STM JOURNALS

Journal of
Polymer & Composites

ISSN: 2321-2810 (Online)
ISSN: 2321-8525 (Print)
Volume 13, Special Issue 1, 2025
DOI (Journal): 10.37591/JoPC

https://journals.stmjournals.com/jopc

Research JoPC

Aerodynamic Influences on Structures with Varied Plan Configurations

Rahul Kumar Meena [1], Abhishek Prakash Paswan [2], Prateek Roshan [3], Manvendra Verma[4,*]

Abstract

The wind-induced effects on tall buildings demonstrate a level of unpredictability, attributed to the intricate influence of the building's plan shape on the distribution of wind loads. The oscillations experienced by tall structures, both longitudinally and laterally, are influenced by factors such as wind direction, building configuration, height, and structural characteristics. This investigation is centred around examining the impact of wind on tall buildings featuring diverse plan cross-sectional shapes. The analysis employs Computational Fluid Dynamics (CFD) through ANSYS CFX version 2023 R2, scaled at 1:200. The study places particular emphasis on exploring aerodynamic modifications to building shapes, delving into their effectiveness and acceptance within the wind engineering community. Noteworthy findings reveal the inherent positivity of wind forces on the windward face and negativity on the leeward face. The graphical presentation of results, including pressure contours and streamlines in both plan and elevation, adds a visual dimension to the research. Significantly, the study establishes a direct correlation between the plan layout and size of the model and wind pressure distribution on the windward face. An intriguing observation is the similar pressure distribution patterns on both windward and leeward faces across all models. These discoveries contribute valuable insights for structural designers and architects, offering guidance on incorporating various plan shapes into tall building designs. Recognizing the nuanced influence of plan layouts on wind pressure distribution becomes crucial in crafting tall building designs that are not only structurally robust but also architecturally viable.

Keywords: Effects of wind; computational fluid dynamics (CFD); pressure distribution; flow patterns; tall buildings

INTRODUCTION

Tall structures have several uses, and there are a number of reasons why they are necessary. These fundamental factors—urbanization and population density, for example—indicate the need for tall structures. As cities expand and the population grows, towering buildings offer a useful way to fit more people and activities into constrained spaces. There could not be as much horizontal room for building due to land shortages in highly populated places. The increasing need for residential and commercial properties, coupled with advancements in construction methodologies, has led to a substantial surge in the construction of tall or high-rise structures globally. With the increasing height of buildings and advancements in construction methodologies leading to lighter and more flexible structures, there is a heightened vulnerability to substantial amplitude vibrations (1). The slender nature of a tall building is characterized by the ratio of its height to the smallest plan dimension. Tall

***Author for Correspondence**
Manvendra Verma

[1]Assistant Professor, Department of Civil Engineering, NIT, Delhi, India
[2]PhD Scholar, Department of Civil Engineering, Delhi Technological University, Delhi, India
[3]PhD Scholar, Department of Civil Engineering, Delhi Technological University, Delhi, India
[4]Assistant Professor, Department of Civil Engineering, GLA University, Mathura, Uttar Pradesh, India

Received Date: April 23, 2024
Accepted Date: August 08, 2024
Published Date: December 18, 2024

Citation: Rahul Kumar Meena, Abhishek Prakash Paswan, Prateek Roshan, Manvendra Verma. Aerodynamic Influences on Structures with Varied Plan Configurations. Journal of Polymer & Composites. 2025; 13(Special Issue 1): S377–S388p.

buildings with a slenderness exceeding five or possessing a fundamental frequency below 0.2 Hz are prone to experiencing significant amplitude vibrations (2). Substantial oscillations of considerable magnitude have the potential to cause harm to structural integrity, possibly resulting in structural damage or, in extreme cases, catastrophic failure (3). Wind-induced vibrations pose concerns for occupant comfort, with perceptions influenced by the level of horizontal acceleration. Various standards exist to gauge acceptable vibrations in tall buildings subjected to wind loads. These standards establish a permissible frequency range for building vibrations, typically ranging from 0.1 to 1.0 Hz (4). Factors like root mean square (RMS) or peak response values, recurrence intervals, and the building's structural characteristics further delineate acceptable frequency levels. Considering the significance of wind-induced vibration in ensuring both structural safety and occupant comfort, it becomes imperative to devise strategies to minimize vibrations in tall buildings subjected to wind forces. High-rise structures often experience cross-dynamic responses caused by vortex-induced vibration and along-wind dynamic response resulting from turbulence-induced buffeting and torsional vibrations originating from uneven loading (5). The predominant factor influencing strength and serviceability design criteria is the cross-wind response. Along-wind vibration is initiated by buffeting loads aligned with the wind direction among the various types of vibrations. Torsional vibration, on the other hand, results from twisting torque along the vertical axis of the building, arising from the uneven distribution of wind loads (6). Traditionally, three main strategies have been used for mitigating wind-induced vibration: adjusting the structural design, incorporating passive or active auxiliary dampers, and making aerodynamic modifications. The goal of modifying the structural design is to reduce wind-induced vibrations by altering properties like mass and stiffness. However, this approach is economically impractical, as it entails increasing the cross-sectional area of the framing system, leading to higher construction costs.

The active control system reduces wind-induced vibrations in tall buildings by modifying structural damping or stiffness through a hydraulic or electro-mechanical actuator system. This system can attenuate vibrations across a broad frequency spectrum, adapting to different excitation conditions. Furthermore, it enables precise, controlled application of force within a short timeframe (4,7). However, a drawback is the substantial energy consumption required to operate the actuators and the potential risk of destabilization if parameters are altered abruptly (8). To mitigate vibrations, a passive control system operates without sensors, actuators, or an external power supply. Damping devices control vibrations induced by wind loads or seismic activity across diverse structures like cables (9), traffic signals(10), and buildings. Among these devices, tuned mass dampers (TMDs) are the preferred choice for managing vibration energy in tall buildings. A TMD comprises a system with a spring and a mass that oscillates out of sync with the structural movement, effectively dissipating vibration energy (11).

