The 2021 World Congress on
Advances in Structural Engineering and Mechanics (ASEM21)
GECE, Seoul, Korea, August 23-26, 2021

Proper orthogonal decomposition analysis of wind-induced pressure coefficients with computational fluid dynamics

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

[1), 2)] *Department of Architecture & Architectural Engineering, Seoul National University, Seoul, Korea*
[2)] *tkang@snu.ac.kr*

ABSTRACT

Research through CFD (Computational Fluid Dynamics) is being actively carried out in various fields including building engineering. The reliability of computer analysis such as finite volume method is gradually improving to the extent that it is indicated that both wind tunnel experiments and CFD analysis results can be used when calculating wind loads of structures in various code provisions (Tominaga et al. 2008). The fact that CFD analysis is not limited to the shape of a structure brings great strength to the initial design phase where frequent design changes can occur. However, there are still some chances for misunderstanding of analysis results. Therefore, CFD analysis should be done with sufficient cross-comparison with statistical analysis and wind tunnel test results. In this study, CFD analysis was performed under the same conditions with the wind tunnel test data provided by Tokyo Polytechnic University. Then, the comparison was conducted with statistical analysis including random vibration theory and proper orthogonal decomposition theory (Tamura et al. 1999).

1. INTRODUCTION

Due to the improvement of computing power and the development of various commercial analysis programs, research through the interpretation of CFD is actively being conducted in various fields. The use of CFD analysis is also increasing in the field of building structures. The reliability or stability of CFD analysis is being 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 2019). 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. Furthermore, aerodynamic properties of atypical structures can be studied with the use of CFD analysis. However, due to the nature of the computational analysis program,

[1)] Graduate Student
[2)] Professor

there is a possibility to produce incorrect results when using computer program. Therefore, cross-validation with wind tunnel test results is needed until a reasonable guideline is established.

In this study, CFD analysis is performed under the same condition as that applied for a wind tunnel experiment. The wind tunnel test data provided by Tokyo Polytechnic University is used. Comparative analysis is conducted using statistical method.

2. EXPERIMENT AND CFD SETUP

2.1 Experiment setup

Fig. 1 shows the sketch of the wind tunnel testing facility. In the testing, model scale of 1/400, wind speed scale of 1/5, and time scale of 1/80 were used to satisfy the law of similarity. To reproduce the atmospheric boundary layer, wind profile using a power law was used in the experiment. Turbulence intensity profile was also introduced to properly express the turbulence component of atmospheric boundary wind. Both wind profile and turbulence intensity profile were created by using the formula given in the AIJ recommendations for loads on buildings.

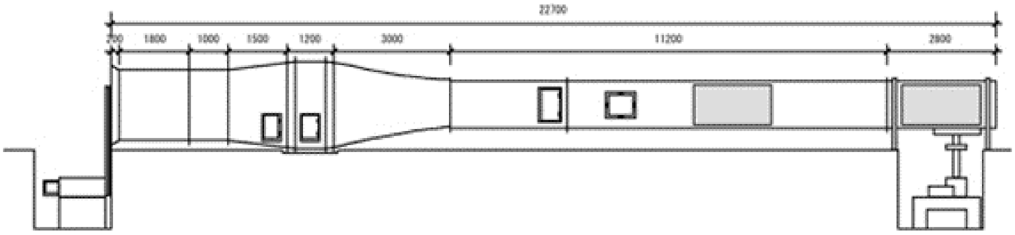

Fig. 1 Sketch of the wind tunnel
(http://wind.arch.t-kougei.ac.jp/system/eng/contents/code/facility_env01)

2.2 CFD setup

To simulate wind tunnel experiment in CFD analysis, a virtual version (domain) of wind tunnel was designed as shown in Fig. 2. The size of domain and blockage ratio were decided by the following recommendations from a previous study (Tominaga et al. 2008). Twenty cells were arranged on the surface of the building to capture the better quality of vortices near the building surfaces and edges. As a result, a total of 5,200,000 cells were designed in the whole domain, and realizable k-epsilon model was utilized for turbulence modeling (Shih et al. 1994). To conduct time history analysis (large eddy simulation) in the CFD analysis, steady-state analysis (Reynolds averaged Navier-Stokes) was executed first, and the results from RANS were used in the large eddy simulation.

To compare the two results (wind tunnel testing and CFD analysis), statistical approach using pressure coefficients was mainly used. It was crucial to get pressure values at the same location as the wind tunnel experiment. In the experiment, a total of 500 pressure taps were attached on all four sides of the building surface. Thus, the same 500 points were assigned at the building surface in the CFD analysis.

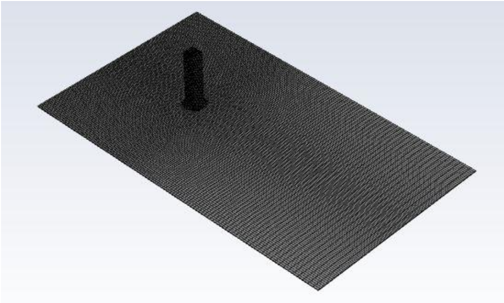

Fig. 2 View of CFD analysis domain

3. CONCLUSIONS

As mentioned in the experiment and CFD setup, this study mainly compares pressure coefficients from wind tunnel experiment and CFD analysis. In Fig. 3, pressure coefficients at the windward surface and leeward surface are presented. Legends in the figure indicate pressure coefficients from the wind tunnel testing, time history analysis, and steady-state analysis. The z and H indicate a height and roof height of the building, respectively.

