

Investigation of Wind Pressure Coefficient Distribution on High-rise Building due to Different Shape Modifications using Computer Simulation

S. N. S. Abd Manap, S. N. Che Deraman*, N. Md Husain, W. N. F. Wan Hassan, S. A. Saad

Department of Civil Engineering, Kulliyyah of Engineering, International Islamic University Malaysia 53100 Gombak Selangor

ABSTRACT - High-rise buildings are rapidly flourishing in various regions due to accelerating industrialization and increasing population density. The limited availability of land has driven this trend. Nonetheless, wind pressure significantly influences the design and structural stability of these tall structures. Therefore, it is crucial to account for wind loads when designing high-rise buildings, particularly in areas prone to strong winds, such as coastal regions, where winds can exert substantial force on the structure. This study focuses on analyzing wind pressure coefficient distribution around high-rise buildings by examining building shapes modifications and wind flow behavior through Computational Fluid Dynamics (CFD) simulations. The suggested tapering ratio and aerodynamic modification factor is 10% was applied to this study. The findings reveal that the maximum negative pressure coefficient (CP) or suction observed is -1.630, associated with the M4 model, which features a tapering design with rounded corners. This high suction is attributed to wind flow behavior for instance the flow separation and reattachment at the edge of the building's top (Zone 2). Except for the M4 model, which has a rounded edge on the leeward side, the thickness of the recirculation eddies is nearly uniform across all models. The M4 model's design results in a smaller wake region and reduced turbulence on the leeward side. Therefore, the M4 shape is the most effective in minimizing the impact of wind on the building structure.

ARTICLE HISTORY

Received : 7 April 2025
Revised : 20 May 2025
Accepted : 9 June 2025
Published : 29 December 2025

KEYWORDS

High-rise building
Wind pressure
Computational fluid dynamics
Computer simulation
Windstorm

1. INTRODUCTION

Tall structures are frequently exposed to wind forces, necessitating cautious consideration of wind interaction in structural engineering and architecture [1]. Wind pressure can greatly affect the design and stability of high-rise buildings, especially in areas prone to strong winds [2]. Engineers must thoroughly evaluate the wind load that the structure encounters during the design and construction phases. Additionally, Mou [3] highlighted that a building's orientation can influence wind flow and behavior. Modifying a building's aerodynamic shape can significantly mitigate wind-induced responses in wind-sensitive structures, particularly tall buildings. Numerous researchers have explored the impact of building shape on wind-induced responses and wind loads on tall structures [4].

Bharat and Ahmed [5] highlighted that airflow patterns around buildings are largely influenced by their geometries. Additionally, the external form of tall buildings plays a crucial role in the development of wind loads, as the flow separates and reattaches around bluff bodies. Thus, optimizing the aerodynamic design of a building's external shape can reduce wind stresses on high-rise buildings [6]. Merrick and Bitsuamlak [7] found that a circular building shape is particularly effective in minimizing torsional loads. Their study, which compared square, circular, triangular, rectangular, and elliptical shapes using a boundary layer wind tunnel (BLWT), revealed that the triangular section is a poor choice for structures sensitive to torsion.

Bairagi and Dalui [8] examined the effects of wind loads on pressure distribution around square and setback-shaped tall buildings. Kumar et al. [9] found that ensuring the functional performance of flexible structures and controlling wind-induced motion in high-rise buildings involve adding structural alternatives and dampening systems. A more effective approach includes aerodynamic modifications such as sculpted building tops, horizontal and vertical openings, and changes to the cross-sectional shape and corner geometry. These modifications alter wind flow patterns and reduce the impact of wind on high-rise buildings. Previous research has primarily focused on individual modifications and their effects on basic models. This study, however, investigates the impact of building shapes on wind pressure distribution around high-rise buildings using Computational Fluid Dynamics (CFD) simulation.

