding Author, E-mail: bitsuamg@fiu.edu

effective mitigation measures. As a first phase of this development effort, the International Hurricane Research Center (IHRC) team at Florida International University (FIU) has built a full-scale 2-fan WoW facility (Figs. 1(a) and 1(b)) for testing small structures and assemblies, including roof fascias, barrel tile roofs, hurricane mitigation products, and Florida Power & Light utilities, (Gan Chowdhury, *et al*. 2009a and 2009b). Building on this experience FIU has subsequently built a larger, more powerful Renaissance-Re 6-fan WoW generating up to category 3 hurricane winds and wind-driven rain (Huang, *et al*. 2009, Bitsuamlak, *et al*. 2009) with sufficient wind field size to engulf a small single-story building (Figs. 1(c) and 1(d)). To house this and future larger WoW systems, a 30.5 × 24.4 × 10.7 m building is under construction at FIU. The North and South faces of the building consist largely of folding doors that will remain open during operation/testing. The WoW will be located on the South end. A dynamically controllable 4.9 m diameter turntable is located 2.7 m downstream of the WoW. Test buildings will be placed on the turntable to allow simulation of the effect of wind directionality. The WoW system forms a large open circuit system during operation. Further expansion and improvements on the current design of WoW using more number of fans are underway with financial support from the State of Florida Legislature. A 1:8 scale replica of the current 6 fan WoW has been built (Fig. 2) to help design flow management components (contraction, airfoil layouts, etc) before testing and implementing them at full-scale on the 6-fan WoW (Huang, *et al*. 2008, 2009). This approach has been found to be efficient and economical. In this study, it is not the intention to use the reduced-scale model of



Two-fan Wall of Wind (a) front isometric view and (b) rear isometric view



Current Six-fan Wall of Wind (c) front view and (d) side view

Fig. 1 Wall of Wind full-scale testing facility

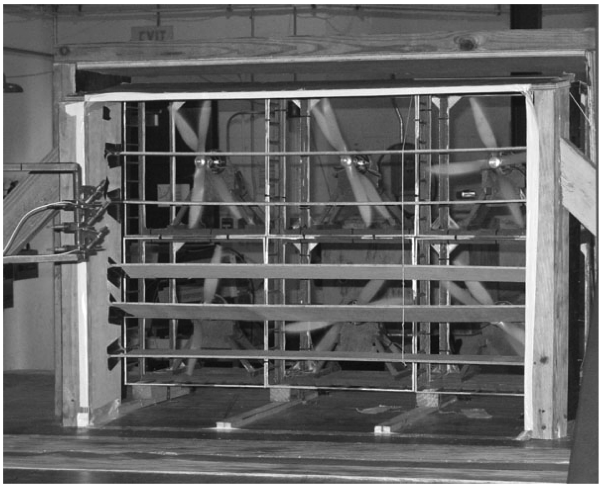


Fig. 2 Six-fan WoW small-scale (1:8) model

the WoW yet.

Testing larger test specimens within the finite WoW wind field, either to achieve Reynolds number similarity or to assess performance of full-scale building components under wind, wind-driven rain, and debris impact resistance, may entail blockage issues. The blockage effect discussed in the present study is concerned with the size of the test specimen compared to the finite size of the wind field generated at the inlet. The need to keep the test specimen as close as possible to the wind simulator in order to subject the test model to strong wind before it diffuses and loses its strength may also affect the quality of the aerodynamic data. The objectives of this study are, therefore, to assess computationally (i) the blockage effect as a function of the size of the test specimens, and develop correction strategies for those cases where those effects are significant, and (ii) the wind simulator proximity effect for various distances between the wind simulator and the test building, and develop proper test guidelines to ensure that this effect is acceptably small. The evolution of computational wind engineering (CWE) based on computational fluid dynamics (CFD) principles is making numerical evaluation of wind effects on built environment a potentially attractive proposition. This is particularly true in light of the positive development trends in hardware and software technology, as well as numerical modeling. Significant progress has been made in the application of CWE to evaluate wind loads on buildings (e.g. Murakami and Mochida 1988; Selvam 1997; Stathopoulos 1997; Wright and Easom 2003; Camarri, *et al*. 2005; Tamura 2006; Tutar and Celik 2007; El-Okda, *et al*. 2008; Tominaga, *et al*. 2008a; Cóstola, *et al*. 2009 and others). Significant progress has also been made on the evaluation of wind load modifications due to topographic elements (Bitsuamlak, *et al*. 2004, 2006). Some countries have already established working groups to investigate the practical applicability of CWE and develop recommendations for their use for wind resistant design of actual buildings and for assessing pedestrian level winds, within the framework of the Architectural Institute of Japan (AIJ) (Tamura, *et al*. 2008, Tominaga, *et al*. 2008b) and the European cooperation in the field of scientific and technical (COST) research, Franke, *et al*. (2007). Further, AIJ provides methods for predicting wind loading on buildings by the Reynolds Averaged Navier Stokes equations (RANS) and LES. Practical applications of CWE are widespread in areas such as pedestrian level wind evaluation Chang (2006), Lam and To (2006), and Blocken and Carmeliet (2008), where only the mean wind speeds are required for evaluating pedestrian comfort Stathopoulos and Hu (2004). CWE applications for wind driven rain are reported by researchers such as Choi (2000), and Blocken and Cameliet (2004). Some CFD wind flow studies for urban neighborhood include Zhang, *et al*. (2006), Huang, *et al*. (2006), and Jiang, *et al*. (2008). While most of studies mentioned above focus on straight winds, studies by Lin and Savory (2006), and Hangan and Kim (2008) focused on simulation of downburst. Other common uses of CWE, to which the present study belong to, include augmentation of experimental wind engineering research: Sengupta and Sarkar (2008) augmented their microburst and tornado wind simulator facility with numerical simulation; Moonen, *et al*. (2006, 2007) used CFD to assess quality of wind tunnel flow conditions and design of wind tunnels. Merrick and Bitsuamlak (2008) used numerical simulation to facilitate selection of an artificial surface roughness length to be applied on curved surfaced buildings during wind tunnel testing as a means to compensate for High Reynolds number effects that is usually missing from low wind speed tunnels. Okajima, *et al*. (1997) computationally assessed the effects of blockage pertaining to the effect of tunnel walls on various aerodynamic features.

