COMPARISONS OF COMPUTATIONAL FLUID DYNAMICS AND WIND TUNNEL EXPERIMENTS FOR PEDESTRIAN WIND ENVIRONMENTS

Chin-Hsien Lin¹ Cheng-Hsin Chang² Chii-Ming Cheng³

¹ Researcher, Department of Water Resources and Environmental Engineering, Tamkang University Tamsui, Taipei County 251, Taiwan, golden@staff.tku.edu.tw
² Associate Professor, Department of Civil Engineering, Tamkang University Tamsui, Taipei County 251, Taiwan, cc527330@mail.tku.edu.tw
³ Director, Wind Engineering Research Center, Tamkang University Tamsui, Taipei County 251, Taiwan, cmchemg@mail.tku.edu.tw

ABSTRACT

CFD (computational fluid dynamics) is being increasingly applied to the prediction of the wind environments around the actual buildings. This study used the commercial software (fluent 6.3) to predict the velocities of the wind environments and compared with the wind tunnel experiments. The results of the correlation coefficients of the experiments for each direction and the CFD predictions were between 0.51 and 0.92, and the average error of the sixteen directions of all measurement points was below 30%. The results of the comfort levels of the Davenport of the wind tunnel experiments were agreed well with the CFD predictions.

KEYWORDS: WIND TUNNEL TEST, CFD, PEDESTRIAN WIND ENVIRONMENT

1. Introduction

Existence of one or a group of tall buildings is known to alter the velocity field surrounding the building or a group of buildings in such a way that the comfort of pedestrians may be affected. Production of corner vortex shedding, horse-shoe vortex, and other wind effects due to interactions between buildings are some of the reasons that areas of high wind velocity and turbulence are generated. Pedestrian acceptability of sidewalks, entrance, plazas, and terraces is often an important design parameter to be considered by the building owner and architect. (He et al[1])

Environmental impact assessment of wind criteria on the pedestrian comfort is becoming a standard requirement in the design of tall buildings in Taiwan. Traditionally, the evaluation of the pedestrian wind environment relies heavily on wind tunnel testing. But the improvements of the computer facilities and the computational fluid dynamics softwares in recent years have enabled predictions of the pedestrian wind environments around buildings in the design stage. The CFD of the pedestrian wind environment can be made more economically and well accurate results. There have been a lot of case studies on the pedestrian level wind environment around actual buildings using the CFD. (Uchida et al [2], Murakami et al[3], Zhang et al [4])

The CFD prediction of the pedestrian wind environments could be made some challenges. First, the geometry of the urban buildings was so complex. If users wanted to draw the geometry of all buildings in urban city, the CFD software hasn’t been successfully resolved yet. It was so important that how to omit the outside of the block buildings. Secondly,
the computation of the flows around buildings required knowledge of the characteristics of atmospheric boundary layer. Besides mean wind speed data, turbulence data were needed to accurately represent the approaching flow of the atmosphere. So, urban meteorological data for atmospheric wind do not provide the level of details needed for CFD prediction. (Kim et al [5])

According to the Architectural Institute of Japan guides, we could get some information of the use of the CFD for investigating the pedestrian wind environments around buildings such as the inflow boundary condition, grid dependence of solution, unstructured grids and the geometry of the urban blocks. The guidelines proposed were mainly based on the high Reynolds number (Re) Reynolds Averaged Navier-Stokes equation (RANS) models, although it is desirable to use a large eddy simulation (LES) and a low Re number model in order to obtain more accurate results. However, it is difficult to use those models for the practical analysis because many computational cases (sixteen wind directions) and a huge number of grids were required for prediction of the pedestrian wind environments under severe time restrictions. (Yoshie et al [6])

In Europe, a 3-year EU-funded project with surveys carried out at different open spaces has finished by 10000 interviews (http://alpha.cres.gr/ruros). The project was an urban design tool to provide architects, engineers, and urban planners to assess effectively the construction of the new buildings and the development cities form human comfort. The European Action C14 was dealing with “Impact of Wind and Storm on City and Built Environment” which was interesting for the parameters of human comfort and evaluation. (ASCE [7])