2. METHODOLOGY

2.1 Flow Field

A schematic view of the building model used for the tests is shown in Figure 1. An isolated high-rise building for dimensions 3000 x 3000 x 6300 mm was placed such that it was perpendicular to the approaching flow. The model could be given one of five (5) different building shapes, specifically M1-basic (without tapering ratio), M2-tapered (with 10% tapering ratio), and three (3) composite modification models namely M3-chamfered (with 10% tapering ratio and 10%

*CORRESPONDING AUTHOR | S. N. C. Deraman | ✉ snoratikah@iiu.edu.my

chamfered), M4-rounded (with 10% tapering ratio and 10% rounded), and M5-recessed (with 10% tapering ratio and 10% recessed). The tapered model has a 10% tapering ratio as suggested by [10]. All five (5) models are designed to represent a high-rise building with a total height of 63 m and a base plan area of 30 m by 30 m. All models are scaled down by a ratio of 1:10 to decrease the computational time.

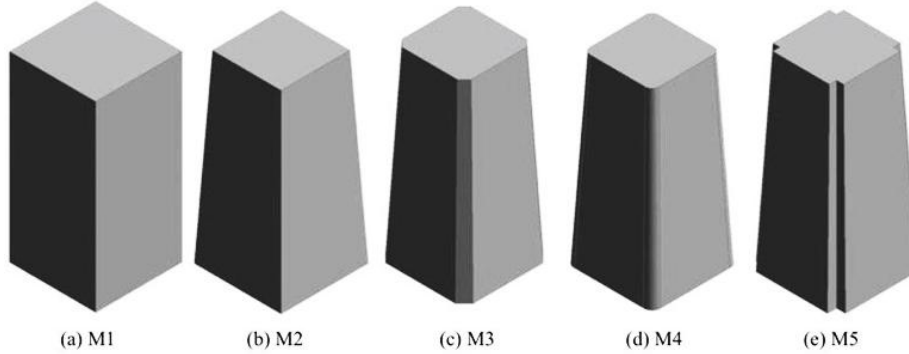


Figure 1. Isometric view of models (a) M1-basic (b) M2- tapered (c) M3- chamfered (d) M4-rounded and (e) M5-recessed

2.2 Computational Method

ANSYS FLUENT 18.5 commercial software was used to perform the steady-RANS computations based on a control volume approach for solving the flow equations. The turbulence models, namely, Reynold’s-averaged Navier–Stokes (RANS) turbulence model k-epsilon ($k-\epsilon$) were used to generate the wind environment in this study. All of the transport equations were discretized using a second-order upwind scheme. Pressure interpolation was of the second order. The SIMPLE algorithm was used for pressure velocity coupling.

Boundary conditions were set using the best practical guidelines [11-12]. The important parameters for the input at the inlet are roughness height, K_s , and roughness constant, C_s . The value of C_s lies in the range of 0 to 1.0 [13]. For uniform sand-grain roughness, the value of the roughness constant, C_s was taken as 0.5 as suggested by [14]. Furthermore, a flat terrain was represented by a power-law profile of 1/7. Since this simulation used the scale-down models with a ratio of 1:10, the roughness length (z_0) also needs to be scaled to $z_0 = 0.035/10$ as recommended by [18]. The boundary conditions were implemented on each surface of the domain. In this case, six (6) boundary conditions were set to the surface of the domain including the model. The wind flow at the inlet was generated using the User Defined Function (UDF) file to create the real wind flow pattern in the simulation using the C-programming language. The UDF enabled the generation of wind flow conditions based on input parameters such as wind speed, wind direction, and turbulence intensity. Symmetry boundary conditions were imposed at the sides and the top of the domain, implying zero normal velocity and zero gradients for all the variables at these boundaries as stated by [11-12,15]. At the outlet from the domain, zero static pressure was imposed.