In the present study, numerical wind flow simulations around parallelepipeds of various sizes, and located at various distances from the wind simulator and engulfed inside the numerical WoW and Atmospheric Boundary Layer (ABL) model have been carried out. In parallel, work is in progress to study the blockage and proximity effects experimentally by using the 1:8 scale small WoW replica in conjunction with the full-scale WoW. When they become available, the test results will be used to validate the Computational Fluid Dynamics (CFD) studies, following the approach of Bitsuamlak (2006). In the mean time, wind tunnel data from literature has been used to validate the present numerical models, resulting in a reasonable agreement with the CFD simulations, as will be discussed in section 3. Previous computational blockage assessments for wind tunnels include studies by Okajima, *et al*. (1997) pertaining to the effect of tunnel walls on various aerodynamic features as mentioned earlier. In the present blockage and wind simulator proximity effect study, however, the focus is on the effect of the size of the test buildings with respect to the finite size of WoW wind field and test building's proximity to the wind simulator. The WoW wind field can for practical purpose be considered to be a wind jet generated by an array of fans with controlled wind-profile characteristics.

2. Numerical modeling

The commercial software FLUENT 6.2 was utilized for the present numerical simulation, and the governing equations employed were the Reynolds Averaged Navier-Stokes (RANS) equations, together with the Renormalization Group (RNG) k-ε turbulence model. For blockage assessment studies, the upstream (U/S), top, downstream (D/S), and two sides of the computational domain (CD) were set to 3.5H, 7H, 10.5H, and 5.5H from the center of the base of the parallelepiped, respectively, as shown in Fig. 3, where H is the parallelepiped height under investigation, as shown in Fig. 4. For wind simulator proximity assessments, cubical buildings with windward faces located at H, 2H, 3H, 4H and 5H from the wind source (fans) were considered, as shown in Fig. 5. For wall bounded flow, Fluent 6.2 provides two different approaches for modeling flows in the inner viscous layer, i.e. use of wall functions or near-wall modeling based on the non-dimensional wall units. The first grid point y_p is placed at 0.01 m from the surface of the test specimen and

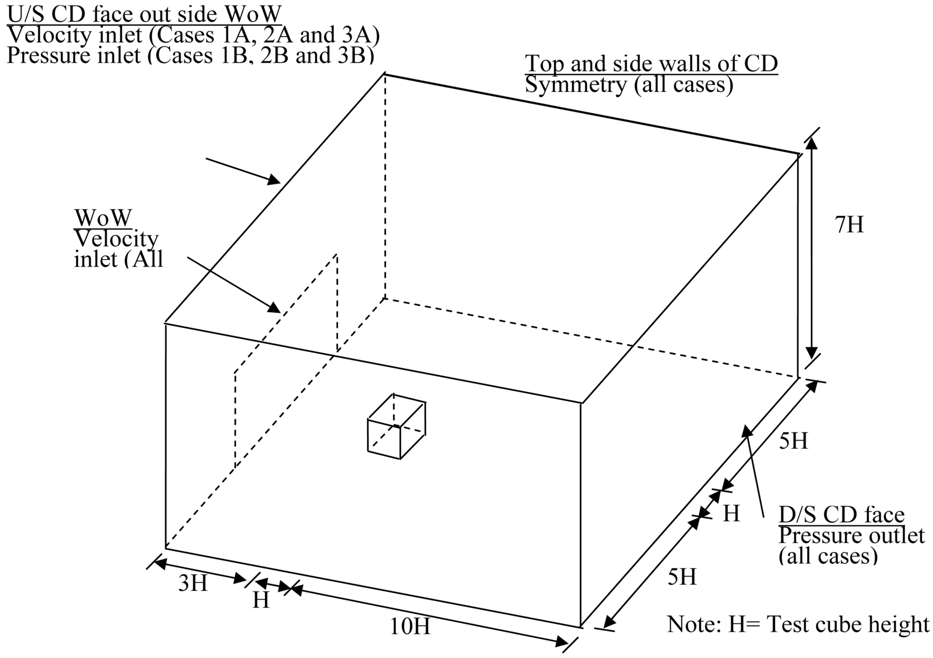


Fig. 3 Computational Domain (CD) and boundary conditions as defined by FLUENT

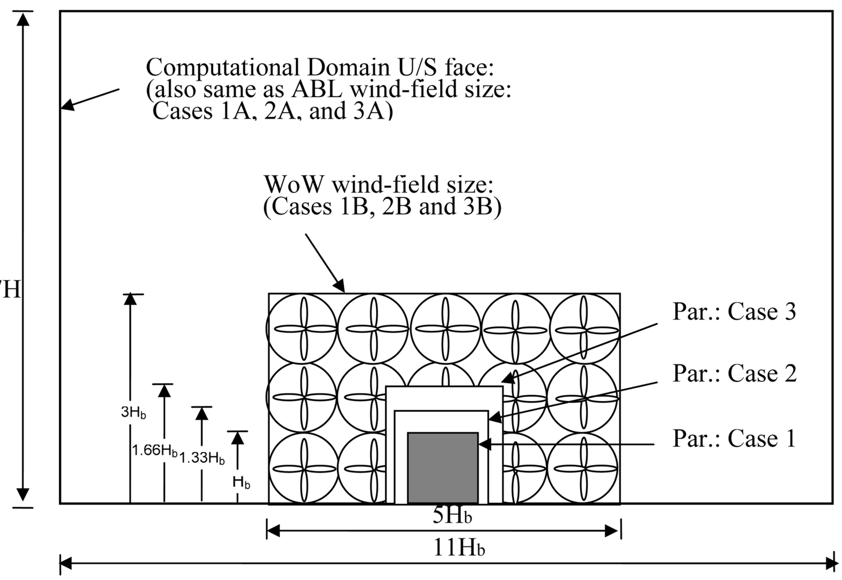


Fig. 4 Sizes of test parallelepipeds and wind-fields at the inlet used for blockage assessment studies. Note that only the grey building has been used for wind simulation proximity assessment

unstructured grids of hexagonal type were used for the CFD simulation. Considering the computational cost in resolving the inner layer, standard wall functions has been used in all present simulations by