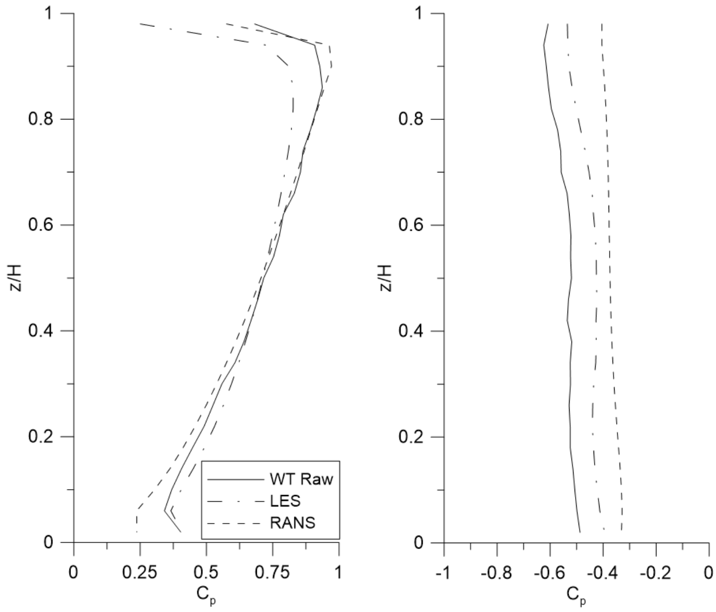

Fig. 3 Comparison of pressure coefficients
(Left: windward surface, right: leeward surface)

Similar trends are shown at both the windward surface and leeward surface. However, underestimation of pressure values by CFD analysis was found at the leeward surface. This is because CFD analysis does not adequately simulate separation of vortex flow at the edge and roof of the building, compared to wind tunnel experiment.

To further analyze the differences between CFD analysis and wind tunnel experiment, a statistical analysis called POD method (proper orthogonal decomposition) was utilized. By decomposing pressure coefficients at four surfaces into several modes, energy contents from multi-dimensional data set can be identified by a few principal modes. In this study, the first three principal modes from CFD analysis and wind tunnel experiment are compared. As shown in Fig. 4, mode shapes of CFD analysis and wind tunnel experiment show generally similar tendencies. However, the CFD analysis produced a slightly more exaggerated or distorted mode shapes than wind tunnel experiment. Further research is required to reduce the distorted consequence by introducing other methodologies or calibrating the results obtained by CFD analysis.

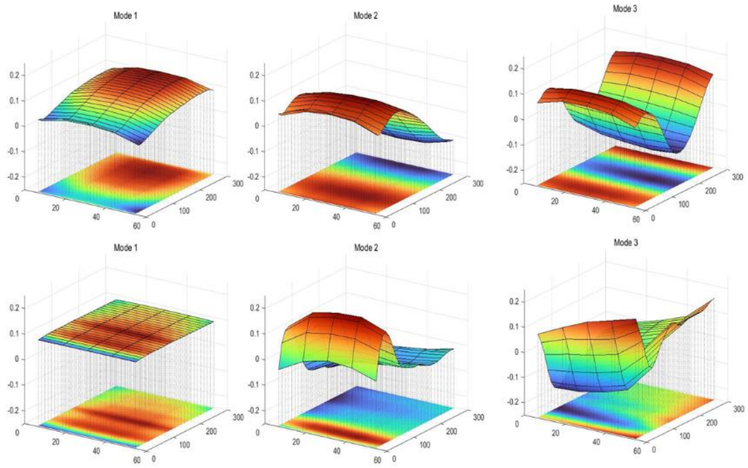

Fig. 4 POD mode shapes on windward surface
(Up: wind tunnel testing, down: CFD analysis)

ACKNOWLEDGEMENT

This research was funded by the National Research Foundation of Korea (NRF) grants funded by the Korean government (MSIT) (No. NRF-2021R1A5A1032433).

REFERENCES

AIJ (2019), "AIJ recommendations for loads on buildings (AIJ-RLB 2015) (English version)," *Architectural Institute of Japan (AIJ)*, Tokyo, Japan.

Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J. (1994), "A new k-epsilon eddy viscosity model for high Reynolds number turbulent flows: Model development and validation." *NASA Sti/recon Technical Report*, **95**, 11442.

Tamura, Y., Suganuma, S., Kikuchi, H. and Hibi, K. (1999), "Proper orthogonal decomposition of random wind pressure field." *Journal of Fluids and Structures*, **13**, 1069-1095.

Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., and Shirasawa, T. (2008), "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." *Journal of Wind Engineering and Industrial Aerodynamics*, **96**, 1749-1761.