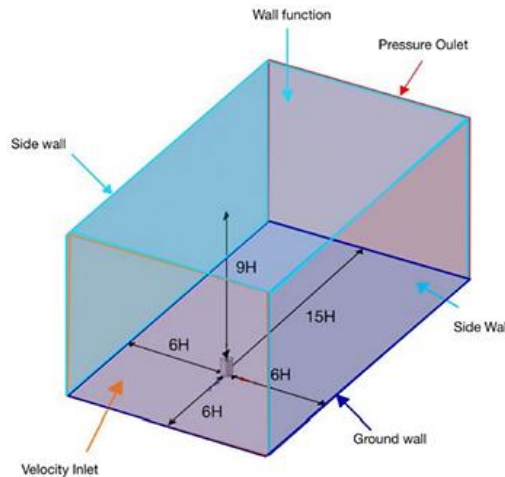


Figure 2. Computational domain and boundary condition for 3D view [11]

The computational domain covered a volume of $12H \times 21H \times 9H$ as shown in Figure 2 as recommended by [16]. The recommendations on the dimension of the computational domains may differ between research but it is important to make sure the blockage ratio must be less than 3% [3,17,10] to eliminate the flow obstacle effect on the inflow and outflow boundary conditions. The computational grid consisted of 728337 cells for the domain of the basic model. Tetrahedron meshing is utilized, and all borders are inflated. Every domain face and edge has been appropriately resized as shown in Figure 3. The grid was based on a grid sensitivity analysis that will be described in the validation model.

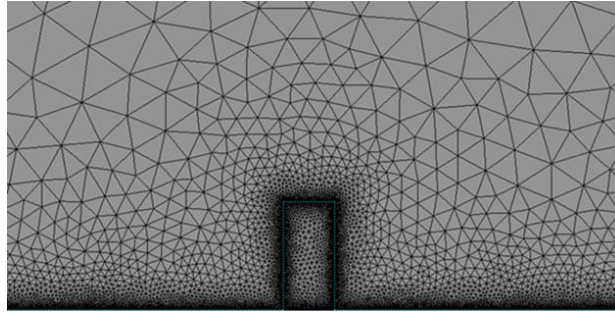


Figure 3. The utilization of tetrahedron meshing for a basic model

3. RESULTS AND DISCUSSION

The current computational study was validated with the previous study by Isdris et al. [19] using a basic model. The validation process involved grid sensitivity analysis and wind flow pattern. The numerical simulation of composite modification high-rise buildings involved the parametric study of the change of shapes.

3.1 Validation of the Model

The wind flow around an isolated basic high-rise building was qualitatively validated by visualizing the streamlines. In the quantitative analysis, the pressure coefficient C_p from the current study were compared with the C_p by [19].

3.1.1 Grid sensitivity analysis

This analytical study was validated by the previous researcher who conducted the square model as the basic structure adopted by [19]. To quantify the impact of grid resolution on the solution, Franke et al. [20] suggested conducting at least three (3) systematically refined grids is recommended. In this study, three (3) different grid sizes namely coarse mesh, medium mesh and fine mesh with the number of elements are 728337, 915925 and 1237101, respectively were generated. The selection of coarse mesh was due to the number of elements gained by [19] for fine grid size is 727997 while the current study gets 728337. The grid sensitivity analysis is important to investigate the level of accuracy with respect to the computational time in order to produce the most economical model as mentioned by Isdris et al. [19]. Furthermore, the creation of the standing vortex and the placement of the stagnation point are comparable in the streamlines pattern result shown in Figure 4.

3.1.2 Wind flow pattern

When compared to the previous researcher, the presence of the vortex area at the leeward face is slightly different. While in the current study, it occurred very close to the leeward face and had a huge radius of vortex while the vortices produced by [19] has a tiny radius of the vortex at the bottom surface that is quite far from the leeward face. This might have happened because the fundamental model's meshing structure was generated differently. Moreover, Isdris et al. [19] obtained a C_p of 0.73 while the current study gained the C_p of 0.65 with a percentage difference of 12%. The parameter utilized by [19] has been set up to produce the result for every model in this study, despite the fact the validation result for the basic model is a bit different from the previous researcher.