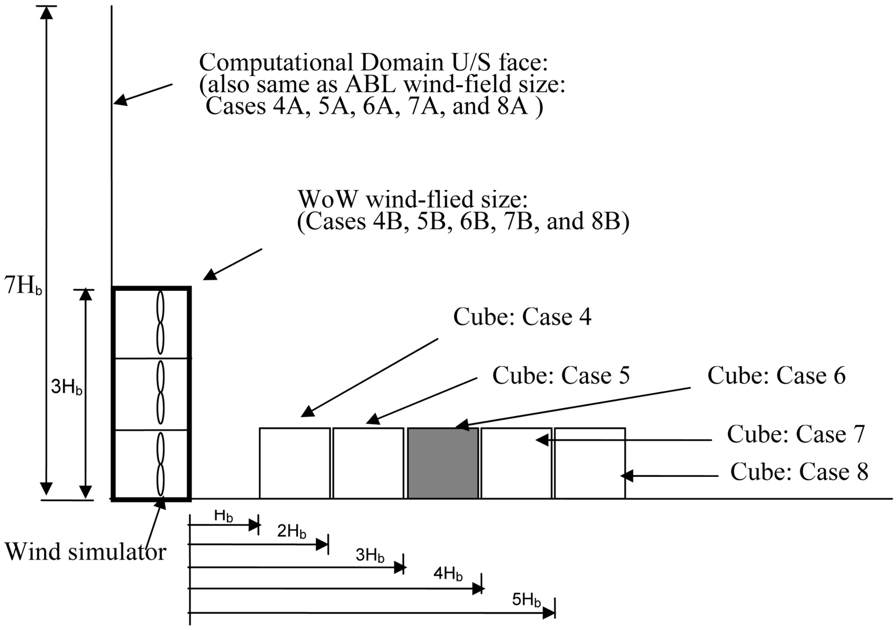


Fig. 5 Test cube windward face distances from the wind simulator (fans) for different simulation cases (Hb, 2Hb, 3Hb, 4Hb, and 5Hb for Cases 4, 5, 6, 7 and 8 respectively)

maintaining the wall unit y+ between 30 and 500. In addition, the inlet power law velocity profile with exponent α=0.25, a turbulence intensity TI=12%, and a 10 m integral length scale were assumed. The latter is less than the typical accepted value for suburban terrain, owing to the need to limit to a minimum the computational domain (CD) size -- assumed to be three times the length scale -- to reduce computational time. These are reasonable assumptions considering the comparative nature of this study. When simulating the ABL, the velocity inlet profile as described above was applied to the whole upstream face of the computational domain. However, when simulating the WoW flow, the application of the velocity inlet was limited to the 12 m × 9 m area of the U/S face of the CD representing the WoW type wind-field condition, as shown in Fig. 3; on the remaining inlet area the atmospheric pressure condition was applied.

A segregated pressure-velocity solver has been used to all the discretization schemes. Pressure interpolation is standard and second order upwind and third order MUSCL schemes were used for convection and momentum terms respectively. The convergence criterion has been limited to 10^{-5}.

For blockage assessment studies, computational models mimicking the WoW and the ABL test model conditions were developed for the three cases shown below. It is to be noted that the blockage effect discussed in the present study is concerned with the size of the test specimen compared to the finite size of the wind field generated at the inlet (see Fig. 4).

Case 1A - Base case for a 3 × 3 × 3 m (height × width × depth) cube placed in ABL wind-field condition (for this case $H=H_b$=3 m);

Case 1B - Same as Case 1A but placed inside WoW wind-field condition;

Case 2A - A 4 × 4 × 3 m (height × width × depth) parallelepiped placed in ABL wind-field condition ($H=1.33H_b$);

Case 2B - Same as Case 2A but placed inside WoW wind-field condition;
Case 3A - A 5 × 5 × 3 m (height × width × depth) parallelepiped placed in ABL wind-field condition ($H=1.66H_b$);
Case 3B - Same as Case 3A but placed inside WoW wind-field condition.

For wind proximity effect studies, computational models mimicking the WoW and the ABL test model conditions for the 3 × 3 × 3 m base cube were developed for the following three cases:

Case 4A - Windward face of base cube located at distance H from the wind simulator and placed in ABL wind-field condition;
Case 4B - Same as Case 4A but placed inside WoW wind-field condition;
Case 5A - Windward face of base cube located at distance 2H from the wind simulator and placed in ABL wind-field condition;
Case 5B - Same as Case 5A but placed inside WoW wind-field condition;
Case 6A - Windward face of base cube located at distance 3H from the wind simulator and placed in ABL wind-field condition (note this case is the same as Case 1A);
Case 6B - Same as Case 6A but placed inside WoW wind-field condition (note this case is the same as Case 1B);
Case 7A - Windward face of base cube located at distance 4H from the wind simulator and placed in ABL wind-field condition;
Case 7B - Same as Case 7A but placed inside WoW wind-field condition;
Case 8A - Windward face of base cube located at distance 5H from the wind simulator and placed in ABL wind-field condition;
Case 8B - Same as Case 8A but placed inside WoW wind-field condition;

Fig. 4 describes the relative size of the parallelepipeds relative to the WoW wind-field ($5H_b \times 3H_b$) and the ABL wind-field (11H × 7H) for Cases 1 to 8, where H_b represents the height of the base cube (H_b=3 m) and H represents the height of the study building for each case. Note that the depth (along the wind flow direction) of all the parallelepipeds considered in the present study is 3 m. Fig. 5 describes the distances from the windward face of the study base cube (3 × 3 × 3 m) from the wind simulator used for cases 4, 5, 6, 7 and 8. In all simulations the wind direction was perpendicular to the upwind face of the parallelepiped. Although the parallelepiped has simple geometry, it represents the complex bluff-body aerodynamic characteristics of a real building. In addition, several experimental studies and results are available for parallelepipeds, which allow the validation of results from the present study against values available in the literature.