An alternative strategy for mitigating wind loads involves the modification of the aerodynamic shape of tall buildings. The shape of an object significantly influences its aerodynamics, playing a crucial role in reducing wind loads on structures. This approach has gained substantial attention in the wind engineering community due to its widespread adoption in recent years. The effectiveness of aerodynamic modifications is typically assessed through wind tunnel experiments and Computational Fluid Dynamics (CFD) simulations. Optimizing aerodynamic shapes has proven to be a key factor in achieving optimal building designs while minimizing stress induced by wind. Current research indicates that the integration of these algorithms with advanced CFD modeling shows promise in overcoming challenges associated with Atmospheric Boundary Layer (ABL) wind tunnel testing. Meena et al.(12) discussed the effects of wind on the bracing system used in various types of multistoried steel buildings. Pal et al.(13) Conducted a comparative analysis of wind-induced mutual interference effects on tall buildings with square and fish plan shapes. Examined the maximum efficiency in terms of wind-induced mutual interference pressure and the exhibited base shear, finding that the square plan-shaped model demonstrated optimal performance under full blockage conditions(14–16). The configuration of a building plays a pivotal role in influencing the dynamic forces induced by wind and subsequent structural responses. Meticulous design of tall building components and shapes is paramount to mitigating wind excitation, ensuring cost-effectiveness. A strategic alteration of flow patterns around structures through passive aerodynamic modifications, often achieved through innovative building shapes, stands out as a potent and versatile construction approach. This technique proves highly

ISSN: 2321-2810 (Online), ISSN: 2321-8525 (Print)

effective in substantially minimizing the impact of wind forces. Despite the abundance of wind effects data for regular building models in various international standards, exploring and implementing novel strategies for shaping buildings offer exciting avenues for further advancements in wind engineering (17–23) However, a notable limitation lies in the presentation of wind load data primarily for rectangular models. The dominance of wind effects is intricately linked to the plan shape of tall buildings. While various studies have explored different plan shapes, there remains a gap in addressing corner configurations or modifications in these areas. The current study aims to fill this gap, providing valuable insights for structural designers in considering comprehensive corner configurations and modifications for tall building designs.

This research delves into the examination of wind effects on a building with eight sides, exploring various configurations like the Z-plan shape and a more intricate complex shape. The investigation employs the Computational Fluid Dynamics (CFD) tool ANSYS CFX, specifically the 2023 R2 version, to analyze wind effects. Simulating the wind profile within a virtual wind tunnel using a power-law approach, the study visually represents the wind-generated effects through graphical elements like pressure contours and streamlines in both plan and elevation. Subsequent sections of this study will thoroughly discuss and interpret the findings derived from this investigation.

NUMERICAL ANALYSIS BY ANSYS CFX

Model

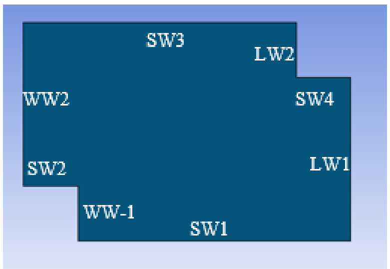
In the ANSYS CFX Design Module, the modeling process for studying wind effects on tall buildings involves the creation of a scale model with a length scale of 1:200, aligning with wind tunnel manual specifications. This meticulous adherence to the length scale ensures accurate representation and consistency with experimental standards. The model is configured with eight sides, a key parameter in capturing the complexity of the building's geometry. This modeling approach facilitates a detailed examination of wind effects on tall structures, providing valuable insights into their aerodynamic behavior and structural response. Fig. 1 shows the building model with the plan and elevation.

Boundary Condition

In the ANSYS CFX Design Module, articulating precise boundary conditions is crucial for establishing a well-defined problem that yields a unique solution. The accuracy of the solution is markedly influenced by the clarity and accuracy of the specified boundary conditions. This entails identifying key elements such as the solver, inlet, outlet, walls, and other parameters essential for effectively defining the problem at hand. The meticulous definition of these boundary conditions ensures a robust foundation for obtaining accurate and meaningful results within the simulation. zidan et al.(24) Proposed recommendations for the numerical simulation involve defining the domain and specifying boundary conditions. In this study, the domain pertains to a virtual wind tunnel, constituting a fluid domain. Within the ANSYS CFX software, the fluid properties are pre-established, while the building faces are designated as walls. Additionally, the ANSYS CFX incorporates pre-defined wall functions to streamline the simulation process. These measures collectively contribute to the effective execution of the numerical simulation. Fig 2 shows the computational domain dimensions and boundary conditions.

