Wind Engineering Joint Usage/Research Center
FY2024 Research Result Report

Research Field: Wind Hazard Mitigation/Wind Resistant design
Research Year: FY2024
Research Number: 24242004
Research Theme: Unsteady aerodynamic characteristics affecting tall buildings with varying corner shapes and side-to-side ratios

Representative Researcher: Dr. Matza Gusto Andika ST, MT.

Budget [FY2024]: 180,000 JPY Yen

1. Research Aim

To investigate the unsteady aerodynamic effect of corner shape, the side ratio, and the different sizes of corners on the characteristic of the wind force by using computational fluid dynamics and wind tunnel testing.

2. Research Method

This research employs Computational Fluid Dynamics (CFD) to systematically evaluate and select the building shape with the most favorable aerodynamic characteristics from several design variations, including basic and corner-modified geometries. Initially, steady-state simulations are conducted using the k-ω SST turbulence model to determine the aerodynamic coefficients and identify the shape that exhibits optimal wind performance. Building upon these results, the study advances to unsteady analysis using the Large Eddy Simulation (LES) turbulence model, focusing on the selected shape to capture detailed turbulence kinetic energy and unsteady characteristics. This two-step CFD approach allows for a comprehensive understanding of both the mean aerodynamic forces and the complex unsteady flow phenomena associated with corner modifications. To validate and complement the numerical findings, wind tunnel experiments are subsequently performed to measure pressure fluctuations around the models. These measurements provide critical insights into the unsteady aerodynamic characteristics and turbulence structures behind the building, which are directly related to vortex-induced vibrations and their potential impact on structural stability.

Modeling a tall building using Computational Fluid Dynamics (CFD) involves creating detailed geometric representations of the building with various corner shapes to study their aerodynamic behavior. In this case, several designs are considered as depicted in Figure 1, including rectangular, corner cut, corner chamfered, rounded corner, and fin corner configurations. These variations help in understanding how different corner geometries influence airflow patterns and wind loads on the structure. The models are scaled down to a geometrical scale of 1/400 to fit within the computational domain, which is set to match the size of the wind tunnel test section. Surface terrain roughness is characterized by a parameter α = 0.27, following the AIJ2015 standard, to simulate velocity profile conditions affecting wind flow around the building.

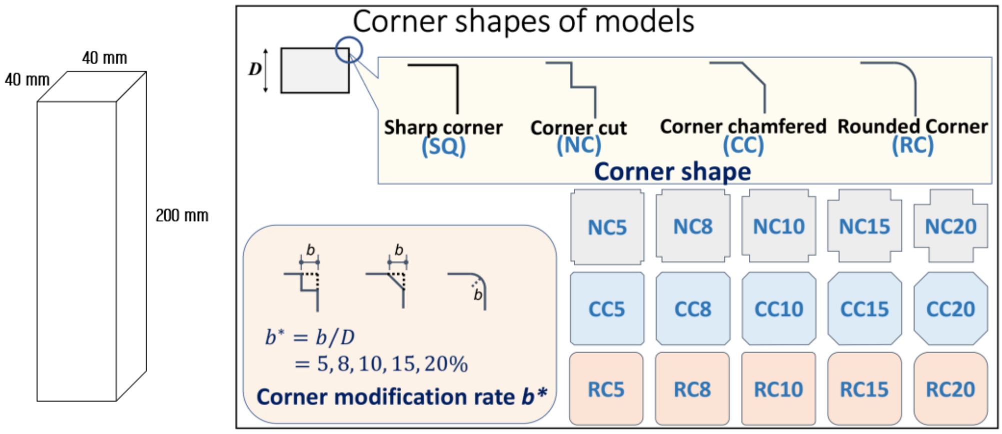


Figure 1. Building model dimension and the modification

The CFD simulation employs the k-ω SST turbulence model, which is well-suited for capturing complex flow features such as separation, recirculation, and vortex shedding around tall structures. This turbulence model combines the advantages of both k-ε and k-ω models, providing accurate predictions of aerodynamic forces under varying flow conditions. The primary objective of this study is to analyze the aerodynamic coefficients—such as drag, lift, and moment coefficients—across the different building shapes. By comparing these coefficients, we can assess how corner modifications impact wind-induced forces and optimize the building design for improved structural performance and wind comfort. This approach enables a detailed understanding of wind-structure interaction, which is critical for the safe and efficient design of tall buildings in urban environments.

Tall building models will be tested in ESWT wind tunnel as depicted in Figure 2. The ESWT wind tunnel is an open-circuit, low-speed wind tunnel as described in Figure 1. The ESWT test section has dimensions of 1.25 m in length and a square cross-section of 0.5 m x 0.5 m, which widens into a rectangle of 0.51 m x 0.5 m (with a wide angle of 0.4° to the side) at the outlet of the test section. The contraction ratio of this wind tunnel is 9. The maximum wind speed that can be achieved is 30 m/s. The main parts of the wind tunnel, which has a length of 8.9 m, include the bellmouth, settling or stilling chamber, contraction, test section, diffuser, and fan housing. (fan house), and a second diffuser (2nd diffuser or exit flow spreader).

