iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 2026 Issue: Conference Month of publication: May 2026

DOI: https://doi.org/10.22214/ijraset.2026.83380

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue V May 2026- Available at www.ijraset.com
National Conference on Integrated Approaches to Sustainable Innovation 2026 (IASI 2026)

Effect of Combined Corner Rounding on Wind-Induced Responses of Irregular Plan Shaped Buildings

Shanku Mandal
Department of Civil Engineering, OmDayal Group of Institutions

Abstract: Present building construction trends mostly rely on architectural considerations and functional necessities. The variation in wind characteristics is primarily controlled by the building's plan shape, height, and the geometry of its outside surface. The present study elucidated the methodology to mitigate wind effects on structures with irregular configurations. The rounded cut has been integrated into the corner of the U-shaped building. The curtailment of corners diminishes wind effect but concurrently decreases the functional floor area of the structure. This study employs a combination method at the corners to mitigate wind impact while preserving substantial floor area. The various models are examined utilizing Computational Fluid Dynamics (CFD) at wind angles of 0° and 90°. The alteration in wind flow patterns has been noted as the cuts at multiple points induce irregularity. The pressure variation has been examined on the surfaces of the models. An increase in the pressure coefficient has been seen at the cut corners in contrast to the uncut corners. So, the walls of the corners must be constructed with precision. The wind forces on the no-cut models have significantly decreased with the cut, although it is unnecessary to cut corners from top to bottom. The usable floor area of the no-cut variant is considerably diminished compared to fully cut counterparts. This study demonstrated a strategy to mitigate wind impact while utilizing plenty available space to enhance the overall economics of the building.
Keywords: Wind Characteristics, Computational Fluid Dynamics, Wind Angle, Pressure Coefficient, Wind Force.

I. INTRODUCTION

The modern trends of building construction mainly depend on both architectural aspects and functional requirements. The variation in wind characteristics influenced mostly by the plan shape, height and the geometry of the outer surface of a building. The irregular plan configuration leads to attracts more wind force than standard plan configuration. To overcome this various wind force resistant techniques has been adopted. Various researchers have incorporated the updating of corners on some regular shapes. Naudascher et al. (1981) minimizes the flow-induced vibration of the square structure by adopting different plates and cuts at the corners. Kwok (1988) explained the influence of the building shape on the wind responses of a tall building. The effect of tapering on a square shape and the change in dynamic responses due to tapering has been explored by Kim and You (2002). Mukherjee et al. (2014) show the pressure differences in Y shape building due to the wind. Elshaer et al. (2017) enhance the performance of wind by changing the sharp corner of a square building with different types of cuts. Kim et al. (2017) considered various complicated shape and presented the comparison of wind impact on them. Sharma et al. (2019) applied Detached-eddy simulation on set-back and tapered square shape to understand the variation in aerodynamic responses. Zheng et al. (2020) explored the pressure on the surface by CFD simulations and also compared the RANS and LES results. Feshalami and He (2021) studied the influence of divergence ratios ranging from 0.2 to 1.2 on the flow behavior around a confined square cylinder at a Reynolds number of 100. The analysis focused on aerodynamic characteristics such as drag and lift coefficients, Strouhal number, and flow patterns through vorticity contours and streamline visualization. Cui et. al. (2023) investigated the shape coefficients, fluctuating wind pressure coefficients, and base bending moment coefficients of buildings under various wind direction angles and spacing ratios. The maximum values of these parameters are also statistically analysed and their corresponding working conditions. Kumar and Verma (2023) examined the aerodynamic behavior of a 69 m tall ‘+’ shaped high-rise building using Computational Fluid Dynamics (CFD) simulations in ANSYS CFX under different wind incidence angles ranging from 0° to 75°. Yadav and Roy (2024) investigated the aerodynamic behavior and structural responses of prismatic and tapered high-rise buildings under extreme wind conditions using CFD simulations.

They analysed the effects of turbulent inflow and varying wind incidence angles on wind-induced forces, pressure distribution, vortex formation, and moments. Gan et. al. (2024) evaluated wind flow patterns around rows of high-rise residential buildings arranged in convex and concave curvilinear forms. This study gives special attention to the effect of different central angles. Parametric building models were developed using Rhino Grasshopper, and their aerodynamic performance was evaluated through experimentally validated CFD simulations. Singh et. al. (2024) studied the effectiveness of aerodynamic corner modifications in reducing wind loads on tall square buildings using CFD simulations in ANSYS Fluent. Meena et. al. (2024) carried out a numerical investigation on regular and irregular tall building models subjected to wind loads using ANSYS CFX. The study evaluated wind performance through pressure contours and mean pressure distributions at different heights and find that rectangular chamfered buildings and Y-shaped models with corner cuts exhibited better wind resistance compared to other plan configurations. Rezaei et. al. (2025) concluded that wind turbine performance is highest in low-roughness terrains, such as coastal areas, while the maximum pressure coefficient decreases in rougher terrains. The study also showed that a VAWT installed on a dome-shaped building generates 50.7% more power than one placed on a cubic building of the same height. Mazarakou et. al. (2025) investigated the wind-induced response of tall buildings with different structural forms using finite element analysis. This study compared maximum displacement of twelve building models with varying heights and configurations under equivalent static wind loads. Gora et. al. (2025) evaluated the impact of climate change on the wind-induced performance of a 151 m tall building under historical and future wind conditions using probabilistic vibration analysis. The results showed that structural acceleration responses may increase significantly in future scenarios, particularly for higher return period wind events. Wang et. al. (2026) studied a dense cluster of tall buildings in the City of London within a 700 m diameter area using three methodologies to analyze flow and dispersion effects. Real-world sonic anemometer data at 191 m height were used to identify near-neutral stationary urban boundary layer flow and evaluate integral time and length scales. Yadav and Roy (2026) evaluated the wind-induced vibration characteristics and dynamic responses of a tall X-shaped building and compared them with a setback-modified configuration.

