Effects Of Wind Load On Hexagonal High Rise Building Having Different Corner Modifications

Raja Babu[1], Rohit Kumar[2], Shivam Singh[3], Siddharth Gautam[4]
[1, 2, 3] Dept of Civil Engineering
[4]Assistant Professor, Dept of Civil Engineering
[1, 2, 3, 4] MGM's College of Engineering and Technology, Noida, India

Abstract- *This project investigates the impact of wind loads on hexagonal-shaped high-rise buildings using Computational Fluid Dynamics (CFD). With the growing trend of complex building geometries and the limitations of existing wind load standards, the study compares three structural forms: a regular hexagon, and hexagonal structures with fillet and chamfered corners. Using CFD analysis in ANSYS CFX (k-ε model), wind effects are examined at angles from 0° to 75° in 15° intervals. Key parameters analysed include force coefficients, drag and lift coefficients and moment coefficients. The results show that chamfered corners experience significantly higher wind-induced forces and turbulence, highlighting the importance of design modifications to improve structural performance. The study emphasises CFD's role as a cost-effective alternative to wind tunnel testing and provides valuable insights into optimising corner configurations in tall hexagonal buildings, based on IS 875-3-2015 standards.*

Keywords- Aerodynamics, Computational Fluid Dynamics (CFD), Drag Coefficients, High Rise Buildings, Hexagon, Corner Modifications

I. INTRODUCTION

Tall buildings have become increasingly essential in modern urban environments due to rapid urbanisation, population growth, and limited land availability. These structures, often viewed as symbols of progress and pride, are now more prevalent in residential, commercial, and mixed-use developments. Technological advancements in construction methods, materials, and structural systems have made tall buildings both safe and economically viable. However, as buildings grow taller, they become more susceptible to environmental forces—especially wind, which can significantly impact structural stability, occupant comfort, and overall building performance.

To assess wind effects, engineers typically rely on wind tunnel testing and Computational Fluid Dynamics (CFD). While wind tunnel testing is highly accurate, it is also expensive and time-consuming. CFD has emerged as a faster, cost-effective alternative, offering detailed visualisations and eliminating issues related to physical scaling and probe interference. It is especially effective in areas with low tomoderate turbulence, making it suitable for early-stage design analysis.

This study examines the wind effects on 40-storey high-rise buildings with hexagonal cross-sections, focusing on variations such as chamfered and filleted corners. Unlike previous research on regular or irregular shapes, this investigation explores how subtle geometric modifications affect wind behaviour. Using CFD analysis, the study evaluates aerodynamic parameters, including pressure distribution, streamline patterns, and turbulence across vertical and horizontal planes. The results highlight the significant impact of corner modifications on wind response, offering valuable insights for optimising high-rise designs to enhance wind resistance and ensure structural safety.

II. NUMERICAL MODEL DEVELOPMENT

The development of numerical models involves constructing and refining mathematical or statistical representations that simulate real-world phenomena and numerical data. This process includes several key activities, such as examining data sets to identify patterns, correlations, and trends; formulating mathematical equations or expressions to describe and forecast the behaviour of specific systems or processes; and applying statistical techniques to interpret data for better decision-making and predictive analysis. Additionally, numerical models are used to replicate complex real-life systems, enabling detailed experimentation and evaluation of various scenarios. These models also assist in identifying optimal solutions to challenges such as efficient resource management or cost minimisation. Numerical model development plays a vital role across diverse disciplines, including physics, engineering, and economics, where quantitative data and analytical modelling are essential tools for understanding and solving real-world problems.

Numerical Model

ANSYS CFX, a computational fluid dynamics (CFD) tool, is utilised for conducting numerical analysis. In the field of wind engineering, the k-ε turbulence model is commonly applied in computational studies. This model operates with greater efficiency and produces results that are closely comparable to those obtained from wind tunnel experiments. One of the main advantages of the k-ε model is that it requires less computational power to simulate wind responses in engineering problems. The model consists of two transport equations—one for turbulent kinetic energy (k) and another for the turbulence dissipation rate (ε), both of which are defined in equation (2) and interpreted through the k-ε framework.