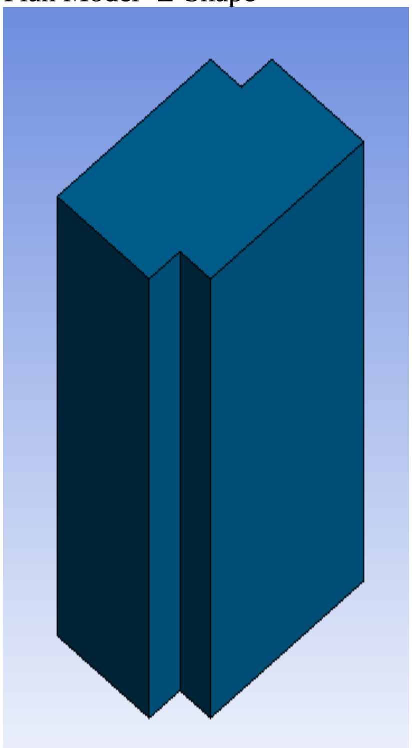
Meshing

Crafting a high-quality mesh is not a guarantee of an optimal solution, but a poorly constructed mesh invariably leads to suboptimal outcomes. The meshing technique, illustrated in Figure. 3, includes inflation, as depicted in Figure. 3(b) for the Z-shaped building and Figure. 3(c) for the U-shaped model. This inflation is strategically implemented to account for the effects of air in the separation zone. The challenge lies in generating an appropriate mesh, as its intricacy is contingent upon the complexity of the problem at hand. Due to the limitations of governing equations in applying to arbitrary shapes, the computer simulation cannot directly operate on the actual geometry. Mesh elements play a critical role in enabling the predictable and mathematical solution of governing equations within defined volumes. Figure. 3(a) showcases full domain meshing, with (b) and (c) demonstrating building meshing, which is finer compared to other meshing methods. Additionally, Figure. 3(d) illustrates ground meshing, strategically introduced to mitigate unusual flow patterns and ensure the simulation's accuracy. In the context of investigating wind effects using ANSYS CFX, the meshing strategy plays a pivotal role in achieving accurate and reliable simulation results. Meshing is the process of discretizing the geometric domain into smaller elements, allowing for the numerical solution of governing equations that describe fluid flow and other physical phenomena. In the ANSYS CFX simulation for wind effects on structures, a carefully crafted mesh is crucial for capturing the intricate details of the flow field and ensuring computational efficiency. To facilitate an effective representation of the complex geometry associated with tall buildings, a structured meshing approach is often employed. Structured meshing involves dividing the computational domain into logically arranged elements, providing a systematic grid that aligns with the geometry. This type of meshing is particularly advantageous for simulations involving architectural structures, as it enables a refined representation of surfaces and boundaries. Furthermore, the meshing process involves considerations such as element size, especially in areas of interest where the wind effects are significant, such as around corners, edges, and other aerodynamically sensitive regions. Fine meshing in these critical zones allows for a more accurate capture of pressure distribution and flow patterns. Inflation layers are another essential aspect of meshing, especially in the vicinity of building surfaces. These layers help resolve the boundary layer, where the flow experiences significant changes in velocity and pressure. By incorporating inflation layers, the simulation can more effectively capture the boundary layer effects, enhancing the overall accuracy of the wind effects prediction (25–30).

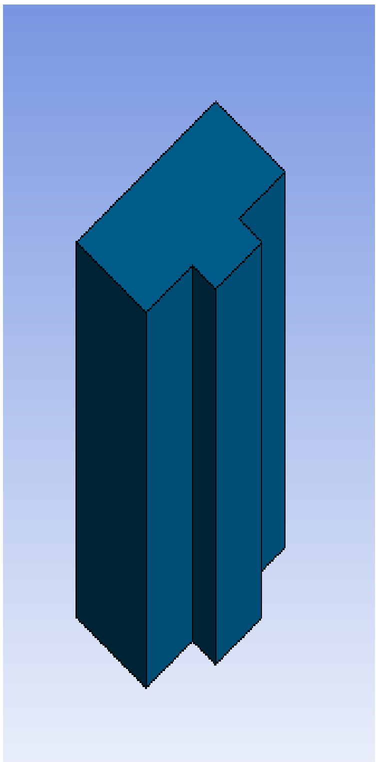


Plan Model- Z-Shape

Plan Model- T Shape



Elevation model - Z Shape



Elevation model- T Shape

Figure 1. Building Model with plan and elevation

Journal of Polymer & Composites
Volume 13, Special Issue 1
ISSN: 2321-2810 (Online), ISSN: 2321-8525 (Print)

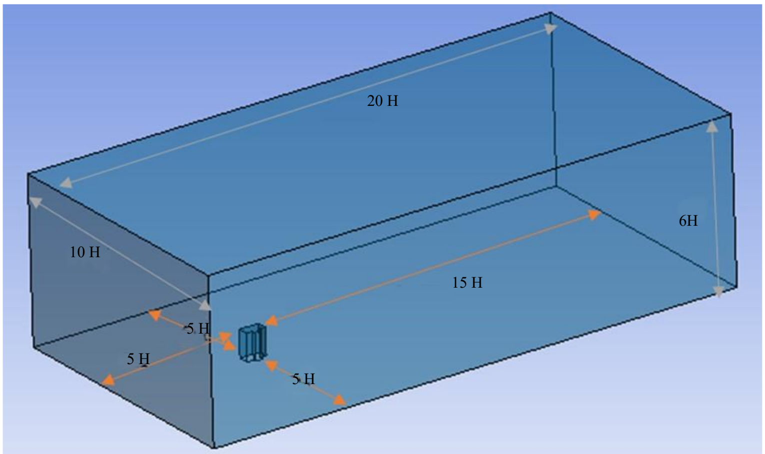
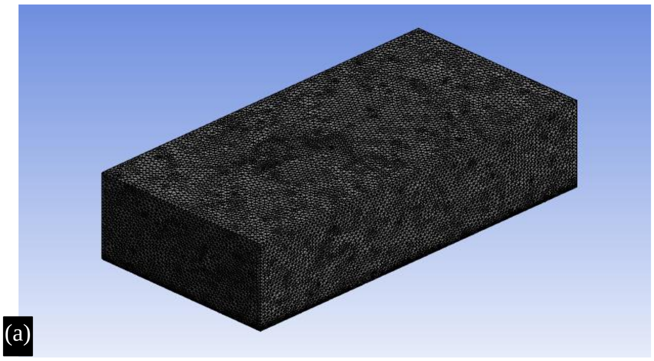
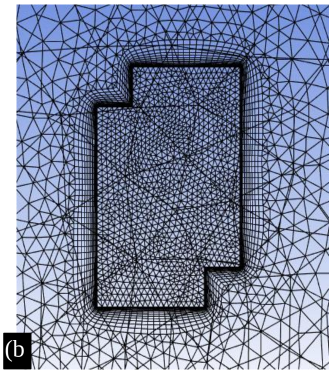


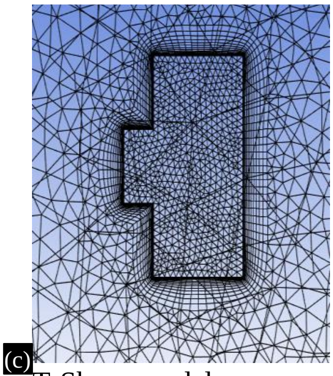
Figure 2. Computational domain dimensions and boundary condition.



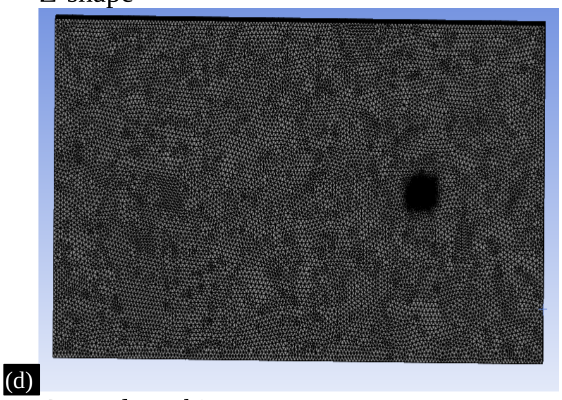
Domain meshing



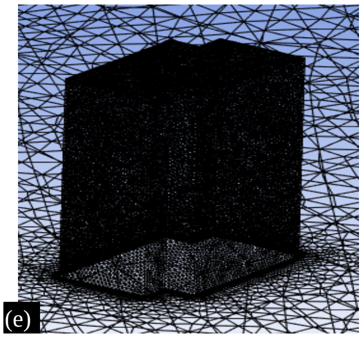
Z-shape



T-Shape model



Ground meshing



Building meshing

Figure 3. Meshing for different component of the building.