Figure 2. ESWT wind tunnel

3. Research Result

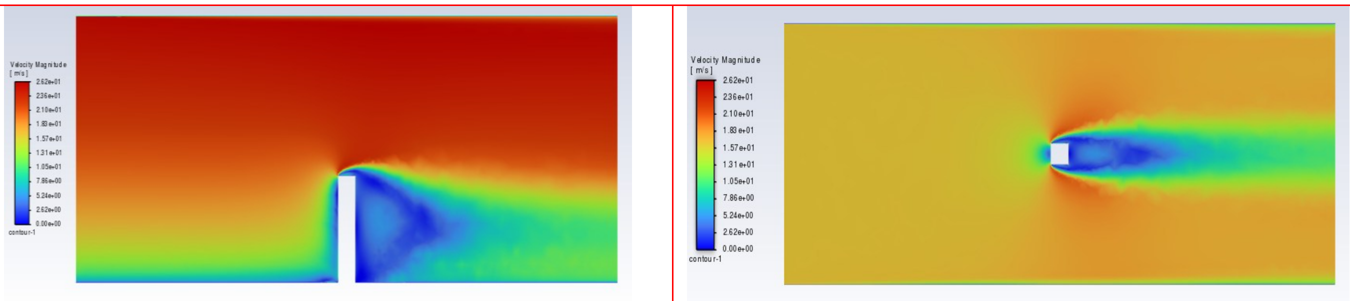
The results of this study begin with a thorough and systematic aerodynamic analysis of various corner modifications applied to a basic sharp-cornered building shape, evaluated across different side ratios to capture the influence of building proportions on aerodynamic performance. This investigation employs steady-state computational fluid dynamics (CFD) simulations using the k-ω SST turbulence model, chosen for its robustness in accurately predicting flow separation and near-wall effects critical to aerodynamic force calculations. By examining how different corner geometries interact with varying side ratios, the study aims to identify the optimal configuration that effectively reduces the mean aerodynamic forces acting on the building, with a particular focus on minimizing the drag coefficient (Cd). The drag force is a key parameter influencing wind loads, making its reduction essential for improving structural resilience and sustainability. The comparative results of these simulations, detailing the aerodynamic performance of each corner modification at different side ratios, are comprehensively summarized in Table 1, providing clear guidance for selecting corner designs that enhance aerodynamic efficiency under diverse geometric conditions.

Table 1. Aerodynamic force coefficient (Cd) at difference corner modification and side ratio

	Ratio 1	Ratio 1.5	Ratio 2	Ratio 3	Ratio 4	Ratio 5
SQ	0.9449	0.8473	0.8096	0.7297	0.7183	0.7246
NC5	0.7483	0.6588	0.6050	0.5747	0.5716	0.5803
CC5	0.6703	0.5769	0.5240	0.4983	0.5033	0.5082
RC5	0.6703	0.5462	0.4604	0.4591	0.4685	0.4877
NC8	0.6903	0.5989	0.5470	0.5201	0.5234	0.5303
CC8	0.5883	0.4885	0.4602	0.4498	0.4514	0.4631
RC8	0.3993	0.3877	0.3770	0.3657	0.3781	0.3812
NC10	0.6936	0.5999	0.5478	0.5119	0.5245	0.5284
CC10	0.6030	0.4660	0.4459	0.4425	0.4385	0.4441

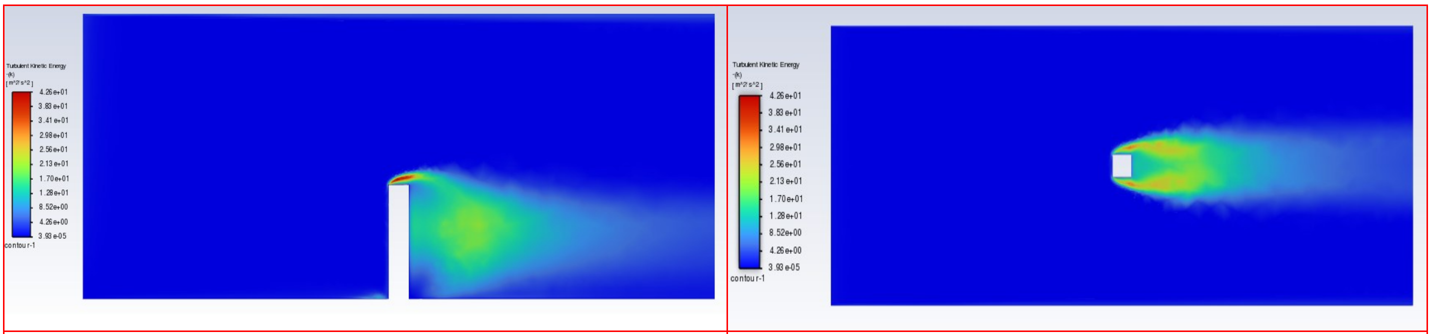
RC10	0. 3758	0. 3835	0. 3773	0. 3502	0. 3683	0. 3748
NC15	0. 6768	0. 5879	0. 5302	0. 5035	0. 5009	0. 5071
CC15	0. 5384	0. 4445	0. 4248	0. 4031	0. 4046	0. 4096
RC15	0. 3747	0. 3462	0. 3355	0. 3096	0. 3292	0. 3357
NC20	0. 6969	0. 6222	0. 5459	0. 5039	0. 5069	0. 5128
CC20	0. 5362	0. 4332	0. 3959	0. 3831	0. 3904	0. 3975
RC20	0. 3384	0. 3092	0. 2971	0. 2945	0. 2914	0. 2973

Among all models, rounded corners (RC) demonstrate the most significant reduction in drag, particularly at higher rounding ratios. The RC20 model shows the lowest drag coefficients, with Cd = 0.3384 at Ratio 1 and decreasing to 0.2973 at Ratio 5, confirming that rounded edges allow airflow to remain more attached to the surface, reducing wake turbulence and aerodynamic resistance. Even at smaller rounding ratios, such as RC10 (Cd = 0.3758 at Ratio 1, reducing to 0.3748 at Ratio 5), the performance improvement over SQ, NC, and CC modifications is evident. This overall trend suggests that larger modifications (higher ratios) lead to greater reductions in drag, with RC modifications being the most effective, followed by NC, CC, and finally SQ as the least aerodynamically efficient design. These findings emphasize the importance of corner modifications in improving wind performance, reducing structural loads, and enhancing overall aerodynamic stability in high-rise buildings. Building upon this initial analysis, the unsteady aerodynamic response was further investigated using Large Eddy Simulation (LES) CFD for both the original sharp-cornered model and the chosen optimal corner-cut configuration, effectively capturing the intricate wake flow patterns and turbulent features that contribute to vibration excitation where depicted in Figure 3 - 10. These approaches demonstrate that corner cut modifications effectively disrupt coherent vortex shedding and reduce turbulence kinetic energy, leading to diminished fluctuating aerodynamic loads.



