A Study on the Variation of Wind Velocity between Buildings with the Change of Building Corner Modification

Jang-Youl You*, Min-Woo Park**, Ki-Pyo You***‡

*Department of Architecture Engineering, Songwon University, Republic of Korea

** Department of Architecture Engineering, Chonbuk National University, Republic of Korea

*** Department of Architecture Engineering, Chonbuk National University, Republic of Korea

(wmjlove1877@hanmail.net, minwoo6050@jbnu.ac.kr, youkp@jbnu.ac.kr)

‡ Corresponding Author; Ki-Pyo You, Department of Architecture Engineering, Chonbuk National University, Republic of Korea, Tel: +82 63 270 4057,

Fax: +82 63 270 2285,youkp@jbnu.ac.kr

Received: 17.12.2018 Accepted:03.02.2019

Abstract- Building Integrated Wind Power (BIWP), the installation of wind power generators on buildings to generate energy, is being attempted in various ways. The advantage of BIWP is that it does not require a support fixture for positioning a wind turbine at a desired height. Furthermore, it allows the energy generated by a turbine to be used directly within the building. Utilizing computational fluid dynamics (CFD), this study examines the variations in wind velocity and turbulence intensity occurring as a result of changes in the morphology of a building's corners to increase the efficiency of wind power generators that are installed between buildings. Results of this study show that buildings with round corners experienced an increase in wind velocity of up to 13%, as compared to regular corners, while corner cuts increased wind velocity by 15%. The morphology of the corners will have less effect on the location where wind velocity is highest where the distance between buildings is greater. However, if the distance between the buildings is shorter and the corners are long enough, wind velocity increases and turbulence intensity decreases.

Keywords Building Integrated Wind Power, Computational Fluid Dynamics, Wind velocity ratio, Turbulence, Tall building

1. Introduction

Many countries have recently seen an increase in the number of high-rise buildings in downtown areas. This is due to increasing population and inflow of people into the inner cities. High-rise buildings in downtown areas are becoming popular because they offer advantages such as addressing the issue of insufficient housing, creating new jobs, and playing the role of landmarks within cities. However, in comparison to regular buildings, high-rise buildings consume more energy for ventilation, elevators, and lighting. In fact, buildings account for approximately 40% of the world's total energy consumption and 21% of the total carbon emissions. This consumption has been reduced by the introduction of new and renewable energy sources for buildings in recent years [1-5]. Wind energy has the advantage of being highly efficient for such purposes, as the wind speed increases in downtown areas due to the wind around buildings. One way to use wind energy to power buildings is to use building-integrated wind power (BIWP), which produces energy using wind turbines installed on or close to buildings [6-10]. BIWP is highly efficient because it does not require supports for positioning the wind turbines at the installation height, and the produced energy can be used in the building without having to be sent long distances. When applying the BIWP method, the effects of wind speed and the turbulence caused by the surrounding buildings should be considered. Surrounding buildings can interfere with the wind flow, decrease the wind speed, and increase the turbulence, thereby reducing the efficiency of wind power generation. The effects of decreases in wind speed should be taken into particular consideration when developing wind power turbines close to high-rise buildings in downtown areas, because the wind speed varies depending on the layout and shape of the building. To address this, the characteristics of air flow through the passage between two buildings have been researched. Ishizaki and Sung (1971) and Wiren (1975) conducted experiments in wind tunnels to measure the average wind speed at the center line of the passage between two block buildings with the same height [11-12]. Stathopoulos and Storms (1986) conducted a wind tunnel experiment to investigate the characteristics of air flow in passages between buildings for various building heights and wind angles [13]. To and Lam (1995) measured the average wind speed and turbulence at pedestrian height in passages between high-rise buildings in a wind tunnel experiment [14]. They found that the wind speed amplification coefficient in the passage between buildings increased up to approximately 1.4. Most previous experiments were related to the air flow between buildings at pedestrian height (i.e., 1.75 m to 2 m from the ground), and did not analyze the characteristics of air flow above pedestrian height in detail.

In this study, we investigated the air current characteristics between two parallel buildings and analyzed the variation in wind speed with respect to the shapes of the building corners by increasing the wind speed. The aim of this study was to improve the power generation efficiency of wind turbines installed near buildings.

2. Validation Test and Analysis

We carried out computational fluid dynamics (CFD) analysis to investigate the variation in wind speed and turbulence for various building corners. We compared the wind speed and turbulence intensity measured in the wind tunnel experiment to the results obtained from the CFD turbulence analysis.

2.1. Wind Tunnel Experiment

We used the Series 100 Cobra Probe (Turbulent Flow Instrumentation) to measure wind speed. The size of the experimental model was 40 m (B) * 40 m (D) * 120 m (H), the distance between the buildings (w) was 60 m, and the model scale was 1/400. The experimental conditions used for the wind tunnel experiment are summarized in Table 1. The experimental buildings were assumed to be high-rise buildings in downtown areas, and we reproduced category B (α = 0.22) surface roughness to simulate a downtown area with scattered mid-rise buildings. In Figure 1, the solid line indicates the theoretical equation of the power law, and the dotted line indicates the wind speed and turbulence intensity as a function of height; this was measured in the wind tunnel. Figure 2 shows the locations at which the wind velocities were measured in the wind tunnel. The measurement locations between the buildings are indicated in red. We measured 40 points in total, with a horizontal spacing of 25 mm and a vertical spacing of 37.5 mm, with the exception of the floor. Figure 3 shows the design of the model installed in the wind tunnel.