3. Results and discussion

To validate the present simulation, results for the base case (i.e. Case 1A) have been compared with experimental results from the literature, as shown in Fig. 6, which also contains numerical results obtained by other researchers. Mean pressure coefficients normalized by reference velocity (U_H=29.43 m/s) at the building height measured at the inlet boundary location for the center vertical lines at U/S and D/S faces of the parallelepiped (i.e. AB and CD) and center horizontal line at the roof (i.e. BC) of the parallelepipeds were used for the comparison. As shown in Fig. 6, the results from previous studies that utilized LES (Lim, *et al*. 2009) or RNG k-ε (Wright and Easom 2003) is in better agreement with the boundary layer wind tunnel BLWT data compared to standard k-ε

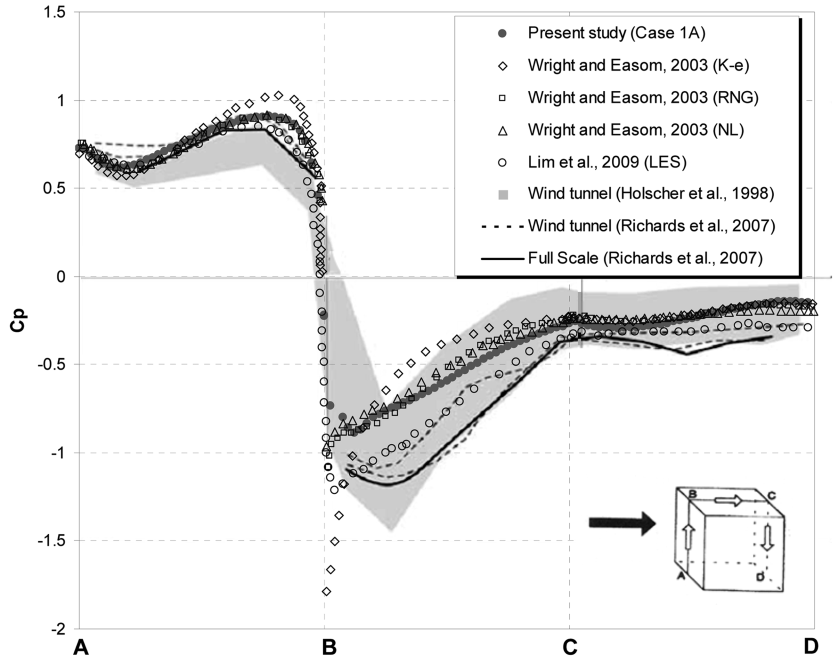
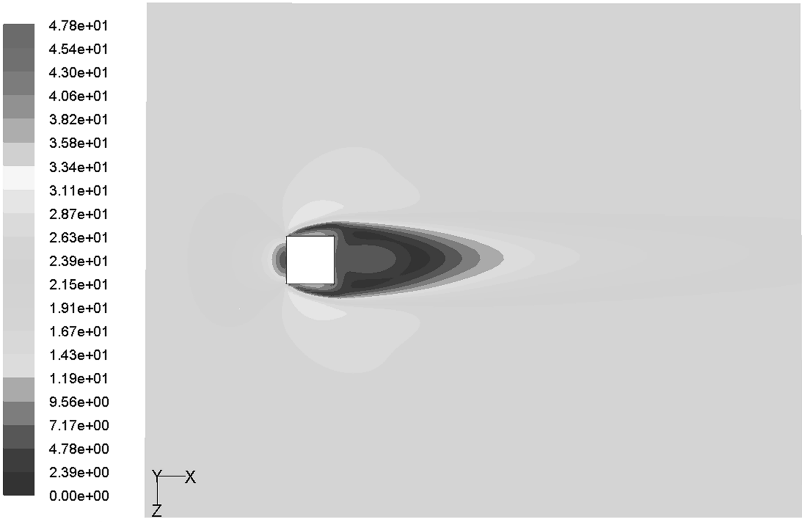


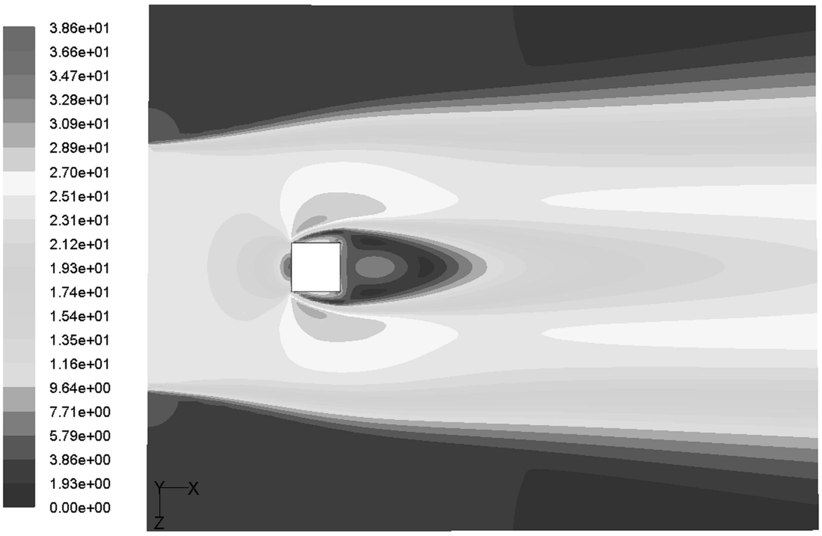
Fig. 6 Comparison of mean wind pressure coefficients: Experimental measurements and numerical simulations by using several turbulence models

model (Wright and Easom 2003). The latter over predicted the pressure coefficients on the windward wall and the roof. In the present study RNG k-ε has been opted due to its relatively good agreement with BLWT compared to Standard k-ε and relatively lower computational resource demand compared to LES. As can be seen from Fig. 6, the present simulation (Case 1A) is in good agreement with boundary layer wind tunnel (BLWT) data from the literature, represented by the grey region. In comparison to full-scale testing (Richards, *et al.* 2007) the numerical result gives less accurate pressure coefficient values for the roof (Root Mean Square Error -RMSE=0.222) compared to the windward wall (RMSE=0.131) and leeward wall (RMSE=0.146) similar to reports in literature (Stathopoulos 2003, Bitsuamlak 2006). However it is to be noted that these errors are in similar order of magnitude with that of the variations observed in pressure coefficients measured in wind tunnels. When examining the CFD results it is necessary to account for the variations within the experimental data, described by the grey region of Fig. 6. It is to be noted that this comparison is made only to give an indicator on the quality of CFD value compared to industry wide accepted wind tunnel measurements from literature no additional effort was made in the present study to verify the quality of the wind tunnel measurements.