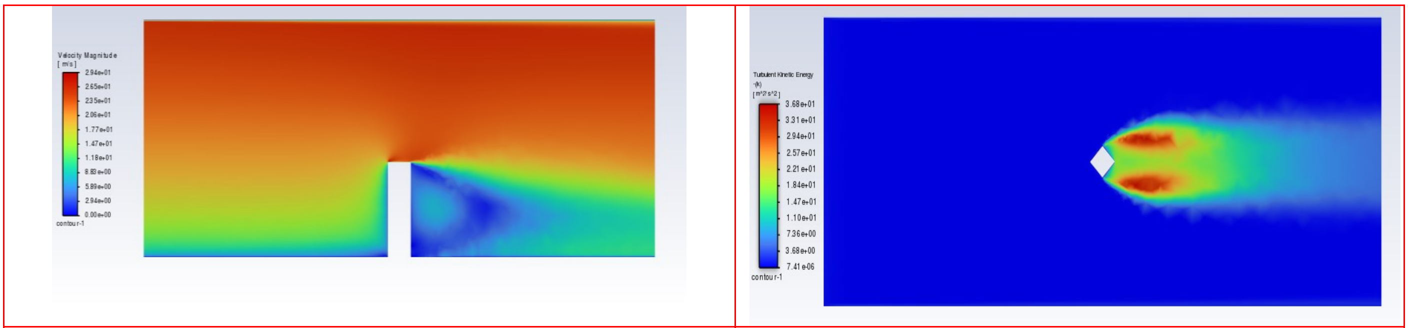
(a) (b)

Figure 3. Velocity magnitude on SQ Model at 0 degree wind direction
(a) Side view (b) Top view



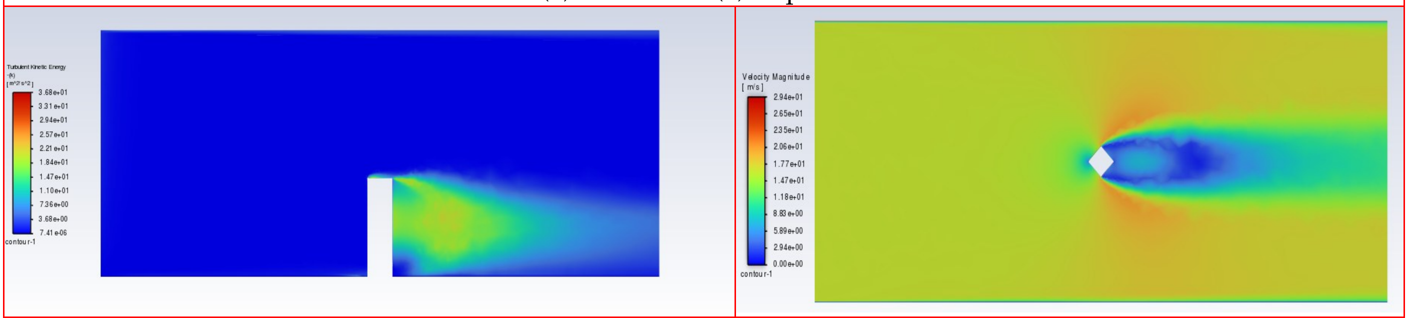
(a) (b)

Figure 4. Turbulence kinetic energy on SQ Model at 0^0 wind direction
(a) Side view (b) Top view



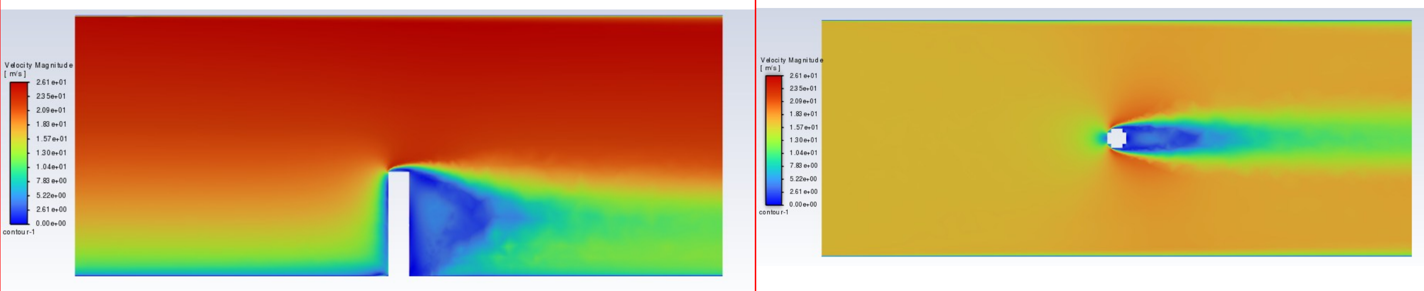
(a) (b)

Figure 5 Velocity magnitude on SQ Model at 45^0 wind direction
(a) Side view (b) Top view