The wind impact on irregular shapes is not widely explored in past studies. In this study, the combined corner rounded configuration has been adopted on U type model to investigate its aerodynamic performance through the analysis of wind flow patterns, drag and lift coefficients, mean pressure coefficients, and pressure contour distributions.

II. DETAILS OF BUILDING MODELS

Figure 1 illustrates the different model configurations used in the study, all developed at a geometric scale of 1:300. Each model maintained identical overall dimensions, including a length of 250 mm, height of 400 mm, limb length of 50 mm, and width of 150 mm, while variations were introduced through different rounded-cut arrangements. Type 1 consisted of a sharpcornered model without any cut along the entire height, whereas Type 2 incorporated a 25% corner cut throughout the full height. In Types 3, 4, and 5, the corner cuts were introduced only above heights of 100 mm, 200 mm, and 300 mm from the ground respectively, and all models were examined under wind incidence angles of 0° and 90°.

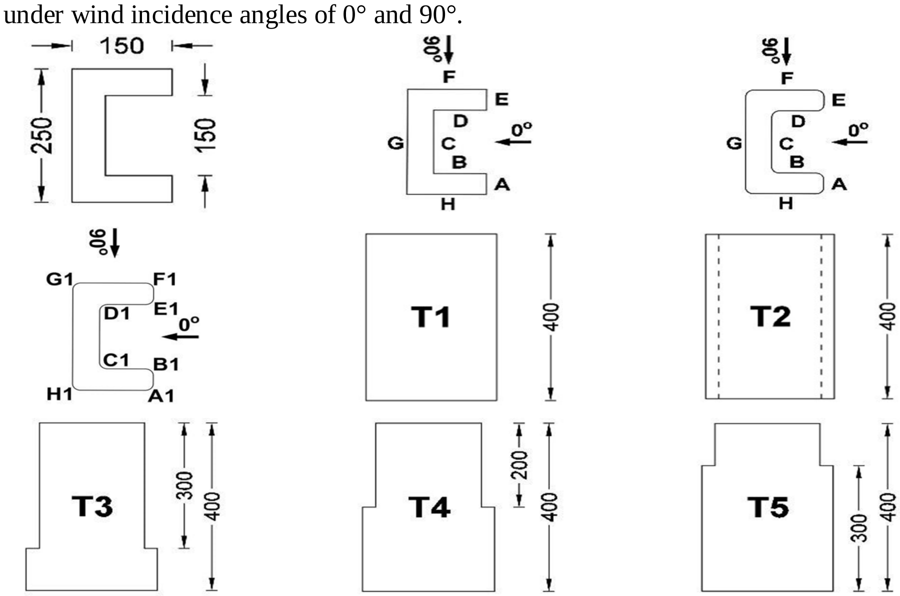


Fig. 1 The combined cut models and the wind angle and face-name details

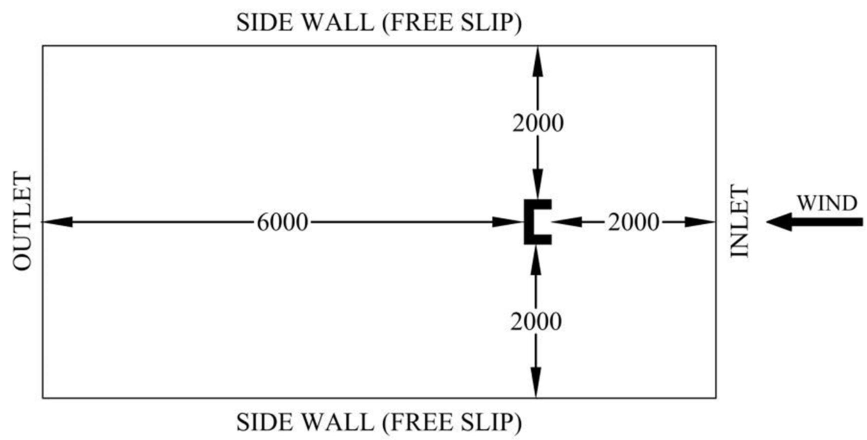
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue V May 2026- Available at www.ijraset.com
National Conference on Integrated Approaches to Sustainable Innovation 2026 (IASI 2026)

III. SOLUTION METHODOLOGY

The CFD module of ANSYS CFX was employed to investigate wind flow behavior. The numerical simulation process involved creating the computational domain, generating an appropriate mesh, applying suitable boundary conditions, and subsequently evaluating the aerodynamic responses of the models.

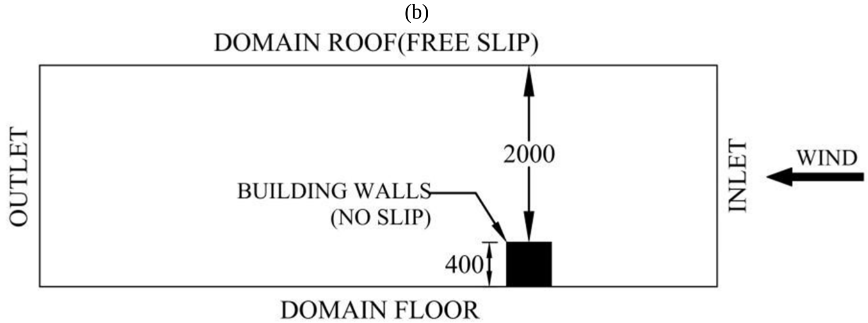
A. Computational Domain and Boundary Conditions

For the analysis, a domain has been taken with 2000 mm, 6000 mm, 2000 mm and 2000 mm distances (Figure 2) from the model in case of the top wall, downstream wall, upstream wall and sidewalls respectively as done by Mukherjee et al. (2014). The power-law equation formed the boundary layer flow and the exponent value has been considered as 0.133. The velocity at the entrance has been taken as 10 m/s. The free slip type boundary has been taken for top and sidewalls. The no slip type is taken for the building and ground surface.