Table 1. Test condition

Device	Series 100 Cobra Probe
Exposure Category	B (α=0.22)
Velocity	5m/s
Scale	1/400
Frequency	150Hz (60secs)

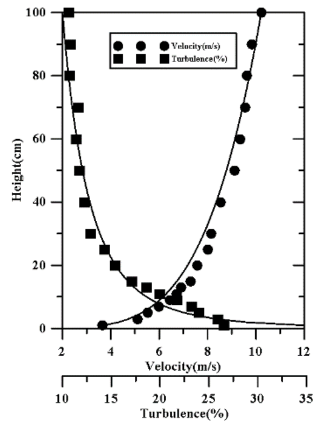


Fig. 1. Distribution of the mean velocity and turbulence intensity in the wind tunnel

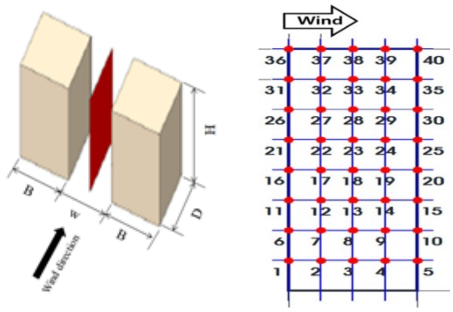


Fig. 2. Measurement locations and points

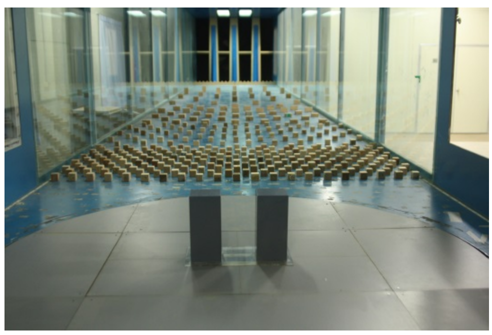
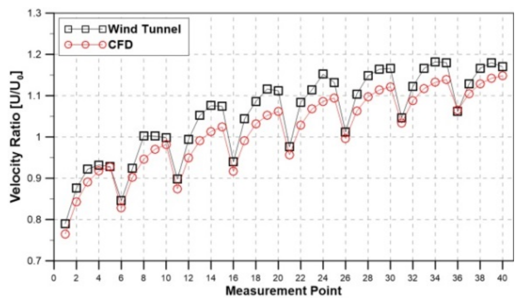


Fig. 3. Model placed in the wind tunnel

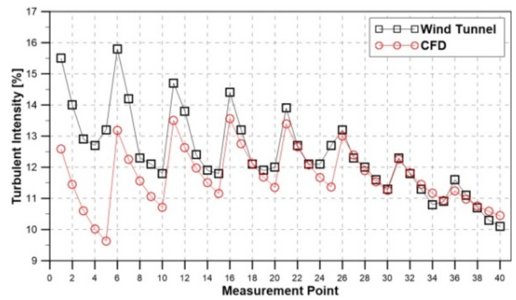
2.2. CFD Validation

We used ANSYS Fluent R16.1 to validate the wind speed measurements, with the same wind speed and turbulence intensity distribution used for the inflow air flow distribution as used in the wind tunnel experiment. We set the renormalization group (RNG) to k-ε in the turbulence model used for the CFD analysis. The total domain size was 2.4 m * 2.1 m * 1.7 m. A mesh independence study was conducted according to the shape of the mesh and the size of the mesh. The shape of the mesh was Tetre, Hexa and the size of the mesh around the cube was 1cm, 1.5cm. As a result, the Hexa mesh of 1.5cm showed the similar tendency as the experimental value and the highest correlation coefficient. So, the number of mesh points was 600,000 using Hexa mesh of 1.5cm around the cube.

Figure 4 shows the wind speed ratio and turbulence intensity at each measurement point in the wind tunnel experiment and the CFD. The wind speed ratio is the wind speed (U) at each point divided by the wind speed (U_0) measured at the highest floor (H). The wind speed ratio distribution was similar to the wind tunnel experiment. Figure 5 shows the correlation between the wind speed ratio and turbulence intensity at each position measured in the wind tunnel experiment and obtained from the CFD analysis. At each measuring point in the wind tunnel experiment and CFD analysis, the correlation coefficient was 0.988 and the turbulence intensity was 0.647. The correlation between the CFD and measured wind speed was very high, whereas the correlation with the turbulence intensity was low. However, the correlation with the turbulence intensity was 0.611 in the lower measurement positions (no.1–20) and 0.930 in the higher measurement positions (no.21–50). Thus, the turbulence intensity was higher in the upper sections. Figure 6 shows the correlations between the turbulence intensities in the lower and upper parts of the buildings obtained from the tunnel experiment and the CFD analysis. It seems that in the CFD analysis, the turbulence intensity in the lower part of the incoming air flow is higher than in the upper part because of the effects of vortices in the lower part of the building.

