ELSEVIER

Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) 1749–1761

JOURNAL OF wind engineering AND industrial aerodynamics

www.elsevier.com/locate/jweia

AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings

Yoshihide Tominaga[a,*], Akashi Mochida[b], Ryuichiro Yoshie[c], Hiroto Kataoka[d], Tsuyoshi Nozu[e], Masaru Yoshikawa[f], Taichi Shirasawa[c]

[a] *Niigata Institute of Technology, Fujihashi 1719, Kashiwazaki-City, Niigata, Japan*
[b] *Tohoku University, Aoba 06, Aramaki, Aoba-ku, Sendai-City, Miyagi, Japan*
[c] *Tokyo Polytechnic University, Iiyama 1583, Atsugi-City, Kanagawa, Japan*
[d] *Technical Research Institute, Obayashi Corporation, Shimokiyoto 4-640, Kiyose-City, Tokyo, Japan*
[e] *Institute of Technology, Shimizu Corporation, Echujima 3-4-17, Koto-ku, Tokyo, Japan*
[f] *Technology Center, Taisei Corporation, Nasecho 344-1, Totsuka-ku, Yokohama, Japan*

Available online 15 April 2008

Abstract

Significant improvements of computer facilities and computational fluid dynamics (CFD) software in recent years have enabled prediction and assessment of the pedestrian wind environment around buildings in the design stage. Therefore, guidelines are required that summarize important points in using the CFD technique for this purpose. This paper describes guidelines proposed by the Working Group of the Architectural Institute of Japan (AIJ). The feature of these guidelines is that they are based on cross-comparison between CFD predictions, wind tunnel test results and field measurements for seven test cases used to investigate the influence of many kinds of computational conditions for various flow fields.

Keywords: CFD; Pedestrian wind environment; Prediction; Guidelines; Benchmark test

1. Introduction

Significant improvements of computer facilities and computational fluid dynamics (CFD) software in recent years have enabled prediction and assessment of the pedestrian

*Corresponding author. Tel./fax: +81 257 22 8176.
E-mail address: tominaga@abe.niit.ac.jp (Y. Tominaga).

0167-6105/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.jweia.2008.02.058

wind environment around buildings in the design stage. However, CFD has been applied with insufficient information about the influence of many factors related to the computational condition on prediction results. There have been several case studies on the pedestrian level wind environment around actual buildings using CFD (Stathopoulos and Baskaran, 1996; Timofeyef, 1998; Westbury et al., 2002; Richards et al., 2002). However, the influence of the computational conditions, i.e., grid discretization, domain sizes, boundary conditions, etc., on the prediction accuracy has not been systematically investigated. Therefore, a set of guidelines is required that summarize important points in using the CFD technique for appropriate prediction of pedestrian wind environment.

Some guidelines on industrial CFD applications have been published in order to clarify the method for validation and verification of CFD results (Roache et al., 1986; AIAA, 1998; ERCOFTAC, 2000). These guidelines provide valuable information on the applications for flow around buildings. However, no guidelines have yet been established on the use of CFD for investigating the pedestrian wind environment around buildings.

Recommendations have recently been proposed on the use of CFD in predicting the pedestrian wind environment by COST (European Cooperation in the field of Scientific and Technical Research) group (Action C14 "Impact of Wind and Storms on City Life and Built Environment" Working Group 2—CFD techniques). These recommendations (hereafter COST) were mainly based on the results published by other authors. They are summarized by Franke et al. (2004) and Franke (2006).

The guidelines for CFD prediction of the pedestrian wind environment around buildings were proposed by the Working Group in the Architectural Institute of Japan (AIJ), which consists of researchers from several universities and private companies. This working group conducted a lot of wind tunnel experiments, field measurements and computations using different CFD codes to investigate the influence of various kinds of computational parameters for various flow fields. A distinctive feature of these guidelines is that they were derived from extensive and numerous cross-comparisons, while those proposed by COST mainly consist of results obtained from a literature review. This paper also discusses similarities and differences to the COST recommendations.

The guidelines proposed here are mainly based on high Reynolds number (*Re*) Reynolds Averaged Navier–Stokes equations (RANS) models, although it is desirable to use a large eddy simulation (LES) and a low *Re* number type model in order to obtain more accurate results. However, it is difficult to use those models for practical analysis because many computational cases and a huge number of grids are required for the prediction and analysis of the pedestrian wind environment under severe time restrictions. In spite of that, these guidelines can also be helpful when using a highly accurate model like an LES or low Re number type model.