A notable benefit of the k-ε model is that it does not rely on geometry-specific parameters to generate accurate predictions, giving it an edge in versatility. Within the system of equations, both turbulent kinetic energy and dissipation rate are additional variables that must be considered. The initial or inlet value of turbulent kinetic energy (k) is calculated using equation (1), serving as a foundational input for the simulation process.

$$k = \frac{3}{2}(U_{avg} I)^2 \tag{1}$$

$$\varepsilon = \frac{C_{\mu} k^2}{v(\frac{\mu_T}{\mu})} \tag{2}$$

Here, C_{μ} is a non-dimensional constant
k is the turbulent kinetic energy
v is the kinematic viscosity
$\frac{\mu_T}{\mu}$ is the eddy viscosity ratio
U_{avg} is mean flow velocity
I is turbulence intensity

Building Model

Ansys Discovery program generated Models Q and R in this study, depicted in Fig. 1. Fig. 1 contrasts Model P, an isolated reference model, with Models Q and R, both featuring fillet and chamfered corners respectively, standing 120 meters tall having a scale ratio of 1:300.

The Ansys analysis encompasses five crucial stages: geometry, mesh, setup, solution, and result. The domain adheres to wind tunnel manual instructions, followed by paras nath and ritu raj [3], preventing flow recirculation. A tetrahedral mesh with a 0.2-meter size covers the extensive domain. Along the boundaries, a mesh with 10 layers inflation facilitates smooth flow integration with the model. Fig. 2 illustrates the domain used in CFD simulation.

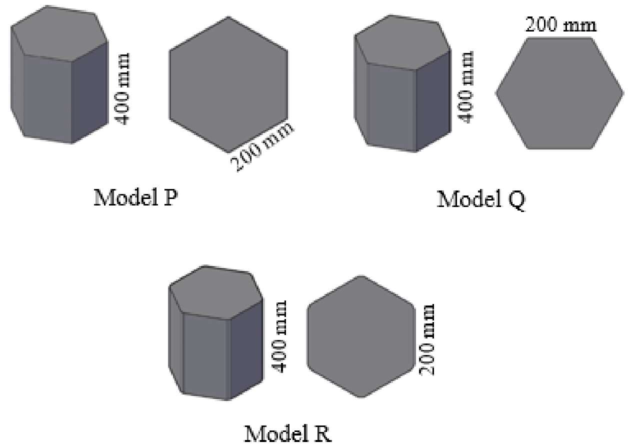


Fig 1: Plan and Elevation of High-Rise Structures

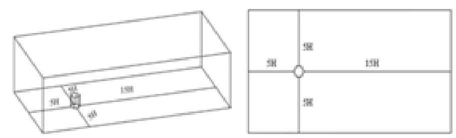


Fig 2: Domain Used in CFD Simulation

Meshing

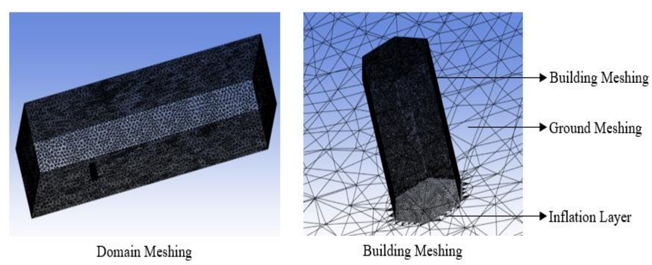


Fig 3: Different types of Meshing

The extensive domain employs tetrahedral meshing with a size of 0.05 m size, featuring 10-layer inflation at the building model’s boundaries to ensure seamless flow integration with domain’s ground. Meshing is applied to various sections of model, as depicted in Fig. 2.

Grid Convergence and Validation

Since it suggests the meshing pattern for the entire numerical simulation, grid convergence analysis is a foundational requirement for CFD programming [3]. For the purposes of this inquiry, model P is used, and testing is done at a wind incidence angle of 0°. Figure 2 shows the domain used for the numerical simulation with its dimensions. In Figure 3, meshing on the domain and meshing on the building with inflation layers are shown.

An independent reference model, known as Model P, has been created and analysed in order to validate the current research project and ensure its compliance with Indian Standard IS: 875 (part-3): 2015. Model P measures 60 meters in radius and has an overall height of 120 meters and scaled down to a ratio of 1:300 (Fig. 1).

