Validation of CFD simulations on the wind loads for tall buildings’ preliminary design

C. L. Fu\textsuperscript{a} S. M. Lee\textsuperscript{b} C. M. Cheng\textsuperscript{c}

\textsuperscript{a} Wind Engineering Research Center, Tamkang University, Taiwan
\textsuperscript{b} Department of Aerospace Engineering, Tamkang University, Taiwan
\textsuperscript{c} Department of Civil Engineering, Tamkang University, Taiwan

ABSTRACT: This paper investigates the appropriateness of the CFD simulation on the design wind loads of a tall building in the preliminary design stage. An isolated 85-story building was selected for this purpose. CFD study was carried out to evaluate structural design wind loads. During the numerical simulation, the V2F turbulence model and commercial CFD software, STAR-CD, was used for the numerical simulation. In order to validate the CFD results, wind tunnel test was performed. This investigation indicates that the CFD technique can be used to predict the mean wind load of an isolated tall building with reasonable accuracy, collaborated with gust response factor would then be suitable for the purpose of preliminary building design.

KEYWORDS: CFD, V2F model, wind tunnel validation, tall building, design wind load

1 INTRODUCTION

The rapid developments in both computer hardware and software have created a possible environment for the practical applications of CFD to simulate flows within and around buildings and other structures in recent years. Murakami [1] performed numerical studied on the wind force of bluff body by k-\(\varepsilon\) and LES model, and showed that the k-\(\varepsilon\) model would have significant discrepancies while the LES model agreed quite well with experiment data. Delaunay [2] and Mikkelson [3] used modified k-\(\varepsilon\) model to predict wind pressures of various shaped models and comparing with the results of wind-tunnel test. For the cube, there are some differences appear near the separation zone and under-estimation at the rear surface. Tamura[4] examined the appropriate CFD techniques for prediction of wind loading, and suggested that modified k-\(\varepsilon\) model could be used on simple problems and unsteady problems should be modeled by LES. Gomes [5] presented the comparison of experiment and numerical results of wind pressure on some irregular-plan shape. Due to the complex of flow field in the vicinity of a building, the past investigations of CFD applications are mainly focused on rather simple geometry or low-rise buildings. In the present work, the wind pressures and design wind load of a genuine tall building located at Kaohsiung were evaluated by both CFD and wind tunnel simulations. The purpose of the investigation is to validate CFD technique as a viable procedure to provide the wind loads for tall buildings’ preliminary wind resistant design.

2 PERFORMANCE OF THE SELECTED CFD MODEL

For the numerical simulation, standard k-\(\varepsilon\) turbulence model is one of the most commonly used turbulence model in a simple flow field. However, in a complex environment such as the flow field surrounding a building which has separating region and re-circulation area, the standard k-\(\varepsilon\) modal would show some deficiency. For the complex flow, the renormalization group (RNG) k-\(\varepsilon\) shows a better agreement with the experimental data[6]. The V2F turbulence model is based on k-\(\varepsilon\) two equation model with two additional equations, i.e., the velocity scale v2 and relax...
redistributed term f. Generally speaking, turbulence models need some modifications in order to simulate the flow near solid boundary in a high Reynolds number flow. However, the V2F model is derived to avoid the use of wall function and damping function. Therefore, the V2F turbulence model becomes attractive in recently years. LES is another widely used turbulence model and performs well for the transient state of complex flow field [7]. However, comparing to the RANS models, the LES approach is considerably much too expensive in term of computational cost. Therefore, it is not a viable practical design tool yet. An example will be presented to illustrate the computing time of LES approach.

Before the CFD simulation fully applied on the target building, a preliminary study was conducted to show the performance of the selected V2F turbulence model with respect to the RNG turbulence model. The experimental results on a cube were chosen as the test problem [7]. Shown in Figure 1 are the predicted velocity profiles along the centerline of a 3D cube. It clearly shows that the results from V2F model are in better agreement with the experiment data from Rodi [7], both in re-circular (x=2.5) and redevelopment(x=4) regions, than the RNG calculations. Secondly, a comparison is made on the predicted reattachment length and the result from V2F model also shows the better resemblance of reattachment region to the experimental result.