2. Experimental design

This study was performed in a complex urban area model. This model consisted of a city block in Taichung city, Taiwan with low-rise buildings. The target building was 130 meters high and the surrounding buildings were between 15 and 40 meters high (Fig.1). The wind tunnel experiments at 1:300 scales were performed on this model in a turbulence boundary layer with a power law exponent of 0.25 and the blockage ratio should be below 3% based on the knowledge of wind tunnel experiments. The real heights of the humans were from 1.5 to 1.8 meters and the velocity of the wind tunnel measurements was at 0.6 mm above the wind test floor by Irwin probes.

3. Computational fluid

The computational domain area was 3.0 m(x) × 3.0 m(y) × 1.2 m(z), which included a whole urban block. The CFD geometries of the buildings were set up by the CAD data which was the same as the wind tunnel models (Fig.2). The geometry of the complex buildings would be bought from the government of the Taichung city. Some problems were needed to solve. First, it would take long time to obtain the locations and the height of all buildings, so we could write the modules of the Microsoft visual basic application (VBA) to get databases of the whole urban buildings. Second, the complex datum of the building would not use user interface to key in the values of the x, y location and height step by step. The gambit software supplied the journal format of the text-file to read the datum and build up whole buildings.

The commercial software of computational fluid dynamic was fluent 6.3. The grid arrangements were unstructured and the numbers were about 1,500,000. The inflow boundary condition was interpolated values of velocity and turbulence intensity from the experimental approaching flow (Fig.3). The outflow boundary condition was pressure outlet. The surface buildings were logarithmic law for smooth wall. The turbulence model of the CFD simulation was adopted the standard kappa-epsilon and renormalization group kappa-epsilon.
4. Results of the wind tunnel experiments and the CFD predictions

The predictions of 16 wind directions were simulated by the CFD software. The results of the wind tunnel test and the CFD predictions were compared with the wind direction N, which was the wind direction that most frequently occurred in Taichung. The dimensionless velocity was the wind velocity that at each measuring point divided by the reference velocity of the approaching flow free stream.

Fig.4 compared the dimensionless velocity at each measuring point with the wind tunnel tests and the CFD simulations. There were the positions of 62 measuring points, see
Fig. 5. The CFD results agreed well with the results of the wind tunnel experiments. The predictions of the near building points were highly accurate. However, several points of the building yard and outside alleys were not matched very well. In the wake regions, the predictions of CFD were smaller than the wind tunnel experiments. The slight differences might be caused by the influences of the different shapes of the geometry buildings that the turbulences of the CFD could not be predicted well on the corner of the separated shear layers.

Fig.4 and Fig.5 represented the different prediction points of the red and blue rectangles. The dimensionless velocities of the measuring points were bigger than the CFD predictions. The blue rectangle region was in the yard of the target building. The red region was in the leeward of the target building and surrounding houses. These two locations were in the wake regions and the dimensionless velocity was smaller by the CFD predictions, see Fig 6 and 7.

Figure 4: Comparison of dimensionless velocity of the wind tunnel test and CFD prediction

Figure 5: The measuring points of the wind tunnel experiment

Figure 6: The vector velocity in North direction

Fig.8 represented that the contour of the dimensionless velocity was in west direction. The correlation coefficient was the best of the sixteen directions. Regions with high dimensionless velocity were between 0.5 and 0.6 and found at the corners on the northwest and leeward of the target building. The lower dimensionless velocity in west-south and north-west of the target building was from 0.1 to 0.2.

It was important to reproduce the characteristics of the separating flows near the buildings. A fine grid arrangement was required to resolve the flow near the corners. It should be confirmed that the prediction result didn’t change significantly with different grid systems. Fig.9 represented that the comparison of the wind tunnel experiment and the CFD prediction of the refine grids (about 4260000) would not be more different than the coarse grids (about 1850000). The correlation coefficient of the refine grids was 0.81. The correlation coefficient
of the coarse grids was 0.78. So the refine grids were better than the coarse grids but the average errors of all measured points wouldn’t be improved very well.

