AIJ Guideline for Practical Applications of CFD to Wind Environment around Buildings

Akashi Mochida[a], Yoshihide Tominaga[b] and Ryuichiro Yoshie[c]

[a]*Tohoku University, Aoba 06, Aramaki, Aoba-ku, Sendai-City, Miyagi, Japan*
[b]*Niigata Institute of Technology, Fujihashi 1719, Kashiwazaki-City, Niigata, Japan*
[c]*Tokyo Polytechnic University, Iiyama 1583, Atsugi-City, Kanagawa, Japan*

ABSTRACT: The guideline for CFD prediction of wind environment around buildings was proposed by Working Group of the Architectural Institute of Japan (AIJ). It is described based on the results of benchmark tests which have been conducted for investigating the influence of many kinds of computational conditions for various flowfields. This paper delineates the guideline proposed by our group.

KEYWORDS: CFD, Wind Environment, Prediction, Guideline, Benchmark test

1 INTRODUCTION

As computer facilities and Computational Fluid Dynamics (CFD) software have been improved in recent years, the prediction and the assessment of wind environment around buildings using CFD have become practical at design stages. Therefore, a guideline which summarizes important points of using CFD technique for appropriate prediction of wind environment is needed. Although there have been the recommendations with similar objectives proposed by COST group [1], those are mainly based on the results published elsewhere. On the other hand, the guideline proposed by AIJ is based on the results of benchmark tests which have been conducted to investigate the influence of many kinds of computational conditions for various flowfields by our own [2-6].

This paper briefly summarizes the guideline and the full version of the guideline will be available in the final paper.

This guideline is mainly based on using high Reynolds (Re) number RANS (Reynolds Averaged Navier-Stokes equations) models. In order to obtain more accurate result, it is desirable to use a Large Eddy Simulation (LES) and a low Re number type model. 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 wind environment under severe time restrictions. If it is necessary to use a highly accurate model like LES or low Re number type model, it is recommended to use it after the calculations and analysis which have been done by high Re number type RANS models following this guideline.

2 COMPUTATIONAL DOMAIN AND REPRESENTATION OF SURROUNDINGS

2.1 *Domain size*

For the size of computational domain, the blockage ratio should be below 3%. In case of the single building model, the inlet, the lateral and the top boundary should be set 5H or more away from the buildings, where H is the building height. The outflow boundary should be set at least 15H or more behind the buildings. In the case including surrounding buildings, the size of computational domain should be covered the area within at least 5H from the outer edges of the target building in horizontal distance [6].

2. 2 *Representation of surroundings*

In the case of the actual urban area, the region where the evaluating points around the target

building exist (generally within a 1H radius) should be represented in the maximum details [5].

3 GRID DISCRETIZATION

3.1 *General notice*

In order to predict the flowfield around buildings with acceptable accuracy, the most important thing is to correctly reproduce the characteristics of separating flows near the roof and the wall of the buildings. Therefore, fine grid arrangement is required to resolve the flows near the corners of the buildings. Many buildings are bluff bodies which have sharp edges, and the separating points always found at the upwind corners, independently of Re numbers. In such cases, there are few problems to use a wall function with certain grid resolution near the walls.

According to the results of the benchmark tests for simple building model [2-4], the minimum number of grids required to divide one side of the building is about ten in order to reproduce the separation flow around the upwind corners.

The shapes of grids should be set up so that the widths of adjacent grids are not much different especially at the region with steep velocity gradient. In these regions, it is desirable to set a stretching ratio of adjacent grids by 1.3 or less.

However, it is desirable to confirm that the results would not change by the different grid systems, since these standard values may change according to the shape of the building and its surroundings.

3.2 *Grid resolution for Actual building complex*

The resolution of the grids should be set at the lowest to about 1/10 of the building scale (about 0.5-5.0m) within the region where the evaluating points around the target building are included. Moreover, the grids should be arranged so that the evaluating height (1.5-5.0m above ground) is located at the 3rd or upward grids from the ground surface [5].

3.3 *grid dependence of the solution*

It should be confirmed that the prediction result does not change much with several different grid systems. The ratio of the grids for consecutive grids should be at least 3.4[1].

3.4 *Unstructured Grid*

It needs to pay attention that the aspect ratios of the grid shapes do not become excessive in the regions adjacent to coarse girds or near the surfaces of complicated geometries. It is good to arrange a layer-shape grids parallel to the walls or ground surfaces for improved accuracy.

4 BOUDNARY CONDITIONS

4.1 *Inflow boundary condition*

In many cases, the vertical velocity profile is given by a power law. The vertical distribution of k is obtained from the wind tunnel experiment or the observation. It is recommended that the values of ε are given by assuming local equilibrium of $P_k=\varepsilon$ (P_k : Production term for k equation).

4.2 *Lateral and upper surfaces of the computational domain*

If the computational domain was large enough (see 2.1), the boundary conditions for lateral and upper surfaces would not have significant influences on the calculated results around the target building [6]. Using the slip wall condition (the normal velocity component and the normal gradients of the tangential velocity components are set to zero) with large computational domain will make the computation more stable.