2. Outline of cross-comparison tests

In order to clarify the major factors affecting prediction accuracy, the Working Group carried out cross-comparisons of wind tunnel experiments, field measurements and CFD results of flow around a single high-rise building placed within the surface boundary layer, flow within a building complex in an actual urban area, and flow around a tree, obtained from various k–ε models, DSM and LES. Fig. 1 illustrates seven test cases for these cross-comparisons. In order to assess the effect of a specific factor, e.g., the performance of a turbulence model, the results were compared under the same computational conditions as

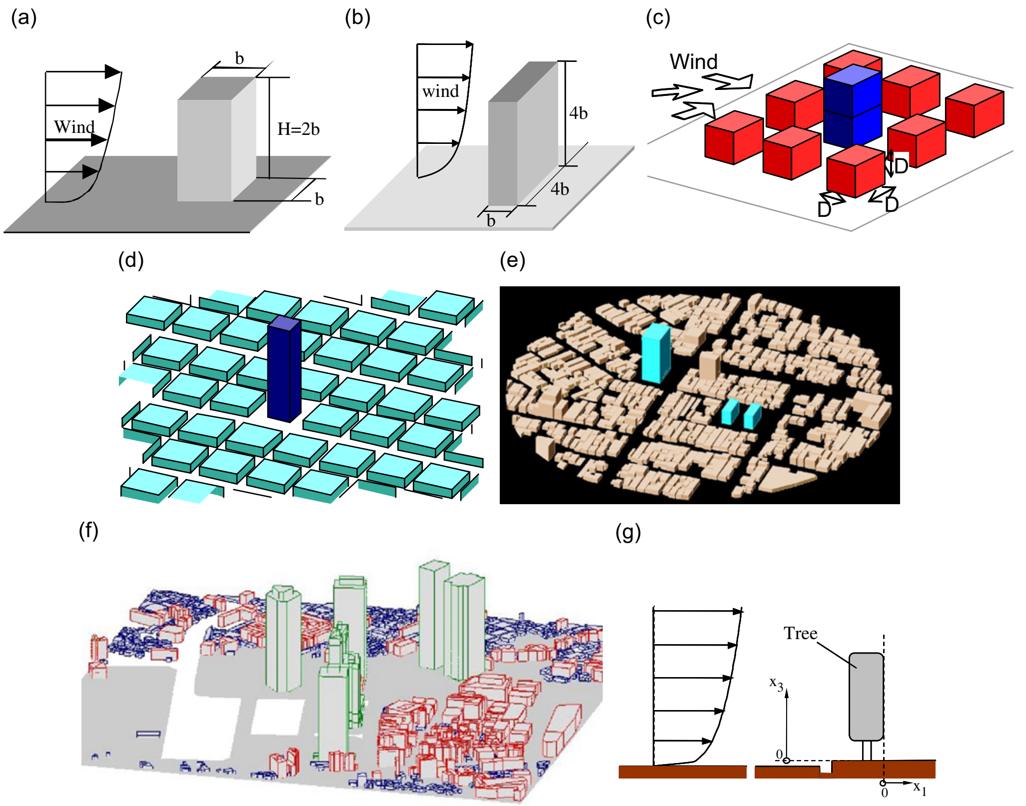

Fig. 1. Seven test cases for cross-comparison. (a) Test case A (2:1:1 square prism); (b) Test case B (4:4:1 square prism); (c) Test case C (Simple city blocks); (d) Test case D (High-rise building in city); (e) Test case E (Building complexes with simple building shapes in actual urban area); (f) Test case F (Building complexes with complicated building shapes in actual urban area); (g) Test case G (Two-dimensional pine tree).

for other factors. Special attention was paid to this point in this project. The basic computational conditions, i.e., grid arrangements, boundary conditions, etc., were specified by the organizers. Contributors were requested to use these conditions. The results of these cross-comparisons have been reported in several papers (Mochida et al., 2002, 2006; Shirasawa et al., 2003; Tominaga et al., 2004, 2005; Yoshie et al., 2005a, b, , 2006).

