臺灣能源期刊　第七卷　第二期　第163-180頁　中華民國109年6月
Journal of Taiwan Energy
Volume 7, No. 2, June 2020, pp. 163-180

A Wind Farm Case Study by the Computational Fluid Dynamics Model and LiDAR Data

Ming-Hong Chen[1*]

ABSTRACT

The purpose of the present study is to verify of the numerical scheme for the evaluation of wind resources by the computational fluid dynamics (CFD). Different turbulence and extended numerical schemes referred in literature were employed and compared against the experimental data. Results showed that the Realizable k-epsilon turbulence model would be the most appropriated choice in the present study. The case study of the wind farm in the Institute of Nuclear Energy Research (INER) campus by the CFD method was conducted. Results were compared with the measured LiDAR (light detection and ranging) data. Similar results were observed within the range from 40 m to 100 m, while larger deviation was shown for the region higher than 100 m. It is also suggested to measure wind data from ground to 40 m by LiDAR for further investigation.

Keywords: computational fluid dynamics (CFD), wind energy, LiDAR.

1. Introduction

The features of weather in Taiwan include the Asian monsoon and tropical cyclones in summer time and northeast trade wind in winter time, leading to strong winds in many places. In the inland and urban region, the topography is generally more complex, and it is not easy to find the place with good potential to install wind turbine. Therefore, reliable assessment for the wind resource is paramount for decision making. For the evaluation of wind energy, one can employed the simplified linear model or conducted the sophisticated non-linear method, i.e., computational-fluid-dynamic (CFD) method to better understand the detail flow field and wind resource of the investigated spot. The CFD technique is a relatively mature method for the evaluation of wind resources. There are several commercial codes available for such analysis. However, the optimal schemes should be examined and verified to obtain the reasonable results for the evaluation of wind resources since the commercial codes are general-purpose software. The purpose of this study is to conduct the verification of the numerical scheme for the evaluation of wind resources by CFD method. Results were also compared with the experimental data, and the optimal numerical model and related parameters were determined. The verified numerical methodology was then employed into the case study for the renewable energy campus of the Institute of Nuclear Energy Research (INER). Results were compared with the measured data by

[1] Assistant Researcher, Mechanical and System Engineering Program, Institute of Nuclear Energy Research.
* Corresponding Author, Phone: +886-3-4711400#3351, E-mail: minghongchen@iner.gov.tw

Received Date: July 22, 2019
Revised Date: December 15, 2019
Accepted Date: April 23, 2020

the light detection and ranging (LiDAR) system installed in INER.

1.1 Literature review

Berge *et al*. (2006) compare the Wind Atlas Analysis and Application Program (WAsP) and two CFD modes in the evaluation of wind in complex terrain in western Norway. Results showed that no improvements in the calculated mean wind speed was observed by the CFD method. The employed turbulence model in the CFD method was the k-epsilon scheme. Balduzzi *et al*. (2012) evaluated the feasibility of Darrieus vertical-axis wind turbine (VAWT) installed in the rooftop of a building by the CFD method. The optimal angle of the slop (8^o) of the installed roof was proposed to increase the wind velocity. The employed turbulence model was the k-epsilon scheme. Song *et al*. (2014) built a CFD model as a tool to assess the wind resources of the investigated wind farm. The proposed model was validated by the measurements of installed anemometers within the analyzed wind farm. This model is able to calculate reliable wind resources distribution for a wind farm with complex terrain without applying meso-scale methods. The employed turbulence model was the k-epsilon model. Wang *et al*. (2016) evaluated the wind energy over the roof of two perpendicular buildings by the CFD method. The employed numerical methods and schemes were verified by introducing a benchmark case of one building with experimental data. The best numerical parameters and models included the k-epsilon turbulence model, double precision for temporal scheme, SIMPLE (semi-implicit method for pressure linked equations) method for pressure-velocity coupling, second-order scheme for pressure discretization, Quadratic Upstream Interpolation for Convective Kinematics (QUICK) scheme for momentum, k and epsilon equations. The proposed numerical models by this study will be examined in the present work as the verification before conducting further numerical analysis. Toja *et al*. (2015) evaluated the effect of roof shape on the flow field by the CFD method in order to determine threshold of turbulence intensity on the horizontal-axis wind turbine (HAWT). Results showed that building with curved shape roof top were the most interesting choice, leading to the highest speed-up and lowest turbulence intensity. The employed turbulence model was the standard k-epsilon scheme in this study. Yang *et al*. (2016) developed a CFD model to evaluate the wind energy of a selected building of urban environment. The employed turbulence model is the Realizable k-epsilon model, and good agreement was observed when comparing to the experimental data. The effect of the geometry of building on the flow field was conducted. Results showed that the rounded roof resulted in smoother corner expansion flow and lower turbulence intensity. At the same time, the power density increased up to 86.5%. Llaguno-Munitxa *et al*. (2017) evaluated the influence of building geometry on street canyon air flow by large eddy simulation (LES) model and wind tunnel experiments. Results showed that reliable agreement with experimental data could be obtained by using the LES model. They also pointed out that larger error was also observed for the case with round and pitched roof due to the difficulties in the prediction of the exact location for the separation. Thus, more research is needed to evaluate the effect of roof geometry on the performance of wall-models.

Hassanli *et al*. (2018) proposed the innovative Double Skin Façade (DSF) system to exploiting wind energy in urban environments. CFD results were validated against a series of wind tunnel tests. Results showed that with newly proposed method, the turbulence could be reduced by up to 30%, and the average wind power density could be increased by a factor of 2 and 4.2, by creating recessed regions and curved walls. The employed turbulent method was the Shear Stress Transport (SST) k-omega scheme. Garcia *et al*. (2018) compared the turbulence model of LES and RANS (Reynolds-averaged Navier-Stokes) in the simulation of urban flow against measurement data. Results showed that the LES model was found to be more accurate in the prediction of turbulence kinetic energy in 80% of the measured sites, while it reduced to 50% in the prediction of mean velocity field. This study also suggested that the inflow uncertainties can be a dominant factor, and its effect should be verified to guarantee the reliability of the results calculated by the LES model. Du *et al*. (2018) evaluated the effect of mesh density, turbulence model and computational domain size on the simulation of wind environment in complex urban area by the CFD modeling process. The case study was conducted by the campus of Hong Kong Polytechnic University. Validation was also made against wind tunnel experimental data, and good agreement was also observed. Four turbulence models were compared, including standard k-epsilon, Re-Normalisation Group (RNG) k-epsilon, MMK k-epsilon, and Realizable k-epsilon. Among the investigated turbulence model, the scheme of RNG k-epsilon yielded the best results in the employed case study. The employed turbulence models in literature were comprehensively compared in Table 1. As shown in Table 1, the most commonly used turbulence model is the k-epsilon model. However, in the study of finding the optimal turbulence model, there was no consistent result so far.

Dadioti and Rees (2017) conducted a case study by the CFD method. The proposed CFD model was built with Detached Eddy Simulation (DES) turbulence method for the full scale campus of De Montfort University (DMU) in UK. Calculated data were compared with data measured

Table 1. Comparison of the employed turbulence models in literature (by author)

Ref.	Standard k-epsilon	RNG k-epsilon	Realizable k-epsilon	SST-k-omega	LES
(Berge *et al*. 2006)	Employed				
(Balduzzi *et al*. 2012)	Employed				
(Song *et al*. 2014)	Employed				
(Wang *et al*. 2016)	Compared Best	Compared	Compared		
(Toja *et al*. 2015)	Employed				
(Yang *et al*. 2016)			Employed		
(Llaguno-Munitxa *et al*. 2017)					Employed
(Hassanli *et al*. 2018)				Employed	
(García-Sánchez *et al*. 2018)	Compared				Compared Better
(Wang *et al*. 2018)			Employed		
(Du *et al*. 2018)	Compared	Compared Best	Compared		

from 3 sites. Good agreement was presented of this study. Such method is then applicable for the analysis of the wind potential in the complex urban environments. Wang *et al*. (2018) proposed a CFD model and compared with data measured from a LiDAR system. For the inlet condition, a semi-log formula was proposed and adjusted, comparing the commonly employed logarithm relation. Comparison was made at different azimuthal position. Larger deviation was observed near the lower grounded region, while good consistency was observed for the rest of the investigated heights. The optimal height and location for the installation of wind turbine were suggested based on the CFD results.