Fig.10 represented that the coefficient correlation of wind dimensionless velocity obtained from CFD simulation and wind tunnel experiments. The predicted results in the CFD code were almost identical. In the wake region of the target buildings, there was a tendency to underestimate the wind speed compared with the results of the wind tunnel experiments. However in other regions, the matching was relatively satisfactory.

Fig.9 represented that the correlation coefficient of the north direction was about 0.78. The low dimensionless velocities (0-0.4) of the wind tunnel test points were obviously higher than the CFD predictions. The bad correlation coefficients of the S and SSW were between 0.5 and 0.6. The average correlation coefficient of 16 directions was 0.81, see Fig.11.

If the turbulence models of the renormalization groups (RNG) $\kappa - \varepsilon$ was used, the results of the correlation coefficient was the same as the standard $\kappa - \varepsilon$. However the RNG $\kappa - \varepsilon$ took long time to simulate and sometimes needed to adjust relaxation factor in order to converge well. According to AIJ guidelines, the RNG $\kappa - \varepsilon$ model was better than the standard $\kappa - \varepsilon$ model in strong regions and matching with the experiments results were improved. However in wake regions, standard $\kappa - \varepsilon$ model was better than the RNG $\kappa - \varepsilon$ model. That was why the AIJ guidelines did not recommend the RNG $\kappa - \varepsilon$ turbulence model to predict the pedestrian wind environments.[6]

The ranked evaluation was performed on the CFD simulation results according to the criteria for assessing the wind environments proposed by Davenport. [8] This was based on the occurrence frequency of the daily mean wind velocity and divided into five rank comfort conditions. The results of the wind tunnel tests and the CFD predictions for each measuring points were summarized in Fig.9 and Fig.10. The wind tunnel test of the measuring point (20) would be fault data, because the criteria of surrounding points (19, 23, 25) were slow walk.

The results at the measuring points with the criteria of the slow walk were in the northeast of the target building. The CFD predictions of points (1, 21, 22, 51, 52) were more conservative than wind tunnel experiments. The coffee regions of the wind tunnel experiments were more serious than the CFD prediction results. However, as described above, there were differences between some of the experimental results and CFD predictions for the measuring points near the target building in wake region. These areas were the north-west side, yard and the alleys surrounding low houses.

Figure 7: The contour velocity in north direction
Figure 8: The contour velocity in west direction
Figure 9: The correlation coefficient of the north direction

Figure 10: The correlation coefficient of the west direction

Figure 11: The correlation coefficient of the different turbulence models

Figure 14 represented that the average error of the sixteen directions was below 30%. In weak regions and alleys such as points 18, 54-62, 31, 34 and 48 were form 40% to 50%. Other points were from 12% to 30%, and had good result with wind tunnel tests and CFD predictions. The CFD prediction of some points were high errors by matching the wind tunnel tests, these results would be queried that the CFD simulation would apply in wind pedestrian environments. But the velocity of the measurement by hot-film, thermal couples and Irwin probes could generate some errors. According to AIJ guidelines, if it was limited to the highest wind region, which is important for evaluation of the pedestrian wind environment around the building, the wind speed difference between CFD prediction and the wind tunnel tests was about 15% at most for the standard $\kappa - \varepsilon$ model.
Conclusions

This study had the good results of the wind tunnel experiments and computational predictions of the wind on people in the urban environments. The average of the correlation coefficients of the 16 direction experiments and the CFD predictions was 0.81. The average error of the sixteen directions of all measuring points was below 30%. The velocity of the wind tunnel experiments measured by Irwin probes or hot-file thermal couples could produce some errors. The application of the CFD can decrease experimental errors and offer engineering designers to adjust the shapes and the directions of the new buildings quickly.

References