RESULT AND DISCUSSION

Velocity Profile and Turbulent Profile

In addition to the Coriolis force, pressure differences in various parts of the Earth cause wind movement. An atmospheric boundary layer forms as a result of the interaction between wind and the Earth's surface, especially close to the ground. Increased wind turbulence is caused by obstructions on the ground. The ground's roughness and the drag produced by nearby projections that impede the wind both affect the vertical profile of wind speed. The elevation at which drag effects stop is indicated by the gradient height, and the accompanying velocity is called the gradient velocity. The atmospheric boundary layer also denotes the height at which terrain affects wind velocity.

In accordance with the Power Law, depicted in Figure 4, the wind speed profile within the atmospheric boundary layer follows the equation (1). This law encapsulates the relationship between wind speed and height, providing a foundational understanding of how wind velocity evolves in the lower atmosphere, considering the impact of various factors on its vertical profile.

$$\frac{U}{U_H} = \left(\frac{Z}{Z_H}\right)^{\alpha} \tag{1}$$

Expressed as U, the horizontal wind speed at elevation Z is represented mathematically. U_H corresponds to the speed at the reference elevation ZH, set at 10 m/s. The parameter α, which fluctuates based on ground roughness and holds a value of 0.147 for terrain category 2, plays a pivotal role in the equation. The reference elevation Z_H is standardized at 1.0 m. This mathematical formulation encapsulates the nuanced relationship between wind speed, elevation, ground roughness, and reference parameters, offering a comprehensive framework for understanding wind behaviour in the specified terrain category.

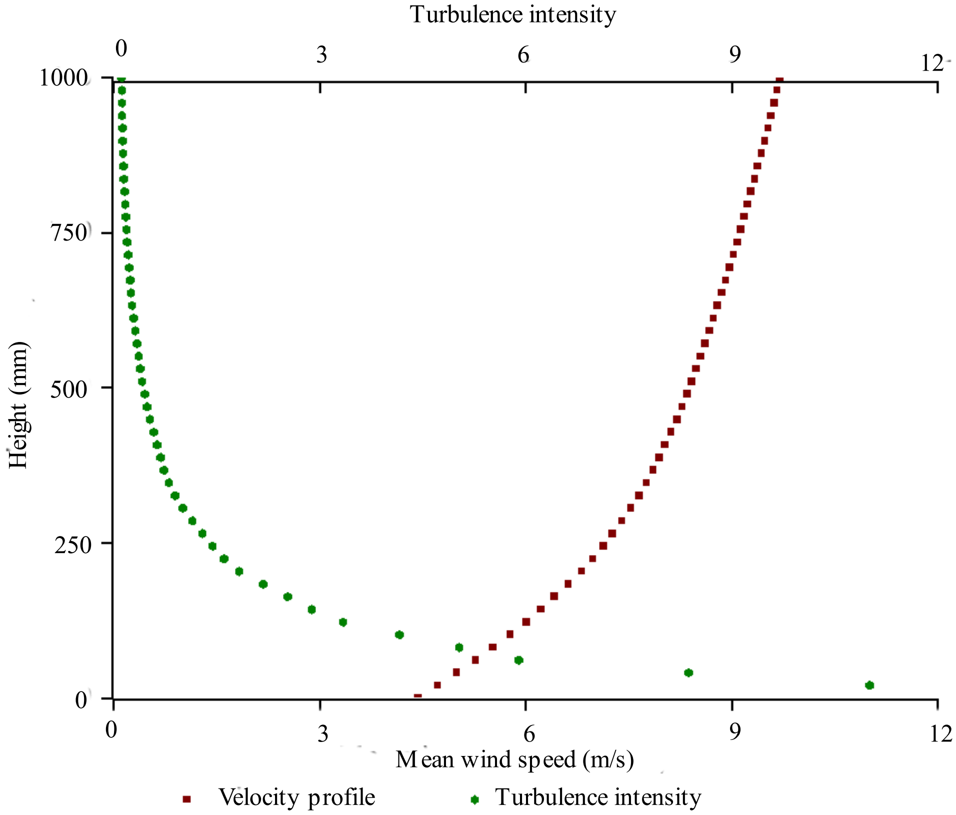


Figure 4. Velocity and turbulence intensity.

Journal of Polymer & Composites
Volume 13, Special Issue 1
ISSN: 2321-2810 (Online), ISSN: 2321-8525 (Print)

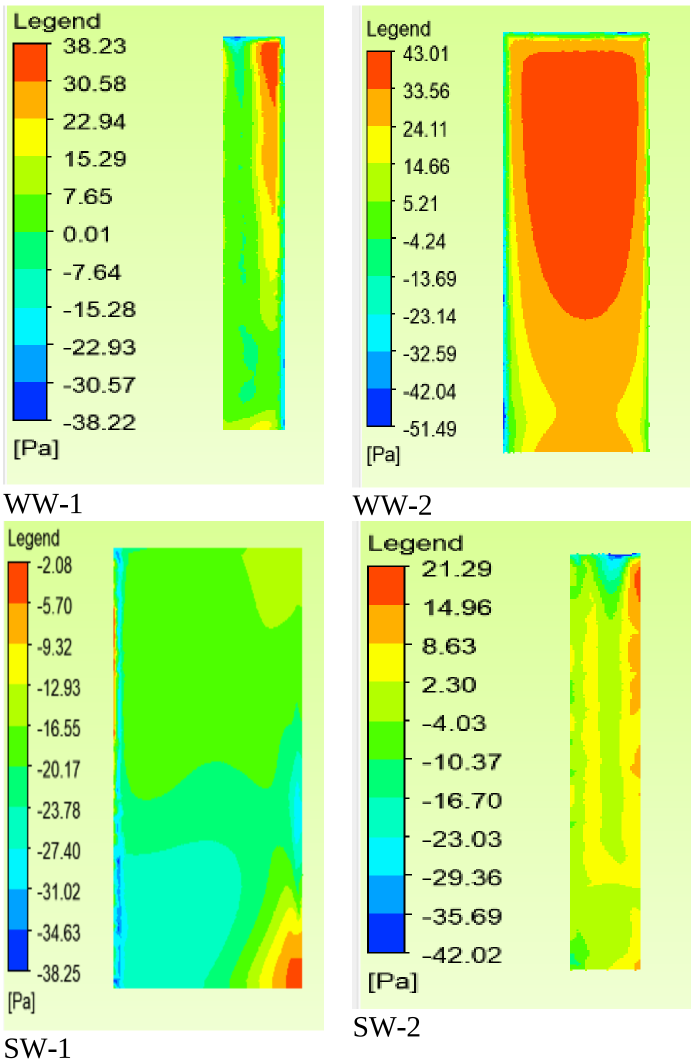
In Figure. 4, it is evident that the wind speed near the surface is notably reduced, primarily due to the friction between the surface and the moving air. The wind profile observed in this study closely resembles the wind profile outlined in ASCE 7 and aligns with findings from prior research in this domain.