(a) Plan view of the domain

(b)



(c) Elevation view of the domain

Fig. 2 The computational domain details

B. Generation of mesh and mesh refinement study

Figure 3 illustrates the computational mesh arrangement used in the study, where tetrahedral elements were employed throughout the entire flow domain. To improve the accuracy of wind pressure prediction near the structures, refined square mesh layers were provided around the building models, enabling dependable simulation results with comparatively lower computational resources.

Mesh refinement study (Figure 4) was carried out for the T1 case at a wind incidence angle of 0° to determine an appropriate mesh size for the simulations.

The along-wind response of the building model was examined through seven successive levels of mesh refinement, progressing from coarse to fine meshes, and the MC6 mesh configuration was finally selected as it provided acceptable computational accuracy with efficient performance.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue V May 2026- Available at www.ijraset.com
National Conference on Integrated Approaches to Sustainable Innovation 2026 (IASI 2026)

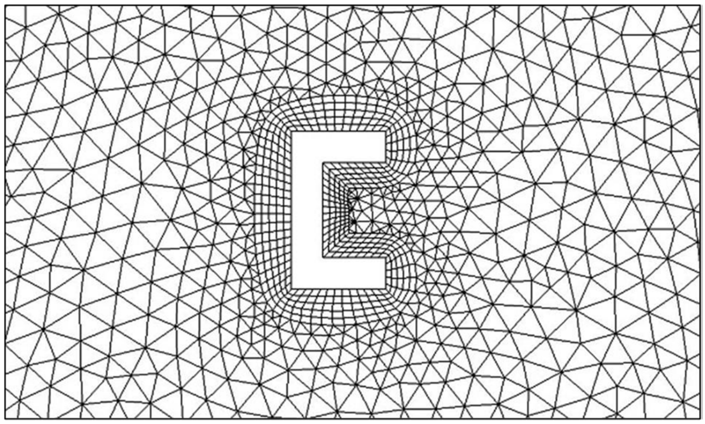


Fig. 3 The meshing pattern around the building model and the remaining domain

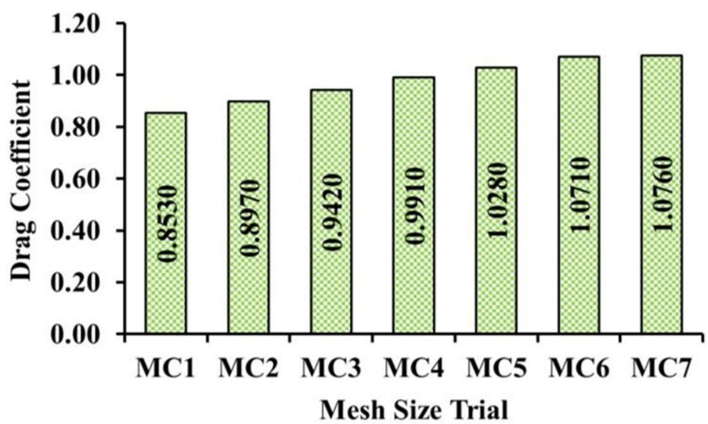


Fig. 4 Mesh Refinement Study for T1 model

C. Validation

The accuracy of the obtained results was validated by comparing the pressure coefficient distributions from the present study with those reported by Gomes et al. (2005), as illustrated in Figure 5. All essential boundary conditions were adopted from this published article to ensure reliable validation. The pressure variation obtained from the ANSYS CFX simulation at Face D showed good agreement with the pressure profile of the article.

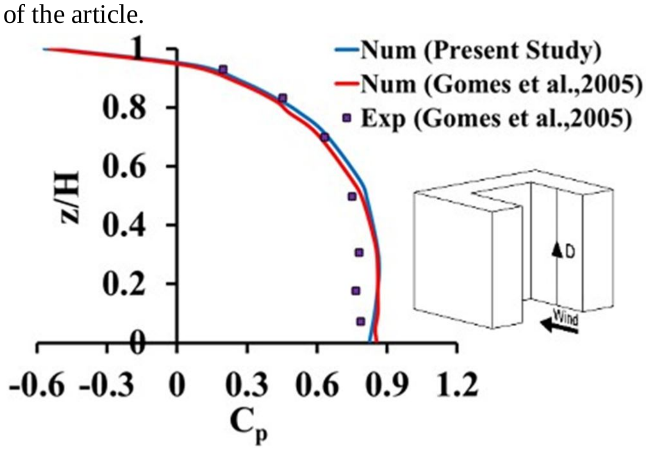


Fig. 5 The validation study of the numerical simulation

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue V May 2026- Available at www.ijraset.com
National Conference on Integrated Approaches to Sustainable Innovation 2026 (IASI 2026)

IV. RESULTS AND DISCUSSION

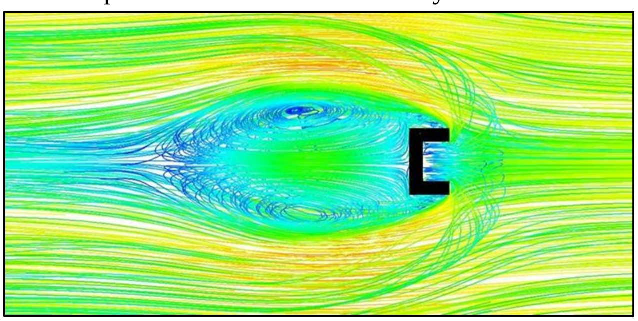
A. Pattern of Wind Flow

The wind flow characteristics around the adopted building models are significantly governed by the plan geometry, corner modification, and wind incidence direction. The presence of the U-shaped configuration generates complex recirculation regions and vortex formations in the wake zone. It is the main cause to alter the overall aerodynamic behavior of the structures. Figures 6 illustrate the streamline patterns developed around the T1 and T4 models for wind directions of 0° and 90°.