From the above literature review, it can be observed that linear model and intricate CFD model were commonly employed in the assessment of wind resources. In the region with simple geographical condition, both models lead to similar results. However, in the complex terrain, only the intricate CFD model can provide better prediction. In the CFD modeling process, it is very important to employ the appropriate turbulence model. From the surveyed studies, there were five turbulence models employed or compared. The most commonly used one is the standard k-epsilon model. In one compared study, the standard k-epsilon model was also verified as the best model (Wang *et al*., 2016). However, another study published in 2018 showed that the RNG k-epsilon model performed better than the standard one (Du *et al*., 2018). Another study showed that the LES model was the better choice in the CFD modeling (García-Sánchez *et al*., 2018).

The subject of the present study is to conduct a case study for wind farm analysis by using the CFD method with verified numerical schemes. More discussion on the effect of turbulent model on the simulation can be found in relevant studies (Sanderse *et al*., 2010; Rezaeiha *et al*., 2019; Stergiannis *et al*., 2016).

Therefore, the most commonly used standard k-epsilon model might not be the best choice in the calculation by the CFD model since there is still no consistent conclusion on this issue. In the present study, the effect of the employed turbulence model on the assessment of wind energy and flow field was verified by the CFD model and published experimental observation in literature. Results were comprehensively compared. The appropriate turbulence model in the simulation of urban environment was proposed accordingly.

With the verified turbulence model, the CFD model of the entire campus of INER was proposed. The inlet wind condition was specified based on the site measurement on the building of INER. A LiDAR system was installed at the same time to measure the wind data at different heights for validation of the proposed CFD model.

2. Numerical Model and LiDAR System

In this chapter, the developed CFD model and employed LiDAR system are introduced.

2.1 CFD model

In this study, the CFD model has been built by using the commercial code ANSYS-FLUENT[1]. Numerical schemes and turbulence models were comprehensively investigated and compared against the experimental data in literature (Yoshie *et al*., 2007). The basic set-up of the CFD model

[1] https://www.ansys.com/zh-tw/products/fluids/ansys-fluen .

was employed from the study of Wang *et al*. (2016). The investigated building was covered by a rectangle box. Grid adaption was employed near the building to capture the complex variation of the flow field. Grid independent test was conducted, with enforcement near the building. The upstream distance from inlet to the building is 125 m, and the downstream distance is 375 m. The width of the computational domain is 500 m. The configuration size of the domain size was adopted from the study of Wang *et al*. (2016).

The CFD model was built by including a block with thickness of b, width of 4b and height of 4b as indicated in Figure 1. This configuration was proposed by Yoshie *et al*. in 2007 (2007) with the wind tunnel experiment with scale of 1 : 100. This wind tunnel experiment was made as a comparative and parametric study on the flow around a square prism. The configuration of 2 : 1 : 1 and 4 : 4 : 1 of the square prism were investigated. In the present study, the case of 4 : 4 : 1 was selected as the validation configuration. Detail measurement was conducted at the place in the turbulent boundary layer. The wind velocity was measured by the split film probe. The average wind speed and the standard deviation of fluctuation were also evaluated.

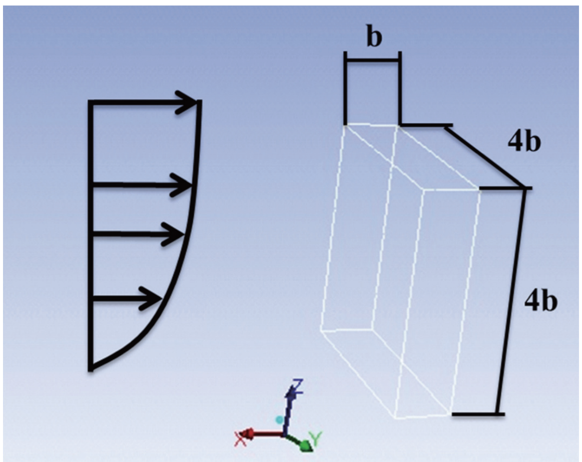

Fig. 1. Schematic diagram of the building of CFD model (by author).

Instead of using the constant profile for the inlet velocity, the employed inlet velocity profile was assumed with the shear exponent of 0.25. At the height of 100 m, the free stream velocity was 7.84 m/s according to the referred study (Wang *et al*., 2016). Comparison was also made for the proposed CFD model with experimental data as shown in Figure 2. As indicated in Figure 2, consistent distribution along the height of the simulation was observed, validating the designated inlet velocity distribution.

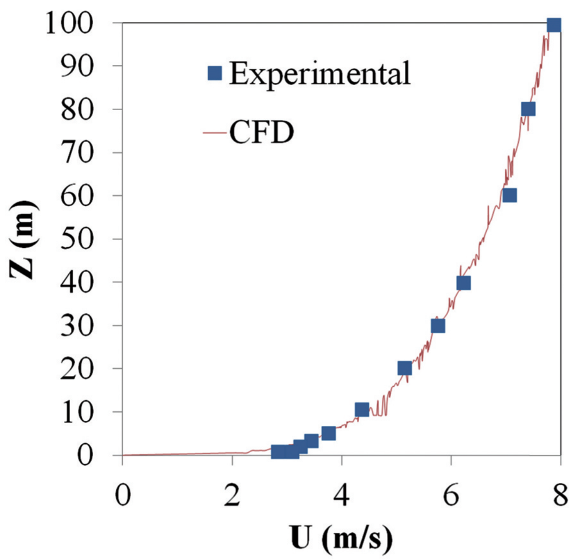

Fig. 2. Comparison of inlet velocity profile (Yoshie *et al*., 2007).

The deployment of the compared experimental measurement (Yoshie *et al*., 2007) was shown in Figure 3. There were 109 points for data collection. For the comparison, the data on top of the building (point 14 to 31), before (point 1-13) and after the building (point 58-70 and 84-96) were employed.

For the parametric study, various turbulence models were employed and results were compared to verify the feasibility for the investigated condition. The investigated turbulence models were the Standard k-epsilon, RNG k-epsilon with Wall function, SST-k-omega, Realizable k-epsilon,

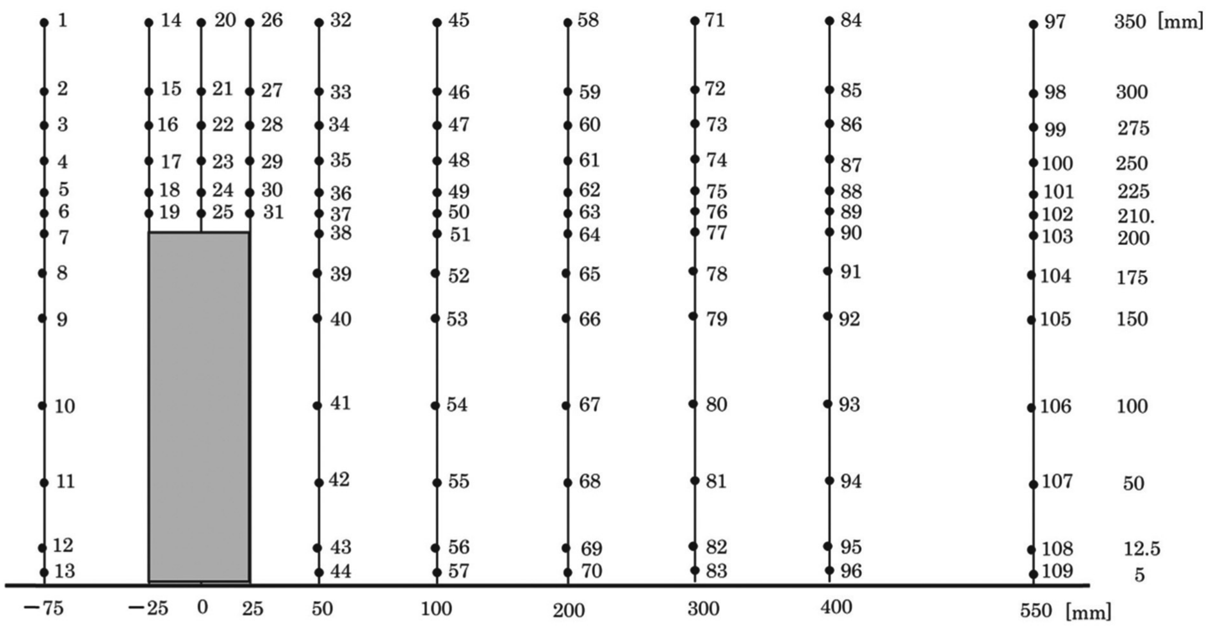

Fig. 3. Wind tunnel measurement distribution (Yoshie *et al*., 2007).

and Transient SST.