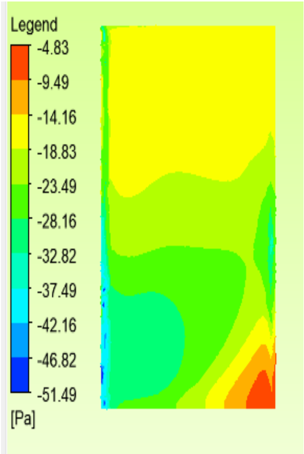
Pressure Contour on Side Face

Pressure contour on Z-shape

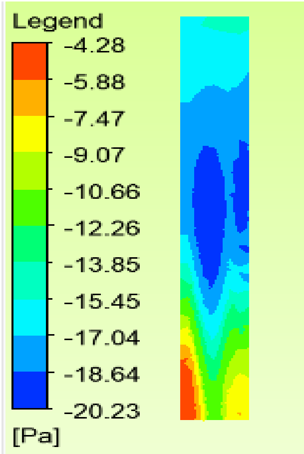
The generation of pressure contours for the Z-plan-shaped building entails the representation of pressure distribution across the entire structure. This visual depiction serves to illustrate the diverse pressure magnitudes at different locations, providing valuable insights into the specific aerodynamic characteristics inherent to the Z-plan shape. The practice of visualizing pressure contours is considered standard in the examination of wind effects on architectural structures and has been widely incorporated in analogous research within the discipline.

In Figure 5, the pressure on the windward face is denoted as "ww," on the leeward face as "lw," and on the side wall as "sw." Notably, the pressures on the windward faces exhibit a nearly identical nature, while on the leeward face, it is characterized by a negative magnitude. These pressure contours, showcasing the variation in pressure across the elevation of the building model, contribute to a comprehensive understanding of the intricate dynamics influenced by the Z-plan shape.

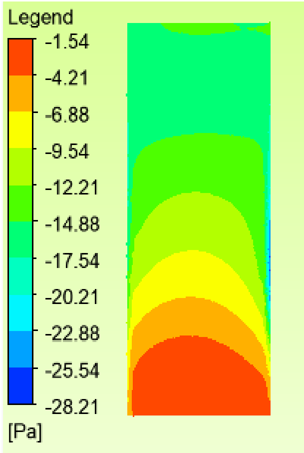




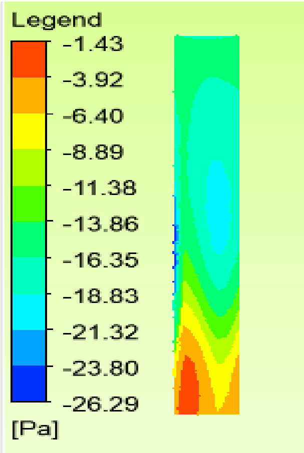
SW-3



SW-4



LW-1



LW-2

Figure 5. Pressure contour for 0 wind for Z plan shape model

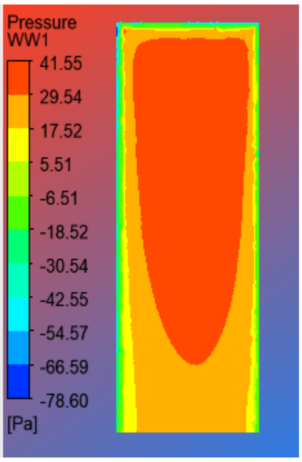
Pressure contour on T-shape:

Examining the wind pressure distribution on an irregular plan-shaped building with a T-type configuration involves analyzing how wind forces vary across the structure. This investigation aims to understand the nuanced aerodynamic effects specific to the irregular T-shaped plan. The exploration of wind pressure distribution on irregular plan shapes is a standard practice in the field of wind engineering and has been widely integrated into analogous studies. This study delves pressure contours on Figure 6,into the intricate details of how wind pressure interacts with the irregular T-shaped building, shedding light on the complex aerodynamic behavior associated with this particular architectural configuration.

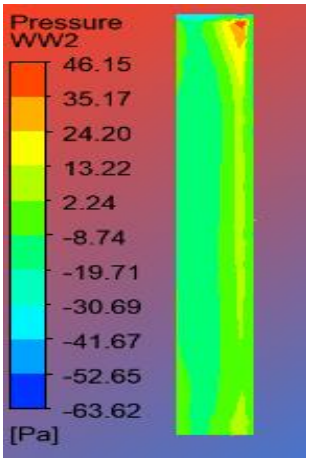
Pressure Contour on Roof Face

Investigating the pressure distribution on the roof of the building model entails a detailed analysis of how pressure varies across this specific architectural element. This examination aims to unravel the intricate dynamics of pressure interactions with the roof, providing valuable insights into the aerodynamic characteristics of the structure. The exploration of pressure distribution on building roofs is a standard practice in the realm of wind engineering and has been extensively utilized in analogous studies within the field. This study contributes to a nuanced understanding of how wind pressure is distributed on the roof, offering unique perspectives on the structural response to wind forces in this particular context. The pressure on the roof is increasing in suction as wind is moving from the wind ward side to leeward side and the contours are presented in figure 7 for both the type building model.