3. Computational domain and representation of surroundings

3.1. Domain size

For the size of the computational domain, the blockage ratio should be below 3% based on knowledge of wind tunnel experiments. For the single-building model, the lateral and the top boundary should be set 5H or more away from the building, where H is the height of the target building (Mochida et al., 2002; Shirasawa et al., 2003). The distance between the inlet boundary and the building should be set to correspond to the upwind area covered by a smooth floor in the wind tunnel. The outflow boundary should be set at least

$10H$ behind the building. Where the building surroundings are included, the height of the computational domain should be set to correspond to the boundary layer height determined by the terrain category of the surroundings (Architectural Institute of Japan, 2004). The lateral size of the computational domain should extend about $5H$ from the outer edges of the target building and the buildings included in the computational domain should not exceed the recommended blockage ratio (3%).

Similar requirements for the inlet and the top boundaries were suggested by COST. However, the recommended lateral boundaries ($2.3W$, where W is the width of built area) and the outflow boundaries ($15H_{max}$, where H_{max} is the height of the tallest building) (Franke, 2006) may be conservative. It should be noted that there is a possibility of unrealistic results if the computational region is expanded without representation of surroundings (Yoshie et al., 2006).

3.2. Representation of surroundings

For the actual urban area, the buildings in the region to be assessed (generally 1–$2H$ radius from the target building) should be clearly modeled. Moreover, at least one additional street block in each direction around the assessment region should also be clearly reproduced (Yoshie et al., 2005a, b). In addition, it is recommended to use some simplified geometries of a cluster of buildings or to specify appropriate roughness lengths z_o for the ground surface boundary condition to represent the roughness of the outer region (from the outer edge of the additional street blocks to the boundary of the computational domain).

COST suggest that the central building, at which wind effects are of main interest, requires the greatest level of detail but its area and resolutions to be represented have not been mentioned (Franke, 2006).

3.3. Treatment of obstacle smaller than grid size

To simulate the aerodynamic effects of small-scale obstacles such as small buildings, sign boards, trees and moving automobiles, etc., it is necessary to add additional terms to the basic flow equations in order to decrease wind velocity but increase turbulence. This is called a canopy model and is based on the k–ε model in which extra terms are added to the transport equations. These extra terms are derived by applying the spatial average to the basic equations (Hiraoka, 1993; Maruyama, 1993; Hataya et al., 2006). A volume fraction technique, e.g. FAVOR (Hirt, 1993), is a simplified technique for considering the effect of an obstacle smaller than the grid.

In particular, tree planting is one of the most popular measures for improving the pedestrian wind environment. Mochida et al. (2006) have classified various tree canopy models and have also compared the predictions of various canopy models with field measurements of flows around trees. It is suggested that users should compare their results with Mochida et al. (2006) when any kind of tree canopy model is used.

4. Grid discretization

4.1. General notice

In order to predict the flow field around a building with acceptable accuracy, the most important thing is to correctly reproduce the characteristics of separating flows near the roof and the walls. Therefore, a fine grid arrangement is required to resolve the flows near the corners. However, it is generally very difficult to resolve the viscous sub-layers near the building walls and it is also difficult to adopt no-slip boundary conditions on the walls. The use of wall functions to represent flow around buildings is basically incorrect, since many wall functions such as logarithmic laws have been developed considering the situations in the attached boundary-layer flows. However, many buildings are bluff bodies with sharp edges and separating points are always found at the leading edges, regardless of the *Re* numbers. In such cases, the decrease of accuracy due to the use of wall functions is not as significant as expected.

According to cross-comparison results for a simple-building model (Mochida et al., 2002; Shirasawa et al., 2003; Yoshie et al., 2005a), the minimum of 10 grids is required on one side of a building to reproduce the separation flow around the upwind corners.

Grid shapes should be set up so that the widths of adjacent grids are similar, especially in regions with a steep velocity gradient. In these regions, it is desirable to set a stretching ratio of adjacent grids of 1.3 or less. However, it is desirable to confirm that the results would not change with different grid layouts, since these recommended stretching ratios may change according to the shape of the building and its surroundings.

COST advises the same limitation for grid stretching ratio, and it recommends that the sensitivity of the results on mesh resolution should be tested (Franke et al., 2004).

4.2. Grid resolution for actual building complex

The minimum grid resolution should be set to about $\frac{1}{10}$ of the building scale (about 0.5–5.0 m) within the region including the evaluation points around the target building. Moreover, the grids should be arranged so that the evaluation height (1.5–5.0 m above ground) is located at the 3rd or higher grid from the ground surface (Yoshie et al., 2005a; Tominaga et al., 2005).

COST suggests that at least 10 cells should be used per building side and 10 cells per cube root of building volume as an initial choice. It also recommends that pedestrian wind speeds at 1.5–2 m height be calculated at the third or fourth cell above the ground (Franke et al., 2004). Those requirements are comparable with the AIJ guidelines.