2.2 LiDAR system

The employed LiDAR system was installed as illustrated in Figure 4. Measured range is from 40 m to 280 m with sampling frequency of 1 Hz. The precision for wind speed is 0.1 m/s from 0 to 60 m/s. The main system is a cubic box with dimension of 54 cm/55 cm/54 cm, and its weight is 45 kg. Wind data were measured from September-17-

Fig. 4. Employed LiDAR system in INER campus (by author).

2018 to December-19-2018.

The observed wind data was scrutinized carefully to allocate a stable 30 minute interval during the measurement. It takes about 3.3 minutes for an average wind of 4 m/s to pass through the campus of INER with dimension of 800 m. The average wind speed of the chosen interval was employed for the comparison and adjustment of the inlet velocity profile as the validation of inlet conditions. After this validation of the inlet condition, the flow field was calculated to investigate the wind distribution and flow field characteristics of the case study.

3. Results and discussion

The grid independent test was conducted for the proposed CFD model as the first step in this study. Secondly, the turbulence models and related numerical schemes were employed and comprehensively examined and compared. Comparison was also conducted against the experimental data. Finally, the most appropriate combination of the turbulence model and numerical schemes were proposed for the CFD model to conduct the simulation of the flow field and the assessment of wind energy in the campus of INER as a case study.

3.1 Grid independent test for the benchmark model

Grid independent test was conducted by comparing the calculated velocities with the experimental data from literature (Yoshie *et al.*, 2007). Data on top of the building were selected for the comparison. As indicated in Figure 3 and Figure 5, selected positions included the front edge (p1~p4 for point of 16~19), middle (z1~z6 for point of 20~25) and rear edge (m1~m6 for point of 26~31) of the building. The computational grid of the CFD model was gradually increased from 32 k, 315 k, to 446 k. Deviation was evaluated by comparing the calculated values with the experimental measurements. The largest deviation was observed in the grid of 32 k at the position of m^3. In overall, velocity difference decreases with the increase of grid number except the position of z1. This might be the insufficient density of

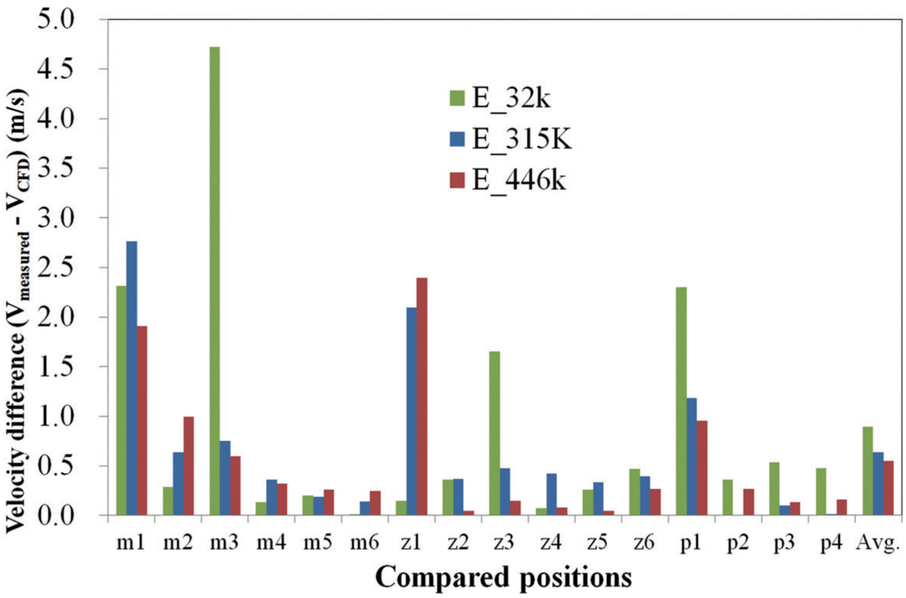

Fig. 5. Grid independent test and comparison of velocity difference (by author).

the measurement in the original experiment. The overall results suggested that the grid system of 32 k was not suitable. The improvement from the grid of 315 k to 446 k was only 2.95%. Therefore, the grid system of 315 k was employed in the further investigation.

3.2 Effect of turbulence models and numerical schemes

Different turbulence models and related numerical schemes referred in literature were employed and compared against the experimental data as shown in Figure 6.

The case of Basic was the model compared in the previous section with sufficient amount of grid number. The case of paper best simply employed the turbulence model and numerical schemes from the study of Yoshie *et al.* (2007), where the standard k-epsilon model was used. The discretization scheme was second order for pressure equation, and QUICK scheme for momentum, k and epsilon equation. Further compared cases were the RNG k-epsilon model with enhanced wall function, Realizable k-epsilon model, SST-k-omega model, and the Transient SST model. As compared in Figure 6, the largest difference was observed for the case using the model of SST-k-omega at the position of p1. The second largest one was also SST-k-omega at position of z2. The overall largest difference was obtained by SST-k-omega, too. The second largest difference in average was obtained in the case of the Transient SST, mainly due to the large deviation at the position of z1 and p1. The average velocity differences for the rest of three cases (paper best, RNG k-epsilon with enhanced wall function, and Realizable k-epsilon) were comparable. Among the compared cases, the Realizable k-epsilon model resulted in the smallest difference, and it was even lower than that of the paper best

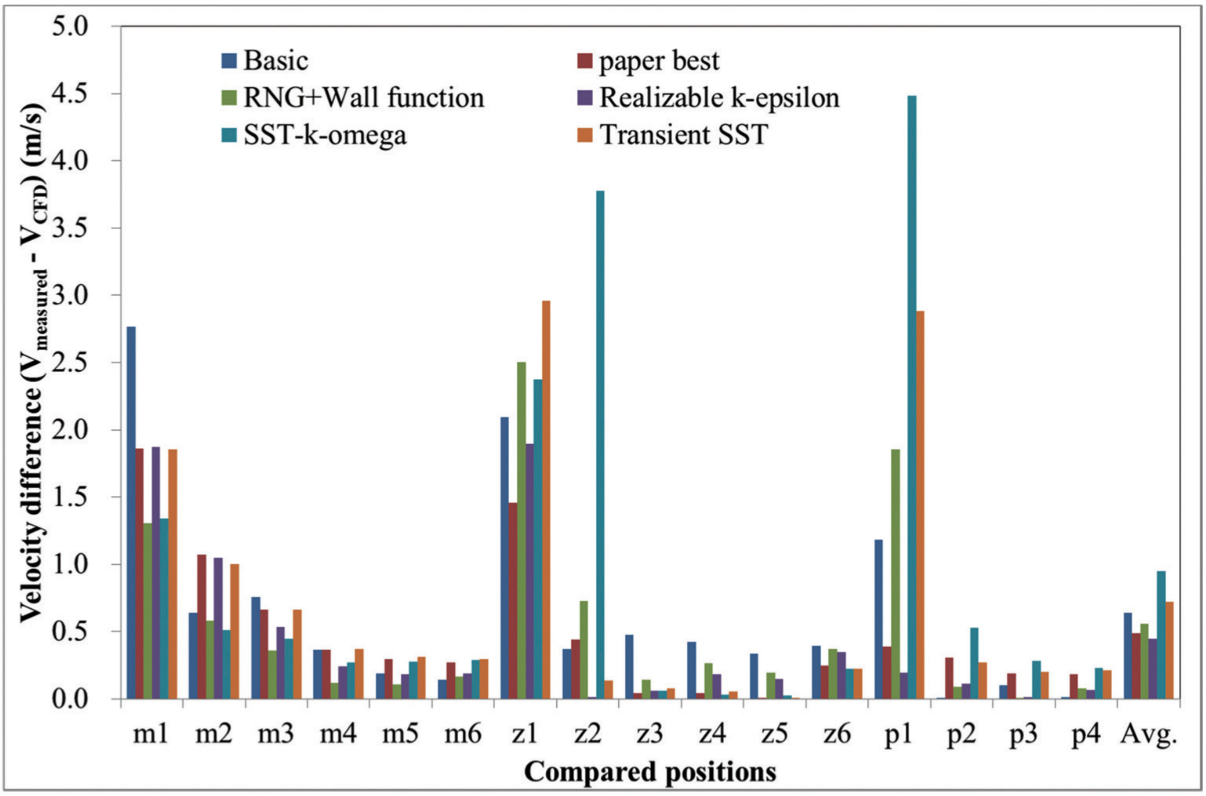

Fig. 6. Comparison of turbulence models and numerical schemes (by author).

case. Therefore, it can be noted that the Realizable k-epsilon turbulence model should be the most appropriated choice in the present study.