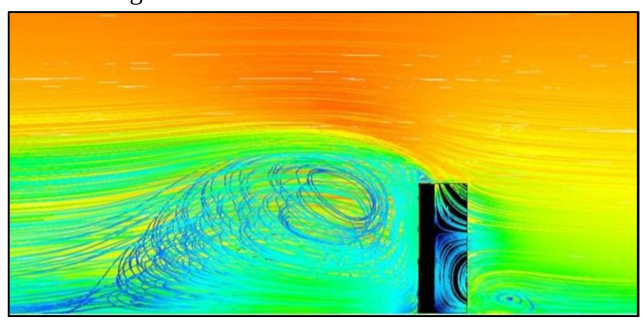
From the streamline patters, it can be observed that the sharp-cornered model (T1) produces comparatively larger wake regions and stronger vortex formations behind the building. Whereas, the rounded-corner in model T4 helps in smoothing the airflow around the outer edges. It reduces the intensity and spread of the recirculation zones. It is also visible that the modification in corner delays the flow separation which allows a comparatively streamlined flow movement around the structure.

For the 0° wind direction, the incoming airflow enters the cavity region of the U-shaped plan and creates noticeable rotational flow inside the two limbs portion. Large vortices are formed behind the building due to sudden flow separation near the sharp edges. For the modified corner model, the span of vortex is reduced and the airflow becomes comparatively smoother near the outer limbs.

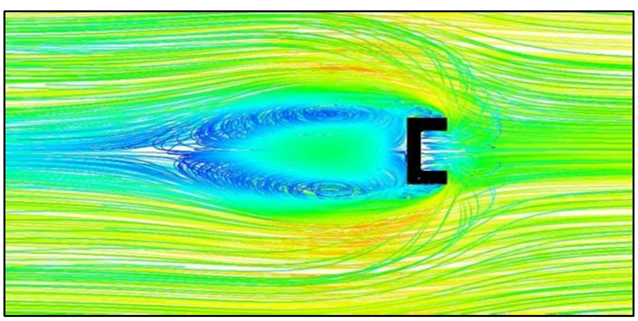
Under the 90° wind direction, the interference of the airflow between the limb becomes more pronounced. Which leads to stronger circulation zones within the zone of the model. Whereas, the rounded corner configuration again contributes to a more controlled flow pattern by minimizing abrupt separation near the edges. These variations in wake formation and streamline distribution directly influence the pressure distribution and aerodynamic forces acting on the buildings.



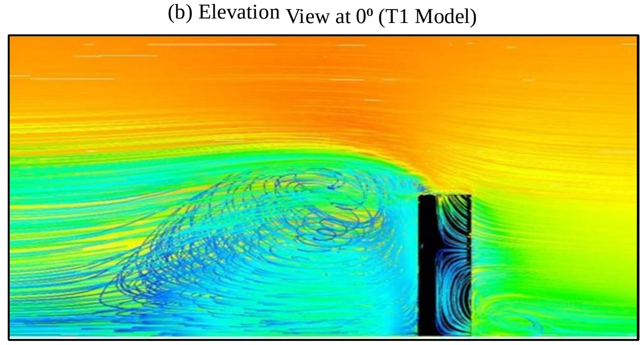
(a) Plan View at 0° (T1 Model)



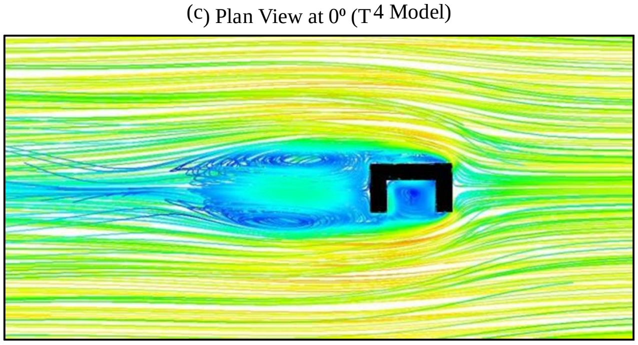
(b) Elevation View at 0° (T1 Model)



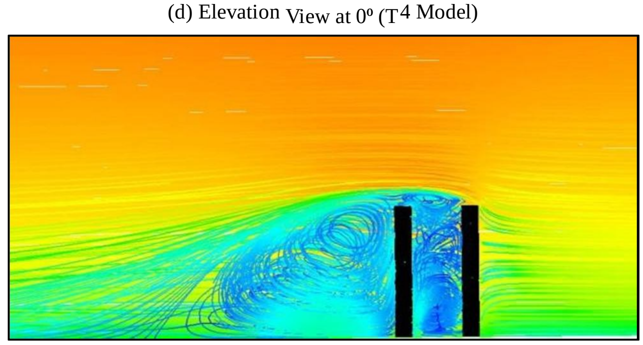
(c) Plan View at 0° (T4 Model)



(d) Elevation View at 0° (T4 Model)

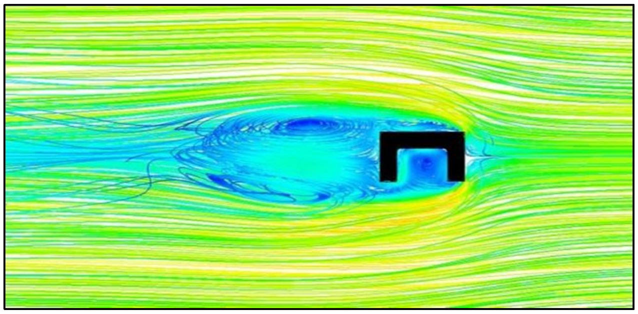


(e) Plan View at 90° (T1 Model)