4.3. Grid dependence of solution

It should be confirmed that the prediction result does not change significantly with different grid systems. The number of fine meshes should be at least 1.5 times the number of coarse meshes in each dimension (Ferziger and Perić, 2002).

COST indicates that at least three systematically and substantially refined grids should be used so that the ratio of cells for two consecutive grids should be at least 3.4 (Franke et al., 2004). The value of 3.4 means finer girds with 1.5 times the grid number in three dimensions, i.e., $1.5^3 = 3.375$.

4.4. Unstructured grid

It is necessary to ensure that the aspect ratios of the grid shapes do not become excessive in regions adjacent to coarse girds or near the surfaces of complicated geometries. For improved accuracy, it is desirable to arrange the boundary layer elements (prismatic cells)

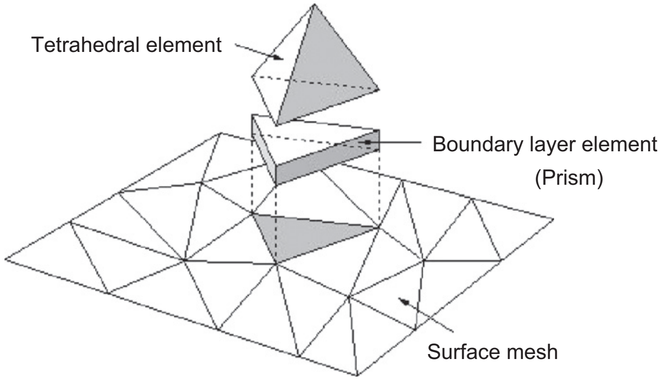

Fig. 2. Arrangement of grid elements near solid surface in unstructured grid.

parallel to the walls or the ground surfaces (Fig. 2). COST also introduces the same technique.

5. Boundary conditions

5.1. Inflow boundary condition

The vertical velocity profile $U(z)$ on flat terrain is usually given by a power law (Architectural Institute of Japan, 2004):

$$U(z) = U_S\left(\frac{z}{z_S}\right) \tag{5}$$

where U_s is the velocity at reference height, z_s, and α is the power-law exponent determined by terrain category.

The vertical distribution of turbulent energy $k(z)$ can be obtained from a wind tunnel experiment or an observation of corresponding surroundings. If it is not available, $k(z)$ can be also given by Eq. (6) based on the estimation equation for the vertical profile of turbulent intensity $I(z)$ proposed by AIJ Recommendations for Loads on Buildings (2004):

$$I(z) = \frac{\sigma_u(z)}{U(z)} = 0.1\left(\frac{z}{z_G}\right)^{(-\alpha-0.05)} \tag{6}$$

where z_G is the boundary layer height determined by terrain category And σ_u the RMS value of velocity fluctuation in stream-wise direction.

In the atmospheric boundary layer, the following relation between $I(z)$ and $k(z)$ can be assumed:

$$k(z) = \frac{\sigma_u^2(z) + \sigma_v^2(z) + \sigma_w^2(z)}{2} \cong \sigma_u^2(z) = (I(z)U(z))^2 \tag{7}$$

It is recommended that the values of ε be given by assuming local equilibrium of $P_k = \varepsilon$ (P_k: production term for k equation):

$$\varepsilon(z) \cong P_k(z) \cong -\overline{u'w'}\frac{\mathrm{d}U(z)}{\mathrm{d}z} \cong C_\mu^{1/2}k(z)\frac{\mathrm{d}U(z)}{\mathrm{d}z} \tag{8}$$

When the vertical gradient of velocity can be expressed by a power law with exponent α,

$$\varepsilon(z) = C_\mu^{1/2} k(z) \frac{U_s}{z_s} \alpha \left(\frac{z}{z_s}\right)^{(\alpha-1)} \tag{9}$$

where $C_\mu$is the model constant (= 0.09).

For the inflow boundary conditions, COST recommends the formulas suggested by Richards and Hoxey (1993), in which the vertical profiles for $U(z)$, $k(i)$ and $\varepsilon(z)$ in the atmospheric boundary layer by assuming a constant shear stress with height are as follows:

$$U(z) = \frac{U^*_{\mathrm{ABL}}}{\kappa} \ln\left(\frac{z+z_0}{z}\right) \tag{10}$$

$$k(z) = \frac{U^{*2}_{\mathrm{ABL}}}{\sqrt{C_\mu}} \tag{11}$$

$$\varepsilon(z) = \frac{U^{*3}_{\mathrm{ABL}}}{\kappa(z+z_0)} \tag{12}$$

where κ is the Karman constant (= 0.4) and $U^*_{\mathrm{ABL}}$the atmospheric boundary layer friction velocity.