The average velocity difference by the best model of the present study was 0.44 m/s, while it was 0.49 m/s for the case of paper best. In the original study for the case of paper best, the proposed model using the standard k-epsilon resulted in the difference of 0.37 m/s (Wang *et al*., 2016). In that study, all the data of the 109 points were compared. By extending the comparison to the position before the building (10 m) and after the building (20 m and 40 m), the average velocity difference was 0.55 m/s for the case using the Realizable k-epsilon model, while it is 0.61 m/s for the case of paper best.

It was also firstly observed that the turbulence model of Realizable k-epsilon resulted in better performance in the model comparison when compared with the surveyed relevant studies as shown in Table 1. However, the standard k-epsilon model was suggested at the same time for the relevant investigation in the future since the resulted difference just 0.06 m/s larger than that of the Realizable k-epsilon. The Realizable k-epsilon model was employed in the following investigation of this article.

3.3 CFD model for the case study of INER campus

With the verified turbulence model, the case study was conducted for the campus of INER by CFD method using the commercial code of ANSYS-FLUENT as indicated in Figure 7.

3.4 Grid independent test for the case study of INER campus

Grid independent test of this case study was also conducted, and results were shown in Table 2. As compared in Table 2, the deviation from M1 to M2 is 3.57%, and it reduces to 0.37% only when applying the mesh of M3. Thus the grid system of M2 was employed for the further calculation.

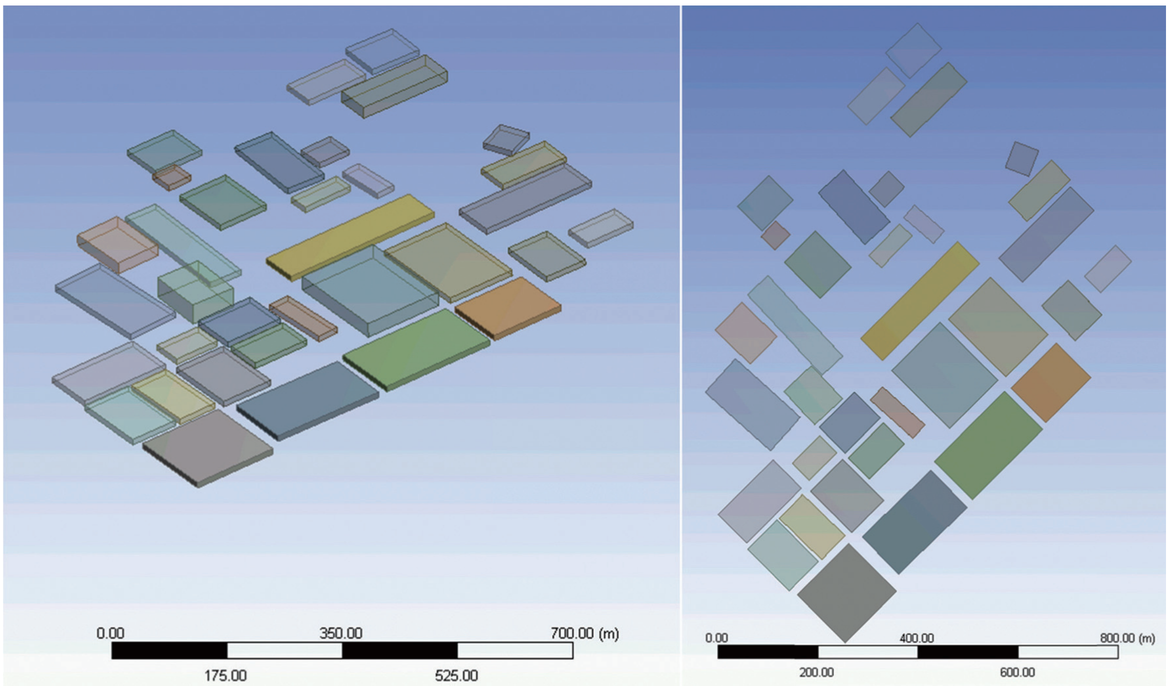

Fig. 7. CFD model of the case study of INER campus (by author).

Table 2. Grid independent test for the model of INER campus (by author)

Name	Cell number	Deviation
M1	687,095	-
M2	2,093,093	3.57%
M3	6,086,270	0.37%

3.5 LiDAR data

The wind data from the Longtan site (the closest weather site to INER) was observed (Figure 8) as the first step for the further analysis of LiDAR data.

Wind data of Longtan site and LiDAR measurement were available for 4 months. By observing the historical value of daily wind speed of Longtan site, 5 days (October 30, 31, November 1, 2, and December 8.) with relatively strong wind speed were identified as shown in Figure 8. LiDAR data were shown in Figure 9. Relatively stronger winds were also identified at October 30, 31, November 1, 2, and December 8.

As shown in Figure 8 and Figure 9, roughly consistent trend was observed at October 30-31, November 1-2, and December 8. Furthermore, the wind speed of December 8 is much higher (9.6 m/s) then those of other identified days (7.1 m/s ~ 7.7 m/s).

Next, the minutely wind data at October 31, November 1 and December 8 were shown in Figure 10, Figure 11, and Figure 12 for the whole day (1,440 minutes). The wind speeds were taken at the height of 100 m, and its variation w.r.t. the 30-minute-average value was also compared.

These time series data was divided into 48 parts for each day with time interval of 30 minutes. For those intervals with smaller variation were marked by the red line circles. The variation for the interval of December 8, 505~649 minute was relatively small with deviation about 2%. However, the corresponding wind speed gradually increases

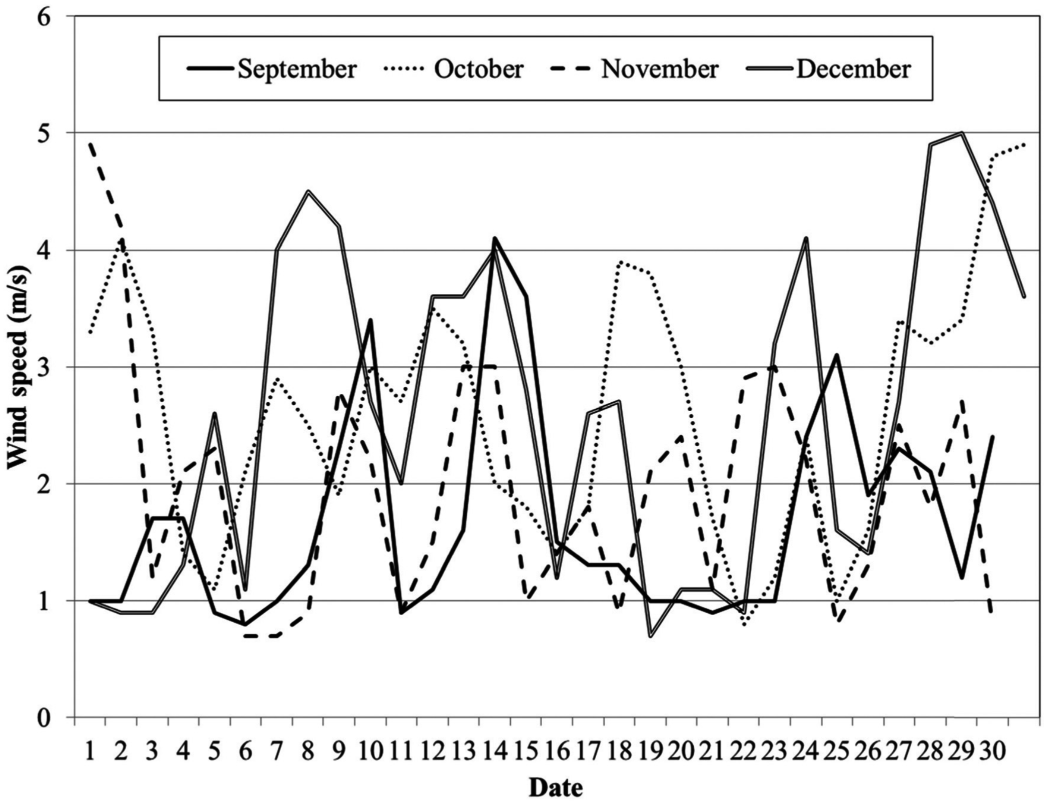

Fig. 8. Collected wind speed history of Longtan site (by author).