CFD Pre-Simulations

To interpret the results produced by the ANSYS setup and solution processes regarding wind effects on a complex building shape, the power law is utilized to make the data meaningful. Figure 4 shows the virtual domain and power law profile utilised in this study to examine the outcomes of this simulation using the following input parameters:

$$\alpha = 0.147$$
$$Z_{ref} = 1\,m$$
$$U_{ref} = 10m/s$$

A power law [3] is utilised in equation (3) in order to calculate pressure.

$$P = U_{ref}\left(\frac{z}{z_{ref}}\right)^{\alpha} \quad (3)$$

The results are analysed after approximately an hour of running the programme. The variation in pressure coefficient [3] is then computed in equation (4).

$$C_P = \frac{P - P_{ref}}{\frac{1}{2}\rho_a V_{ref}} \quad (4)$$

V_{ref} is the reference wind speed, which is retained at 10 *m/s* and calculated in accordance with the instructions in the wind tunnel handbook and Indian Standards [1]. As a result, the density of air is calculated to be $\rho_a = 1.225\,kg/m^3$.Additional scenarios for each face of the wind analysis in ANSYS will be taken into account for wind angles incident at 0°, 15°,30°, 45°,60°,75° and 90° on the z-x plane.

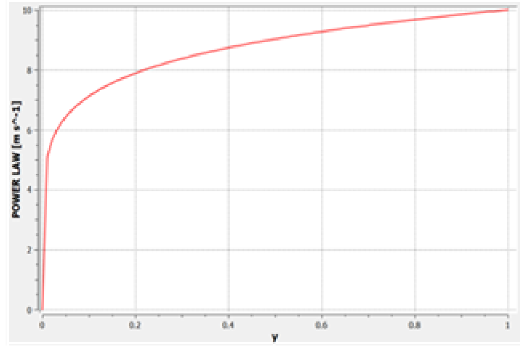


Fig 4: Graph of Power Law

For Model Q and R, the influence of the wind load is examined for 6 faces. The given names of the faces are illustrated in Figure 6. Different angle of attack is also shown in Figure 6. The side facing the incoming wind is known as the windward side. The leeward side signifies the side that is opposing the wind as it comes in. As shown in Figure 6, the categorization of faces depends primarily on six directions, which are collectively referred to as the "front faces": faces A, B, C, D, E and F. The remaining faces are categorized as "side faces" underneath subscripts of the front face.

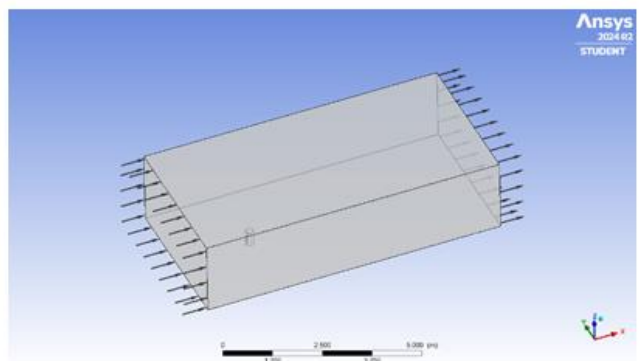


Fig 5: Representation of setup in Ansys

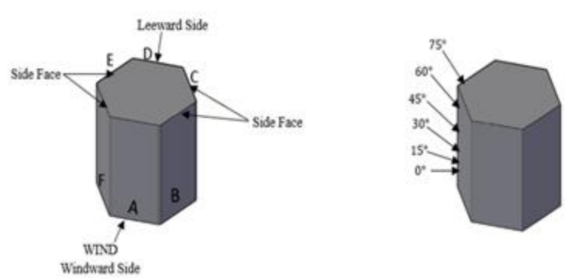


Fig 6: Representation of faces and angle of attack on models

Different scenarios for each face for wind angles incident at $0°$, $15°$, $30°$, $45°$, $60°$, and $75°$ on the *x-y* plane will be taken into consideration for wind analysis in ANSYS.

III. RESULTS AND DISCUSSIONS

Exploring the results of models with regular and asymmetrical shapes and equal plan areas, the findings are displayed in various visualizations.

Force Coefficient

The non-dimensional quantity called a force coefficient [$F/(\rho V^2 S)$], (where *F* is an aerodynamic force and S is an area), is similar to the type often developed and used in aerodynamics.