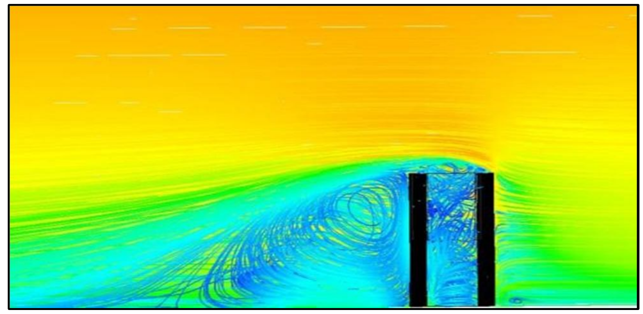


(f) Elevation View at 90° (T1 Model)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue V May 2026- Available at www.ijraset.com
National Conference on Integrated Approaches to Sustainable Innovation 2026 (IASI 2026)



(g) Plan View at 90º (T4 Model)



(h) Elevation View at 90º (T4 Model)

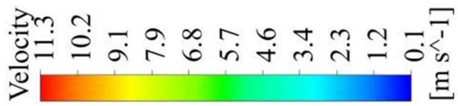


Fig. 6 The velocity streamline distributions for typical building models

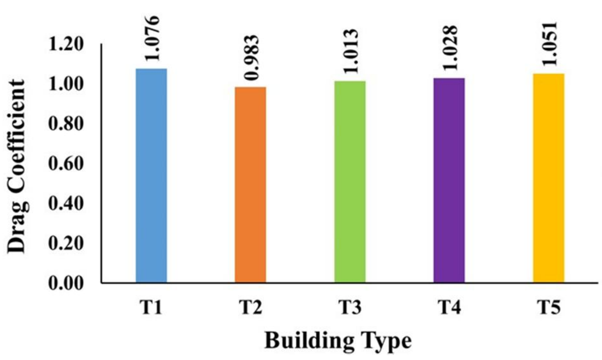
B. Variation in Force Coefficients

The variations in drag coefficient and lift coefficient for the different building models are presented in Figures 7 and 8 for wind incidence angles of 0° and 90°. The results indicate that the aerodynamic response of the buildings is highly dependent on the corner modification and the location at which the alteration is introduced. The sharp-cornered model T1 experiences the maximum drag coefficient at 0° wind angle. But the modified models generally show a reduction in drag force due to smoother airflow separation around the rounded edges. Among all the cases, model T2 exhibits the minimum drag coefficient at 0°. It is also noticed that the continuous corner modification throughout the height effectively reduces wind resistance.

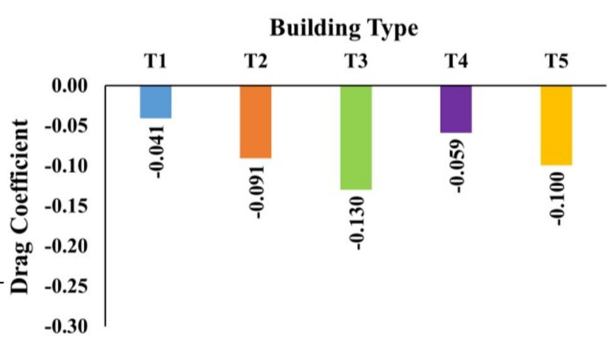
At the 90° wind direction, the drag coefficients become negative for all models. But the magnitude of the force is comparatively smaller than that observed at 0°. Model T3 records the highest negative drag coefficient, while T1 shows the least magnitude. This variation is mainly caused by the change in flow separation and wake formation due to the altered geometry of the models.

The lift coefficient distributions also reveal a significant influence of corner cut on the aerodynamic behavior of the buildings.

At 0° wind angle, all models show relatively small positive lift coefficients. The T2 producing the highest value among all. However, for the 90° wind direction, all the models experience negative lift coefficients because of the asymmetric flow pattern and vortex formation in between the limbs of the U-shaped plan. The sharp-cornered model T1 exhibits the maximum negative lift coefficient, while the modified configurations reduce the magnitude of lift force to some extent. Overall, the results demonstrate that the introduction of rounded corner modifications can improve the aerodynamic performance of U-shaped tall buildings by controlling flow separation and reducing wind-induced forces.

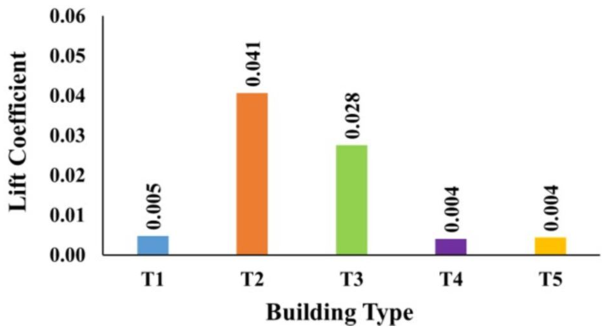


(a) 0º

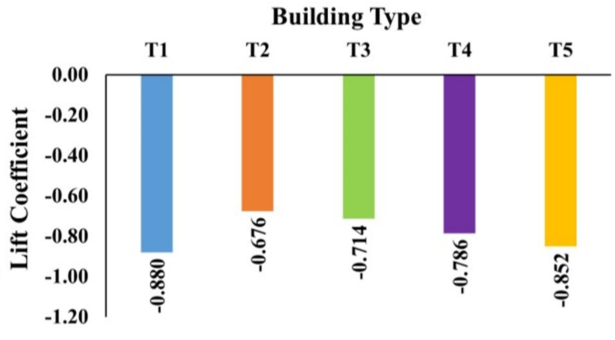


(b) 90º

Fig. 7 Comparison of Drag coefficient of the various building models



(a) 0º



(b) 90º

Fig. 8 Comparison of Lift coefficient of the various building models

C. Variation in Mean Pressure Coefficients

The mean pressure coefficient distributions on different faces of the building models for 0° and 90° wind incidence angles are presented in Table 1 and 2, respectively. The pressure variation on the building surfaces is strongly influenced by the plan geometry, corner modification, and the direction of wind. Significant differences in pressure distribution can be observed between the sharp-cornered and rounded-corner models due to the change in flow separation and recirculation characteristics around the models.

