

Wind Pressure Analysis on the Selected Structure Using IS 875 (Part 3) 2015 and CFD (Computational Fluid Dynamics)

Raj Singh^{1,*}, Sneha Gupta²

Abstract

Wind-induced pressure plays a significant role in the safety and serviceability of tall buildings, slender vertical structures, and signboard hoardings. Currently, computational fluid dynamics (CFD) has become an effective supplemental tool in the analysis of the interaction of wind and structures. This paper presents a unified comparative study integrating multiple investigations various modal (i) Tall rectangular building (50 m × 50 m × 150 m), (ii) Rectangular Building (18 m × 12 m × 51 m) analyzed using the force coefficient method as per IS 875 (Part 3):2015, (iii) Tall Rectangular Building (24 m × 12 m × 96 m) analyzed using the Gust Factor Approach as per IS 875(Part 3):2015, (iv) Signboard Hoarding Structure analyzed using IS 875(Part 3):2015 with the CFD simulations performed in ANSYS Fluent. Three-dimensional (3D) numerical models were developed, and steady-state simulations employing the standard $k-\epsilon$ turbulence model were carried out under realistic boundary conditions. The CFD-predicted wind pressures show close agreement with codal values, with deviations generally within 0.3 - 4% for maximum windward pressures. However, CFD reveals significant spatial variations and higher suction zones on leeward and side faces that are not explicitly represented in codal methods. The study demonstrates that IS 875 and the gust factor method remain reliable for preliminary and global design, whereas CFD provides enhanced insight into localized pressure behavior critical for detailing, cladding design, and failure assessment. A combined use of codal provisions and CFD analysis is therefore recommended for improved accuracy and structural safety in wind-sensitive applications.

Keywords: Wind pressure, gust factor method, CFD, ANSYS Fluent

INTRODUCTION

Wind tunnel testing, numerical simulation, and code-based techniques have been used extensively to study wind-induced effects on tall buildings and similar bluff bodies because wind determines the

*Author for Correspondence

Raj Singh

E-mail: rajsingh.ce23@gmail.com

¹Student, Department of Civil Engineering Madan Mohan Malaviya University of Technology Gorakhpur, Uttar Pradesh, India

²Assistant Professor, Department of Civil Engineering Madan Mohan Malaviya University of Technology Gorakhpur, Uttar Pradesh, India

Received Date: July 23, 2026

Accepted Date: August 07, 2026

Published Date: August 10, 2026

Citation: Raj Singh, Sneha Gupta. Wind Pressure Analysis on the Selected Structure Using IS 875 (Part 3) 2015 and CFD (Computational Fluid Dynamics). Journal of Offshore Structure and Technology. 2026; 13(2): 70–80p.

strength and serviceability of tall, thin, and light structures [1, 2]. Many studies have examined the effects of building aspect ratio, plan shape, and geometric modifications on pressure distribution and global responses using benchmark configurations such as the CAARC tall building to validate computational fluid dynamics (CFD) tools and turbulence models [1, 3, 4]. Recent reviews have demonstrated a sharp increase in the use of CFD for tall building wind design, with applications ranging from straightforward bluff-body aerodynamics to complex fluid-structure interactions and façade performance [2, 5, 6]. Comparing parallel HFBB or CFD results with wind code predictions (such as AS/NZS, AIJ, Eurocode, and ASCE 7) has typically been done [4, 7–15].

Despite these advancements, the integrated use of IS 875 (Part 3):2015 and CFD for the routine structural design of wind-sensitive systems in the Indian context still has significant gaps. Without systematically quantifying the agreement and discrepancies between IS 875 and CFD predictions across multiple structural typologies and analysis methods (force coefficient method, gust factor approach, direct pressure evaluation), many investigations either apply codal procedures alone, perform CFD analysis in isolation, or treat only one idealized building form [15, 16]. Specifically, there is little comparative evidence in the literature regarding the capacity of IS 875 (Part 3):2015 to depict localized peak suctions, three-dimensional corner vortices, and wake-induced pressure fluctuations that govern cladding and connection design, particularly for tall buildings with sharp edges and signboard or hoarding structures [1, 5, 6, 17–22].