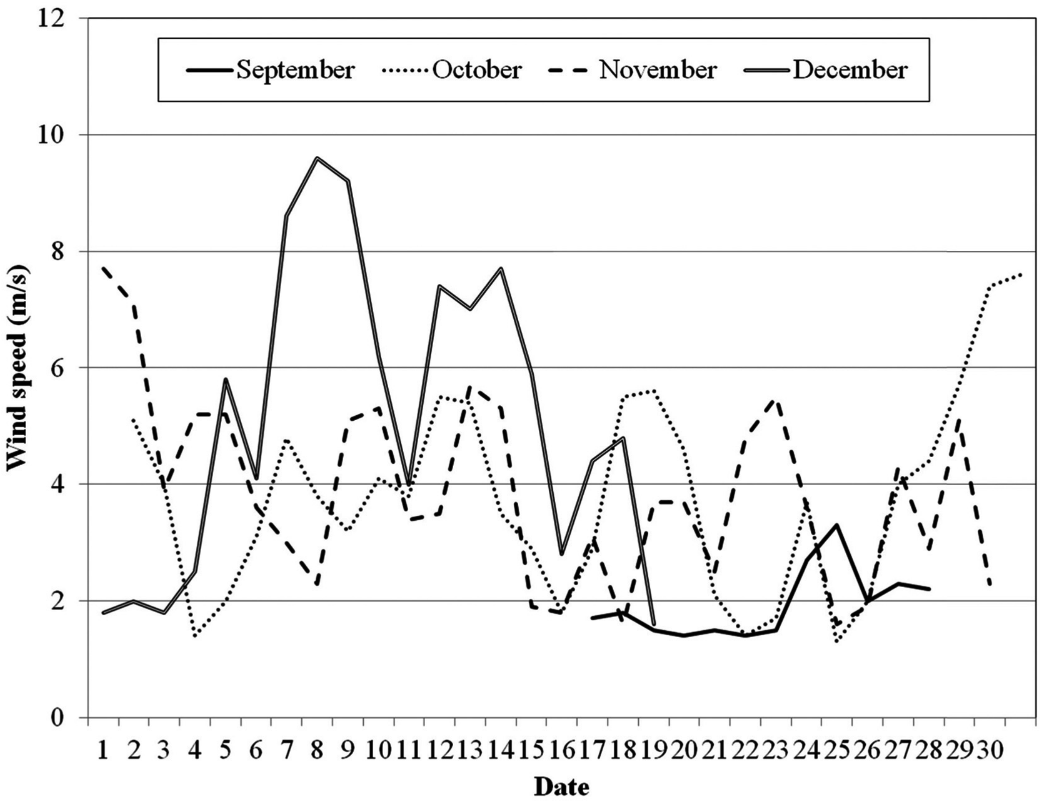

Fig. 9. Collected wind speed history by LiDAR system (by author).

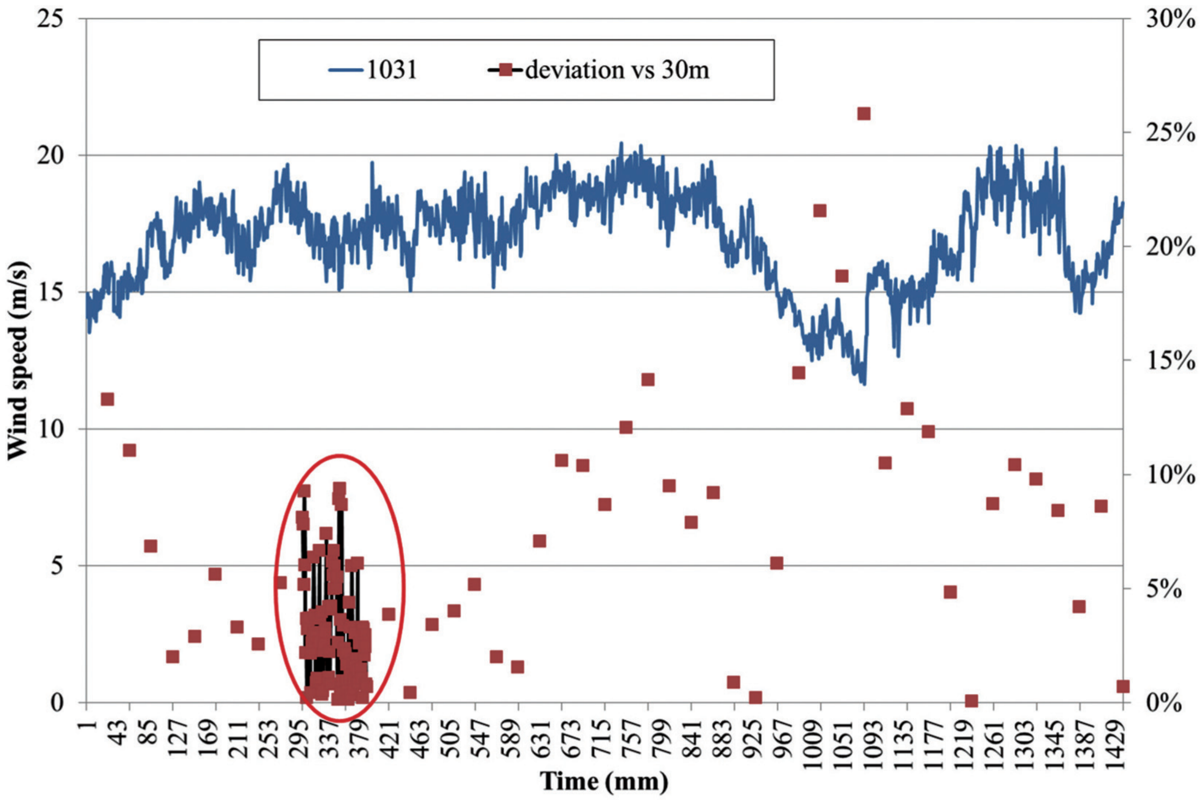

Fig. 10. Collected wind speed history and deviation at October-31 2018 (by author).