For the 0° wind direction, the windward faces such as B, C, and D experience comparatively higher positive pressure coefficients because these surfaces directly obstruct the incoming flow. Among all the faces, Face C records the maximum positive pressure coefficient for most of the models. It indicates the concentration of stagnation pressure at the central recessed region of the U-shaped plan. On the other hand, the leeward and side faces, particularly F, G, and H, experience negative pressure coefficients due to vortex formation and suction effects generated behind the building. The rounded corner models generally reduce the intensity of suction pressure by allowing smoother airflow around the outer edges.

At the 90° wind angle, almost all the faces exhibit negative pressure coefficients because the wind flow primarily interacts with the side limbs and central region of the U-shaped model. Strong suction effects are observed at faces E, F1, G1, and H1 due to intensified flow separation and recirculation within the limb portion of the building. Model T5 shows the highest negative pressure coefficient at Face F1, indicating the formation of a strong vortex near the modified corner region. However, Face F experiences positive pressure coefficients for the modified models because the wind directly hit this face and also for the localized stagnation effects.

The results clearly indicate that the introduction of rounded corner modifications considerably alters the surface pressure distribution by controlling the flow separation and wake formation around the building. The variation in pressure coefficients among different models directly contributes to the changes in drag and lift forces acting on the structures.

TABLE I
THE MEAN PRESSURE COEFFICIENT ON ALL THE FACES OF THE VARIOUS MODEL FOR 0° WIND ANGLE

Models	A	A1	B	B1	C	C1	D	D1	E	E1	F	F1	G	G1	H	H1
T1	0.47		0.83		0.88		0.83		0.47		-0.42		-0.33		-0.42	
T2	0.44	-0.67	0.83	0.70	0.88	0.86	0.83	0.86	0.44	0.70	-0.50	-0.67	-0.34	-0.43	-0.50	-0.43
T3	0.36	-0.66	0.80	0.86	0.85	0.83	0.80	0.83	0.36	0.86	-0.53	-0.66	-0.34	-0.47	-0.53	-0.47
T4	0.41	-0.63	0.81	0.87	0.85	0.76	0.81	0.76	0.41	0.87	-0.53	-0.63	-0.34	-0.50	-0.53	-0.50
T5	0.43	-0.60	0.81	0.74	0.86	0.52	0.81	0.52	0.43	0.74	-0.53	-0.60	-0.33	-0.57	-0.53	-0.57

TABLE 2
THE MEAN PRESSURE COEFFICIENT ON ALL THE FACES OF THE VARIOUS MODEL FOR 90° WIND ANGLE

Models	A	A1	B	B1	C	C1	D	D1	E	E1	F	F1	G	G1	H	H1
T1	-0.40		-0.58		-0.54		-0.55		-0.71		0.58		-0.51		-0.24	
T2	-0.31	-0.26	-0.60	-0.53	-0.59	-0.54	-0.59	-0.54	-0.87	-0.64	0.66	-0.94	-0.49	-0.92	-0.22	-0.25
T3	-0.32	-0.30	-0.62	-0.59	-0.59	-0.61	-0.60	-0.57	-0.88	-0.66	0.52	-1.01	-0.46	-0.94	-0.22	-0.30
T4	-0.37	-0.34	-0.56	-0.52	-0.52	-0.53	-0.54	-0.52	-0.75	-0.58	0.51	-0.88	-0.48	-0.86	-0.24	-0.33
T5	-0.39	-0.34	-0.56	-0.53	-0.54	-0.57	-0.54	-0.54	-0.74	-0.58	0.55	-1.05	-0.46	-0.77	-0.28	-0.35