The Force Coefficient for Y direction (C_{FY}) is calculated using the formula shown in equation (5), similarly for X direction (C_{FX}) is calculated from equation (6) and lastly the resultant force coefficient (C_{Fr}) from equation (7). Now for the reference model P shown in Fig 19, the C_{FY} maximum value lies at 0.024 at $15°$ angle of attack which creates positive pressure and the minimum value lies at -0.009 at $45°$ angle of attack which generates suction. For C_{FX} maximum value lies at 0.875 at $0°$ angle of attack which creates positive pressure while minimum value lies at -0.038 at $60°$ angle of attack which generates suction. For C_{Fr} maximum value lies at 0.875 at $0°$ angle of attack and the minimum value lies at 0.038 at $45°$ angle of attack which creates positive pressure. Secondly for model Q shown in Fig16, the C_{FY} maximum value lies at 0.046 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.024 at $75°$ angle of attack which generates suction. For C_{FX} maximum value lies at 1.484 at $75°$ angle of attack while the minimum value lies at -0.021 at $60°$ angle of attack which creates suction. For C_{Fr} maximum value lies at 1.484 at $75°$ angle of attack which creates positive pressure while minimum value lies at 0.022 at $60°$ angle of attack which creates positive pressure. Thirdly for model R shown in Fig17, the C_{FY} maximum value lies at 0.039 at $15°$ angle of attack which creates positive pressure while the minimum value lies at -0.225 at $75°$ angle of attack which generates suction. For C_{FX} maximum values lies at 0.513 at $0°$ angle of attack which creates positive pressure while the minimum value lies at -0.056 at $45°$ angle of attack which generates suction. For C_{FR} maximum value lies at 0.515 at $0°$ angle of attack which creates positive pressure and the minimum value lies at 0.025 at $30°$ angle of attack which creates positive pressure.

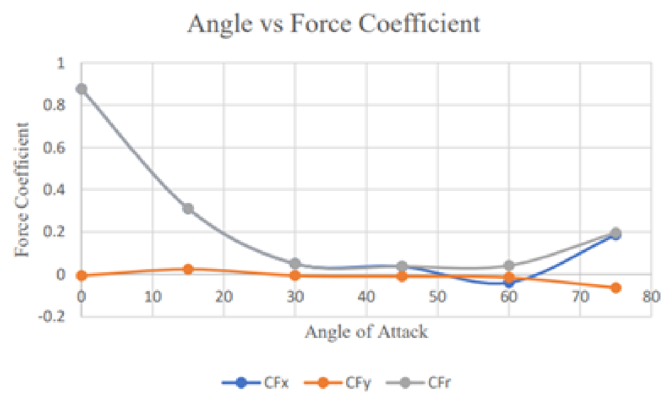


Fig 7: Force Coefficient for Model P

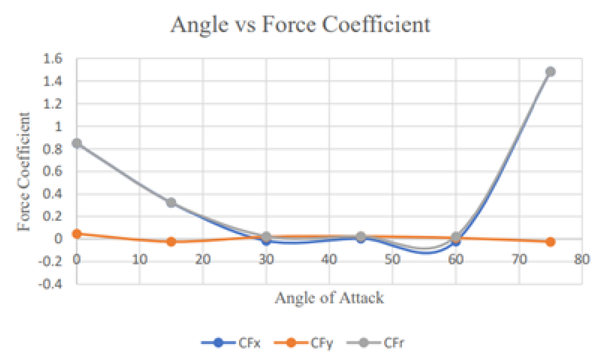


Fig 8: Force Coefficient for Model Q

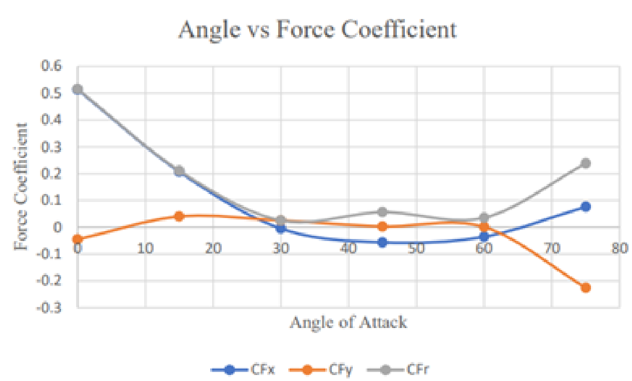


Fig 9: Force Coefficient for Model R

The Force Coefficient is calculated using the formula:

$$(y - direction)C_{FY} = \frac{\sum F_y}{0.5\rho V^2 L_x H} \quad (5)$$

$$(x - direction)C_{FX} = \frac{\sum F_x}{0.5\rho V^2 L_y H} \quad (6)$$

$$Resultant\ Force\ Coefficient, C_{Fr} = \sqrt{(C_{Fy})^2 + (C_{Fx})^2} \quad (7)$$

Drag & Lift Coefficient

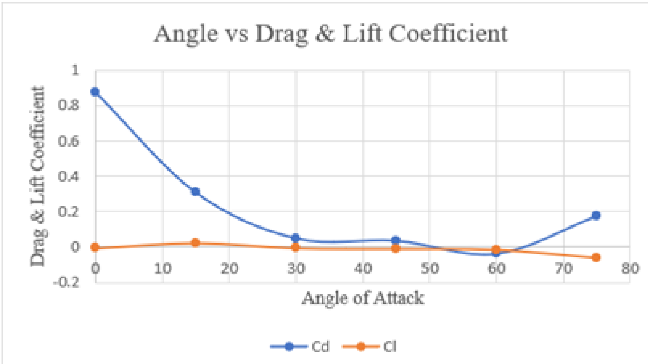


Fig 10: Drag & Lift Coefficient on Model P

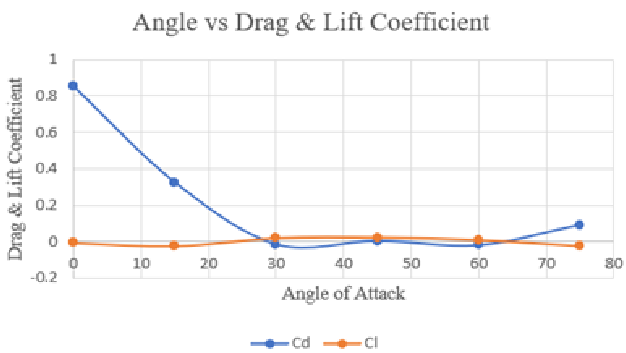


Fig 11: Drag & Lift Coefficient on Model Q

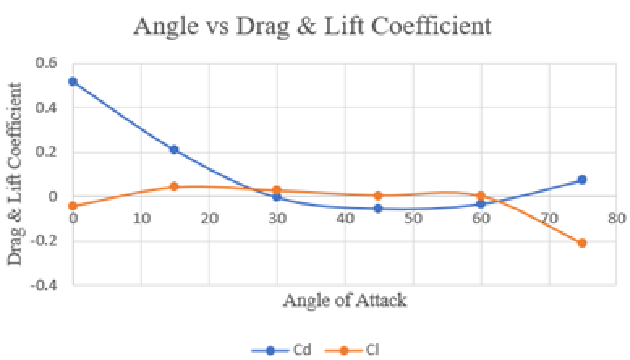


Fig 12: Drag & Lift Coefficient on Model R

The drag coefficient, also known as: in fluid dynamics, is a dimensionless parameter that is used for determining an object's resistance or drag in a fluid environment, such as air or water. A lower drag coefficient means an item will have less aerodynamic or hydrodynamic drag, in accordance to the drag equation. A specific surface area is constantly linked to the drag coefficient. Engineers utilise the lift coefficient to represent all of the intricate relationships between shape, tilt, and certain flow conditions and lift. The ratio of the lift force to the force generated by the dynamic pressure times the area is then expressed by the lift coefficient.

