Estimation of wind pressure coefficients for high-rise building by CFD analysis

*Kim, Min Kyu[1)] and Thomas Kang[2)]

[1), 2)] *Department of Architecture & Architectural Engineering, Seoul National University, Seoul, Korea*

[2)] *tkang@snu.ac.kr*

ABSTRACT

Recently, Architectural Institute of Japan (AIJ 2015) allowed using CFD (Computational Fluid Dynamics) analysis for the calculation of wind loads. This means that computer based analysis has reached to similar level with the wind tunnel test due to the improvement of the computing power and CFD analysis theory. This study conducted the analysis based on the AIJ guideline and the recommendation by the European Cooperation in Science and Technology (COST). Wind tunnel test data provided by Tokyo Polytechnic University were used to verify the analysis results. Wind loads with aspect ratios of 3, 4 and 5 were compared to assess the similarity between the analysis results and the wind tunnel test data.

1. INTRODUCTION

Due to the improvement of computing power of computers and the development of various commercial programs, research through the interpretation of CFD is actively being conducted in various fields. The use of CFD is also increasing in the field of building structures, and the reliability and robustness of CFD analysis is gradually improved to the point where it is stated that both wind tunnel experiments and CFD analysis results can be used in calculating wind loads in Japan (AIJ 2015). Because CFD analysis is not subject to any restrictions on the modeling of structures, it has significant advantages in the initial design phase where frequent design changes occur. However, due to the nature of the analysis program, there is a room for misinterpretation such that inaccurate results are considered to be correct. Therefore, cross-validation with the wind tunnel test results is needed until a reasonable guideline is established.

In this study, CFD analysis was performed under the same conditions as wind tunnel experiments only if the aspect ratio was 3, 4, or 5 and the angle of the wind direction was 0° among the wind tunnel test data provided by Tokyo Polytechnic

---

[1)] Graduate Student
[2)] Professor

University. A comparative analysis of the results was conducted.

2. CFD SIMULATION MODELING

When conducting CFD analysis in wind engineering, the virtual wind tunnel is designed and wind speed and turbulence data are entered in the form of a profile at the beginning of the analysis area. The wind tunnel experiment reproduces the characteristics of the wind within the atmosphere boundary layer in the course of the wind blowing in the wind tunnel using the spire and the light block, but in the case of CFD, the intended wind velocity and turbulence intensity are inserted at the beginning of the virtual wind tunnel through numerical data of turbulent kinetic energy and turbulent dissipation ratio.