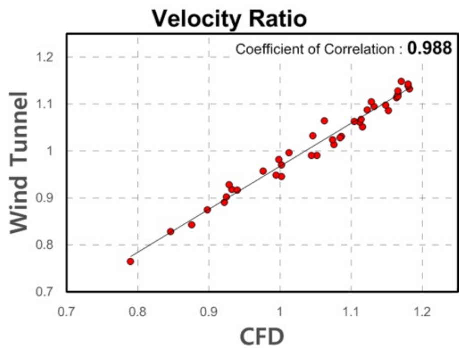


(a) Velocity ratio

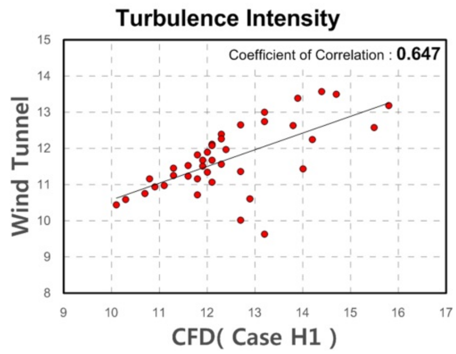


(b) Turbulence intensity

Fig. 4. Data obtained from the wind-tunnel and computational fluid dynamics analyses

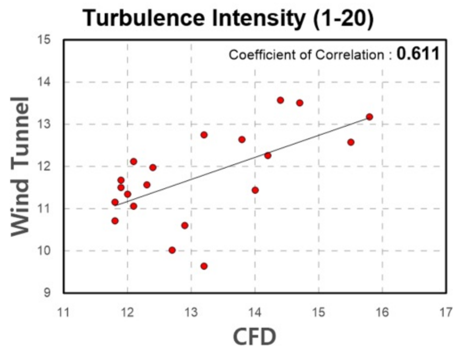


(a) Velocity ratio (Correlation Coefficient : 0.988)

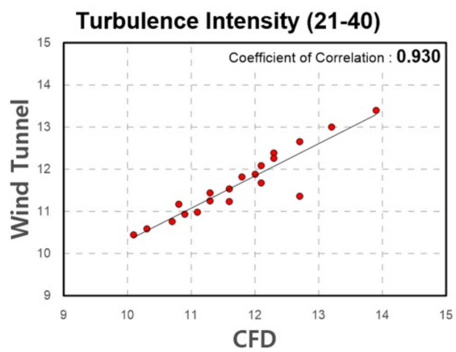


(b) Turbulence intensity (Correlation Coefficient : 0.647)

Fig. 5. Correlation between the wind-tunnel and computational fluid dynamics analysis



(a) Lower part (Correlation Coefficient : 0.611)



(b) Upper part (Correlation Coefficient : 0.930)

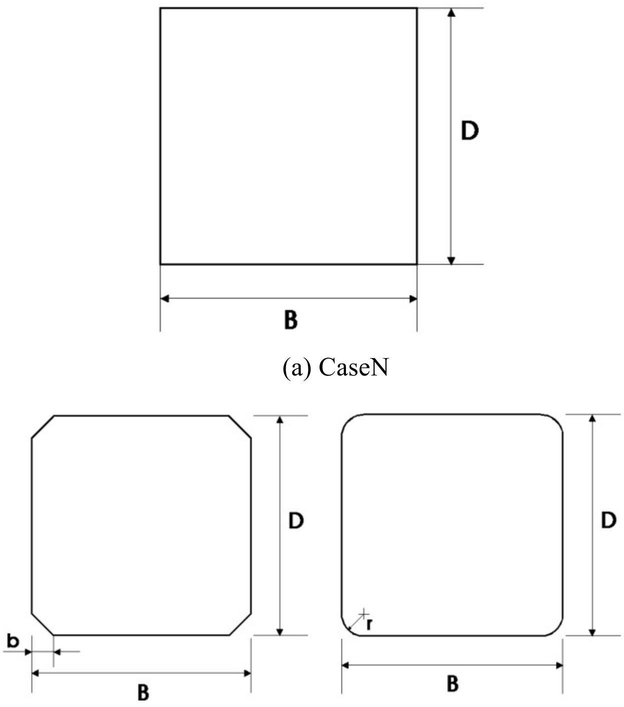
Fig. 6. Correlation of turbulence intensity in the lower and upper parts of the buildings

3. CFD Analysis

3.1. Model Specifications

We used CFD to analyze the aspects of the air flow pertinent to installing a wind turbine between neighboring buildings. We considered five distances (w) between buildings, three types of building corners, and two building corner lengths (b), and analyzed 25 cases in total. Figure 7 shows the building corner shapes tested, which were classified into three types: normal, corner-cut, and rounded.

Figure 8 shows the dimensions of the analyzed buildings. Tables 3 to 5 list the names and dimensions of the three buildings. We used a square high-rise building with length (B) and width (D) of 40 m and height (H) of 120 m as our model for the CFD analysis. The smallest distance between buildings was 20 m, and we increased it in 10 m increments up to 60 m, analyzing five cases in total. Furthermore, we tested corner-cut and rounded corners of lengths 2 and 6 m. We used a building corner length (b) of 2 m in cases C1 and R1, which had cut and rounded corners, respectively. Cases C2 and R2, which have cut and rounded corners, respectively, had a building corner length (b) of 6 m. In each case, the numbers −1 to −5 represent the distances between the buildings, from 20 to 60 m.