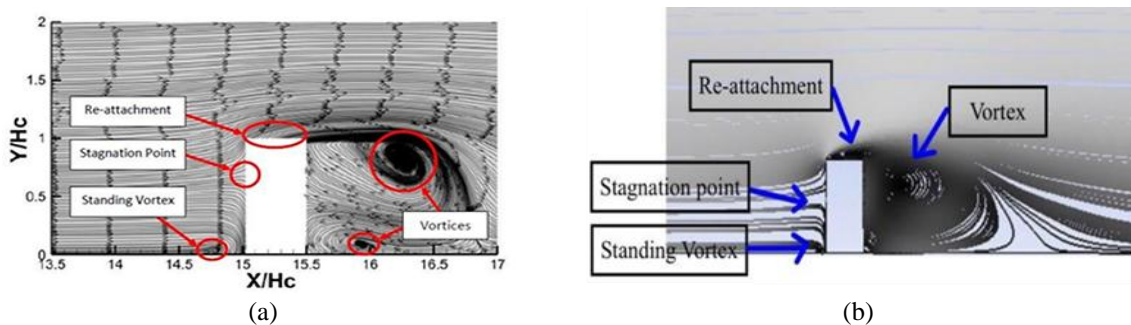


Figure 4. Streamline pattern of the basic model by (a) Isdris (2019) (b) current study

3.2 Effects of Building Shapes on Airflow Properties on the Building

The effect of modification of building shapes can be illustrated using contour and distribution graph as well as streamlines.

3.2.1 Wind pressure coefficient, C_p contour

Figure 5 compares the spatial distributions of static pressure, represented by the pressure coefficient (C_p), on the walls and roof surfaces of five different high-rise building shapes: M1-basic, M2-tapered, M3-chamfered, M4-rounded, and M5-recessed. The maximum C_p value indicates locations on the building's surface where the pressure is significantly

higher than the surrounding air. Positive pressure was observed on all parts of the windward side, causing high pressure in front of the wall due to the direct impingement of the flow onto the building and the façade behind it, resulting in a large stagnation area. Conversely, negative pressure (suction) developed at the edges and sides of the building due to the sharp edges, as shown by the blue and green colors. The findings suggest that sharp edges (M1, M3, M5) cause high suction values at the sides of the building. Negative C_p values represent areas where the wind pulls or sucks the building in those directions.