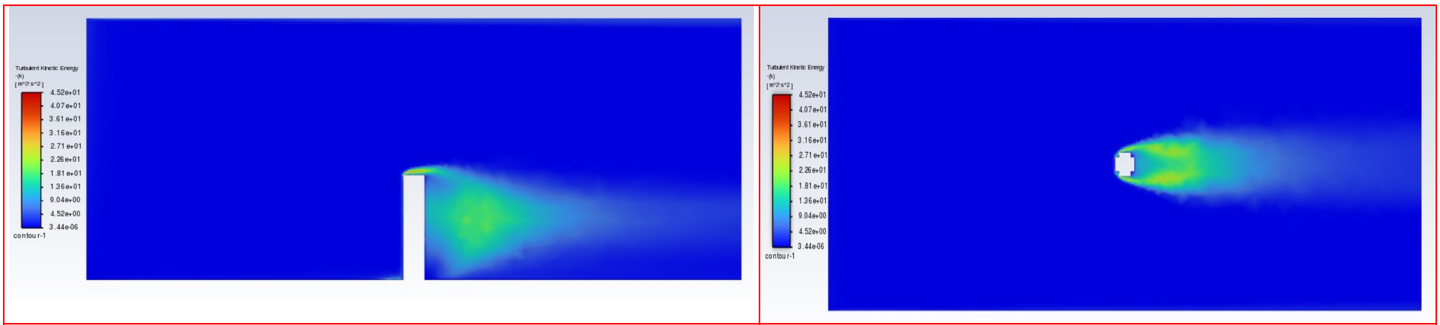
(a) (b)

Figure 6. Turbulence kinetic energy on SQ Model at 45^0 wind direction
(a) Side view (b) Top view



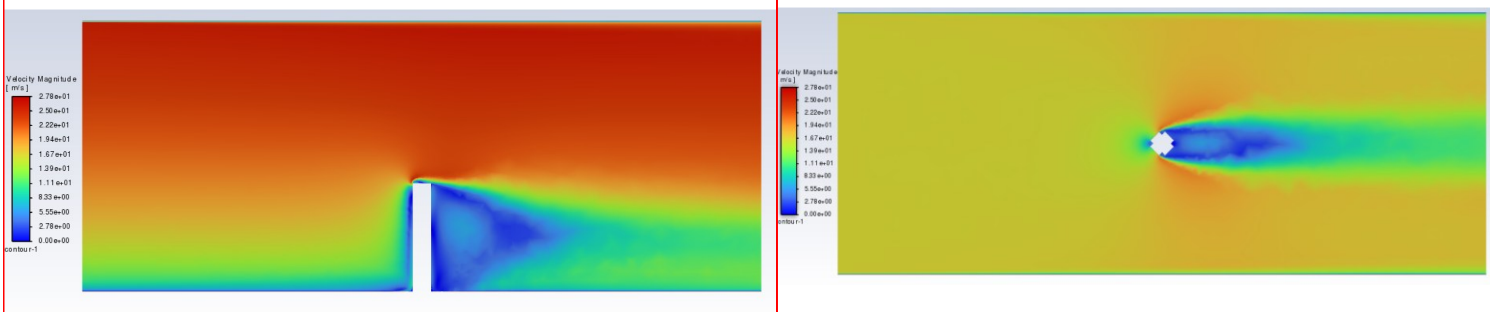
(a) (b)

Figure 7. Velocity magnitude on NC20 Model at 0^0 wind direction
(a) Side view (b) Top view



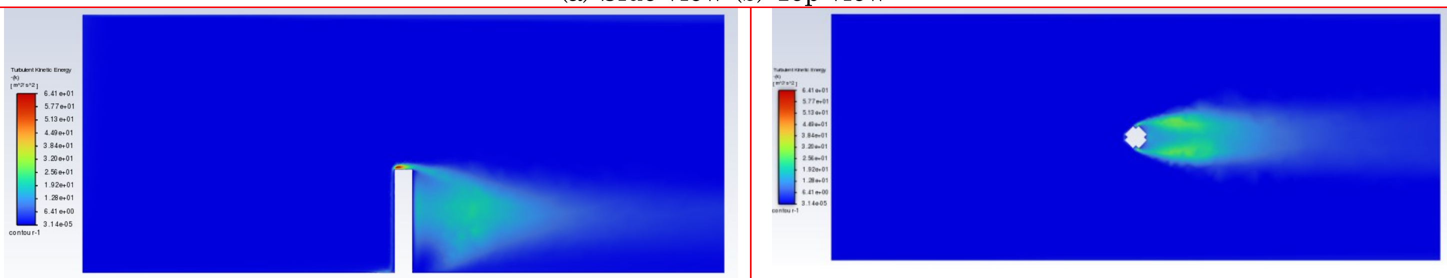
(a) (b)

Figure 8. Turbulence kinetic energy on NC20 Model at 0⁰ wind direction
(a) Side view (b) Top view



(a) (b)

Figure 9. Velocity magnitude on NC20 Model at 45⁰ wind direction
(a) Side view (b) Top view



(a) (b)

Figure 10. Turbulence kinetic energy on NC20 Model at 45⁰ wind direction
(a) Side view (b) Top view

Turbulence Kinetic Energy (TKE) plays a critical role in understanding wind-induced structural responses in high-rise buildings. It represents the energy contained in turbulent wind eddies, which can significantly influence aerodynamic stability, vortex shedding, and wind-induced vibrations. By analyzing the SQ (Sharp-Edged) model and the NC20 (Notched Corners) model at 0° and 45° wind directions, we can evaluate how corner modifications affect the dissipation of turbulent energy and the aerodynamic performance of buildings under varying wind conditions. The differences in TKE between these models provide insights into their stability, susceptibility to oscillations, and ability to mitigate wind-induced forces.