Following the comparisons of the numerical simulations with results from the literature, the blockage assessments were pursued. The velocity contours for Case 1 are shown in Fig. 7. Figs. 7(a) and 7(b) show the contours on a horizontal plane at mid-height of the cube. Similarly, Figs. 7(c) and 7(d) show the contours on a vertical plane passing through the center of the cube. Fig. 8 shows the path-lines for the recirculation zones for Case 1. Qualitatively, there is generally good agreement in terms of size of recirculation length behind the parallelepipeds. Quantitatively, Fig. 9



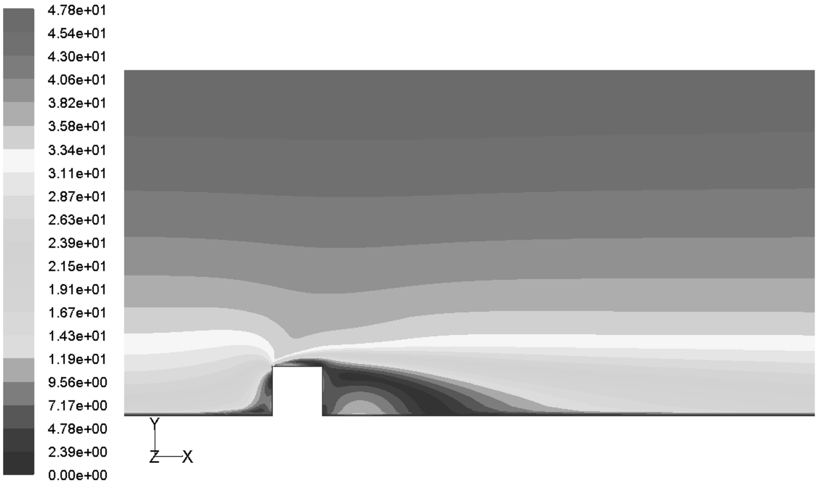
(a) Case 1A (ABL) – Horizontal plane at mid-height of the parallelepiped.



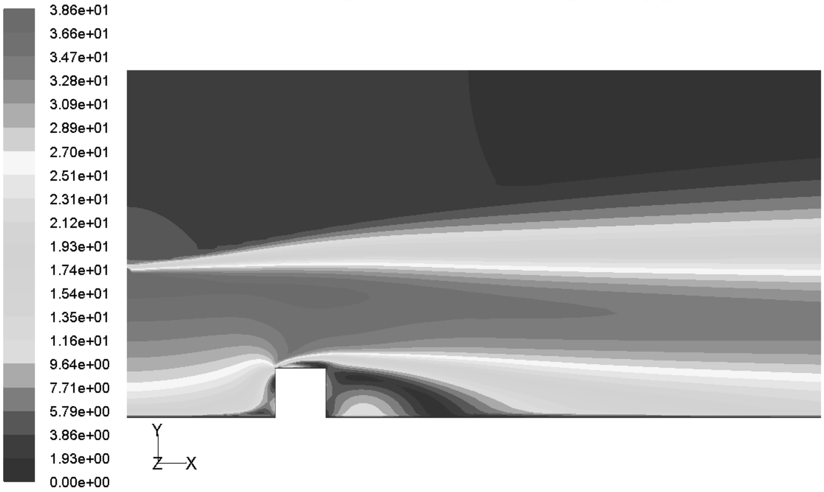
(b) Case 1B (WoW) – Horizontal plane at mid-height of the parallelepiped.

Fig. 7 Wind velocity contour plots for ABL and WoW simulation

shows mean pressure coefficient comparisons for Cases 1A (ABL) and 1B (WoW). As can be seen from the figure, there is a very good agreement between the two, confirming the viability of using a proper wind-jet flows generated by using the WoW system with proper turbulence and boundary layer generation schemes representing ABL conditions. Figs. 10 and 11 show similar results for Cases 2 and 3, respectively. Slight differences in mean pressure coefficients (Cp values) were observed for Cases 2 and Case 3 at the roof and leeward wall. These differences could be due to blockage or inadequacy of the basic type of turbulence model used in the present study. This remains to be verified through use of better numerical models such as Large Eddy Simulation (LES). The authors are currently working on an experimental investigation using the 1:8 scale WoW



(c) Case 1A (ABL) – Vertical plane at the center of the parallelepiped.