The Drag Coefficient is calculated using the formula shown in equation (8), similarly Lift coefficient is calculated from equation (9). Firstly, for model P shown in Fig16, the C_d maximum value lies at 0.875 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.037 at $60°$ angle of attack which generates suction. For C_l maximum value lies at 0.023 at $15°$ angle of attack which creates positive pressure and the minimum value lies at -0.059 at $75°$ angle of attack which generates suction. Secondly for model Q shown in Fig16, the C_d maximum value lies at 0.850 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.020 at $60°$ angle of attack which generates suction. For C_l maximum value lies at 0.022 at $45°$ angle of attack which creates positive pressure and the minimum value lies at -0.024 at $15°$ angle of attack which generates suction. Thirdly for model R shown in Fig16, the C_d maximum value lies at 0.513 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.055 at $45°$ angle of attack which generates suction. For C_l the maximum value lies at 0.039 at $15°$ angle of attack which generates positive pressure and the minimum value lies at -0.214 at $75°$ angle of attack which generates suction.

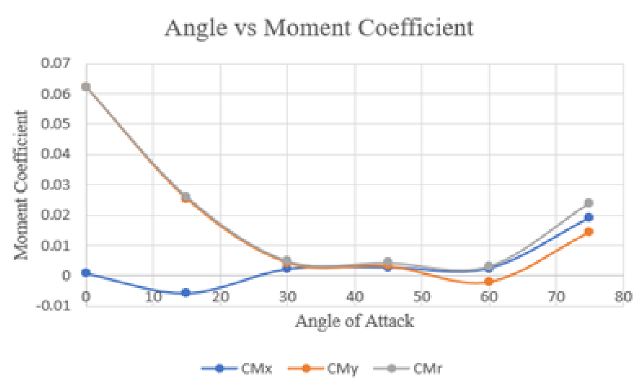
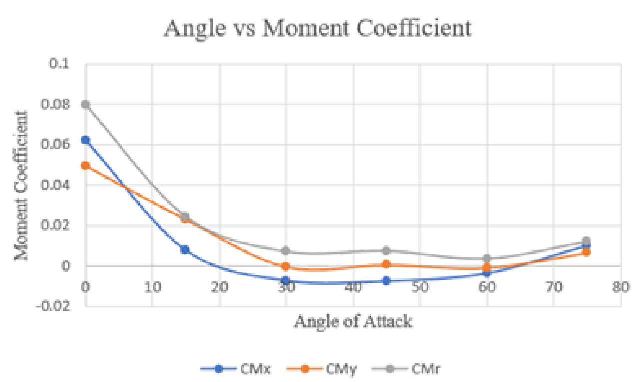


Fig 13: Moment Coefficient on Model P



$$\textbf{Drag Coefficient} = \frac{\sum F \text{ along wind direction}}{0.5\rho V^2 L_x H} \quad (8)$$

$$\textbf{Lift Coefficient} = \frac{\sum F \text{ across wind direction}}{0.5\rho V^2 L_x H} \quad (9)$$

Moment Coefficient

The moment that is directly caused by the aerodynamics force—primarily the lift force acting on the wing—is referred to as the moment coefficient. It is possible to compute a force's moment about any point on the chord, even outside of it.

The moment coefficient for Y direction (C_{My}) is calculated using the formula shown in equation (10), similarly for X direction (C_{Mx}) is calculated from equation (11) and lastly the resultant moment coefficient (C_{Mr}) from equation (12). Firstly, for reference model P shown in Fig16, the C_{My} maximum value lies at 0.062 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.0019 at $60°$ angle of attack which generates suction. For C_{Mx} maximum value lies at 0.019 at $75°$ angle of attack which generates positive pressure and the minimum value lies at -0.0057 at $15°$ angle of attack which generates suction. For C_{Mr} maximum value lies at 0.062 at $0°$ angle of attack which generates positive pressure and the minimum value lies at 0.0032 at $60°$ angle of attack which generates suction. Secondly for model Q shown in Fig16, the C_{My} maximum value lies at 0.049 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.0007 at $60°$ angle of attack which generates suction. For C_{Mx} maximum value lies at 0.062 at $0°$ angle of attack which generates positive pressure and the minimum value lies at -0.0074 at $45°$ angle of attack which generates suction. For C_{Mr} maximum value lies at 0.079 at $0°$ angle of attack which generates positive pressure and the minimum value lies at 0.0037 at $60°$ angle of attack which generates suction. Thirdly for model R shown in Fig16, the C_{My} maximum value lies at 0.037 at $0°$ angle of attack which creates positive pressure and the minimum value lies at -0.0038 at 45° angle of attack which generates suction. For C_{Mx} maximum value lies at 0.051 at $75°$ angle of attack which generates positive pressure and the minimum value lies at -0.017 at $15°$ angle of attack which generates suction. For C_{Mr} maximum value lies at 0.051 at $75°$ angle of attack which generates positive pressure and the minimum value lies at 0.0024 at $60°$ angle of attack which generates suction.