At a 0° wind direction, the SQ model demonstrates a narrow, elongated wake region, characterized by highly concentrated turbulence directly behind the building. With a peak TKE of approximately 42.6 m^2/s^2, the sharp edges cause abrupt flow separation, leading to a strong and structured vortex street. This coherent vortex shedding creates sustained aerodynamic forces, increasing the risk of periodic oscillations that may amplify wind-induced vibrations. Furthermore, the wake remains structured for a longer distance, indicating that turbulence energy is not efficiently dissipating, making the SQ model more susceptible to resonance effects.

In contrast, the NC20 model at 0° wind direction shows a slightly higher peak TKE of ~45.2 m^2/s^2, but with a significantly broader and more diffused wake region. The notched corners disrupt vortex formation, preventing the development of coherent and alternating vortex structures. This results in faster dissipation of turbulence energy as the flow moves downstream. Instead of forming a narrow, high-energy wake like in the SQ model, the turbulence in NC20 spreads out more uniformly, reducing the impact of concentrated aerodynamic forces on the building. The broader turbulence dispersion in NC20 helps lower the risk of sustained oscillatory motion, enhancing the structure's aerodynamic stability.

When the wind direction shifts to 45°, the SQ model exhibits a distinct change in turbulence behavior. The peak TKE drops to ~36.8 m^2/s^2, but the wake becomes more irregular, with turbulence spreading over a larger area compared to the 0° case. This angular wind interaction leads to asymmetrical vortex shedding, which reduces coherence but increases lateral wind loads. While the overall TKE is lower, the oscillatory nature of the aerodynamic forces remains strong, meaning the SQ model still experiences wind-induced vibrations, but in a less structured manner. However, since energy dissipation is still slow, periodic aerodynamic forces continue to act on the structure, potentially leading to long-term fatigue effects.

The NC20 model at 45° wind direction further demonstrates the effectiveness of corner modifications in dissipating turbulence energy. With a peak TKE of ~64.1 m^2/s^2, turbulence is initially higher near the wake region, but it dissipates more quickly than in the SQ model. The notched corners create localized turbulence that helps disrupt vortex coherence, leading to weaker periodic aerodynamic forces. Unlike the SQ model, where vortex shedding remains a major factor in wind-induced responses, the NC20 model at 45° benefits from less structured turbulence, meaning wind loads are distributed more randomly, preventing significant oscillations.

One of the most important observations from the TKE distribution across different wind directions is that the SQ model generally retains turbulence energy longer, which results in higher aerodynamic instability. Whether at 0° or 45°, the wake remains structured, and vortex shedding is strong, making the SQ model prone to sustained wind-induced vibrations. This indicates that buildings with sharp-edged designs may require additional damping mechanisms or secondary modifications to improve aerodynamic performance and occupant comfort.

The NC20 model, however, consistently outperforms the SQ model in terms of aerodynamic stability. While it does experience localized increases in TKE, the notched corners help disperse turbulent energy faster, preventing long-lasting aerodynamic forces from acting on the structure. This faster dissipation of turbulence means lower overall wind-induced oscillations, making NC20 a more stable design for high-rise buildings. The broader turbulence dispersion and disrupted vortex shedding in NC20 suggest that corner modifications are highly effective at mitigating wind-induced forces, even under varying wind directions.

The comparative analysis of TKE in SQ and NC20 models at 0° and 45° wind directions clearly highlights the benefits of corner modifications. The SQ model's sharp edges promote structured vortex shedding, leading to long-lasting turbulence energy that increases wind-induced oscillations. In contrast, the NC20 model reduces vortex coherence and disperses TKE more effectively, preventing strong periodic aerodynamic forces from acting on the building. This enhanced aerodynamic stability makes NC20 the preferred design for mitigating wind-induced vibrations in tall buildings. Future high-rise structures should consider corner modifications like NC20 to optimize wind resistance, energy dissipation, and overall structural safety.

The wind tunnel tests were conducted using three building models from the basic sharp-cornered model (SQ) and two corner-modified models (RC20 and NC20), which were selected based on their superior aerodynamic performance identified in prior CFD analysis. The RC20 model, featuring 20% rounded corners, and the NC20 model, incorporating 20% notched (cut) corners, had previously demonstrated significant improvements in reducing aerodynamic forces, turbulence intensity, and vortex shedding coherence compared to the unmodified SQ model. The objective of this experimental setup was to validate the CFD-predicted aerodynamic benefits by measuring and comparing the fluctuating velocity fields and unsteady wake characteristics behind each model in the wind tunnel, thereby providing a comprehensive understanding of how effective these corner modifications are in mitigating wind-induced vibrations in tall buildings.

(a)	(b)	(c)

Figure 11. Wind Tunnel Model (a) SQ (b) RC20 and (c) NC20

In this wind tunnel experiment, the wind velocity profile at the center of the test section was carefully adjusted by installing a turbulence generator at the upstream part of the test section. The purpose of this adjustment was to ensure that the incoming flow accurately replicated an atmospheric boundary layer representative of urban conditions. Specifically, the vertical velocity profile was tuned to comply with a power-law exponent (α) of 0.27, which characterizes the rate of wind speed increase with height in a typical urban environment. By calibrating the turbulence generator and verifying the resulting profile through preliminary measurements, we ensured that the wind flow entering the building models was realistic, providing a consistent and reliable basis for analyzing aerodynamic forces, wake development, and unsteady flow characteristics related to wind-induced building vibrations.