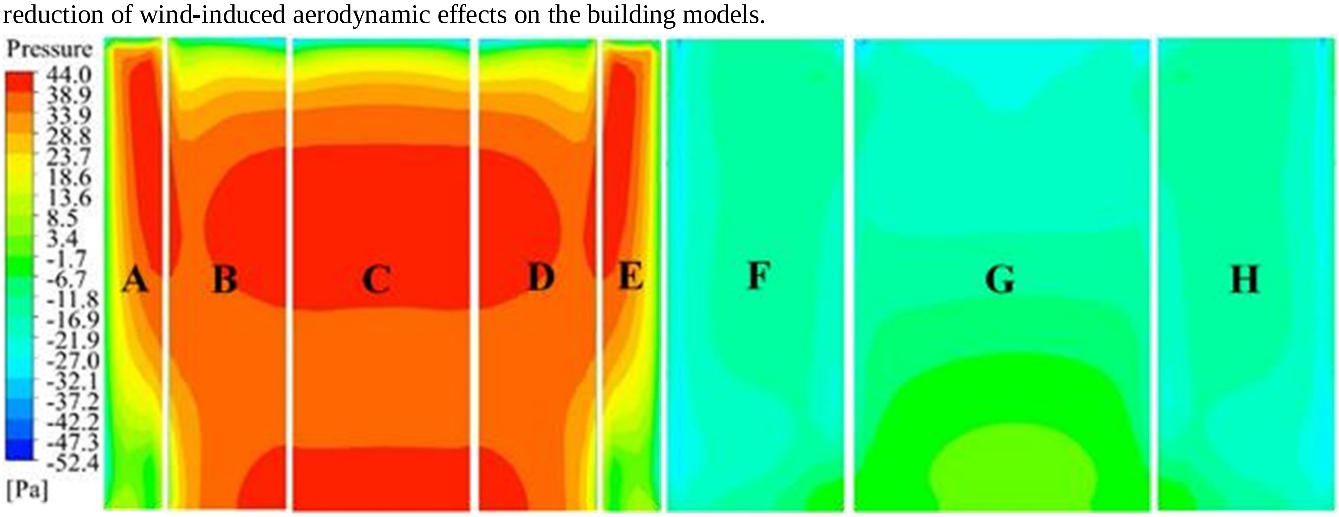
D. Variation in Pressure Contour

The pressure contour distributions for the building models (T1 and T4) at a 0° wind angle are illustrated in Figure 9. The pressure variation around the building surfaces is greatly influenced by the geometry of the structure and the presence of corner modifications. Distinct regions of positive pressure and suction can be observed on different faces due to flow impact, separation, and wake formation around the U-shaped model.

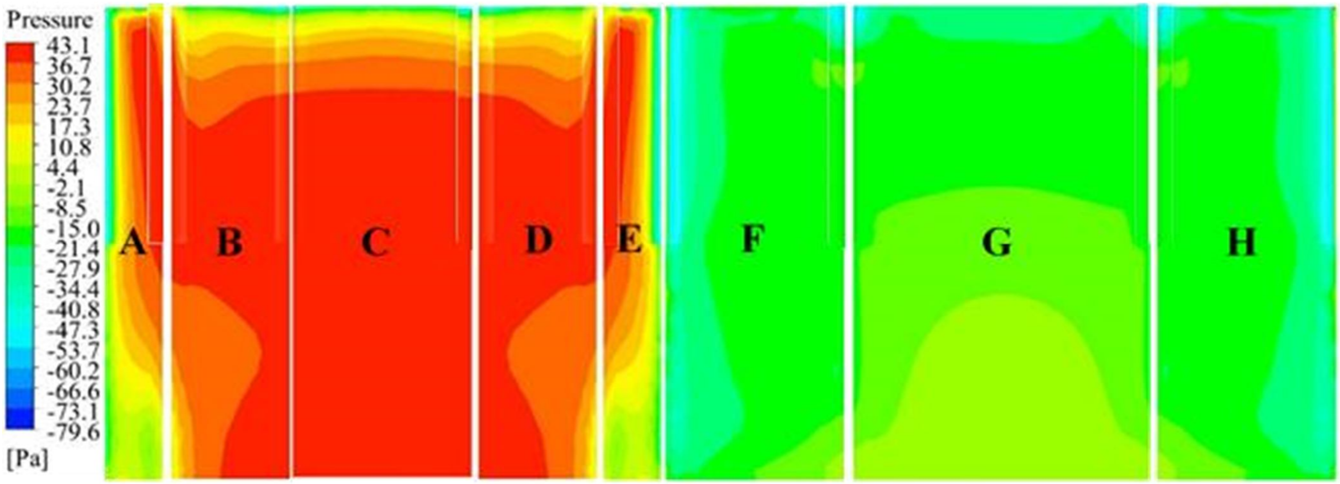
From the contour plots, it can be noted that the windward faces A, B, C, D, and E experience comparatively higher positive pressure because these surfaces directly oppose the incoming airflow. Among these faces, the Faces B, C, and D records the maximum pressure concentration, indicating the formation of stagnation zones within the limb portion of the model. The pressure gradually decreases towards the side and leeward faces due to flow separation and vortex development behind the structure.

For the sharp-cornered model T1, stronger pressure rise and abrupt contour transitions are observed near the outer edges because of sudden flow separation. Whereas, the modified corner model T4 exhibits comparatively smoother pressure distribution patterns around the building surfaces. The rounded corner configuration reduces the intensity of localized pressure concentration and weakens the suction regions near the side faces.

The leeward faces F, G, and H mainly experience negative pressure or suction effects generated by the wake region behind the building. The extent of these suction zones is comparatively reduced in model T4 due to improved airflow movement around the rounded edges. Thus, the corner modification significantly influences the pressure distribution characteristics which contributes to the reduction of wind-induced aerodynamic effects on the building models.



(a) Pressure contour plots for T1 building model



(b) Pressure contour plots for T4 building model

Fig. 9 Comparison of pressure contour for typical building models

V. CONCLUSIONS

The present study investigated the aerodynamic behavior of U-shaped tall building models with different combined corner rounding configurations using Computational Fluid Dynamics (CFD) simulations in ANSYS CFX. The influence of corner modification on wind flow pattern, drag coefficient, lift coefficient, mean pressure coefficient, and pressure distribution was examined for 0° and 90° wind incidence angles.

The results revealed that the geometry of the building and the location of corner modification considerably affect the windinduced responses. The sharp-cornered model generated larger wake regions, stronger vortices, and comparatively higher aerodynamic forces due to abrupt flow separation around the edges. In contrast, the rounded corner configurations helped in smoothing the airflow around the structure and reducing the intensity of recirculation zones behind the building.

Among the building models, the fully modified model exhibited better aerodynamic performance due to drag reduction at normal wind incidence. However, the study also demonstrated that it is not always necessary to introduce corner cuts throughout the entire building height to achieve satisfactory wind performance. Partial corner modification in selected regions can also effectively control the wind aerodynamics and also preserve a greater amount of usable floor area.

The pressure coefficient and contour distributions indicated that the central region of the U-shaped plan experiences significant stagnation pressure under direct wind exposure. The side and leeward faces are mainly dominated by suction effects. The rounded corner reduced the impact of localized pressure concentration and minimized strong suction zones near the outer edges. Nevertheless, comparatively higher-pressure variation was observed near the modified corner regions, suggesting that these locations require careful structural and cladding design consideration.