U^*_{ABL} is calculated from a specified velocity U_h at reference height h as

$$U^*_{\mathrm{ABL}} = \frac{\kappa U_h}{\ln((h+z_0)/z)} \tag{13}$$

where U_h is the specified velocity at a reference height h.

The vertical profiles expressed in Eqs. (5)–(9) are given by assumimg the power-law exponent α and are consistent with the wind load estimation method in Japan (Architectural Institute of Japan, 2004). However, the recommended profiles in COST, as described by Eqs. (10)–(13), are based on an assumed value of the roughness parameter z_0. Eqs. (10)–(13) assume that the height of the computational domain is much lower than the atmospheric boundary layer height because the assumption of constant shear stresses is only valid in the lower part of the atmospheric boundary layer. Therefore, it is necessary to pay attention to the relationship between the height of the computational domain and the atmospheric boundary layer.

5.2. Lateral and upper surfaces of computational domain

If the computational domain is large enough (see Section 3.1), the boundary conditions for lateral and upper surfaces do not have significant influences on the calculated results around the target building (Mochida et al., 2002; Shirasawa et al., 2003; Yoshie et al., 2005a). Using the inviscid wall condition (normal velocity component and normal gradients of tangential velocity components set to zero) with a large computational domain will make the computation more stable.

5.3. Downstream boundary

It is common to set the normal gradients of all variables to zero for the outflow boundary condition. The outflow boundary needs to be placed far from the region where the influence of the target building is negligible (see Section 3.1).

5.4. Solid surface boundary conditions for velocities

5.4.1. Ground surface for single-building model for comparison with experimental result

When choosing the ground surface boundary conditions, the most important principle is that the computational trail of a simple boundary layer flow without a building should be first assessed. The vertical profile of wind velocity gradually changes near the turntable floor in a wind tunnel as the flow proceeds downstream. Boundary conditions that reproduce this gradual change in velocity profile should be used.

A logarithmic law for a smooth wall surface or a logarithmic law with roughness parameters z_0 or k_s (k_s: sand-grain roughness height) can be used for the boundary condition.

The logarithmic law for a smooth wall surface is expressed as follows:

$$\frac{U_P}{(\tau_w/\rho)^{1/2}} = \frac{1}{\kappa}\ln z_n^+ + A = \frac{1}{\kappa}\ln\frac{(\tau_w/\rho)^{1/2} z_P}{\nu} + A \tag{14}$$

where U_P is the tangential component of velocity vector at near-wall node, τ_w the shear stress at the wall, z_n^+ the wall unit, z_p the distance between the definition point of U_p and wall, and A the universal constant (= 5–5.5).

To obtain τ_w without iterative calculation, one can use its generalized form proposed by Launder and Spalding (1974). Murakami and Mochida (1988) have applied this generalized log-law boundary conditions to investigate flows around a building.

The logarithmic law with roughness parameter z_0 is expressed as follows:

$$\frac{U_p}{(\tau_w/\rho)^{1/2}} = \frac{1}{\kappa}\ln\left(\frac{z_p}{z_0}\right) \tag{15}$$

If the boundary layer formed near the ground can be regarded as the constant flux layer, the value of z_0 can be assumed from the logarithmic law using the relation $(\tau_w/\rho)^{1/2} = U^* = C_\mu{}^{1/4}k^{1/2}$ and the measured values of velocity and k near the ground surface.

$$z_0 = \frac{z_p}{\exp(\kappa U_p / C_\mu^{1/4} k_p^{1/2})} \tag{16}$$

where k_p is the k value at z_p.

In order to check whether the given boundary condition is appropriate, it should be confirmed that the velocity profile near the ground surface is similar to wind tunnel observations at a few measured locations. This can be done by 2D computation of boundary layer flow with the same grids in the vertical plane of the 3D grid system. It was thus confirmed that the condition with Eqs. (15) and (16) could minimize the changes in the vertical profiles obtained by Eqs. (5–9) for test case A (Mochida et al., 2002). COST also emphasizes verification of the assumption of an equilibrium boundary layer corresponding to the prescribed approach flow by performing a simulation in an empty domain with the same grid and boundary conditions as the final computation (Franke, 2006).