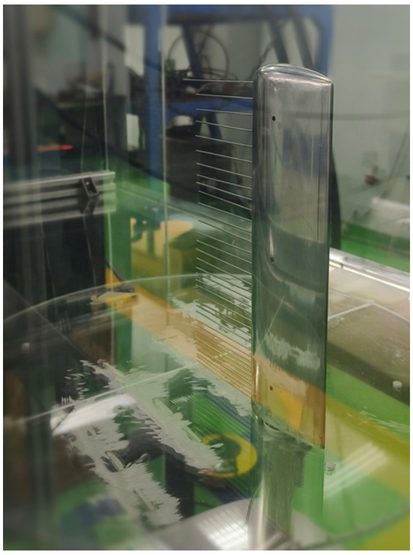


Figure 12. Velocity profile measurement

In this wind tunnel study, we employed a rake system to measure the fluctuating velocity components on the leeward side of the building models. The rake was strategically positioned at a distance of 1.5 times the characteristic width (D = 40 mm) behind the model, which corresponds to 60 mm downstream from the rear face of the building. The measurement height was set at three-quarters (3/4) of the total building height, meaning 150 mm from the base, considering the model height of 200 mm. This setup allowed for capturing detailed information about the unsteady flow field immediately in the wake zone, where the influence of vortex shedding and turbulent fluctuations is most pronounced. The measurement points were distributed horizontally to capture the variation of velocity fluctuations across the wake width, providing valuable insights into the nature of flow separation and reattachment behind the structure.

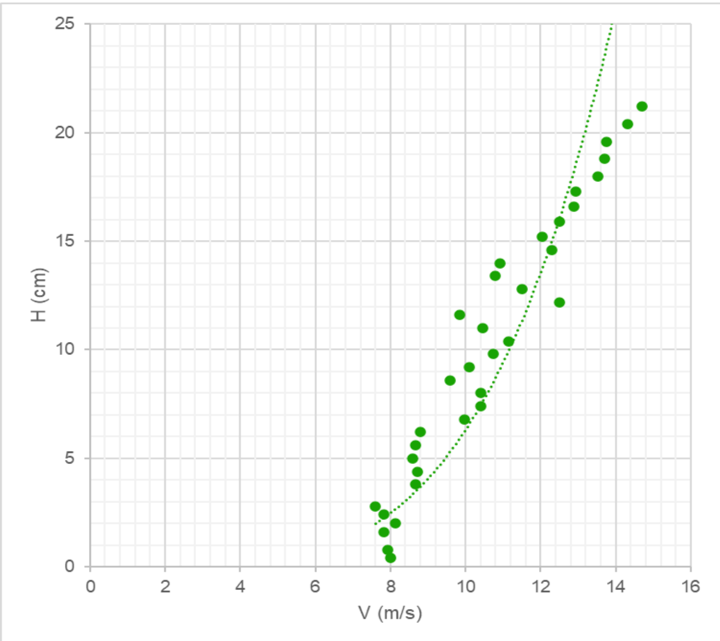
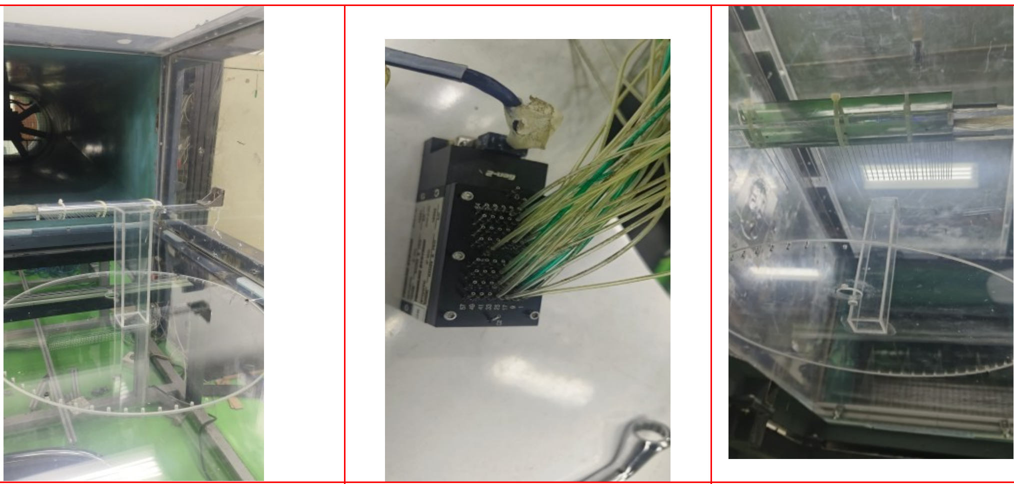


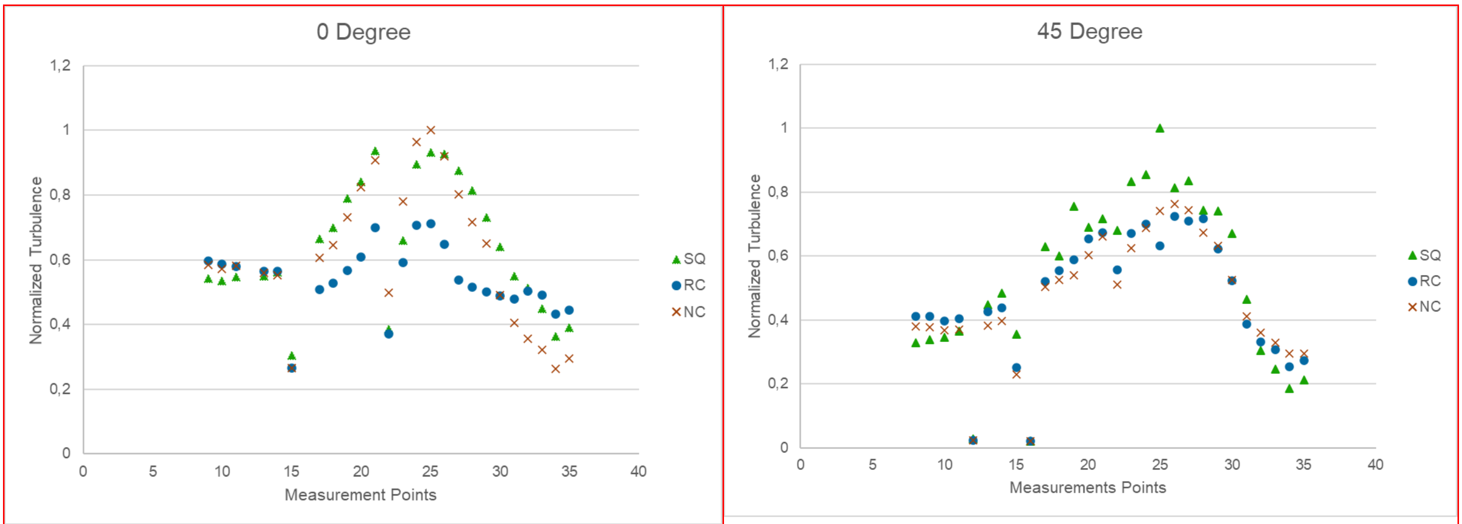
Figure 13. Velocity profile at the center of test section