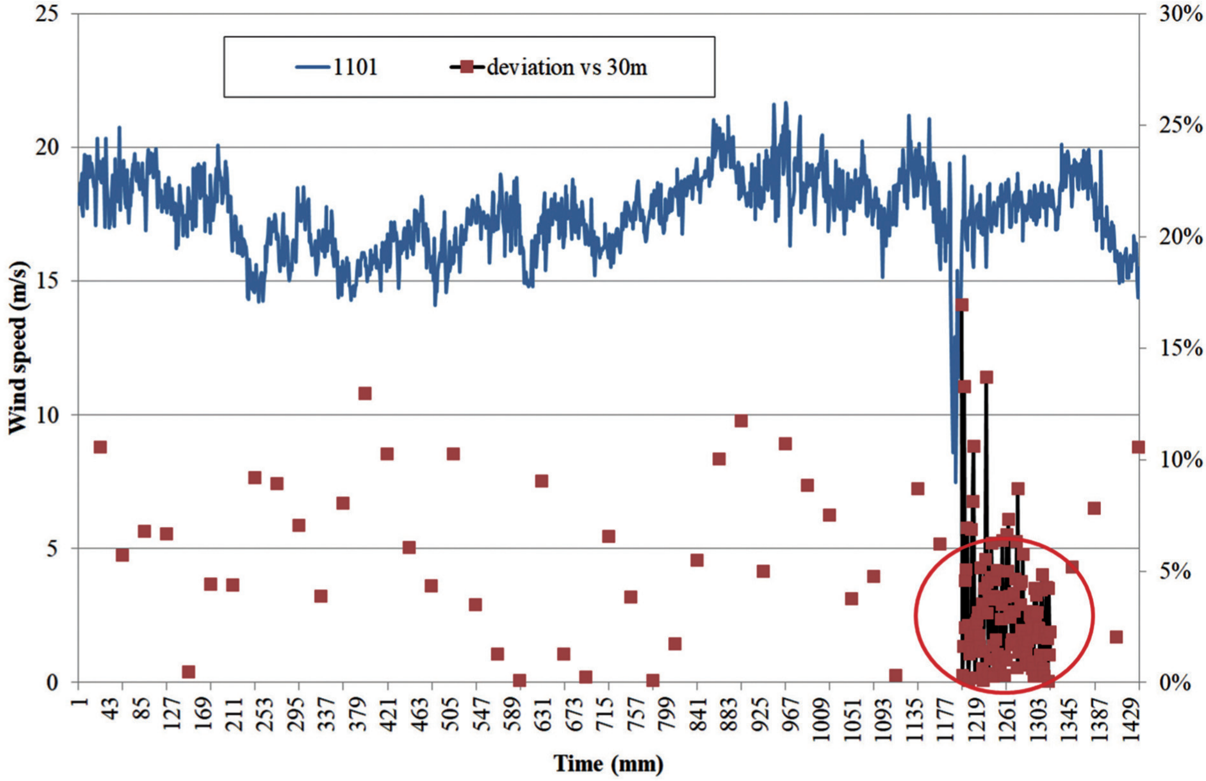

Fig. 11. Collected wind speed history and deviation at November-1 2018 (by author).

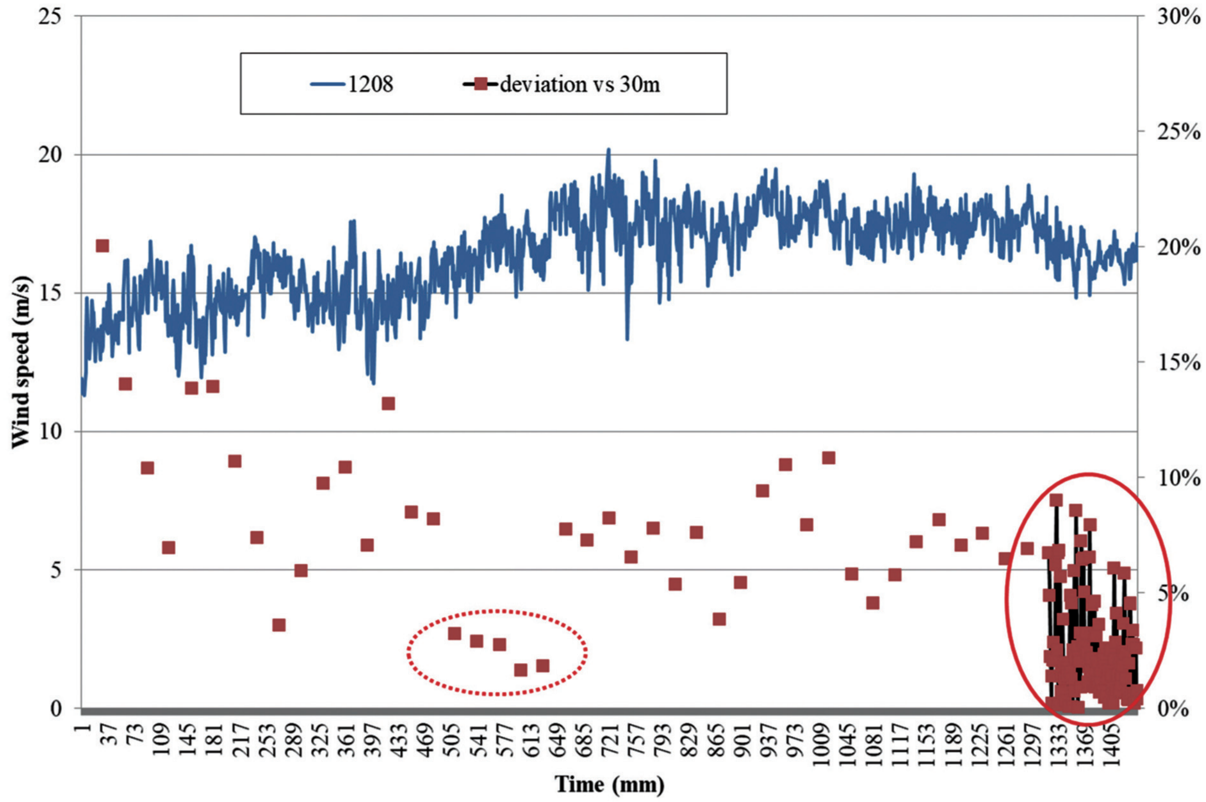

Fig. 12. Collected wind speed history and deviation at December-8 2018 (by author).

within this time interval. The second last interval of December 8 (1,381~1,410 minute) showed the minimum deviation within whole day (just 1.83%). Thus, this time interval was chosen for further investigation.

Sensitivity study of the wind shear exponent for the inlet condition was conducted and comprehensively compared. As indicated in Figure 13, smaller deviation within the height of 100 m and overall was obtained with wind shear value of 0.25. The purpose of the present study is to investigate the flow filed within the height from 40 m to 100 m. Thus, larger deviation from the height of 100 m to 140 m won't affect the conclusion of the present study. With the wind shear exponent of 0.25, the deviation within the height of 40 m to 100 m is only 1.37%. It is 3.9% above the height of 100 m. Therefore, the wind shear exponent of 0.25 was chosen for the following calculation.

3.6 Comparison of CFD and measurement data

Results by the CFD model (blue line) were compared with the measured LiDAR data (dots) as shown in Figure 14. Similar variation was observed within the range from 40 m to 100 m, while larger deviation was shown for height higher than 100 m.

For LiDAR data, the variation of velocity was observed within the height of 40 to 80 m, and it kept constant higher than 80 m, indicating the range for the effect of boundary layer in the velocity profile for the investigated site. For CFD results, however, velocity variation was observed from 5 to 140 m. The comparison showed close profile within the range of 40 to 60 m and larger deviation above 60 m. The velocity difference for higher domain might due to the employed velocity profile (power law) in the present study. It was suggested to introduce different inlet velocity profile to improve the prediction of velocity for higher domain in the future.

In the present study, the hub heights of the considered wind turbine are within 40 m to 60 m. For the range from ground to 40 m, the comparison and reliability of CFD model cannot be investigated without the measured data. Thus,

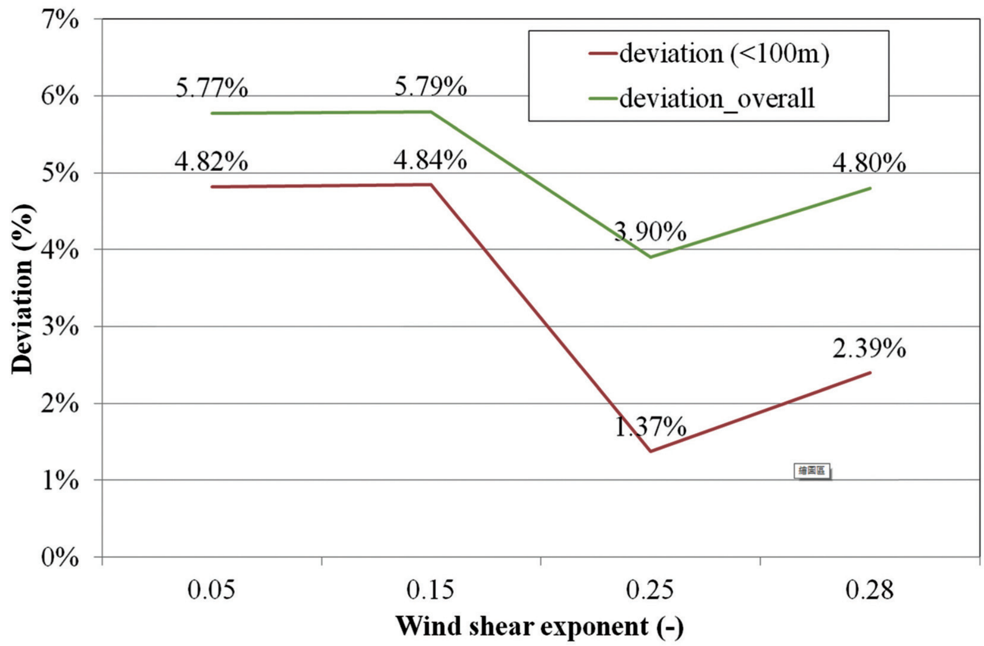

Fig. 13. Effect of wind shear exponent (by author).