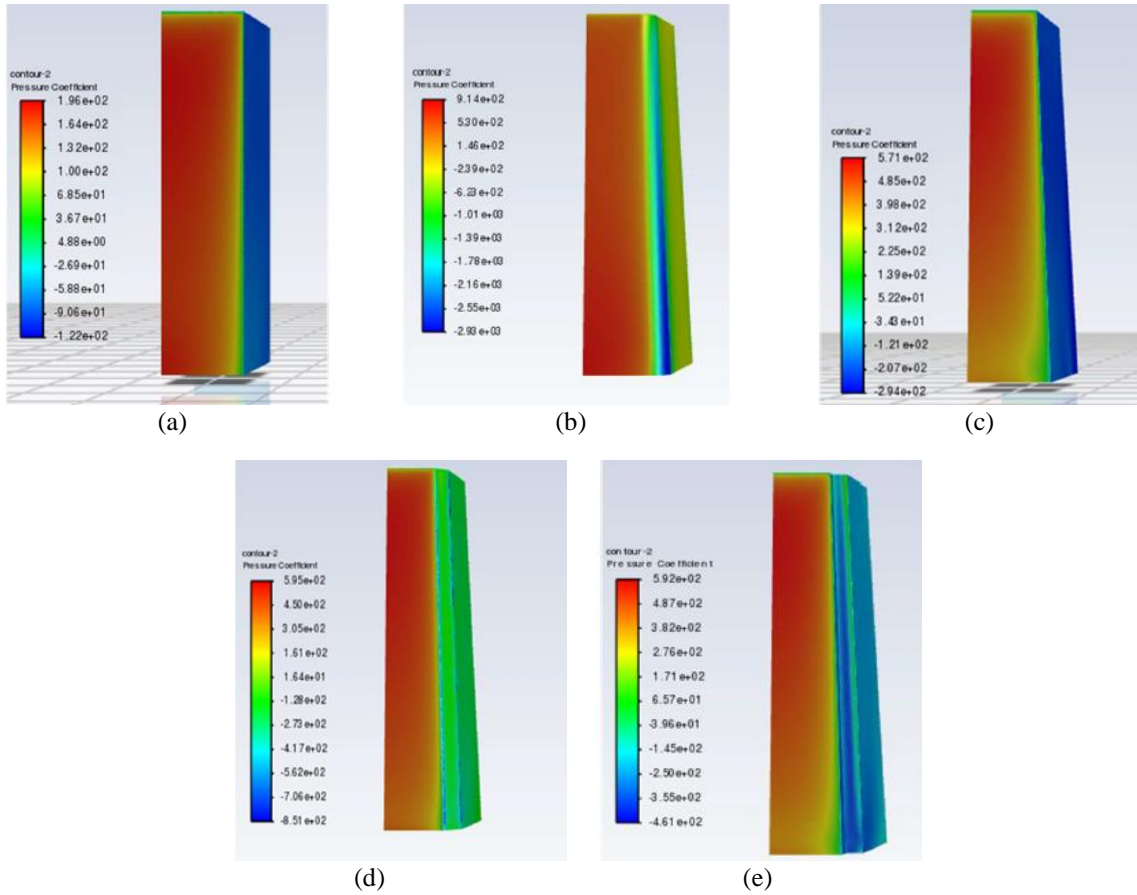


Figure 5. Pressure coefficient contour on the walls and roof surfaces of building a) M1-basic b) M2-tapered c) M3-chamfered, d) M4-rounded, and d) M5-recessed

3.2.2 Wind pressure coefficient, C_p distribution graph

The pressure coefficient (C_p) is a non-dimensional parameter that describes the loading effect on buildings and other structures [21]. Figure 6 compares the C_p profiles for different building shapes across three zones: Zone 1 (windward wall), Zone 2 (top of the building), and Zone 3 (leeward wall). The distribution pattern of the pressure coefficient is similar for all models. The C_p values were extracted along the building's mid-axis. In Zone 1, the C_p starts with positive values, with M4 showing the highest value of 1.790, indicating that this model experiences more intense pressure changes and wind forces than the others [22].

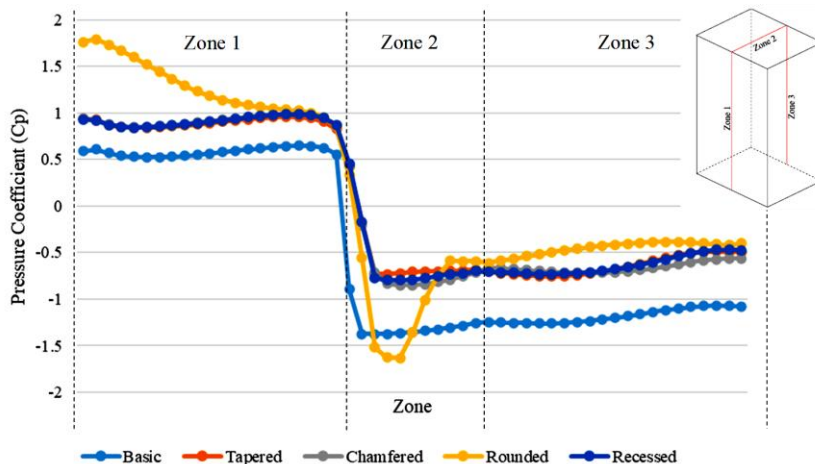


Figure 6. Comparison of C_p profiles for different shapes of buildings according to the zones

As the C_p approaches Zone 2, there is a sudden drop in positive pressure, with the minimum values located near the edge of the building top due to reattachment, resulting in negative pressure (suction). The maximum suction occurs near the edge of the building top, with a value of -1.630 for the M4 model. The negative C_p decreases as it approaches the leeward wall in Zone 3, eventually becoming constant, which weakens the suction effect.