The primary objective of this experimental investigation was to analyze the unsteady flow characteristics in the wake regions of three building configurations: the sharp-cornered model (SQ), the 20% rounded corner model (RQ20), and the 20% notched corner model (NC20). By comparing the fluctuating velocity profiles across these different models, we aimed to understand how corner modifications influence the strength, distribution, and dissipation of wake turbulence. This analysis directly relates to vortex-induced vibrations (VIV) in buildings, as higher levels of unsteady wake turbulence are typically associated with stronger fluctuating aerodynamic forces on the structure. Through this approach, the study provides important experimental validation to complement the CFD findings and offers a deeper understanding of the aerodynamic stability improvements achieved through building shape modifications.



(a) (b) (c)

Figure 14. Wind Tunnel Setting (a) Front-view (b) Data Acquisition and (c) back-view



(a) (b)

Figure 15. Normalized turbulence characteristics in difference wind direction (a) 0^0 wind direction (b) 45^0 wind direction

The SQ (Sharp-Cornered) model exhibits the highest peak turbulence intensity among the three models, particularly around the central region of the wake (measurement points 20-25). The sharp corners of the SQ model lead to strong flow separation and vortex shedding, which contribute to a high turbulence region directly behind the building. The normalized turbulence values exceed 1.0, indicating that the flow remains highly unsteady and energetic. Additionally, turbulence levels remain relatively high over a broader range of measurement points, suggesting that the wake is more structured and persists over a longer distance, which could lead to strong periodic vortex shedding and increased wind-induced vibrations.

In contrast, the RC (Rounded Corner) model demonstrates a smoother turbulence profile with lower peak values compared to the SQ model. The absence of sharp edges in the RC model results in a more gradual flow separation, allowing the airflow to remain attached for longer before transitioning into the wake. This effect leads to a more diffused turbulence distribution with lower intensity peaks (~0.8 max) and a faster dissipation of turbulent energy. Additionally, the turbulence levels decrease more rapidly after the peak, indicating that the wake behind the RC model recovers more quickly, leading to less aerodynamic instability and lower wind-induced vibrations compared to the SQ model.
The NC (Corner Cut) model shows an intermediate turbulence behavior between the SQ and RC models. The NC model reduces peak turbulence levels (max ~0.9), which is lower than the SQ model but slightly higher than the RC model. The cut corners disrupt the formation of strong vortices, leading to weaker but still noticeable turbulence peaks. However, unlike the RC model, turbulence dissipation is not as rapid, meaning that some vortex structures remain active in the wake region, though at a lower magnitude than SQ. This suggests that while the NC modification effectively reduces turbulence intensity compared to SQ, it does not perform as well as the RC model in terms of completely stabilizing the wake. In summary, RC is the most aerodynamically stable, followed by NC, while SQ experiences the highest turbulence intensity and the most prolonged wake effects.

At a 45-degree wind direction, the SQ (Sharp-Cornered) model continues to exhibit the highest peak turbulence intensity, similar to its behavior at 0-degree wind direction. However, compared to the 0-degree case, the peak turbulence level is slightly reduced. The sharp edges of the SQ model still promote strong vortex shedding, but the oblique wind direction alters the wake structure, leading to asymmetric turbulence distribution. The maximum turbulence value is slightly above 1.0, confirming that the SQ model experiences significant turbulence intensity due to strong and persistent flow separation. Additionally, the spread of turbulence remains wide, showing that the wake of the SQ model extends further downstream, which could contribute to larger aerodynamic forces and stronger wind-induced oscillations.

The RC (Rounded Corner) model exhibits a more stable and distributed turbulence pattern, as observed in the 0-degree wind case. Compared to the SQ model, the peak turbulence level is lower, reaching around 0.8, and turbulence dissipates more quickly. The rounded corners reduce sharp flow separation, allowing the wind to remain attached for longer before transitioning into the wake. This behavior results in lower turbulence intensity and a more evenly distributed wake region, reducing the likelihood of strong periodic vortex shedding. The quicker dissipation of turbulence in the RC model suggests that aerodynamic loads are more evenly distributed, reducing peak wind loads on the structure and leading to improved wind stability for the building.

The NC (Corner Cut) model shows an intermediate turbulence response, aligning with its behavior in the 0-degree wind case. The peak turbulence intensity is lower than in the SQ model but remains slightly higher than in the RC model. The corner cuts in the NC model disrupt vortex shedding, reducing the coherence of turbulent structures while still maintaining some level of asymmetry in the wake. Compared to the SQ model, the wake recovers faster, but not as efficiently as the RC model. This indicates that while NC modifications help in reducing wind-induced turbulence, they are not as effective as RC modifications in completely minimizing turbulence intensity and stabilizing wake structures. In conclusion, the RC model remains the most aerodynamically stable, followed by NC, while SQ experiences the highest turbulence levels and most prolonged wake effects under oblique wind conditions.