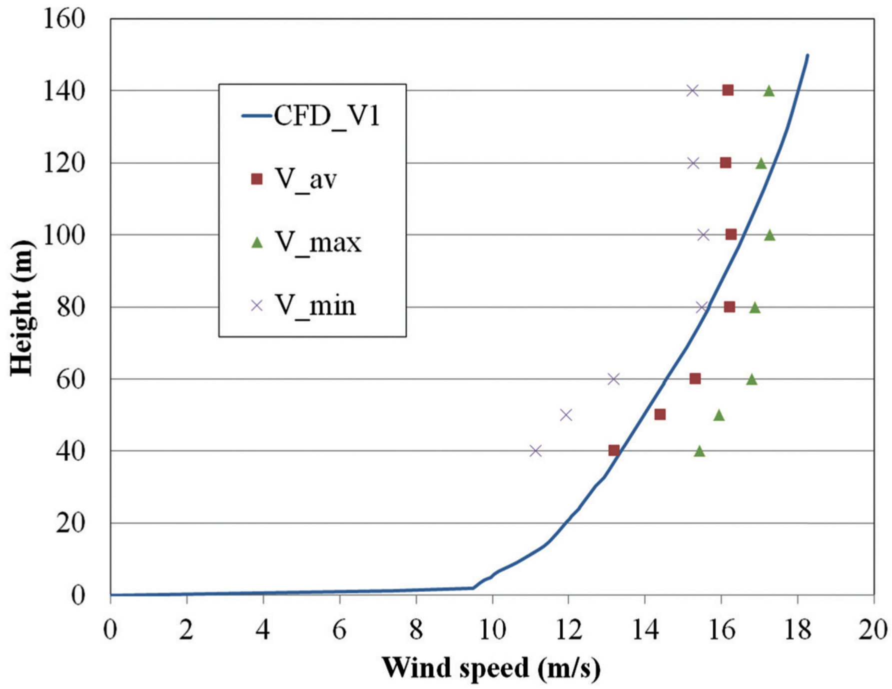

Fig. 14. Comparison of CFD results and LiDAR data (by author).

it was also suggested to conduct the measurement from ground to 40 m by the LiDAR system in the future.

Velocity distributions were drawn from 6 m to 100 m. The maximum and minimum velocity for all velocity distribution plots were the same for comparison as shown in Figure 15.

The height of 6 m is specifically for the wind turbine with rated power less than 5 kW. As indicated in Figure 15, low wind speed was observed in the most locations due to the effect of building and ground boundary layer. Therefore, it is not recommended to install the wind turbine with hub height lower than 6 m within the campus of INER. For the height of 12 m, lower buildings were not observed, while low speed region downstream still existed. Speed-up was observed in the tunnel between buildings. There is a region in right, lower corner with higher wind speed where the 25 kW and 150 kW wind turbines were installed. At the height of 25 m, only 2 buildings were observed, and low wind speed region still existed downward. At the height of 50 m, the effect of 3 height buildings still existed and resulted in low speed regions. At the height of 100 m, there is no effect of building on the flow field.

4. Conclusion

The CFD technique is relatively mature method for the evaluation of wind resources. However, the optimal schemes should be tested and verified to obtain the reasonable results for the evaluation of wind resources since the commercial codes are general-purposed software. The purpose of the present study is to conduct the verification of the numerical schemes for the evaluation of wind resources by the CFD method. Results were also compared with the experimental data to find the optimal numerical model and related parameters for the case study of INER campus.

The CFD model has been built by using the commercial code ANSYS-FLUENT. The grid independent test was conducted for the proposed

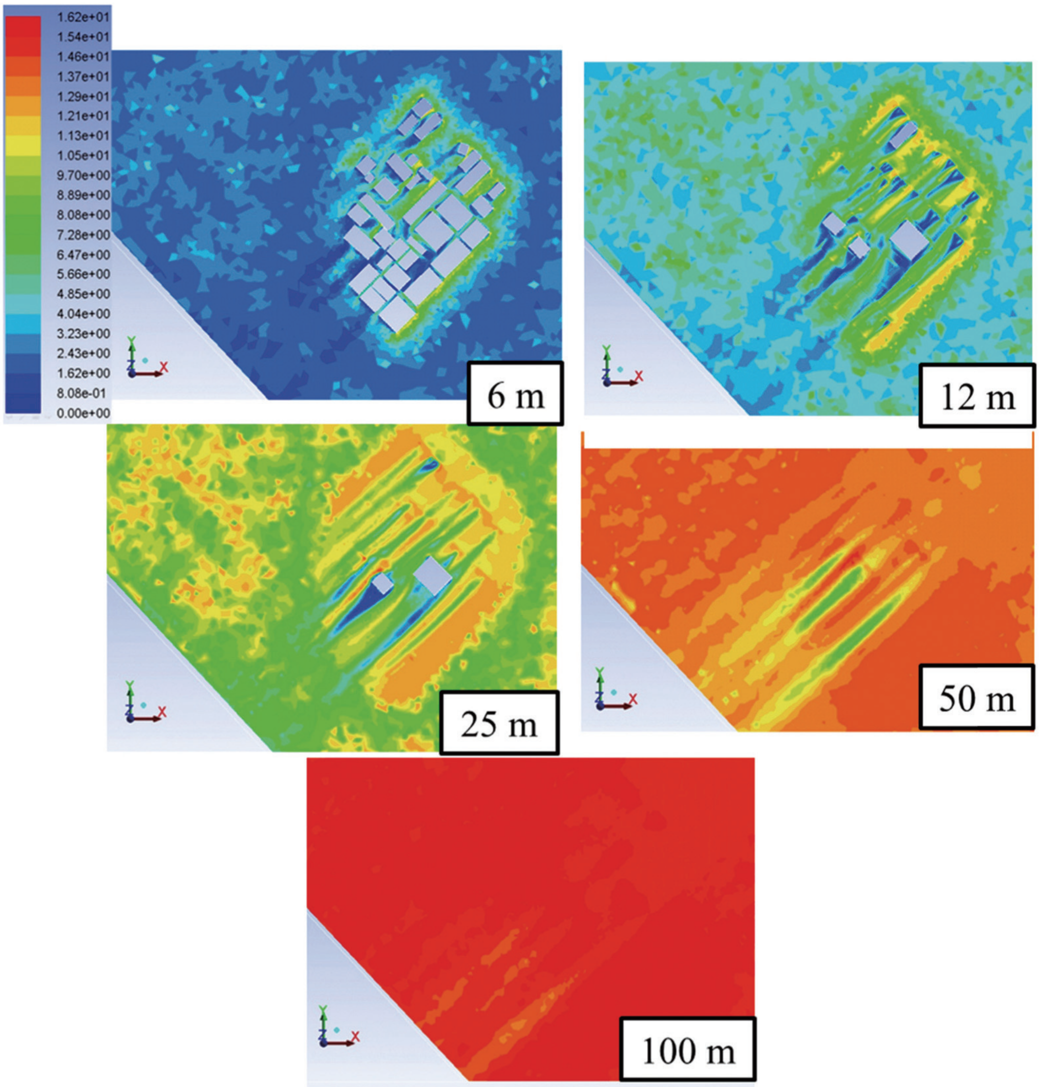

Fig. 15. Velocity distributions at selected height (by author).