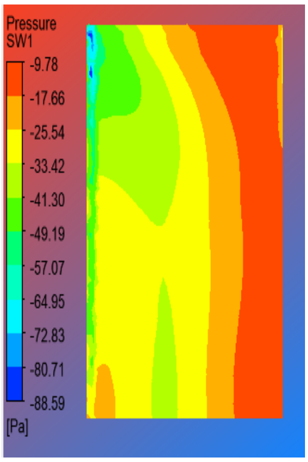
Journal of Polymer & Composites
Volume 13, Special Issue 1
ISSN: 2321-2810 (Online), ISSN: 2321-8525 (Print)



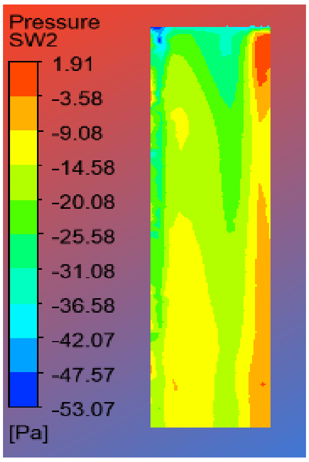
WW1



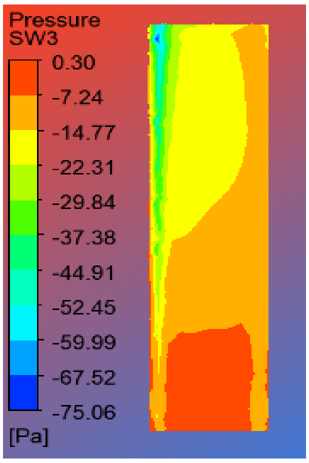
WW2



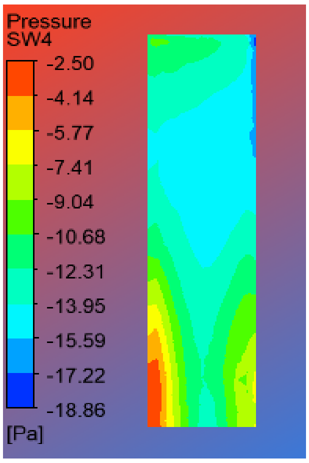
SW1



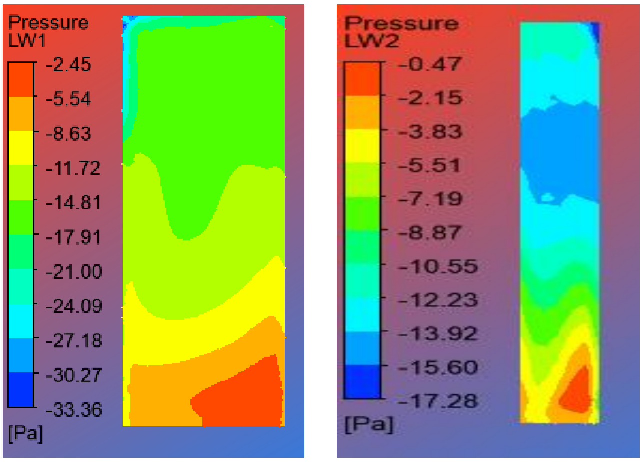
SW2



SW3

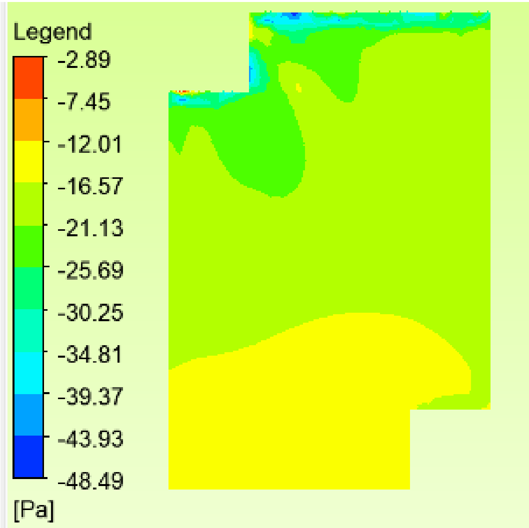


SW4

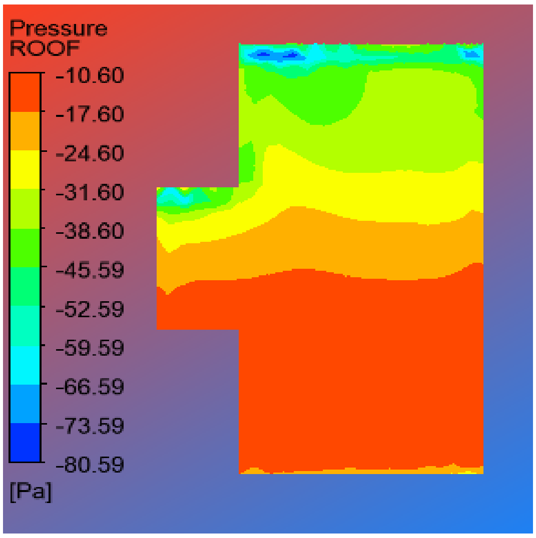


LW1 LW2

Figure 6. Pressure Contours for the T Shape Model

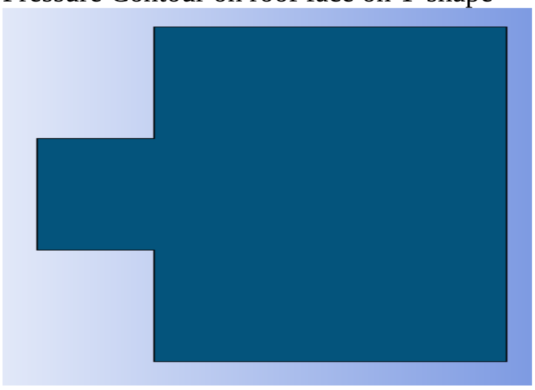


Pressure Contour on roof face on Z-shape



Pressure Contour on roof face on T-shape

Z shape building roof



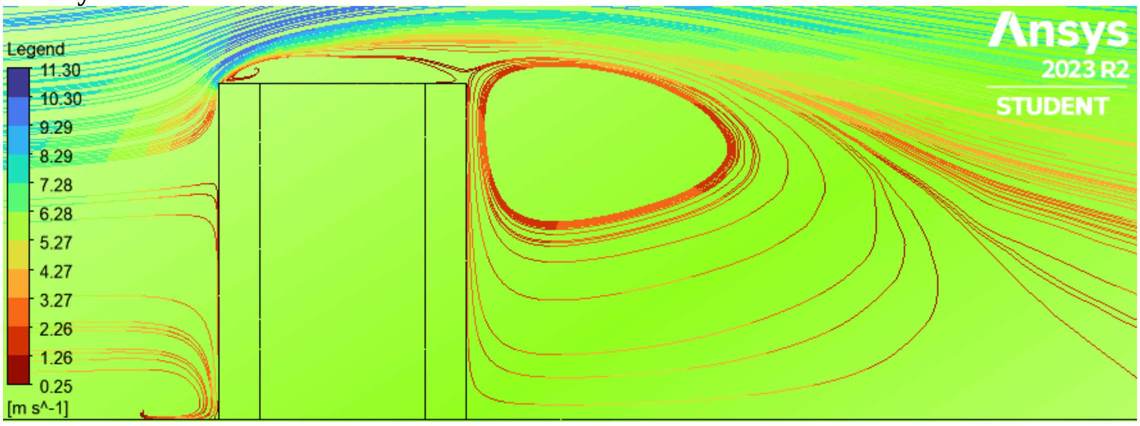
T Shape building roof

Figure 7. Pressure contours on building roof.

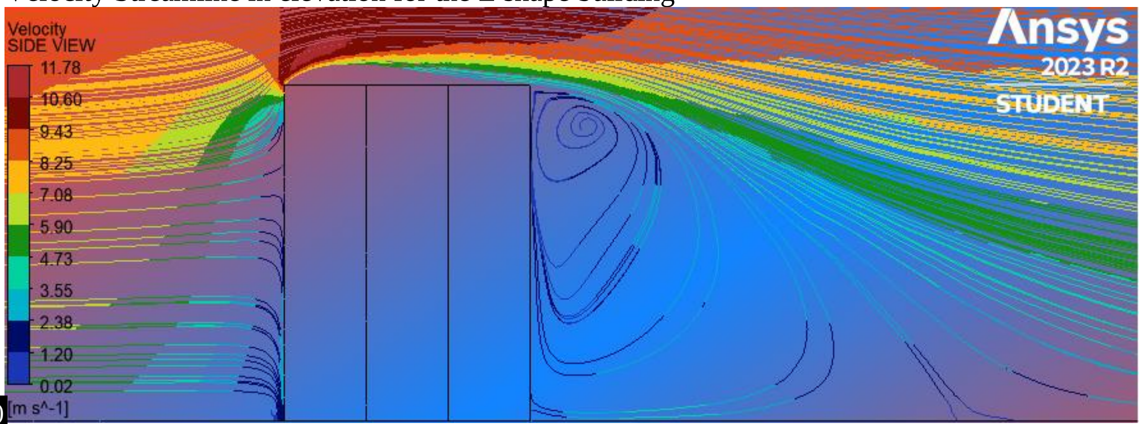
Velocity Streamlines

Exploring the velocity streamlines on both the elevation and plan of a high-rise building model serves as a crucial component in the investigation of wind effects. This analysis involves a detailed examination of how wind flows along the surfaces of the building, providing essential insights into the aerodynamic behaviour of the structure. The visualization of velocity streamlines is presented in elevation in figure 8 and while in the plan in figure 9, it is a standard and widely accepted practice in wind engineering studies, contributing to a comprehensive understanding of how wind interacts with high-rise buildings. This study uniquely contributes to the field by offering detailed insights into the complex flow patterns and velocities on both the vertical and horizontal aspects of the building, enriching our understanding of the wind effects on high-rise structures.

Velocity Streamline in Elevation



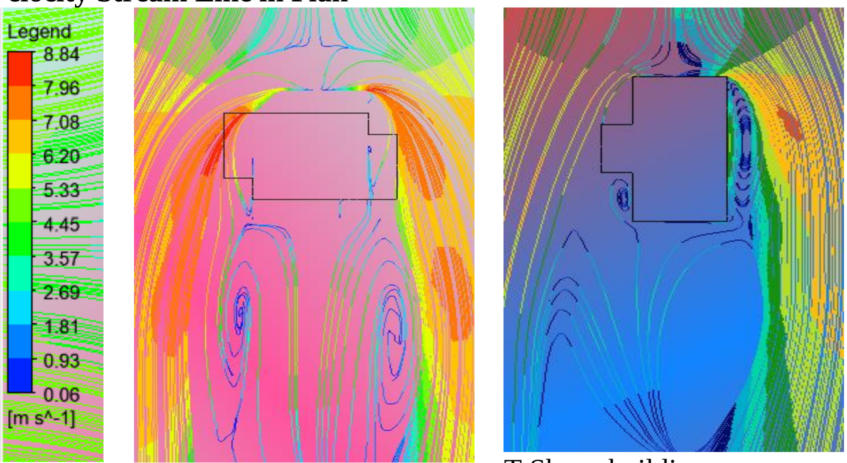