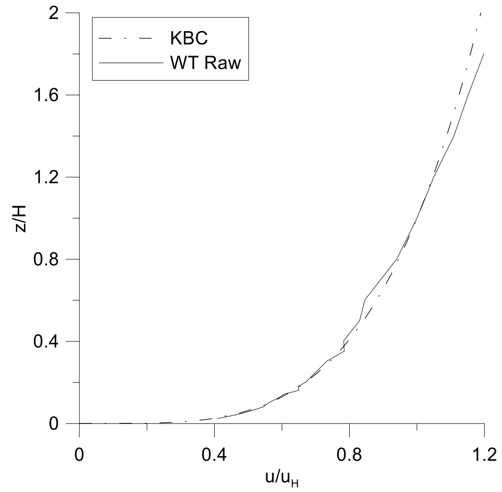


Fig. 1 Velocity profile

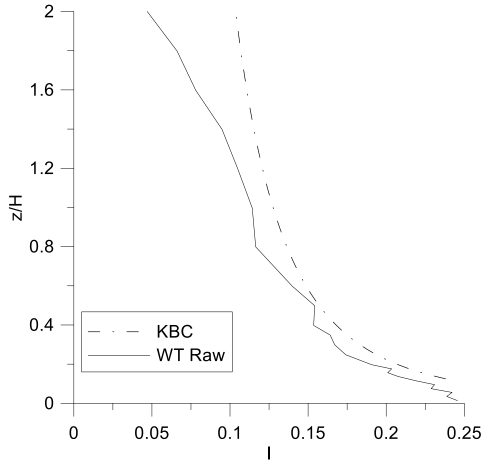


Fig. 2 Turbulence intensity profile

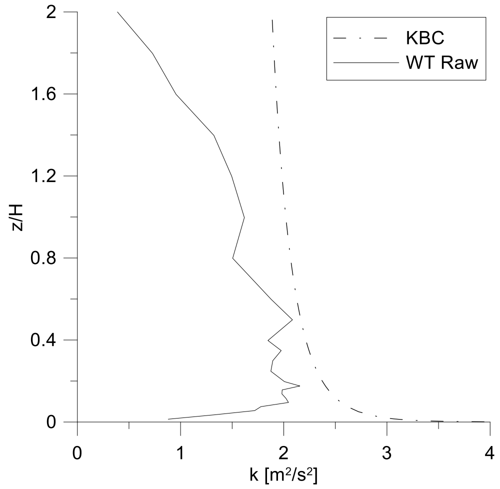


Fig. 3 Turbulence kinetic energy profile

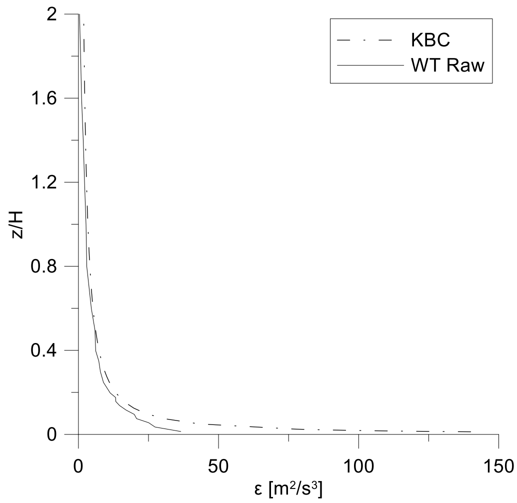


Fig. 4 Turbulence dissipation rate profile

Tominaga et al. (2008) suggested using theoretical equations as an inflow boundary condition. Because these equations are similar to KBC 2019, this study used following equations to simulate the wind tunnel experiment. As a result, the inflow condition of wind tunnel experiment and simulation such as turbulence intensity could be adjusted similarly.

In Figs. 1 through 4, legend WT Raw means wind tunnel test data, and KBC means data calculated by the proposed formula of KBC 2019. In this study, proposed formula from KBC 2019 was used to calculate wind speed and turbulence intensity because KBC 2019 and AIJ 2015 use essentially same formula.

In CFD analysis, it is important to properly size the analytical zone and adjust the mesh size around the building in order to obtain the analysis results as intended. In this study, the specifications of wind tunnel used in experiments were used to design the analysis domain. In addition, downstream area was set at least 15 times the building height as suggested by COST and AIJ. In this study, COST indicates the recommendation was from the European Cooperation in Science and Technology (COST), which was initially suggested by Franke (2006).

The mesh design was done with a software called ICEM CFD, and in this case also, following the recommended methods in COST and AIJ, there are at least 10 meshes on one side of the building for clear observation of vortex shedding. The blockage ratio also met the recommendation that the analytical area should be set to 3~5% or less, with a total of 1.2% or less for the three aspect ratios. As a result, about 5 million cells were used at this simulation. To check the consistency of the analysis results according to the size of the mesh, the mesh sizing was reduced or increased by 1.5 times in x, y, or z direction. In either case, the mesh size was determined by verifying that it did not cause much difference in the analysis results.