(b) CaseC (c) CaseR

Fig. 7. Edge types used in the model

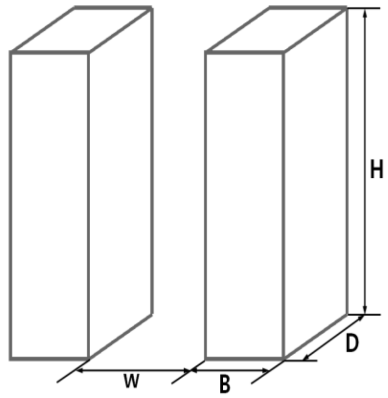


Figure 8. Dimensions used in the model

Table 3. Model size (Case N)

Case	B(m)	D(m)	H(m)	w(m)
CaseN1				20
CaseN2				30
CaseN3	40	40	120	40
CaseN4				50
CaseN5				60

Table 4. Model size (Case C)

Case	B(m)	D(m)	H(m)	b(m)	w(m)
CaseC1-1					20
CaseC1-2					30
CaseC1-3	40	40	120	2	40
CaseC1-4					50
CaseC1-5					60
CaseC2-1					20
CaseC2-2					30
CaseC2-3	40	40	120	6	40
CaseC2-4					50
CaseC2-5					60

Table 5. Model size (Case R)

Case	B(m)	D(m)	H(m)	r(m)	w(m)
CaseR1-1					20
CaseR1-2					30
CaseR1-3	40	40	120	2	40
CaseR1-4					50
CaseR1-5					60
CaseR2-1					20
CaseR2-2					30
CaseR2-3	40	40	120	6	40
CaseR2-4					50
CaseR2-5					60

3.2. Analysis Conditions

Table 6 shows the conditions tested in the CFD analysis, for which we used ANSYS Fluent R16.1. We used RNG k-ε for the turbulence model and standard wall function. The SIMPLE algorithm was used for the Pressure-Velocity coupling. Spatial discretization was taken as the Second order in Pressure and Momentum and First order in Turbulent Kinetic Energy and Turbulent Dissipation Rate. Regarding the domain size, the upstream region was set to 0.8 m (8B) and the downstream region was 1.5 m (15B), based on a building width (B) of 0.1 m. The length and height were modeled to be identical to those of the Chonbuk National University Wind Tunnel Laboratory, at 2.1 and 1.7 m, respectively. The total domain size was 2.4 m * 2.1 m * 1.7 m, and the total number of analysis meshes was approximately 600,000. Figure 9 shows the dimensions of the analysis domain. The reference wind speed (U_0) was 5 m/s at the highest floor of the building, and we used a category B ($\alpha = 0.22$) surface roughness. We input the experimental value of the incoming air flow into the CFD analysis, which varied according to height. This was assigned by a user-defined function, based on the input conditions summarized in Table 6. Figure 10 shows a graph of the incoming air flow distribution as a function of height.

Table 6. Computational fluid dynamics conditions

CFD Program	ANSYS Fluent R16.1	
Scale	1/400	
Velocity(U_0)	5m/s (Reference height = 120m)	
Exposure category	B (α =0.22)	
Domain size	2.4m * 2.1m * 1.7m	
Inflow condition (User Defined Function)	Turbulence kinetic energy (k)	$k = (U \cdot I)^2$
	Turbulence eddy dissipation (ε)	$\varepsilon = \sqrt{C_\mu k \frac{U_0}{H} \alpha (\frac{z}{H})^{(\alpha-1)}}$

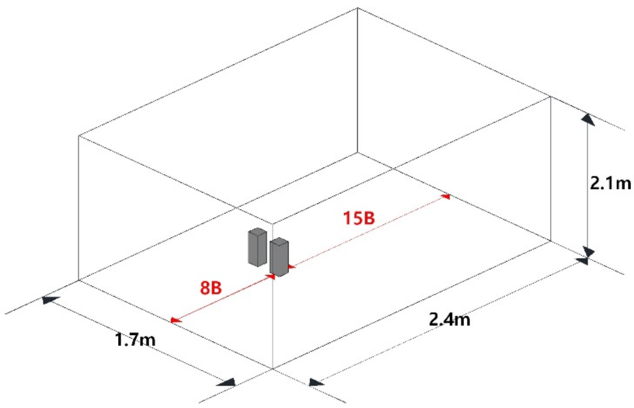
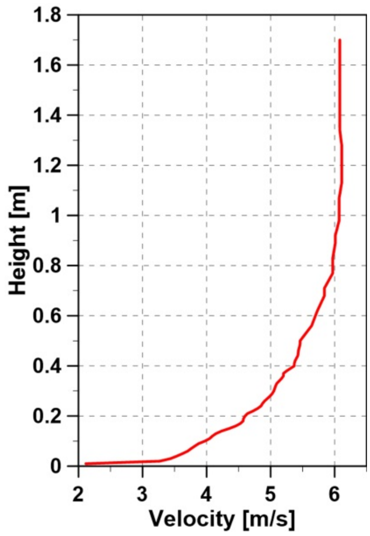
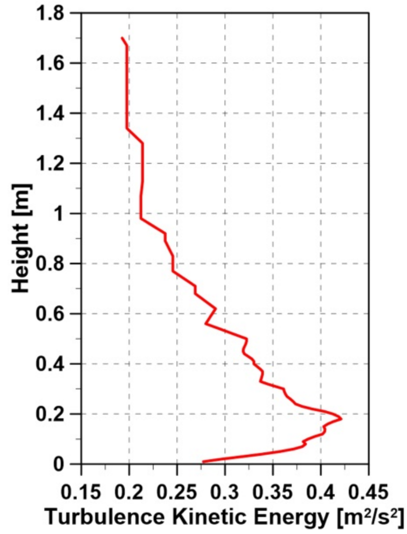