(a)
Velocity Streamline in elevation for the Z shape building



(b)
Velocity Streamline in elevation for the Z shape building

Figure 8. Velocity streamline in elevation for the high-rise building

Velocity Stream Line in Plan



Legend Z shape building T Shape building

Figure 9. Velocity Streamline in the plan for the high-rise building.

CONCLUSIONS

In this paper, a comparative analysis is conducted on the pressure contour, mean pressure coefficients, and velocity streamlines for high-rise building models with Z and T Shapes at a wind incidence angle of zero degree. The simulation for this research is performed using the k-ε Model. The key discoveries derived from this study are outlined below.

- With the use of CFD, wind impacts can be virtually simulated, offering an economical and effective way to investigate different design options and evaluate how they might affect both low- and high-rise structures.
- Because of CFD's versatility, it is possible to optimise building forms and configurations to improve performance under various wind conditions through iterative design revisions.
- Changes to the building's layout have a major effect on the models' leeward and side faces' suction.
- For both Z-shaped and T-shaped building styles, the windward face is under the most strain, while the side face is under the least.
- The supplied graphics provide a visual representation of velocity streamlines in both plan and elevation.
- The correctness of the results depends on how well the geometry model is meshed and how precisely flow physics is defined, simulating boundary layer wind tunnel conditions.
- The leeward face wind pressure distribution in this research emphasises the formation of vorticity, suggesting higher degrees of turbulence.