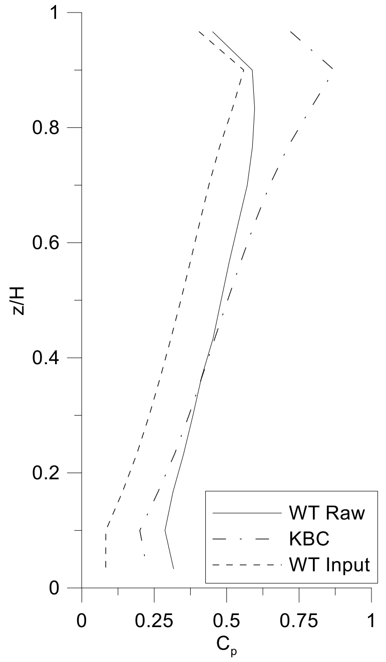
In this study, the SIMPLE algorithm was used for pressure-velocity coupling, and the 3D steady RANS (Reynolds-Averaged Navier-Stokes) equation was solved. Second order pressure interpolation and second order discretization were used for both the convection term and the viscous term. The convergence of the analytical results was determined to obtain when the residual error of the continuous equation decreased below 1.0×10^{-4}.

For wind tunnel experiments, wind pressure data provided by Tokyo Polytechnic University were used. The wind tunnel experiment setting included the model scale of 1/400, wind speed scale of 1/5 and time scale of 1/80. Wind speed altitude distribution index used in the experiment was 0.25, which is similar to 0.27 for surface roughness category IV of KBC 2019.

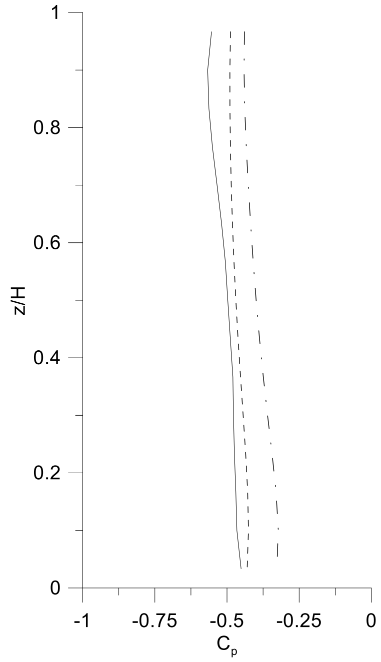
3. DISCUSSION AND CONCLUSION

In Figs. 5 through 7, legend WT Raw means wind tunnel test data, KBC means CFD analysis data using proposed formula of KBC 2019, and WT Input means CFD analysis data using a profile of wind tunnel experiment. Referring to the results, wind tunnel test data and CFD analysis data showed similar tendency. However, windward pressure coefficient was generally conservatively calculated, while leeward pressure coefficient was not conservatively calculated. Considering both pressure coefficients comprehensively, the results of CFD analysis are judged to be more conservative than those of wind tunnel experiments.

In this study, it was confirmed that the CFD analysis could sufficiently derive the wind pressure coefficient, if some factors for correcting errors with wind tunnel experiments and other peak factors are conservatively considered.