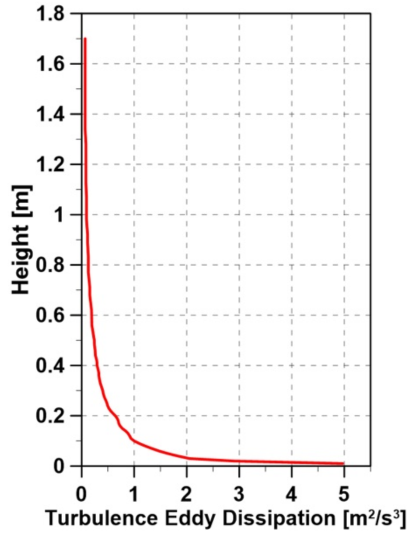
Fig. 9. Domain size



(a) Velocity



(b) Turbulence kinetic energy



(c) Turbulence eddy dissipation

Fig. 10. Distribution of inflow by height

3.3. Analysis Result

Figure 11 shows the modeling of the CFD analysis.

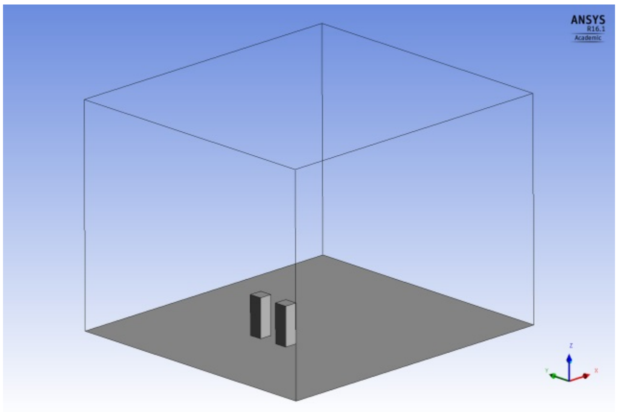
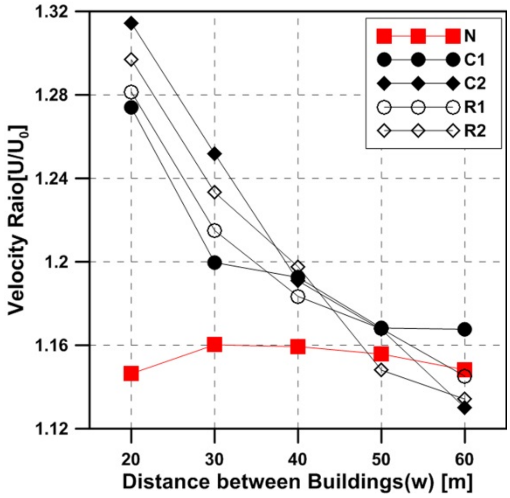


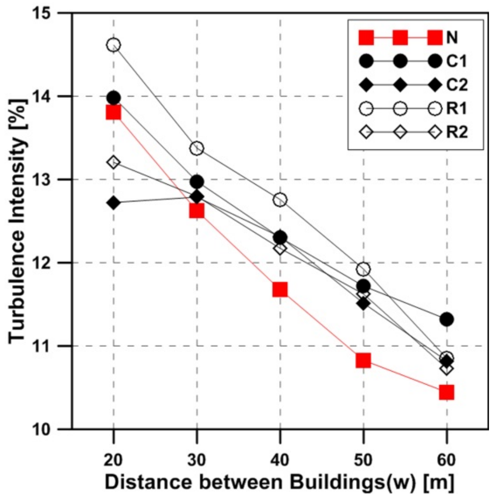
Fig. 11. Modeling of the CFD analysis

3.3.1. Variation with respect to corner shape (N, C, R)

Figure 12 shows the wind speed ratio and turbulence intensity as a function of the distance between the buildings and the building corner shape. The wind speed ratio for each position is the maximum average wind speed (U) at that position divided by the wind speed (U_0) measured at the height of the building of interest. We then measured the turbulence intensity at the position where the maximum average wind speed occurred. The wind speed ratio was higher than 1.0 at the point measured regardless of the corner shape, which means that the wind speed was always higher than the reference wind speed. The wind speed ratio measured in the case of the basic model, which had a square building corner, changed by approximately 1.15 when the distance between buildings was increased from 20 m to 60 m. This indicates that the wind speed increased by more than 15% with respect to the reference wind speed. The turbulence intensity decreased as the gap between the buildings increased. The turbulence intensity in Case N5 was approximately 24% lower than that of N1. The wind speed ratio also varied with the distance between buildings when the corners of the buildings were rounded or corner-cut. The effect of the corner shape increased as the distance between buildings increased. The wind speed ratio was approximately 13% higher when the corners were rounded than in the basic model, and approximately 15% higher when the buildings had cut corners. However, the wind speed ratio decreased as the distance between the buildings increased. The turbulence intensity increased by approximately 10% with respect to the basic model when the corner shape was changed. Among the corner shapes, the turbulence intensity was higher when we used a rounded corner shape than when we used a cut corner. Compared to the basic model, the turbulence intensity between buildings with rounded corners increased by approximately 10%, and it increased by up to 8% when we used corner-cuts.