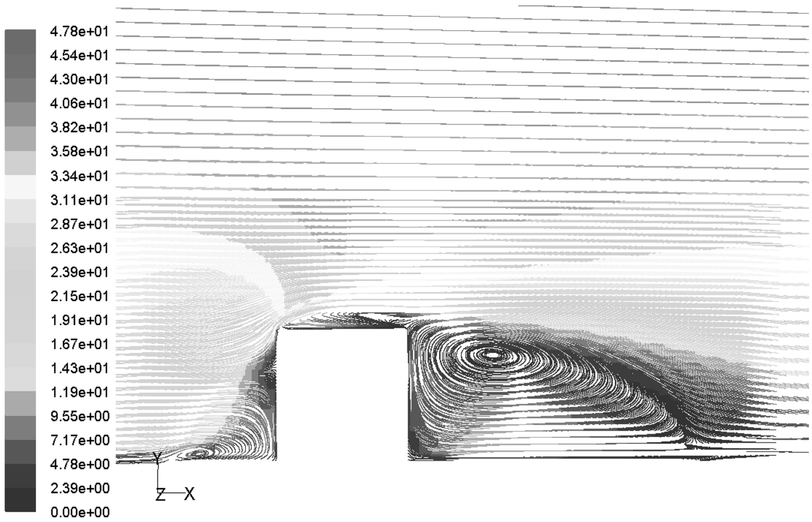


(d) Case 1B (WoW) – Vertical plane at the center of the parallelepiped

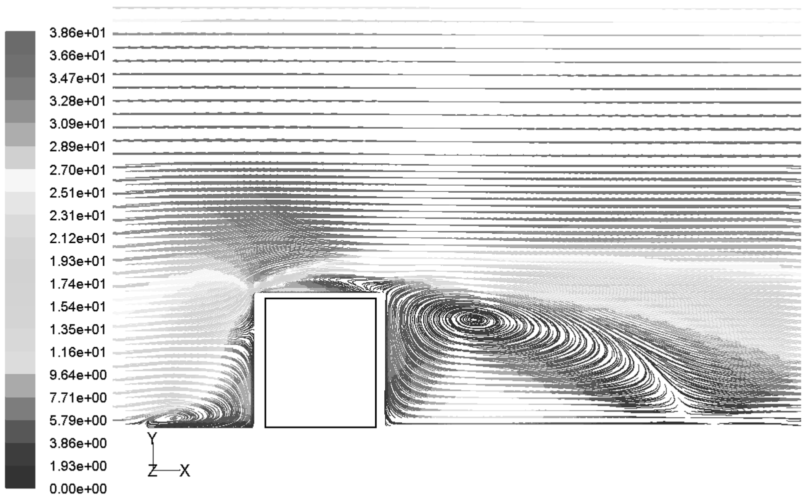
Fig. 7 Continued

replica and the full-scale WoW. Once the sources of these differences are identified, proper corrections can be applied when testing larger models.

Finally, the wind simulator proximity assessments were pursued. Similar to the blockage assessments, mean Cp values extracted from the center vertical lines at U/S and D/S faces of the parallelepiped (i.e., AB and CD) and the center horizontal line of the roof (i.e., BC) were used for comparison purposes. The mean Cp values for Cases 4, 5, 6, 7 and 8 were compared with the wind tunnel data obtained from the literature as shown in Fig. 12. There is generally good agreement between the CFD and the wind tunnel data for Cases 5, 6, 7 and 8. For Case 4, however, the comparison revealed exaggerated Cp values in the windward wall. This means that the pressure coefficients at the windward wall were created by higher wind speed than the wind speed used to obtain the pressure coefficients. It is to be recalled that wind speed measured at H



Case 1A (ABL)



Case 1B (WoW)

Fig. 8 Wind velocity path-lines and recirculation zones

ft from ground (H= cube height) before placing the cube in the testing position has been used to obtain the pressure coefficients. This is believed to be due to the close proximity of the test cube to the wind simulator blocking the flow before it expands upwards and to the sides thus resulting in a higher velocity that created the pressure system compared to the wind speed used for obtaining the pressure coefficients. Compared to windward wall, the roof Cp values were less sensitive to test building proximity to the wind simulator as can be seen in Fig. 11. The insensitivity of the roof pressures to the proximity of the wind simulator is believed to be due to localized flow effect such as flow separation at the roof level, which is less independent of the proximity parameter. For Cases 5, 6, 7 and 8, where the test cube was placed at >2H distance from the wind simulator, the exaggerated positive pressure disappeared. Thus, it may be concluded

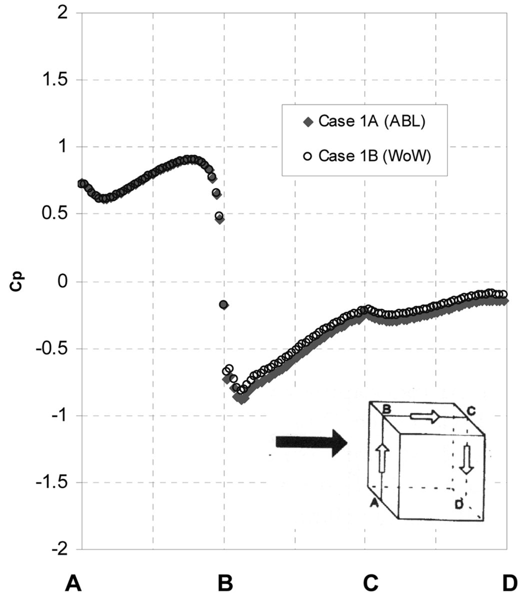


Fig. 9 ABL and WoW mean pressure coefficient comparisons for Case 1 (3×3×3 m cube)

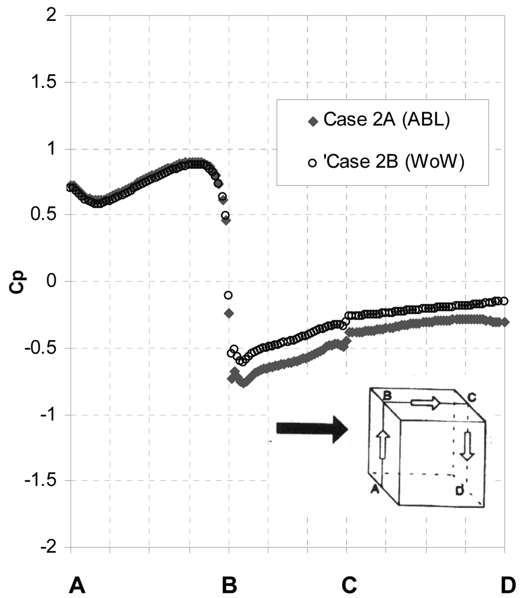


Fig. 10 ABL and WoW mean Cp comparisons for Case 2 (4×4×3 m parallelepiped)

that to obtain a good quality aerodynamic data on walls, the models needs to be placed at a distance of more than 2H from the wind simulator. However, for roof or roof top equipment tests the aerodynamic data are less sensitive to the proximity of the test-specimen to the wind simulator.