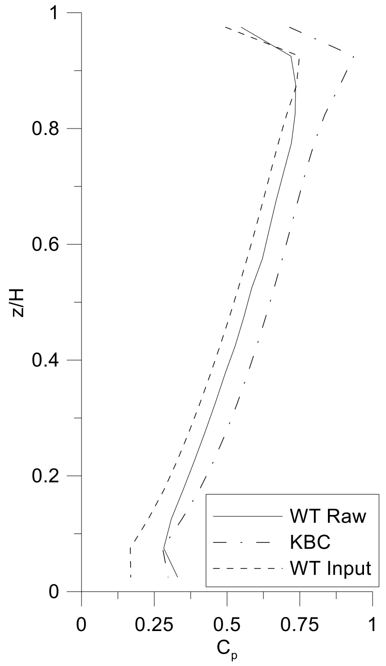


(a) Windward

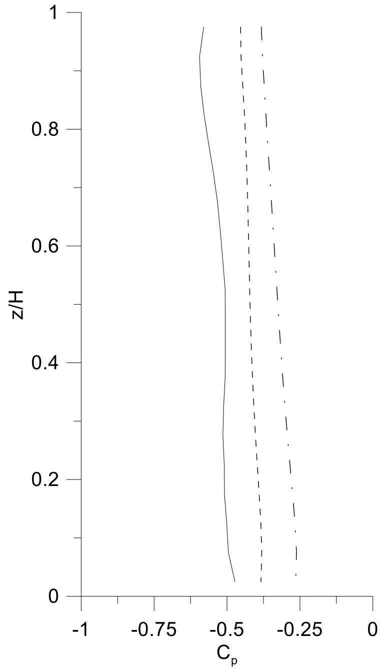


(b) Leeward

Fig. 5 Pressure coefficient for aspect ratio of 3



(a) Windward



(b) Leeward

Fig. 6 Pressure coefficient for aspect ratio of 4

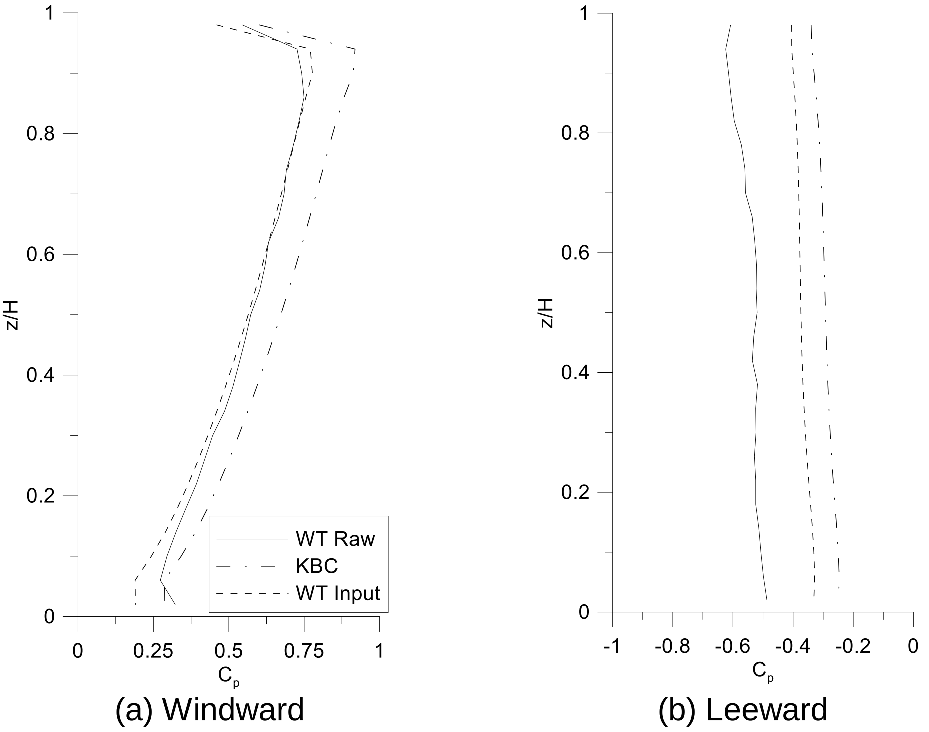


Fig. 7 Pressure coefficient for aspect ratio of 5

REFERENCES

AIJ, “Recommendations for loads on buildings”, Architectural Institute of Japan, Tokyo, Japan, 2015.

Tominaga, Y., Mochida, A., Shirasawa, T., Yoshie, R., Kataoka, H., Harimoto, K., and Nozu, T., "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." Journal of wind engineering and industrial aerodynamics 96.10-11 (2008): 1749-1761.

Ministry of Land, Infrastructure and Transport (Korea), “Korea Building Code (KBC 2019)”, 2019. (in Korean)

Franke, J., "Recommendations of the COST action C14 on the use of CFD in predicting pedestrian wind environment." The fourth international symposium on computational wind engineering, Yokohama, Japan. 2006.