![Figure 1. comparison of mean velocity profile of cube centerline(x=0.5,2.5,4)](image1)

3 WIND TUNNEL AND NUMERICAL SIMULATIONS
The wind tunnel tests were conducted in a boundary layer wind tunnel. A 1/350 scale pressure model of 85-story high-rise building was built as shown in Figure 2. More than 700 pressure taps were installed on the model. The pressure taps are basically uniformly distributed for the derivation of structural wind loads. Maximum of 384 channels of pressure signals could be sampled.
simultaneously through the ZOC pressure scanner system. The structural wind loads are obtained by integrating the simultaneously measured surface pressure in either x or y axes.

During the CFD simulation, both the flow field and the tall building are simply the digital version of the wind tunnel scale-down model test. The numerical computation domain was 3m×3m×2m, whereas the target building was placed 1.5m from the inlet. The simulation used a variable-space grid as shown in Figure 3. All simulations presented here were carried out by the commercial software package STAR-CD and a Pentium 4 Dual-core 3.2GHz desktop computer.

Shown in Figure 4(a) and 4(b) are the comparisons of mean pressure coefficient distribution on the windward and leeward walls, respectively. On the windward side, the wind pressures calculated by the V2F model agree quite well with the wind tunnel measurements. The pressure coefficients on the upper half of the building are 0.6–0.8, and the surface pressure rapidly decays near the edges and the corners. On the lower half of the building, the pressure coefficients are 0.2–0.5. On the leeward side, the CFD tends to under-estimate the wind pressure by roughly 30%. Based on the wind tunnel test, the pressure coefficients are -0.6 ~ -0.7 near the top of the building height; -0.5 ~ -0.6 in the most of the building leeward side. However, the pressure coefficient obtained by numerical simulation are -0.3 ~ -0.4 for most of the leeward side, and slight less (more negative) near the edges of the center cavity.

![Figure 4. Pressure coefficient distribution on the external walls](image)

**Figure 4. Pressure coefficient distribution on the external walls**

### 4 COMPARISON OF DESIGN WIND LOAD

The building design wind load was estimated by multiplying the Gust Response Factor to the mean wind load produced by CFD simulation. The Displacement based GRF was used to be consistent with the current building wind code. The design wind load based on wind tunnel test was derived based on direct pressure measurements and the complete random vibration theory. Based on Taiwan building wind code, the Gust Response Factor of the target building was found to be 2.41. Shown in Figure 5 are the comparisons of mean and maximum design wind loads among CFD prediction, building wind code and estimation based on wind tunnel data.

For the mean wind load, the CFD procedure tends to under-estimate the actual wind load by 13%, whereas the building wind code gives fairly reasonable prediction. As for the building design wind load, the building wind code tends to over-estimate and the CFD gives a better estimation. It should be pointed out that the overall better performance of CFD over wind code, at least in this particular case, is the combined result of lower mean wind load and a conservative Gust Response Factor. However, through the aforementioned numerical example, the CFD approach
with V2F turbulence model has indeed shown a promising potential to be used as a tool for providing design wind load for tall buildings preliminary wind resistant design.

![Figure 5. Comparisons of building design wind load.](image)

5 CONCLUDING REMARKS
In this paper, both CFD numerical simulation and wind tunnel physical simulation were carried out to investigate the appropriateness of using the CFD approach to provide wind loads for building’s preliminary design. Through studies on a benchmark cube problem, it was found that V2F model has a better performance among the various RANS turbulence model. The estimation on the wind pressures and wind loads of a tall building indicate that the CFD procedure can provide accurate mean wind pressure on the windward side but it tends to underestimate the pressure on the leeward side. Nevertheless, the numerical example results suggest that the CFD simulation with V2F turbulence model can provide a reasonable design wind load for the purpose of tall buildings’ preliminary design.

6 REFERENCES