5.4.2. Ground surface for actual building complex

The boundary condition corresponding to the actual ground surface should be used. For example, for a smooth ground surface, the logarithmic law for a smooth wall (Eq. (14)) can be used.

For a rough ground surface, which can be expressed by a roughness length z_0, a logarithmic law including a roughness parameter (Eq. (15)) is applicable.

COST points out that the rough wall condition with k_S leads to a very bad resolution of the flow close to the wall, because the first calculation node of the wall should be placed at least one k_S away from the wall. Therefore, the use of the smooth wall condition for a built area is recommended (Franke et al., 2004). More detailed investigation was reported by Blocken et al. (2007).

5.4.3. Building wall

For the building walls, the boundary condition according to the above principle is used.

5.5. Solid surface boundary for turbulent energy k and dissipation rate ε

5.5.1. Turbulent energy k

The transport equation of k is solved with the condition that the normal gradient of k is zero.

5.5.2. Dissipation rate

The dissipation rate ε at the first grid point, ε_p, is given by

$$\varepsilon_p = \frac{C_\mu^{3/4} k_p^{3/2}}{\kappa z_p} \tag{17}$$

6. Solution algorithm, spatial discretization

6.1. Solution algorithm

Basically, steady and unsteady calculations using the RANS model should result in the same solutions if unsteady fluctuation does not occur in the calculation and if both are sufficiently convergent. However, in real situations, unsteady periodic fluctuation usually occurs behind high-rise buildings. This fluctuation essentially differs from that of turbulence, and cannot be reproduced by a steady calculation. This periodic fluctuation is not reproduced in many cases using a high *Re* number type k–ε model, although the unsteady calculation is conducted. It may be reproduced when highly accurate turbulence models and boundary conditions are used (Mochida et al., 2002; Tominaga et al., 2003). For this case, the time-averaged values of each variable need to be calculated, because the solution changes with time.

6.2. Scheme for convection terms

The first-order upwind scheme is not appropriate for all transported quantities, since the spatial gradients of the quantities tend to become diffusive due to a large numerical viscosity.

COST also does not recommend the use of first-order methods like the upwind scheme except in initial iterations (Franke et al., 2004).

7. Convergence of solution

7.1. Criteria for convergence

Calculation needs to be finished after sufficient convergence of the solution. For this purpose, it is important to confirm that the solution does not change by monitoring the variables on specified points or by overlapping the contours among calculation results at different calculation steps. The default values for convergence in most commercial codes are not strict because code vendors want to stress calculation efficiency. Therefore, stricter convergence criteria are required to check that there is no change in the solution.

When the calculation diverges or convergence is slow, the points below should be examined:

- The aspect ratio and the stretching ratio of the grids may be too large.
- The relaxation coefficient of the matrix solver may be too small.
- Periodic fluctuations such as a vortex shedding may be occurring.

COST suggests that scaled residuals should be dropped 4 orders of magnitude (Franke, 2006). However, these values are largely dependent on flow configuration and boundary conditions, so it is better to check the solution directly using different convergence criteria, as mentioned above.

7.2. Initial conditions

To obtain the converged solution quickly, an appropriate physical property of initial condition should be given. The inflow profiles extended to the whole domain or the results obtained by laminar flow computation are often used for the initial condition.

8. Turbulence models

The well-known problem of the standard k–ε model is that it cannot reproduce the separation and reverse flow at the roof top of a building due to its overestimation of turbulence energy k at the impinging region of the building wall. Although this problem does not appear near the ground surface as much as it does on the roof, it may affect the prediction accuracy of the value and the location of high velocity. However, many revised k–ε models and differential stress model (DSM) have mitigated this problem and enhanced the prediction accuracy for the strong wind region near the ground surface (Mochida et al., 2002; Shirasawa et al., 2003; Tominaga et al., 2004; Yoshie et al., 2005a).

Concerning the choice of turbulence models, COST concludes that the standard k–ε model should not be used in simulation for wind engineering problems, but recommends the improved two-equation models within the linear eddy viscosity assumption (Franke, 2006). This investigation of the turbulence model corresponds with the finding of the Working Group. Although COST also mentions that preferably non-linear models or Reynolds-stress models should be used (Franke, 2006), there are presently very few examples of the prediction accuracy of these models applied to pedestrian wind problems in order to evaluate their performance. Therefore, it is expected that further investigation will be carried out by COST in the near future.