(a) Velocity ratio



(b) Turbulence intensity

Fig. 12. Velocity ratio and turbulence intensity

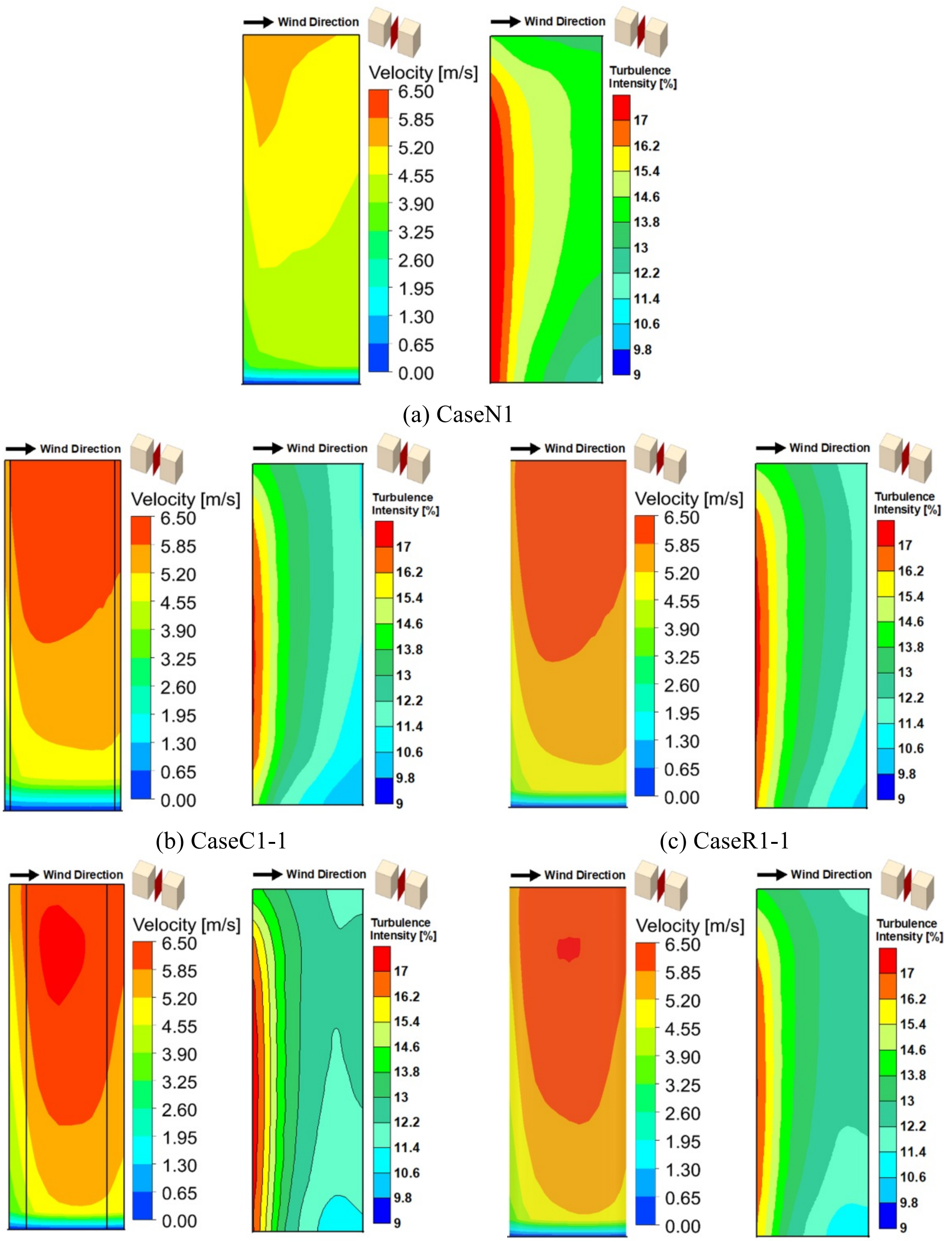
3.3.2. Variation with respect to the corner length(b) of each shape (C, R)

The wind speed ratio for corner-cut and round corners with different lengths increased by up to 4% for a length of 6 m, and by up to 4% when the length was 2 m. However, when the distance between the buildings was greater than 40 m, the wind speed ratio was 3% higher when the length of the corner was 2 m than when it was 6 m. The effect of the corner length also decreased as the distance between the buildings increased. The 2 m corner-cut had an effect at every distance between buildings, up to a distance of between 40 and 50 m.