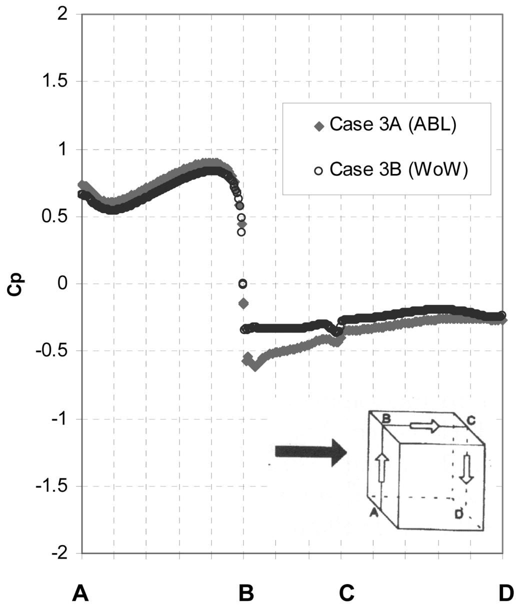


Fig. 11 ABL and WoW mean Cp comparisons for Case 3 (5×5×3 m parallelepiped)

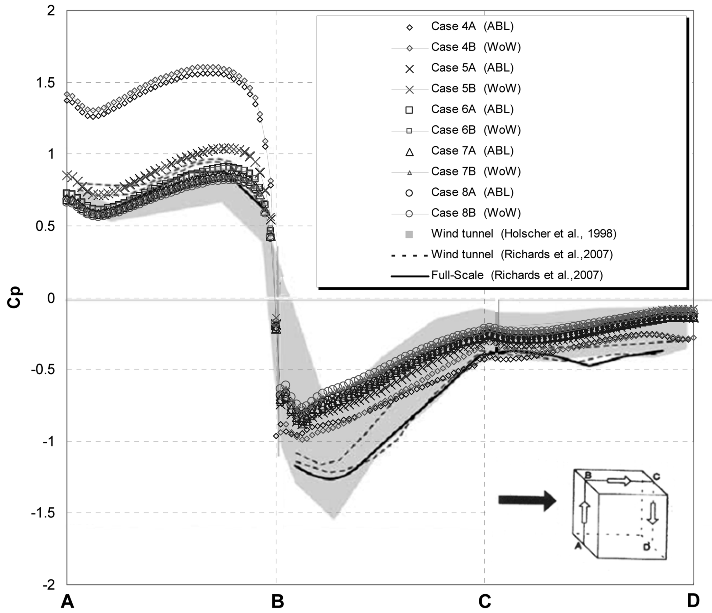


Fig. 12 ABL and WoW mean Cp comparisons for Cases 4, 5, 6, 7 and 8 with wind tunnel data from literature

4. Conclusions

The following guidelines and observations based on the present study are warranted:

(i) Pressure coefficients were reasonably reproduced. It was noted that the RMSE at the wind- and lee-ward walls (0.131 and 0.146 respectively) were better than the RMSE at the roofs (0.222).

(ii) For large test models, i.e. for cases where the height of the test model is larger than one third of the wind field height, carrying out proper blockage assessments is necessary.

(iii) Test buildings shall be preferably located at least 3H from the wind source (fans). If the model is placed closer than 3H, the quality of the aerodynamic data particularly in the windward wall can be compromised and appropriate correction needs to be applied. The roof aerodynamic data appears less sensitive compared to the windward wall.

These guidelines may be followed when conducting similar studies. The present study is limited to mean characteristics. In the future detailed validation focusing on the transient characteristics as well will be carried out by comparing the CFD results with 1:8 WoW replica and full scale WoW blockage and wind source proximity data.

Acknowledgments

The support from National Science Foundation (through NSF Career award to the first author) and Florida Department of Emergency Management is gratefully acknowledged. Dr Emil Simiu's comments during the preparation of the manuscript are also gratefully acknowledged.

References

Bitsuamlak, G.T. (2006), "Application of computational wind engineering: A practical perspective", *Third National Conf. in Wind Engineering*, January 5-7, Kolkata, India.

Bitsuamlak, G.T., Gan Chowdhury, A. and Sambare, D. (2009), "Application of a full-scale testing facility for assessing wind-driven rain intrusion", *Build. Environ.*, **44**, 2430-2441.

Bitsuamlak, G.T., Stathopoulos, T. and Bédard, C. (2006), "Effect of upstream hills on design wind load: a computational approach", *Wind Struct.*, **9**(1), 37-58.

Bitsuamlak, G.T., Stathopoulos, T. and Bédard, C. (2004), "Numerical evaluation of turbulent flows over complex terrains: A review", *J. Aerospace Eng.*, **17**(4), 135-145.

Blocken, B. and Carmeliet, J. (2008), "Pedestrian wind conditions at outdoor platforms in a high-rise apartment building: generic sub-configuration validation, wind comfort assessment and uncertainty issues", *Wind Struct.*, **11**(1), 51-70.

Blocken, B. and Carmeliet, J. (2004), "A Review of Wind-driven Rain Research in Building Science", *J. Wind Eng. Ind. Aerod.*, **92**(13), 1079-1130.

Camarri, S., Salvetti, M.V., Koobus, B. and Dervieux, A. (2005), "Hybrid RANS/LES simulations of a bluff-body flow", *Wind Struct.*, **8**(6), 407–426.

Chang, C. (2006), "Computational fluid dynamics simulation of pedestrian wind in urban area with the effects of tree", *Wind Struct.*, **9**(2), 147-158.

Choi, E.C.C. (2000), "Variation of Wind-driven Rain Intensity with Building Orientation", *J. Arch. Eng.*, **6**, 122–130.

Cóstola, D., Blocken, B. and Hensen, J.L.M. (2009), "Overview of pressure coefficient data in building energy simulation and airflow network programs", *Build. Environ.*, **44**, 2027-2036.

El-Okda, Y.M., Ragab, S.A. and Hajj, M.R. (2008), "Large-eddy simulation of flow over a surface-mounted prism using a high-order finite-difference scheme", *J. Wind Eng. Ind. Aerod.*, **96**(6-7), 900-912.

Franke, J., Hellsten, A., Schlunzen, H. and Carissimo, B. (2007), *Best practice guideline for the CFD simulation of flows in the urban environment*, COST Office Brussels, ISBN 3-00-018312-4.