9. Validation of user's CFD model

Users should conduct calculations for at least one case of a single high-rise building and at least one case of a building complex in an actual urban area using their CFD code, and compare the results with those carried out by the AIJ group. These experimental results are available on web page http://www.aij.or.jp/Jpn/publish/cfdguide/index_e.htm.

10. Conclusions

The guidelines for practical application of CFD to the pedestrian wind environment around buildings proposed by the working group of the AIJ have been delineated. They are based on the results of cross-comparison between CFD predictions, wind tunnel test results and field measurements for seven test cases, which have been conducted to investigate the influence of many kinds of computational conditions for various flow fields. They summarize important points in using CFD techniques to predict the pedestrian wind environment. The authors believe that the guidelines presented here give useful information for predicting and assessing the pedestrian wind environment around buildings using CFD. The results of cross-comparisons for the seven test cases conducted within this project will be utilized to validate the accuracy of CFD codes used in the practical applications of wind environment assessments.

Acknowledgments

The authors would like to express their gratitude to the members of the working group for CFD prediction of the pedestrian wind environment around buildings. The working group members are: A. Mochida (Chair, Tohoku University), Y. Tominaga (Secretary, Niigata Inst. of Tech.), Y. Ishida (I.I.S., Univ. of Tokyo), T. Ishihara (Univ. of Tokyo), K. Uehara (National Inst. of Environ. Studies), H. Kataoka (Obayashi Corporation), T. Kurabuchi (Tokyo Univ. of Sci.), N. Kobayashi (Tokyo Polytechnic Univ.), R. Ooka (I.I.S., Univ. of Tokyo), T. Shirasawa (Tokyo Polytechnic Univ.), N. Tsuchiya (Takenaka Corp.), Y. Nonomura (Fujita Corp.), T. Nozu (Shimizu Corp.), K. Harimoto (Taisei Corp.), K. Hibi (Shimizu Corp.), S. Murakami (Keio Univ.), R. Yoshie (Tokyo Polytechnic Univ.), M. Yoshikawa (Taisei Corp.)

We also give our sincere thanks to Prof. T. Stathopoulos (Concordia Univ.) and Dr. J. Franke (Univ. of Siegen) for their valuable comments and information on this work.

This manuscript was written with the assistance of Dr. Y. F. Lun (Tohoku Univ.) and Dr. C.-H. Hu (Tokyo Polytechnic Univ.), whose contributions are gratefully acknowledged.

References

AIAA, 1998. Guide for the Verification and validation of computational fluid dynamics simulations, AIAA G-077–1998.

Architectural Institute of Japan, 2004. Recommendations for loads on buildings. Architectural Institute of Japan (in Japanese).

Blocken, B., Stathopoulos, T., Carmeliet, J., 2007. CFD simulation of the atmospheric boundary layer: wall function problems. Atmos. Environ. 41, 238–252.

ERCOFTAC. Best practices guidelines for industrial computational fluid dynamics, Version 1.0, January 2000.

Ferziger, J.H., Perić, M., 2002. Computational Methods for Fluid Dynamics, third ed. Springer, Berlin, Heidelberg, New York.

Franke, J., 2006. Recommendations of the COST action C14 on the use of CFD in predicting pedestrian wind environment. In: The Fourth International Symposium on Computational Wind Engineering, Yokohama, Japan, July 2006.

Franke, J., Hirsch, C., Jensen, A.G., Krüs, H.W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A., Wright, N.G., 2004. Recommendations on the use of CFD in wind engineering. In: van Beeck, J.P.A.J. (Ed.), COST Action C14, Impact of Wind and Storm on City Life Built Environment. Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics, 5–7 May 2004. von Karman Institute, Sint-Genesius-Rode, Belgium.

Hataya, N., Mochida, A., Iwata, T., Tabata, Y., Yoshino, H., Tominaga, Y., 2006. Development of the simulation method for thermal environment and pollutant diffusion in street canyons with subgrid scale obstacles. In: The Fourth International Symposium on Computational Wind Engineering, July 2006, Yokohama, Japan.

Hiraoka, H., 1993. Modelling of turbulent flows within plant/urban canopies. J. Wind Eng. Ind. Aerodyn. 46&47, 173–182.

Hirt, C.W., 1993. Volume-fraction techniques: powerful tools for wind engineering. J. Wind Eng. Ind. Aerodyn. 46&47, 327–338.

Launder, B., Spalding, D., 1974. The numerical computation of turbulent flows. Comput. Methods Appl. Mech. Eng. 3, 269–289.