Conclusion

The analysis of unsteady turbulence characteristics using CFD simulations and wind tunnel testing has revealed significant differences between sharp-cornered (SQ) buildings and modified corner designs such as Rounded Corners (RC) and Corner Cuts (NC).

Sharp-Cornered (SQ) Buildings Exhibit the Highest Turbulence and Wake Instability

- Strong vortex shedding and high turbulence intensity were observed in the SQ model, leading to a structured and prolonged wake region.
- Delayed turbulence dissipation increases aerodynamic loads and enhances the risk of wind-induced oscillations.
- **Rounded Corners (RC) Significantly Reduce Turbulence and Improve Stability**
- The RC model showed the lowest turbulence intensity, indicating that smooth flow transition minimizes vortex shedding.
- Faster wake recovery and reduced aerodynamic forces make RC the most effective modification for wind-resistant buildings.

Corner Cuts (NC) Provide Moderate Turbulence Reduction but Retain Some Vortex Coherence

- The NC model disrupts vortex formation but does not fully eliminate wake turbulence.
- Although it reduces wind loads compared to SQ, it is less effective than RC in stabilizing the wake.

Vortex-Induced Vibrations (VIV) Are More Severe in SQ Buildings Due to Periodic Shedding

- Resonant oscillations occur when vortex shedding frequency aligns with the building's natural frequency, posing a risk of structural fatigue and excessive motion.
- RC modifications effectively mitigate VIV by reducing periodic vortex shedding, while NC reduces but does not fully eliminate oscillatory forces.

Corner Modifications Are Essential for Aerodynamic Performance and Structural Safety

- RC and NC modifications improve aerodynamic stability, reduce wind loads, and enhance building durability.
- Future designs should explore adaptive morphing corners or smart aerodynamic modifications to further optimize building response under varying wind conditions.

Future Research

Building upon the findings of this study, future research should focus on the integration of artificial intelligence (AI) and machine learning (ML) for predicting aerodynamic characteristics in wind tunnel testing. Recent advancements in AI have shown promising capabilities in capturing complex nonlinear relationships between building geometry, wind direction, and aerodynamic responses. This presents an opportunity to shift from resource-intensive physical testing and computational fluid dynamics (CFD) simulations to data-driven predictive models that can provide rapid and reliable estimations of aerodynamic characteristics. By training AI algorithms on large datasets—including experimental results from TPU’s aerodynamic database and validated CFD simulations—we can develop robust models capable of generalizing across various building shapes, corner modifications, and urban wind scenarios.

The objective of this future research is to establish AI-driven tools that support early-stage aerodynamic design decisions, significantly reducing time and cost while maintaining high accuracy. These models will be designed to predict aerodynamic characteristics and wind load profiles based on inputs such as building geometry, height-to-width ratio, wind incidence angle, and corner treatments (e.g., NC20, RC20). Furthermore, incorporating AI will enable real-time parametric studies and optimization processes, allowing us to explore aerodynamic performance trade-offs before physical prototyping. Ultimately, this approach addresses current technology gaps in wind engineering by enhancing the efficiency, scalability, and precision of wind load evaluations, especially for complex or dynamically evolving building forms in urban environments.

4. Published Paper etc.

[Draft paper]
1. Integrated CFD and Wind Tunnel Investigation of Corner Cut Modifications for Reducing Unsteady Turbulence-Induced Vibrations in High-Rise Buildings

5. Research Group
1. Dr. Matza Gusto Andika (Representative Researcher)

2. Collaborate Researchers
1. Basir S.T., M.Eng
2. Sulistiya, S.Si., MT

6. Abstract

Research Theme
Representative Researcher (Affiliation)
Summary・Figures

Integrated CFD and Wind Tunnel Investigation of Corner Cut Modifications for Reducing Unsteady Turbulence-Induced Vibrations in High-Rise Buildings

Dr. Matza Gusto Andika (Representative Researcher)

Abstract

Wind-induced structural vibrations remain a critical concern for the design and performance of high-rise buildings, particularly those subjected to unsteady aerodynamic forces arising from turbulent flow and vortex shedding. Despite extensive research into mean wind load mitigation, a significant gap persists in strategies specifically targeting the unsteady turbulence characteristics responsible for aerodynamic excitation and serviceability issues. To address this gap, this study investigates the efficacy of corner cut modifications as a passive aerodynamic solution for reducing turbulence-induced vibrations. A comprehensive methodology integrating high-fidelity Computational Fluid Dynamics (CFD) simulations and wind tunnel experiments was employed. Steady-state simulations using the k-ω SST model provided baseline aerodynamic force predictions, while Large Eddy Simulation (LES) captured the unsteady wake dynamics for both the baseline sharp-cornered (SQ) model and the modified NC20 configuration. Complementary wind tunnel measurements utilizing a rake system quantified the fluctuating velocity field at the leeward side of the models under simulated atmospheric boundary layer conditions.
The results demonstrate that corner cut modifications significantly alter wake structure by disrupting coherent vortex formation, leading to a reduction in turbulence kinetic energy (TKE) and fluctuating aerodynamic forces. The findings highlight the strong agreement between CFD predictions and experimental observations, affirming the potential of corner cutting as an effective strategy for aerodynamic stabilization. This study advances the understanding of unsteady wind-structure interactions and provides a validated framework for the aerodynamic optimization of high-rise buildings exposed to complex urban wind environments.