$$(\boldsymbol{y\ direction})C_{M_y} = \frac{\sum M_y}{0.5\rho V^2 L_y H^2} \quad (10)$$

$$(\boldsymbol{x\ direction})C_{M_x} = \frac{\sum M_x}{0.5\rho V^2 L_x H^2} \quad (11)$$

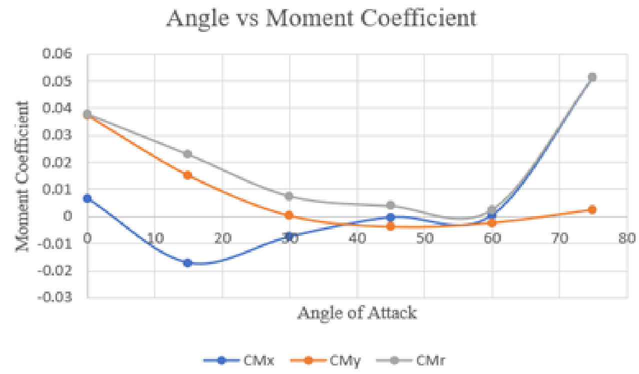


Fig 15: Moment Coefficient on Model R

IV. CONCLUSION

The comparison analysis results indicate that hexagonal building model with chamfered corners outperforms the other two models, namely the regular hexagonal structure and the hexagonal structure with fillet corners. This superiority is particularly evident when examining the Angle vs Force Coefficient graph. This graph shows that the hexagonal building with fillet corners is more effective in withstanding the impacts of wind forces, as it exhibits lower force coefficient. But, the lowest drag coefficient (C_d) was observed for model R at $45°$ angle of attack and the lowest lift coefficient (C_l) is also observed for model R at $75°$ angle of attack. In simpler terms, a lower positive C_d means that the building experiences less wind resistance, which is advantageous for stability during strong winds. On the other hand, a lower negative C_l indicates that the building generates less lift force, which can prevent it from being lifted or overturned by the wind. Therefore, the hexagonal structure with chamfered corners is the most resilient design when it comes to wind effects, making it a suitable choice for ensuring structural integrity and safety in windy conditions. The result and suggestions from this research have a direct influence on the examination of corner configurations design of buildings. This paper provides valuable new insights into how different corner configurations affects the pressure caused by wind, especially the drag and lift coefficients (C_d&C_l). This knowledge can be applied for tall buildings in minimizing the effect of wind that may occur on tall structures also it is beneficial for assessing the structural integrity of different types of structures that is from rectangular to tapered shape under windy conditions, ultimately aiding to the development of cost-effective designs. Understanding how wind interacts with large structures where improvements are implemented along the height of the building is challenging due to the turbulent and capricious nature of wind. Therefore, it is of the utmost importance to look into the wind-flow characteristics of such kinds of constructions. In this piece of work, we reach the following conclusions:

- When assessing the negative effects of wind on tall buildings that have intricate shapes, international standards have often been the go-to approach. However, in this research, a different method was used. This study employed a combination of Computational Fluid Dynamics (CFD) analysis and the k-turbulence model within the ANSYS: CFX software to evaluate the impact of wind on a high-rise building with a more intricate cross-sectional design, referred to as "Model P."
- With change in the corner configuration the C_d and C_l value decreases and it is seen that the hexagonal building with chamfered corners that is model R performs better in resisting the effect of wind in high rise structures.
- Moment coefficient in *x* and y-direction decrease with changing the corner configuration of the hexagonal building.

There are numerous benefits of using Computational Fluid Dynamics (CFD) instead of traditional wind tunnel testing:

- First, it accelerates up the design procedure through minimising down on the amount of time required to conduct analysis and testing. Conventional wind tunnel testing necessitates time-consuming physical model building and complex testing environments.
- Contrarily, CFD uses computer simulations in order for engineers to quickly explore and optimize solutions.
- Furthermore, CFD is a more affordable option.
- While there are initial software and hardware expenses, they are often far less than the ongoing costs associated with maintaining a wind tunnel facility and creating physical models.
- Overall, CFD provides a more efficient and economical means of assessing aerodynamic performance and fluid dynamics in engineering and design.
- In conclusion, this study sheds light on the complicated behaviour of wind flow on tall structures with standard cross sections and emphasizes the significance of taking different corner configurations and cross sections into account when constructing stable structure.