4.3 *Downstream boundary*

Although it is common to use the condition that the normal gradients of all variables set to zero, the outflow boundary needs to be set far away from the region where is influenced by the target building (see 2.1).

4.4 *Ground surface boundary*

(1)Single building model for comparing with the experimental result
When choosing the ground surface boundary conditions, the most important principle is that the vertical profiles of velocity and turbulent energy at inflow boundary are maintained until the outflow boundary for the computation of a simple boundary layer flow without building.

For this purpose, using a logarithmic law including roughness parameter z_0 is recommended [2] for the boundary condition. The value of z_0 can be assumed by the logarithmic law using the relation $u_* = C_\mu^{1/4} k^{1/2}$ and the measured values of velocity and k near the ground surface.

In order to check whether the given boundary condition is appropriate, it should be confirmed that the velocity profile near the ground surface is maintained by 2D computation of boundary layer flow with the same grids in vertical plane of the grid system to be used.

(2)Actual Building Complex
The boundary condition corresponding to the actual condition of the ground surface should be used. For example, if it is a smooth surface, the generalized logarithmic law can be used. For the wall boundary of the buildings, the boundary condition according to this principle is used.

5 SOLUTION ALGORITHM, SPATIAL DISCRETIZATION

5.1 *Solution algorithm*

Basically, steady and unsteady calculations should result in the same solutions if both of them secured sufficient convergences. However, an unsteady periodic fluctuation often occurs behind the high-rise building. This fluctuation essentially differs from that of turbulence, and could not be reproduced by a steady calculation. This periodic fluctuation is not reproduced in many cases using high Re number type k-ε model, although the unsteady calculation were conducted. But it may be reproduced when highly accurate turbulence models and boundary conditions are used. For this situation, the time averaged values of each variable need to be calculated because the solution depends on the computational time.

5.2 *Scheme for convective term*

A first order upwind scheme should not be used for all transported quantities, since the spatial gradients of the quantities tend to become blunt due to a large numerical viscosity.

6 CONVERGENCE OF SOLUTION
Calculation needs to be finished after checking the 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. In the case that the convergence is judged by the spatial average of residuals by default setup, the computation may be finished while a convergence of the solution is not enough near the building. It is because that the residual at upper region of computational domain is much smaller than that at region near the buildings. Therefore, it should be checked that there is no change in the solution by more strict convergence criteria.

7 TURBULENCE MODELS

The well-known problem of the standard k-ε model is that it cannot reproduce the separation and the reverse flow at the roof top of the 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 notably as it does on the roof, it may affect the prediction accuracy of the peak value of velocity distribution. On the other hand, many revised k-ε models and DSM (Differential Stress Model) improve this drawback of the standard k-ε model and they improve the prediction accuracy of the strong wind region near ground surface. However, they tend to underestimate the turbulent energy k around the building. Therefore the attention should be paid to the fact that this underestimation of k cause over-prediction of the recirculation region behind the building compared with the standard k-ε model [2-4].

8 VALIDATION OF CFD MODEL

It will confirm that the results obtained by the present calculations are within the range of the previous experimental and computational results through solving at least 1 case of benchmark test carried out by AIJ group [7]. When using a new CFD code, the analysis sets all the calculation conditions as the same as an old one should be firstly conducted and, both of the new calculation and old one should bring up the same results.

REFERENCES

[1]J. Franke, C. Hirsch, A.G. Jensen, H.W. Krüs, M. Schatzmann, P.S. Westbury, S.D. Miles, J.A. Wisse, N.G. Wright : Recommendations on the Use of CFD in Predicting Pedestrian Wind Environment, COST Action C14 "Impact of Wind and Storms on City Life and Built Environment", 2004

[2]A. Mochida, Y.Tominaga, S.Murakami, R.Yoshie, T.Ishida and R.Ooka : Comparison of various k-εmodel and DSM applied to flow around a high-rise building - report on AIJ cooperative project for CFD prediction of wind environment -, Wind & Structures -An International Journal-, Vol.5, No.2-4, 227-244, 2002

[3]Y. Tominaga, A. Mochida, T. Shirasawa, R. Yoshie, H. Kataoka, K. Harimoto and T. Nozu : Cross Comparisons of CFD Results of Wind Environment at Pedestrian Level around a high-rise Building and within a Building Complex, Journal of Asian Architecture and Building Engineering, 63-70, May 2004

[4]R. Yoshie, A. Mochida, Y. Tominaga, H. Kataoka, K. Harimoto and T. Nozu, T. Shirasawa : Cooperative project for CFD Prediction of Pedestrian Wind Environment in Architectural Institute of Japan, The Fourth European & African Conference on Wind Engineering, Prague, Jul., 2005.

[5]Y. Tominaga, R. Yoshie, A. Mochida, H. Kataoka, K. Harimoto and T. Nozu : 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 6th Asia-Pacific Conference on Wind Engineering, Seoul, Sep., 2005.

[6]R. Yoshie, A. Mochida, Y. Tominaga, H. Kataoka, Masaru Yoshikawa : 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, The 6th Asia-Pacific Conference on Wind Engineering, Seoul, Sep., 2005.

[7]http://www.niit.ac.jp/abehtml/tomilab/benchmark/