3.2.3 Wind streamlines

Figure 7 illustrates the pattern of streamlines along the vertical profile for all the building models. Stagnation pressure was generated on the windward wall as the wind flow impinged on the building models. A similar pattern was observed for nearly all the models, except for M4-rounded, due to its rounded edges [23]. The windward wall deflected the wind flow, leading to circulation, as highlighted in the red rectangle. This recirculation results in high pressure in the windward region, as confirmed by the C_p distribution graph in Figure 6. The streamlines that separated from the windward edges reattached to the top of the building models; a phenomenon known as reattachment. In this scenario, separation zones primarily involved the top of the building, from the leading edge to the reattachment point. Suction occurred within the separation zone, and upon reattachment, the C_p recovered from its low values in these zones. The reattachment zone experienced much less suction than the separation zone.

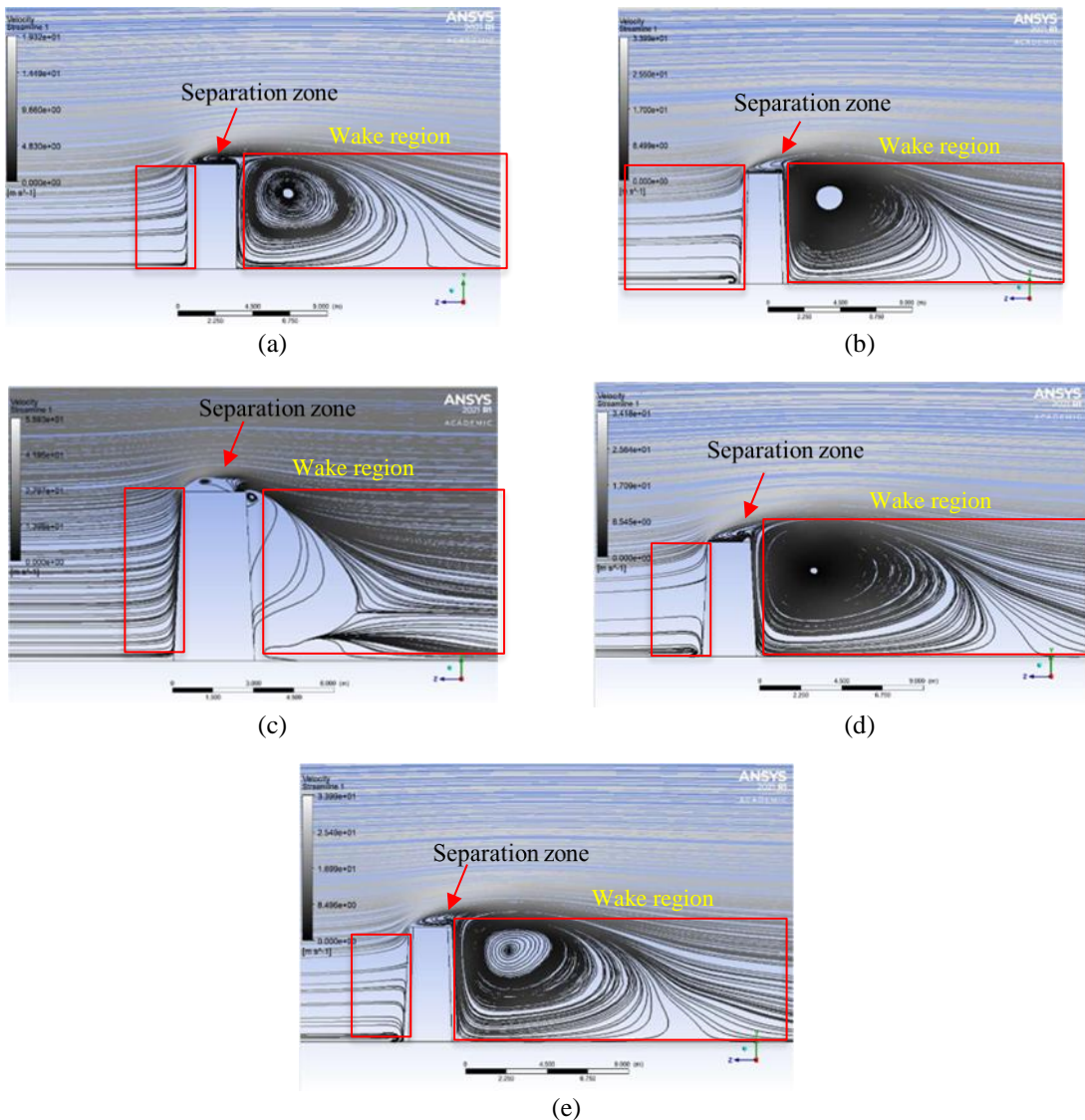


Figure 7. Streamline on the walls and roof surfaces of building (a) M1-basic (b) M2-tapered (c) M3-chamfered, (d) M4-rounded, and (e) M5-recessed