V. FUTURE SCOPE OF PAPER

The current study provides valuable insights into the aerodynamic behaviour of hexagonal high-rise structures with varying corner configurations under wind loading using CFD simulations. However, there remains significant potential for future research and development in this domain:

- Exploration of Additional Geometrical Variations
- Dynamic Wind Effects
- Height Variation and Slenderness Ratio Analysis
- Impact of Surrounding Buildings

VI. ACKNOWLEDGEMENT

We sincerely thank Mr. Siddharth Gautam, Assistant Professor, Department of Civil Engineering, MGM’s College of Engineering and Technology, NOIDA, for his consistent guidance, encouragement, and valuable insights throughout the project. We also gratefully acknowledge Dr. Ram Prakash, Head of the Department, for his unwavering support and cooperation, which significantly contributed to the successful execution and completion of our project work.

REFERENCES

[1] IS: 875 (2015), Indian Standard design loads (other than earthquake) for buildings and structures code of practice, part 3(wind loads), 3rd ed. New Delhi, 2015.

[2] Hong Kong Building Department, Code of practice on wind effects in Hong Kong 2019. Hongkong: Buildings Department, Hong Kong, 2019.

[3] Paras Nath, Sudhanshu Pandey, Nitesh Kumar Thakur, Deepak Sharma, Ritu Raj, “Effect of Wind Load on High Rise Building Having Chamfer Edge Corner”, The 10th National Conference on Wind Engineering.

[4] Nikhil Gaur, Ritu Raj, “Aerodynamic Mitigation by Corner Modification on Square Model Under Wind Loads employing CFD and Wind Tunnel”, Ain Shams Engineering Journal 13 (2022).

[5] Kasun Wijesooriya, Damith Mohotti, Chi-King Lee, Priyan Mendis, “A Technical Review of Computational Fluid Dynamics (CFD) Applications on Wind Design of Tall Buildings and Structures: Past, Present and Future”, Journal of Building Engineering 74 (2023).

[6] Rahul Kumar Meena, Ritu Raj, S Anbukumar, “Effect of Wind Load on Irregular Shape Tall Buildings Having

Different Corner Configuration", Indian Academy of Sciences (2022).

[7] Rahul Kumar Meena, Ritu Raj, S Anbukumar, "Wind Excited Action Around Tall Building Having Different Corner Configurations", Hindawi Advances in Civil Engineering (Vol. 2022).

[8] Prasenjit Sanyal, Sujit Kumar Dalui, "Effects of Side Ratio for 'Y' Plan Shaped Tall Building Under Wind Load", Building Simulation (2022).

[9] Najah Assainar, Sujit Kumar Dalui, "Aerodynamic Analysis of Pentagon-Shaped Tall Buildings", Asian Journal of Civil Engineering (2020).

[10] Gavane Vaishali, Prof. A. N. Shaikh, "Analysis of High-Rise Buildings with Different Shapes by Corner Modification & Considering Aerodynamic effect", IJARIIE-ISSN(O) (Vol.8 2022).

[11] Prasenjit Sanyal, Sujit Kumar Dalui, "Effect of Corner Modifications On 'Y' Plan Shaped Tall Building Under Wind Load", Research Gate (2022).

[12] Yong-gui Li, Peng Liu, Yi Li, and Jia Quan, "Wind Loads Characteristics of Irregular Shaped High-Rise Buildings", Advances in Structural Engineering (Vol.26 2023).

[13] Pradeep K Goyal, Sonia Kumari, Shivani Singh, Rahul Kumar Saroj, Rahul Kumar Meena, Ritu Raj, "Numerical Study of Wind Loads on Y Plan-Shaped Tall Building Using CFD", Civil Engineering Journal (Vol.8 2022).

[14] Dr. P.N Modi, Dr. S.M Seth, "Hydraulics & Fluid Mechanics including Hydraulic Machines" 21st Edition (2017).