REFERENCES

1. Jafari M, Alipour A. Methodologies to mitigate wind-induced vibration of tall buildings: A state-of-the-art review. J Build Eng. 2021;33:1–60.
2. Kayvani K. Design of high-rise buildings : past , present and future. 23rd Australas Conf Mech Struct Mater. 2014;I:15–20.
3. Baskaran A. Wind engineering studies on tall buildings-transitions in research. Build Environ. 1993;28(1):1–19.
4. Ankireddi S, Yang HTY. Simple ATMD Control Methodology for Tall Buildings Subject to Wind Loads. J Struct Eng. 122(1).
5. Mendis P, Ngo T, Haritos N, Hira A, Samali B, Cheung J. Wind loading on tall buildings. Electron J Struct Eng. 2007;7:41–54.
6. Isyumov N, Case PC. Wind-Induced torsional loads and responses of buildings. Struct Congr 2000 Adv Technol Struct Eng. 2004;103(April 2000).
7. Tse KT, Kwok KCS, Hitchcock PA, Samali B, Huang MF. Vibration control of a wind-excited benchmark tall building with complex lateral-torsional modes of vibration. Adv Struct Eng. 2007;10(3):283–304.
8. Kim HS, Kang JW. Semi-active fuzzy control of a wind-excited tall building using multi-objective genetic algorithm. Eng Struct. 2012;41:242–57.
9. Ni YQ, Chen Y, Ko JM, Cao DQ. Neuro-control of cable vibration using semi-active magneto-rheological dampers. Eng Struct. 2002;24(3):295–307.
10. Jafari M, Sarkar PP, Alipour AA. A numerical simulation method in time domain to study wind-induced excitation of traffic signal structures and its mitigation. J Wind Eng Ind Aerodyn. 2019;193(July):103965.
11. Elias S, Matsagar V. Wind response control of tall buildings with a tuned mass damper. J Build Eng. 2018;15(February 2017):51–60.
12. MEENA RK, Awadhiya GP, Paswan AP, Jayant HK. Effects of Bracing System on Multistoryed Steel Building. 2018;
13. Pal S, Raj R, Anbukumar S. Comparative study of wind induced mutual interference effects on square and fish-plan shape tall buildings. Sādhanā. 2021;0123456789.

14. Meena RK, Raj R, Anbukumar S. Effect of wind load on irregular shape tall buildings having different corner configuration. Sadhana - Acad Proc Eng Sci [Internet]. 2022;47(3). Available from: https://doi.org/10.1007/s12046-022-01895-2
15. Meena RK, Raj R, Anbukumar S. Numerical Investigation of Wind Load on Side Ratio of High-Rise Buildings Numerical Investigation of Wind Load on Side Ratio of High-Rise Buildings. Springer Singapore; 2021.
16. Meena RK, Raj R, Anbukumar S. Wind Excited Action around Tall Building Having Different Corner Configurations. Adv Civ Eng. 2022;2022.
17. ASCE: 7-16(2017). Minimum Design Loads and Associated Criteria for Buildings and Other Structures. Structural Engineering Institute of the American Society of Civil Engineering, Reston. AMERICAN SOCIETY OF CIVIL ENGINEERS,Reston. 2017. 1–330 p.
18. ASCE: 49-12(2012). Wind Tunnel Testing for Buildings and Other Structures.Structural Engineering Institute of the American Society of Civil Engineering, Reston. American Society of Civil Engineers 1801 Alexander Bell Drive Reston, Virginia 20191 www.asce.org/pubs; 2012.
19. AS/NZS:1170.2(2011). Structural Design Actions - Part 2: Wind actions. Standards Australia/Standards New Zealand, Sydney. 2011. 101 p.
20. IS: 875 (2015). Indian Standard design loads (other than earthquake) for buildings and structures-code of practice,part 3(wind loads). BIS, New Delhi. 2015. 51 p.
21. ETHIOPIAN STANDARD. ES ISO 4354 (2012) (English): Wind actions on structures. Vol. 2012, Ethiopian Standards Agency(ESA) Ethiopia, Addis Ababa. 2012. 76 p.
22. MNBC. MYANMAR NATIONAL BUILDING CODE 2020. International relation and legal Section Department of Building Minis; 2020.
23. GB 50009-2001. NATIONAL STANDARD OF THE PEOPLE’S REPUBLIC OF CHINA. 2002.
24. Abu Zidan Y, Mendis P, Gunawardena T. Optimising the computational domain size in CFD simulations of tall buildings. Heliyon.
25. Sharma U, Gupta N, Verma M. Prediction of compressive strength of GGBFS and Flyash-based geopolymer composite by linear regression, lasso regression, and ridge regression. Asian J Civ Eng [Internet]. 2023;24(8):3399–3411. Available from: https://doi.org/10.1007/s42107-023-00721-2
26. Sharma U, Gupta N, Verma M. Prediction of Compressive Strength of Geopolymer Concrete using Artificial Neural Network. Asian J Civ Eng [Internet]. 2023;24(8):2837–2850. Available from: https://doi.org/10.1007/s42107-023-00678-2
27. Verma M. Prediction of compressive strength of geopolymer concrete using random forest machine and deep learning. Asian J Civ Eng [Internet]. 2023;24(7):2659–68. Available from: https://doi.org/10.1007/s42107-023-00670-w
28. Verma M. Prediction of compressive strength of geopolymer concrete by using ANN and GPR. Asian J Civ Eng [Internet]. 2023;24(8):2815–2823. Available from: https://doi.org/10.1007/s42107-023-00676-4
29. Verma M, Upreti K, Khan MR, Alam MS, Ghosh S, Singh P. Prediction of Compressive Strength of Geopolymer Concrete by Using Random Forest Algorithm. In: ICACIS 2022. 2023. p. 170–9.
30. Verma M, Upreti K, Dadhich P, Ghosh S, Khatri V, Singh P, et al. Prediction of Compressive Strength of Green Concrete by Artificial Neural Network. In: ICACIS 2022. 2023. p. 622–32.