However, the recirculation behind the building models displayed different patterns for each model, as highlighted in the red rectangle. The thickness of the recirculation eddies was almost the same for all models except for M4, due to its rounded edge. M4 had a smaller wake region, resulting in less turbulence. In the wake region, the air pressure is below ambient pressure, generating strong turbulence that causes fluctuations in the negative C_p (suction).

4. CONCLUSION

This paper focused on the variation of wind pressure coefficients on different building shapes, particularly those with tapering and corner modifications. The analytical simulations revealed that the highest-pressure coefficient occurs on M4's windward face, due to the ineffectiveness of the rounded corner cut on the building. It was found that a 10% corner modification is sufficient to reduce the pressure coefficient on high-rise buildings. M4 also had a smaller wake region, but the leeward side developed a wake region that led to high turbulence. In this region, the air pressure is below ambient pressure, generating strong turbulence that causes fluctuations in the negative C_p (suction). Overall, building geometry and shape significantly impact how wind interacts with its surface. These elements must be strategically designed to withstand wind loads, ensuring the building's stability and safety. Wind, as a real phenomenon, affects neighboring buildings before impacting a structure's façade. This study demonstrated that a rounded edge provides a beneficial effect on wind flow around high-rise buildings

ACKNOWLEDGEMENTS

The authors would like to express their deep gratitude to the staff from the Department of Civil Engineering and Department of Mechanical Engineering for their help in finishing this study, International Islamic University Malaysia for providing the facilities.

FUNDING

The financial support of the Research Management Centre (RMC) Grant 2020 (RMCG20-071-0071) from International Islamic University Malaysia.

CONFLICT OF INTEREST

The authors declare no conflicts of interest.

AUTHORS' CONTRIBUTION

The author's contribution role(s) following, such as:

Siti Nur Solehah Abdul Manap (Methodology; Investigation; Formal analysis; Writing - original draft)

Siti Noratikah Che Deraman (Software; Data curation; Resources; Supervision)

Nadiah Md Husain (Writing - review & editing)

Wan Nur Firdaus Wan Hassan (Writing - review & editing)

Siti Asmahani Saad (Writing - review & editing)

AVAILABILITY OF DATA AND MATERIALS

The data used to support the findings of this study are included within the article.

ETHICS STATEMENT

'Not applicable' in this section.

REFERENCES

- [1] G. Vinayagamurthy and K. B. Rajasekarababu, "Experimental and computational simulation of an open terrain wind flow around a setback building using hybrid turbulence models," *Journal of Applied Fluid Mechanics*, vol. 12, no. 1, pp. 145-154, 2018.
- [2] R. Yanalagaran, N. I. Ramli, and P. J. Ramadhansyah, "Overview of monsoon induced coastal erosion disaster in Peninsular Malaysia based on Mass-Media Reports," in *IOP Conference Series: Earth and Environmental Science*, 2019.
- [3] B. Mou, B. J. He, D. X. Zhao, K. W. Chau, "Numerical simulation of the effects of building dimensional variation on wind pressure distribution," *Engineering Applications of Computational Fluid Mechanics*, vol. 11, no. 1, pp. 293-309, 2017.
- [4] L. Wang, and W. Zhang, "The influence of chamfered and rounded corners on vortex-induced vibration of super-tall buildings," *Applied Science*, vol.13, no. 2, pp. 1049, 2023.
- [5] A. Bharat, and A. S. Ahmed "Effects of high-rise building complex on the wind flow patterns on surrounding urban pockets," *International Journal of Engineering Research and Development*, vol. 4, no. 9, pp. 21-26, 2012.
- [6] R. R. Ahirwar "Experimental study of wind-induced pressures on tall buildings of different shapes," *Wind and Structure*, vol. 31, no. 5, pp. 441-453, 2021