Maruyama, T., 1993. Optimization of roughness parameters for staggered arrayed cubic blocks using experimental data. J. Wind Eng. Ind. Aerodyn. 46&47, 165–171.

Mochida, A., Tominaga, Y., Murakami, S., Yoshie, R., Ishihara, T., Ooka, R., 2002. Comparison of various k–ε models and DSM applied to flow around a high-rise building—Report on AIJ cooperative project for CFD prediction of wind environment. Wind Struct. 5 (2–4), 227–244.

Mochida, A., Yoshino, H., Iwata, T., Tabata, Y., 2006. Optimization of tree canopy model for CFD prediction of wind environment at pedestrian level. In: The Fourth International Symposium on Computational Wind Engineering, Yokohama, Japan, July 2006.

Murakami, S., Mochida, A., 1988. 3-D numerical simulation of airflow around a cubic model by means of the k–ε model. J. Wind Eng. Ind. Aerodyn. 31, 283–303.

Richards, P.J., Hoxey, R.P., 1993. Appropriate boundary conditions for computational wind engineering models using the k–ε turbulence model. J. Wind Eng. Ind. Aerodyn. 46&47, 145–153.

Richards, P.J., Mallinson, G.D., McMillan, D., Li, Y.F., 2002. Pedestrian level wind speeds in downtown Auckland. Wind Struct. 5 (2–4), 151–164.

Roache, P.J., Ghia, K., White, F., 1986. Editorial policy statement on the control of numerical accuracy. ASME. J. Fluids Eng. 108 (1), 2.

Shirasawa, T., Tominaga, T., Yoshie, R., Mochida, A., Yoshino, H., Kataoka, H., Nozu, T., 2003. Development of CFD method for predicting wind environment around a high-rise building part 2: the cross comparison of CFD results using various k- models for the flowfield around a building model with 4:4:1 shape. AIJ J. Technol. Des. 18, 169–174 (in Japanese).

Stathopoulos, T., Baskaran, A., 1996. Computer simulation of wind environmental conditions around buildings. Eng. Struct. 18 (11), 876–885.

Timofeyef, N., 1998. Numerical study of wind mode of a territory development. In: Proceedings of the Second East European Conference on Wind Engineering, September 7–11, Prague, Czech Republic.

Tominaga, Y., Mochida, A., Murakami, S., 2003. Large eddy simulation of flowfield around a high-rise building. Conference Preprints of 11th International Conference on Wind Engineering, vol. 2, June 2–5, Lubbock, Texas, USA, pp. 2543–2550.

Tominaga, Y., Mochida, A., Shirasawa, T., Yoshie, R., Kataoka, H., Harimoto, K., Nozu, T., 2004. Cross comparisons of CFD results of wind environment at pedestrian level around a high-rise building and within a building complex. J. Asian Archit. Build. Eng. 3 (1), 63–70.

Tominaga, Y., Yoshie, R., Mochida, A., Kataoka, H., Harimoto, K., Nozu, T., 2005. Cross comparisons of CFD prediction for wind environment at pedestrian level around buildings, Part 2: Comparison of results for flowfield around building complex in actual urban area. The Sixth Asia-Pacific Conference on Wind Engineering, September 2005, Seoul, Korea.

Westbury, P.S., Miles, S.D., Stathopoulos, T., 2002. CFD application on the evaluation of pedestrian-level winds. In: Workshop on Impact of Wind and Storm on City Life and Built Environment, Cost Action C14, CSTB, June 3–4, Nantes, France.

Yoshie, R., Mochida, A., Tominaga, Y., Kataoka, H., Harimoto, K., Nozu, T., Shirasawa, T., 2005a. Cooperative project for CFD prediction of pedestrian wind environment in Architectural Institute of Japan. J. Wind Eng. Ind. Aerodyn. 95, 1551–1578.

Yoshie, R., Mochida, A., Tominaga, Y., Kataoka, H., Yoshikawa, M., 2005b. Cross comparisons of CFD prediction for wind environment at pedestrian level around buildings, Part 2: Comparison of results of flow-field around a high-rise building located in surrounding city blocks. In: The Sixth Asia-Pacific Conference on Wind Engineering, September 2005, Seoul, Korea.

Yoshie, R., Mochida, A., Tominaga, Y., 2006. CFD prediction of wind environment around a high-rise building located in an urban area. In: The Fourth International Symposium on Computational Wind Engineering, July. 2006, Yokohama, Japan.