Overall, the study confirms that combined corner rounding is an effective aerodynamic treatment for irregular U-shaped tall buildings. The adopted modification strategy not only improves wind performance by reducing drag, lift, and wake intensity, but also provides a practical balance between structural safety and architectural usability. The findings of the present investigation may assist designers and researchers in developing wind-resistant irregular tall buildings with improved functional efficiency and economic feasibility.

REFERENCES

[1] A. Elshaer, G. Bitsuamlak, and A. El Damatty, "Enhancing wind performance of tall buildings using corner aerodynamic optimization," Engineering Structures, vol. 136, pp. 133–148, April 2017.

[2] A. Gora, M. Huang, C. Wang, and R. Zhang, "Wind-induced dynamic performance evaluation of tall buildings considering future wind climate," Applied Sciences, vol. 15, no. 9, pp. 1-27, May 2025.

[3] A. Kumar and A. Verma, "CFD Analysis of the Wind Loads on the '+' Plan Building Structure Including Interference Condition Using ANSYS CFX," Journal of Xidian University, vol. 17, no. 10, pp. 50–63, Oct. 2023.

[4] A. Singh, D.-P. N. Kontoni, and S. Mandal, "CFD analysis of the impact of corner adjustments in tall square buildings for wind load mitigation," Engineering Research Express, vol. 6, no. 4, pp. 1-19, Nov. 2024.

[5] A. Sharma, H. Mittal, and A. Gairola, "Aerodynamics of tapered and set-back buildings using detached-eddy simulation," Wind and Structures: An International Journal, vol. 29, no. 2, pp. 111–127, Aug. 2019.
[6] B. Forouzi Feshalami and S. He, "Suppression of Vortex Shedding behind a Square Cylinder Confined in a Diverging Channel," Ocean Engineering, vol. 235, pp. 1-11, July 2021.
[7] C. Wang, A. Mishra, D. Clements, D. Bi, M. Placidi, D. Lasagna, O. Coceal, J. Barlow, M. Carpentieri, S. Grimmond, A. Robins, and Z.-T. Xie, "Flow and dispersion over a 'real-world' tall buildings cluster," Urban Climate, Vol. 67, pp. 1-25, April 2026.
[8] E. Naudascher, J. R. Weske, and B. Fey, "Exploratory study on damping of galloping vibrations," Journal of Wind Engineering and Industrial Aerodynamics, vol. 8, nos. 1–2, pp. 211–222, July 1981.
[9] F. Rezaei and M. Paraschivoiu, "Evaluating roof-mounted VAWT performance with CFD simulations under varying terrain roughness, turbine placement, and building shapes," Transactions of the Canadian Society for Mechanical Engineering, vol. 49, no. 2, pp. 350–367, April 2025.
[10] M. G. Gomes, A. Moret Rodrigues, and P. Mendes, "Experimental and numerical study of wind pressures on irregular-plan shapes," Journal of Wind Engineering and Industrial Aerodynamics, vol. 93, pp. 741–756, Oct. 2005.
[11] H. Cui, H. An, M. Ma, Z. Han, S. C. Saha and Q. Liu, "Experimental study on wind load and wind-induced interference effect of three high-rise buildings," Journal of Applied Fluid Mechanics, Vol. 16, No. 11, pp. 2101-2114, Nov. 2023.
[12] H. Yadav and A. K. Roy, "Wind-induced aerodynamic responses of triangular high-rise buildings with varying cross-section areas," Buildings, vol. 14, no. 9, pp. 1-25, Aug. 2024.
[13] H. Yadav and A.K. Roy, "Wind-induced vibration and dynamic response of tall X-Shaped plan building," Journal of Vibration Engineering & Technologies, vol. 14, pp. 1-22, Jan. 2026.
[14] K. C. S. Kwok, "Effect of building shape on wind-induced response of tall building," Journal of Wind Engineering and Industrial Aerodynamics, vol. 28, pp. 381–390, August 1988.
[15] P. Mazarakou and A. Papalou, "Effect of Structural Forms on Wind-Induced Response of Tall Buildings: A Finite Element Approach," Eng, vol. 6, pp. 125, June 2025.
[16] R. K. Meena, A. Pratap, R. Raj, and S. Anbukumar, "Influence of Corner Geometry on Wind-Induced Forces in Tall Building Models," Asian Journal of Civil Engineering, vol. 26, pp. 593–616, Oct. 2024.
[17] S. Mukherjee, S. Chakraborty, S. K. Dalui, and A. K. Ahuja, "Wind induced pressure on 'Y' plan shape tall building," Wind and Structures: An International Journal, vol. 19, no. 5, pp. 523–540, Nov. 2014.
[18] W. Gan, H. Guo, H. Zhang, F. Zhao, J. Li, S. Peng, and Y. He, "Wind-driven dynamics around building clusters: impact of convex and concave curvilinear morphologies and central angles," Atmosphere, vol. 15, no. 12, pp. 1-22, Dec. 2024.
[19] W. Kim, J. Yi, Y. Tamura, and A. Yoshida, "Wind-induced responses of super-tall buildings with various complicated cross-sectional shapes," in Proceedings of the World Congress on ASEM, Seoul, South Korea, 2017.
[20] X. Zheng, H. Montazeri, and B. Blocken, "CFD simulations of wind flow and mean surface pressure for buildings with balconies: Comparison of RANS and LES," Building and Environment, vol. 173, pp. 1–14, April 2020.
[21] Y. M. Kim and K. P. You, "Dynamic responses of a tapered tall building to wind loads," Journal of Wind Engineering and Industrial Aerodynamics, vol. 90, nos. 12–15, pp. 1771–1782, Dec. 2002.