- [7] R. Merrick, and G. Bitsuamlak, "Shape effects on the wind-induced response of high-rise buildings," *Journal of Wind and Engineering*, vol. 6, no. 2, pp. 1-18, 2009.
- [8] A. K. Bairagi, and S. K. Dalui "Distribution of wind pressure around different shape tall building," in *Lecture Notes on Multidisciplinary Industrial Engineering*, Springer Nature, vol. Part F252, pp. 31–38. 2020.
- [9] R. Kumar, V. K. Mewada, M. T. Scholar, "Analysis of aerodynamic modification of the shape of the tall structure-A review," *International Journal of Advance Research and Innovative Ideas in Education*, vol. 7, no. 6, pp. 576-581, 2021.
- [10] M. M. Khan, and A. K. Roy, "CFD simulation of wind effects on industrial RCC chimney," *International Journal of Civil Engineering and Technology*, vol. 8, no. 1, pp. 1008–1020, 2017.
- [11] J. Franke, A. Hellsten, H. Schlünzen, and B. Carissimo, "Best practice guideline for the CFD simulation of flows in the urban environment," *COST Action 732, Quality assurance and improvement of microscale meteorological models*. 2007.
- [12] Y. Tominaga, A. Mochida, R. Yoshie, H. Kataoka, T. Nozu, M. Yoshikawa, et al., "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, pp.1749-1761, 2008.
- [13] Fluent "ANSYS Workbench User's Guide," *ANSYS Incorporated*, USA. 2010.
- [14] W. Zhang, and P. P. Sarkar, "Influence of buildings on tornado-induced wind loads of a low-rise building," *Procedure for the 11th Americas Conference on Wind Engineering*, Puerto Rico. 2019.
- [15] Y. Tominaga, "Flow around a high-rise building using steady and unsteady RANS CFD: Effect of large-scale fluctuations on the velocity statistics," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 142, pp. 93-103, 2015.
- [16] Y. Kim, and J. Kanda "Characteristics of aerodynamic forces and pressures on square plan buildings with height variations," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 98, no. 8–9, pp. 449–465, 2010.
- [17] F. Baetke, and H. Warner "Numerical simulation of turbulent flow over surface-mounted obstacle with sharp edges and corners," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 35, pp. 129-147, 1990.
- [18] H. Irtaza, M. A. Javed, and A. Jameel, "Effect of wind pressures by variation of roof pitch of low-rise hip-roof building," *Asian Journal of Civil Engineering*, vo. 16, no. 6, pp. 869-889, 2015.
- [19] S. R. M. Isdris, S. S. Zaini, M. H. H. Ishaik, M. S. A. Aziz, and N. A. Rahman, "Numerical investigation on the distribution of pressure coefficients of modified building shapes," *Pertanika Journal of Science and Technology*, vol. 31, no. 6, pp. 3139–3155, 2023.
- [20] J. Franke, C. Hirsch, A. G. Jensen, H. W. Krus, M. Schatzmann, P. S. Westbury, et al., "Recommendations on the use of CFD in wind engineering," *COST Action C14: The Use of CFD in Predicting Pedestrian Wind Environment*, 2004.
- [21] Z. Zhao, "Wind flow characteristics and their effects on low-rise buildings," Ph.D. Thesis, Faculty of Texas Tech University, 1997.
- [22] S. N. C. Deraman. "Distribution of pressure coefficient along the external profile of rural house exposed to windstorm in Malaysia using Computational Fluid Dynamics (CFD)," Ph.D. Thesis, Universiti Sains Malaysia, Malaysia, 2019.
- [23] J. J. A. Amin, and A.K. Ahuja "Aerodynamic modifications to the shape of the buildings: A review of the state-of-the-art," *Asian Journal of Civil Engineering (Building and Housing)*, vol. 11, no. 4, pp. 433-450, 2010.