Gan Chowdhury, A., Bitsuamlak, G.T. and Simiu, E. (2009a), "Aerodynamic, hydro-aerodynamic, and destructive testing", *J. Struct. Build*., accepted for publication.

Gan Chowdhury, A., Simiu, E. and Leatherman, S.P. (2009b), "Destructive Testing under Simulated Hurricane Effects to Promote Hazard Mitigation", *Nat. Hazards Review J.* ASCE, **10**(1), 1-10.

Hangan, H. and Kim, J.D. (2008), "Swirl ratio effects on tornado vortices in relation to the Fujita scale", *Wind Struct*., **11**(4), 291-302.

Holscher, N. and Niemann, H.J. (1998), "Towards quality assurance for wind tunnel tests: A comparative testing program of the Windtechnologische Gesellschaft", *J. Wind Eng. Ind. Aerod*., **74**, 599-608.

Huang, H., Kato, S. and Ooka, R. (2006), "CFD analysis of ventilation efficiency around an elevated highway using visitation frequency and purging flow rate", *Wind Struct*., **9**(4), 297-313.

Huang, P., Gan Chowdhury, A., Bitsuamlak, G.T. and Liu, R. (2009), "Development of Devices and Methods for Simulation of Hurricane Winds in a Full-Scale Testing Facility", *Wind Struct.*, **12**(2), 151-177.

Huang, P., Liu, R., Gan Chowdhury, A., Bitsuamlak, G., Erwin, J. and Ahmed, S.S. (2008), "Turbulence Simulation of Small-Scale Wall of Wind Flows", *Proc. of the 4th Int. Conf. on Advances in Wind and Structures*, Jeju, Korea.

Jiang, D., Jiang, W., Liu, H. and Sun, J. (2008), "Systematic influence of different building spacing, height and layout on mean wind and turbulent characteristics within and over urban building arrays", *Wind Struct.*, **11**(4), 275-289.

Lam, K.M. and To, A.P. (2006), "Reliability of numerical computation of pedestrian-level wind environment around a row of tall buildings", *Wind Struct.*, **9**(6), 473-492.

Lin, W.E. and Savory, E. (2006), "Large-scale quasi-steady modelling of a downburst outflow using a slot jet", *Wind Struct.*, **9**(6), 419-440.

Lim, C.H., Thomas, T.G. and Castro, I.P. (2009), "Flow around a cube in a turbulent boundary layer: LES and experiment", *J. Wind Eng. Ind. Aerod*., **97**, 96-109.

Merrick, R. and Bitsuamlak, G.T. (2008), "Control of flow around a circular cylinder by the use of surface roughness", *4th Int. Conf., Advances on Wind and Structures (AWAS08)*, Jeju, Korea.

Moonen, P., Blocken, B. and Carmeliet, J. (2006), "Numerical modeling of the flow conditions in a closed-circuit low-speed wind tunnel", *J. Wind Eng. Ind. Aerod*., **94**, 966-23.

Moonen, P., Blocken, B. and Carmeliet, J. (2007), "Indicator for the evaluation of wind tunnel test section flow quality and application to a numerical closed-circuit wind tunnel", *J. Wind Eng. Ind. Aerod.*, **94**, 1289-1314.

Murakami, S. and Mochida, A. (1988), "3-D numerical simulation of airflow around a cubic model by means of the k-ε model", *J. Wind Eng. Ind. Aerod*., **31**, 283-303.

Okajima, A., Yi, D., Sakuda, A. and Nakano, T. (1997), "Numerical study of blockage effects on aerodynamic characteristics of an oscillating rectangular cylinder", *J. Wind Eng. Ind. Aerod*., **67&68**, 91-102

Richards, P.J., Hoxey, R.P., Connell, B.D. and Lander, D.P. (2007), "Wind-tunnel modelling of the Silsoe Cube", *J. Wind Eng. Ind. Aerod*., **95**, 1384-1399.

Selvam, S.P. (1997), "Computation of pressures on Texas Tech university building using large eddy simulation", *J. Wind Eng. Ind. Aerod*., **67&68**, 647-657.

Sengupta, A. and Sarkar, P.P. (2008), "Experimental measurement and numerical simulation of an impinging jet with application to thunderstorm microburst winds", *J. Wind Eng. Ind. Aerod*., **96**(3), 345-365.

Stathopoulos, T. (1997), "Computational wind engineering: Past achievements and future challenges", *J. Wind Eng. Ind. Aerod*., **67-68**, 509-532.

Stathopoulos, T. (2003), "Wind loads on low buildings: in the wake of Alan Davenport's contributions", *J. Wind Eng. Ind. Aerod*., **91**(12-15), 1565-1585.

Stathopoulos, T. and Wu, H. (2004), "Using Computational Fluid Dynamics (CFD) for pedestrian winds", *Proc. of the 2004 Structures Congress*, Nashville, TN.

Tamura, T., Nozawa, K. and Kondo, K. (2008), "AIJ guide for numerical prediction of wind loads on buildings", *J. Wind Eng. Ind. Aerod.*, **96**, 1974-1984.

Tamura, T. (2006), "Towards practical use of LES in wind engineering", *The fourth Int. Symp. in Computational Wind Engineering (CWE2006)*, Yokohama, Japan.

Tominaga, Y., Mochida, A., Murakami, S. and 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", *J. Wind Eng. Ind. Aerod*., **96**(4), 389-411.

Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu,T., Yoshikawa, M. and Shirasawa, T. (2008b), "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings", *J. Wind Eng. Ind. Aerod.*, **96**, 1749-1761.
Tutar, M. and Celik, I. (2007), "Large eddy simulation of a square cylinder flow: Modelling of inflow turbulence", *Wind Struct.*, **10**(6), 511-532.
Wright, N.G. and Easom, G.J. (2003), "Non-linear k-ε turbulence model results for flow over a building at fullscale", *Appl. Math. Model.*, **27**(12), 1013-1033.
Zhang, N., Jiang, W. and Miao, S. (2006), "A large eddy simulation on the effect of buildings on urban flows", *Wind Struct.*, **9**(1), 23-35.

CC