CFD model as the first step in this study. Secondly, the turbulence models and related numerical schemes were employed and comprehensively examined and compared. Comparison was also conducted against the experimental data. Finally, the most appropriate combination of the turbulence model and numerical schemes were proposed for the CFD model to conduct the simulation of the flow field and the assessment of wind energy in the case study. Results of grid test indicated that the grid system of 32 k was not suitable. The improvement on the evaluation of velocity from the grid of 315 k to 446 k was only 2.95%. Therefore, the grid system of 315 k was employed in the further investigation. Different turbulence and extended numerical schemes referred in literature were employed and compared against the experimental data. Among the compared cases, the Realizable k-epsilon model resulted in the smallest difference, and it was even lower than that of the best case proposed in literature. Therefore, in can be noted that the Realizable k-epsilon turbulence model would be the most appropriated choice in the present study.

For the case study of the INER campus, the measurement data by the Longtan site were taken as the reference to identify the representative wind data of the LiDAR system for the CFD model. By focusing on the higher wind speed, 3 days has been chosen as the second step. The wind speeds were taken at the height of 100 m, and its variation w.r.t. the 30-minute-average value was also compared. Results showed that the wind speed at December 08. (1,381~1,410 minute) was observed with the minimum deviation within whole day (just 1.83%). Thus, this time interval was selected for following investigation. Sensitivity study of wind shear for the inlet condition was conducted and deviation was compared. With the wind shear exponent of 0.25, the deviation within the height of 40 m to 100 m is only 1.37%. Therefore, the wind shear exponent of 0.25 was chosen for the following calculation.

Results by the CFD model were compared with the measured LiDAR data. Similar variation was observed within the range of 40 m to 100 m, while larger deviation was shown for height higher than 100 m. In the present study, the hub heights of the considered wind turbines are within 40 m to 60 m. For the height from ground to 40 m, the comparison and reliability of CFD model cannot be investigated in the present study.

Detail velocity distributions were presented from 6 m to 100 m. The height of 6 m is specifically for the wind turbine with rated power less than 5 kW. Low wind speed was observed in the most regions due to the effect of building and the ground boundary layer. Furthermore, it is not recommended to install the wind turbine with hub height lower than 6 m within the campus of INER. For the height of 12 m, lower buildings were not observed, while low speed region downstream still existed. Speed-up was observed in the tunnel between buildings. There is a region in right-lower corner with higher wind speed where the 25 kW and 150 kW wind turbines were installed. With higher height, the effect of buildings on the flow field vanished gradually. At the height of 100 m, there is no obvious effect of building on the flow field within the investigated INER campus.

In the present study, the employed wind data were measured within the height from 40 m to 200 m. However, more sophisticated variation is expected from ground to 40 m. Thus, it is suggested to measure wind data within this region for further investigation.

References

Balduzzi, Francesco, Alessandro Bianchini, Ennio Antonio Carnevale, Lorenzo Ferrari and Sandro Magnani, 2012. "Feasibility Analysis of a Darrieus Vertical-Axis Wind Turbine Installation in the Rooftop of a Building." *Applied Energy* 97: 921-29.

Berge, E., R.Gravdahl, J.Schelling, Lars Tallhaug and Ove Undheim, 2006. "Wind in Complex Terrain. A Comparison of WAsP and Two CFD-Models." *Proceedings from EWEC*, January 2006.

Dadioti, Rallou and Simon Rees, 2017. "Performance of Detached Eddy Simulation Applied to Analysis of a University Campus Wind Environment." *Energy Procedia* 134: 366-75.

Du, Yaxing, Cheuk Ming Mak and Zhengtao Ai, 2018. "Modelling of Pedestrian Level Wind Environment on a High-Quality Mesh: A Case Study for the HKPolyU Campus." *Environmental Modelling and Software* 103: 105-19.

García-Sánchez, C., J.van Beeck and C. Gorlé, 2018. "Predictive Large Eddy Simulations for Urban Flows: Challenges and Opportunities." *Building and Environment* 139 (February): 146-56.

Hassanli, Sina, Gang Hu, David F. Fletcher and Kenny C. S. Kwok, 2018. "Potential Application of Double Skin Façade Incorporating Aerodynamic Modifications for Wind Energy Harvesting." *Journal of Wind Engineering and Industrial Aerodynamics* 174 (January): 269-80.

Llaguno-Munitxa, Maider, Elie Bou-Zeid and Marcus Hultmark, 2017. "The Influence of Building Geometry on Street Canyon Air Flow: Validation of Large Eddy Simulations against Wind Tunnel Experiments." *Journal of Wind Engineering and Industrial Aerodynamics* 165 (December 2016): 115-30.

Rezaeiha, Abdolrahim, Hamid Montazeri and Bert Blocken, 2019. "On the Accuracy of Turbulence Models for CFD Simulations of Vertical Axis Wind Turbines." *Energy* 180: 838-57.

Sanderse, B, S. P. Van Der Pijl and B. Koren, 2010. "Review of CFD for Wind-Turbine Wake Aerodynamics." *Physical Review Letters*, 1-28.

Song, M. X., K. Chen, Z. Y. He and X. Zhang, 2014. "Wind Resource Assessment on Complex Terrain Based on Observations of a Single Anemometer." *Journal of Wind Engineering and Industrial Aerodynamics* 125: 22-29.

Stergiannis, N., C.Lacor, J. V. Beeck and R. Donnelly, 2016. "CFD Modelling Approaches against Single Wind Turbine Wake Measurements Using RANS." *Journal of Physics: Conference Series* 753: 032062.

Toja, Francisco, Carlos Peralta, Oscar Lopez-Garcia, J. Navarro and Ignacio Cruz, 2015. "On Roof Geometry for Urban Wind Energy Exploitation in High-Rise Buildings." *Computation* 3: 299-325.

Wang, B, L. D. Cot, L. Adolphe, S. Geoffroy and J. Morchain, 2016. "Estimation of Wind Energy over Roof of Two Perpendicular Buildings." *Energy and Buildings* 88 (2015): 57-67.

Wang, Qiang, Jianwen Wang, Yali Hou, Renyu Yuan, Kun Luo and Jianren Fan, 2018. "Micrositing of Roof Mounting Wind Turbine in Urban Environment: CFD Simulations and Lidar Measurements." *Renewable Energy* 115: 1118-33.

Yang, An-Shik, Ying-Ming Su, Chih-Yung Wen, Yu-Hsuan Juan, Wei-Siang Wang and Chiang-Ho Cheng, 2016. "Estimation of Wind Power Generation in Dense Urban Area." *Applied Energy* 171: 213-30.

Yoshie, R., A. Mochida, Y. Tominaga, H. Kataoka, K. Harimoto, T. Nozu and T. Shirasawa, 2007. "Cooperative Project for CFD Prediction of Pedestrian Wind Environment in the Architectural Institute of Japan." *Journal of Wind Engineering and Industrial Aerodynamics* 95 (9-11): 1551-78.

以計算流體力學模式與光達數據進行風場案例分析

陳銘宏[1*]

摘 要

本研究之目的在於分析適用於風能評估之計算流體力學之數值模型與參數設定。其中針對紊流模型之測試結果發現Realizable k-epsilon之紊流模型之結果與實驗數據之誤差最小，將列於後續之案例分析當中。在以核研所為標的之案例分析中，採用計算流體力學方法建立分析模型，並搭配先前分析找出之最佳數值模型及參數進行分析，再與光達量測之數據進行比對。在高度從40公尺到100公尺之間呈現一致之結果，而超過100公尺以上的區域之差異則較大。後續建議再以光達設備在相同地點進行從地面到40公尺高度的範圍進行量測，並做進一步的探討與分析。

關鍵詞：計算流體力學，風能，光達

[1] 核能研究所機械及系統工程專案計畫 助理研究員
*通訊作者電話: 03-4711400 #3351, E-mail: minghongchen@iner.gov.tw

收到日期: 2019年07月22日
修正日期: 2019年12月15日
接受日期: 2020年04月23日