The turbulence intensity increased when the length of the corner was 2 m, regardless of the gap between the buildings, and increased by up to 10% with respect to the basic model. In the case of corners of length 6 m, the turbulence intensity was approximately 7% more than in the basic model when there was 30 m or more between buildings. However, when the distance between buildings was 20 m, the turbulence intensity decreased by up to 8% compared to the basic model, regardless of the corner shape.

3.3.3. Analysis of the vertical air flow between buildings

Figures 13 and 14 show the results of the analysis of the vertical air flow between buildings for each corner shape tested. The wind speed ratio and turbulence intensity with respect to the corner shape and the distance between buildings, which we increased from 20 m (which had the largest deviation) to 60 m (where the difference in the wind speed ratio was small). The wind speed increased from the middle point upwards, which we attributed to the change in corner shape. Compared to Case N, the turbulence intensity decreased on the downstream side of the building. There was a marked increase in wind speed on the upper part of the building as the corner length increased from 2 to 6 m. This is considered to be the position at which we can minimize the effects of turbulence. This is important because, when a wind turbine is installed between buildings, the turbulence between the buildings may cause the wind turbine motor to rotate in an unstable manner. The larger the distance between the buildings, the less the position at which the wind speed increased varied. In general, the turbulence intensity decreased as the distance between the buildings increased.



(a) CaseN1

(b) CaseC1-1 (c) CaseR1-1

(d) CaseC2-1 (e) CaseR2-1

Fig. 13. Vertical profile of the wind flow for different shaped corners (w = 20m)

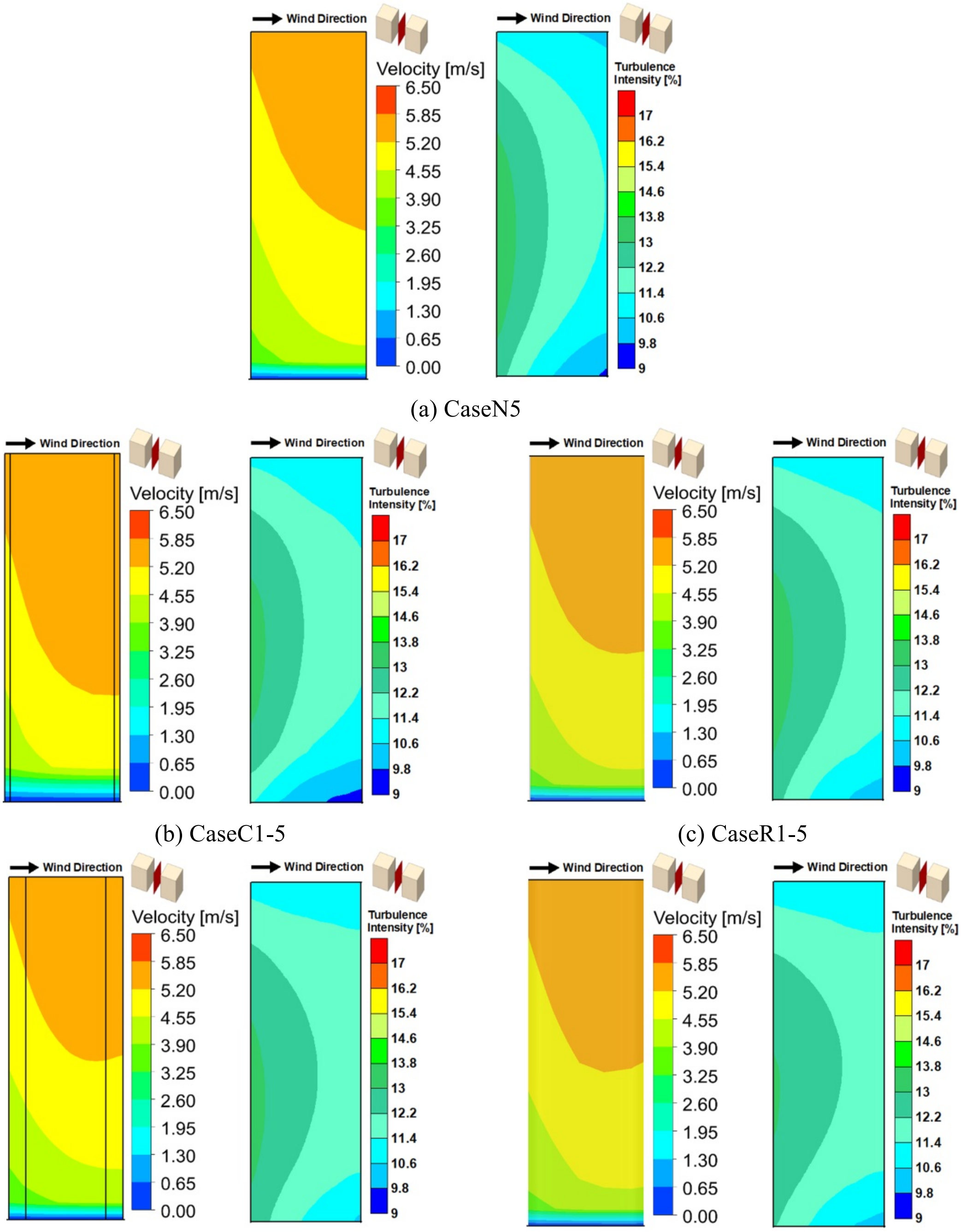


Fig. 14. Vertical profile of the wind flow for different shaped corners (w = 60m)

4. Conclusion

We carried out a CFD analysis to determine how varying the corner shape and distance between buildings affects the wind speed and turbulence intensity, with the aim of increasing the wind speed and, accordingly, the power generation efficiency of wind turbines installed between buildings. We obtained the following results from the CFD analysis:

1) When the distance between the buildings was 20 m, which was the smallest distance measured, the wind speed ratio increased by up to 13% in the case of rounded corners, and by up to 15% in the case of corner-cut corners. However, when the distance between buildings was 60 m, which was the largest distance measured, the wind speed ratio decreased by up to 1.2% compared to the basic model for rounded corners, and by up to 1.5% in the case of corner-cut corners. As such, we concluded that the wind speed ratio decreased as the distance between the buildings increased, regardless of the corner shape.

2) When the corner length was 6 m and the distance between the buildings was 20 m, the turbulence intensity at the position of the maximum wind speed ratio decreased by up to approximately 8% for corner-cut corners, and by up to approximately 4% for rounded corners, compared to the basic model. However, when the distance between the

buildings was increased to 30 m or more, the turbulence intensity increased by approximately 7%, regardless of the corner shape. When the corner length was 2 m, the turbulence intensity increased by up to approximately 10% compared to the basic model, regardless of the distance between buildings.

3) The maximum wind speed between the buildings, and the largest decrease in turbulence intensity, occurred at the top of the building (0.8H).

4) The wind speed ratio increased and the turbulence intensity decreased when the distance between the buildings decreased and the corner was long enough, regardless of the corner shape. As such, because wind turbulence between buildings can decrease the efficiency of wind turbines, we believe that their efficiency can be increased by positioning buildings close together and thus reducing the turbulence.

Acknowledgements

This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (NRF-2016R1A2B4015364)

References

[1] R.F. Smith and S. Killa, "Bahrain World Trade Center(BWTC): The First Large-scale Intergration of Wind Turbines in a Building", The Structual Design of Tall Special Buildings, vol. 16, pp. 429-439 , 2007

[2] O. Kaplan and M. Temiz, "The Analysis of Wind Speed Potential and Energy Density in Ankara", International Conference on Renewable Energy Research and Application, pp. 919-923, 2016

[3] Devashish and A. Thakur, "A Comprehensive Review on Wind Energy System for Electric Power Generation: Current Situation and Improved Technologies to Realize Future Development", International Journal of Renewable Energy Research, vol. 7, pp. 1786-1805, 2017

[4] S. Kahla, Y. Soufi, M. Sedraoui and M. Bechouat, "Maximum Power Point Tracking of Wind Energy Conversion System Using Multi-objective Grey Wolf Oprimization of Fuzzy-Sliding Mode Controller", International Journal of Renewable Energy Research, vol. 7, pp.926-936, 2017

[5] C. Smuthkalin, T. Murayama and S Nishikiazawa, "Evaluation of The Wind Energy Potential of Thailand considering its Environmental and Social Impacts using Geographic Information Systems", International Journal of Renewable Energy Research, vol. 8, pp. 575-584, 2018

[6] A.S. Bahaj, L. Myers and P.A.B. James, "Urban energy generation: Influence of micro-wind turbine output on electricity consumption in buildings, Energy and Buildings, vol. 39, pp. 154-165, 2007

[7] L. Lu and K.Y. Ip, "Investigation on the feasibility and enhancement methods of wind power utilization in high-rise buildings of Hong Kong", Renewable and Sustainable Energy Reviews, vol. 13, pp. 450-461, 2009

[8] B. Rezaie, E. Esmailzadeh and I. Dincer, "Renewable energy options for buildings: Case studies, Energy and Buildings, vol. 43, pp. 56-65, 2011

[9] D. Ayhan and Sa˘glam, "A technical review of building-mounted wind power systems and a sample simulation model, Renewable and Sustainable Energy Reviews, vol. 16, pp. 1040-1049, 2012

[10] D. Ayhan and S. Sa˘glam, "A technical review of building-mounted wind power systems and a sample simulation model", Renewable and Sustainable Energy Reviews, vol. 16, pp. 1040-1049, 2012

[11] H. Ishizaki and I.W. Sung, "Influence of adjacent buildings to wind", In: Proceedings of the Third International Conference Wind Effects on Buildings and Structures, pp. I.15.1–I.15.8, 1971

[12] B.G. Wiren, "A wind tunnel study of wind velocities in passages between and through buildings", In: Proceedings of the Fourth International Conference Wind Effects on Buildings and Structures, pp. 465–475, 1975

[13] T. Stathopoulos and R. Storms, "Wind environmental conditions in passages between buildings", Journal of Wind Engineering and Industrial Aerodynamics., vol. 24, pp. 19–31, 1986

[14] A.P. To and K.M. Lam, "Evaluation of pedestrian-level wind environment around a row of tall buildings using a quartile-level wind speed descripter", Journal of Wind Engineering and Industrial Aerodynamics. vol..54–55, pp. 527–541, 1995

[15] Ansys. Inc., ANSYS FLUENT 12.0 User's Guide, 2009.