Furthermore, most CFD-based studies on tall buildings focus on advanced turbulence closures, such as LES or hybrid RANS–LES, and isolated high-rise buildings in relatively idealized surroundings [19]. CFD methods can reproduce detailed flow structures and peak pressures, which are computationally demanding and not yet widely adopted in everyday design practice. There is a need to clarify how far more accessible steady RANS models with standard k – ϵ turbulence, when carefully set up with realistic atmospheric boundary layer profiles and validated domain/mesh configurations, can support or refine codal design based on IS 875 (Part 3):2015 for typical Indian building geometries. Simultaneously, very few studies extend such comparisons to ancillary components, such as signboard hoardings, which are highly exposed, lightweight, and often critical in wind damage scenarios [3, 4, 23, 24].

In light of this, the current study compares wind pressures across four representative structural models:

1. A tall rectangular structure measuring 50 m $\times$ 50 m $\times$ 150 m
2. A rectangular structure measuring 18 m $\times$ 12 m $\times$ 51 m was examined using the force coefficient method
3. A tall rectangular building measuring 24 m $\times$ 12 m $\times$ 96 m was examined using the gust factor method
4. A signboard hoarding structure that was created in compliance with IS 875 (Part 3):2015

For every scenario, three-dimensional CFD simulations were run in ANSYS Fluent using a steady RANS framework with the standard k – ϵ model under realistically modeled atmospheric boundary layer conditions and systematically compared with wind pressures derived from codal procedures. The results of three-dimensional (3D) computational fluid dynamics (CFD) simulations using ANSYS Fluent were systematically compared with wind pressures derived from IS 875 (Part 3):2015 [25]. The primary goal of this study is to assess the degree of agreement between codal and numerical approaches, draw attention to the shortcomings of oversimplified analytical techniques, and illustrate the importance of CFD in capturing localized aerodynamic effects. The primary goals of this study are to examine the degree of agreement between codal and numerical approaches, highlight the drawbacks of simplified analytical techniques, and highlight the importance of CFD in capturing localized aerodynamic effects for safe, cost-effective, and dependable structure design [14, 17, 26–29].

METHODOLOGY

The current study evaluated wind-induced pressure on different structural models using an integrated analytical and numerical framework. The results of three-dimensional (3D) CFD simulations are routinely compared with wind pressure values computed in compliance with IS 875 (Part 3) 2015 [7, 8]. To ensure reliability and uniformity, identical boundary conditions, wind characteristics, and assessment parameters were maintained for all cases [11]. Throughout this section, relevant Figure references are included to clearly illustrate the modeling process, associated geometries, meshing details, and CFD simulation results [14–16, 27–29].

Structural Configurations Considered

Four representative structural configurations were selected to cover a broad range of geometries and

codal wind load evaluation methods prescribed in IS 875 (Part 3):2015 [7, 26]. The overall geometry and placement of the models within the computational domain are illustrated in Figures 1 to 4.

- *Tall building*: A $50\text{ m} \times 50\text{ m} \times 150\text{ m}$ [9] model as a rectangular prism subjected to oblique wind incidence (geometry shown in Figure 1).
- *Rectangular building* with dimensions of $18\text{ m} \times 12\text{ m} \times 51\text{ m}$ was analyzed using the force coefficient method [9] (Figure 2).
- *Tall rectangular building*: $24\text{ m} \times 12\text{ m} \times 96\text{ m}$ [10] was analyzed using the gust factor approach, with representative pressure contours shown in Figure 8.
- *Signboard hoarding structure*: [9] A flat rectangular panel supported on vertical members, representative of roadside or rooftop hoardings (geometry shown in Figure 4) [2, 25].

These configurations represent typical buildings, slender structures, and lightweight exposed systems that are commonly encountered in urban environments.

Click an object double-click to select an edge loop triple-click to select a solid

Ansys
2024 R1

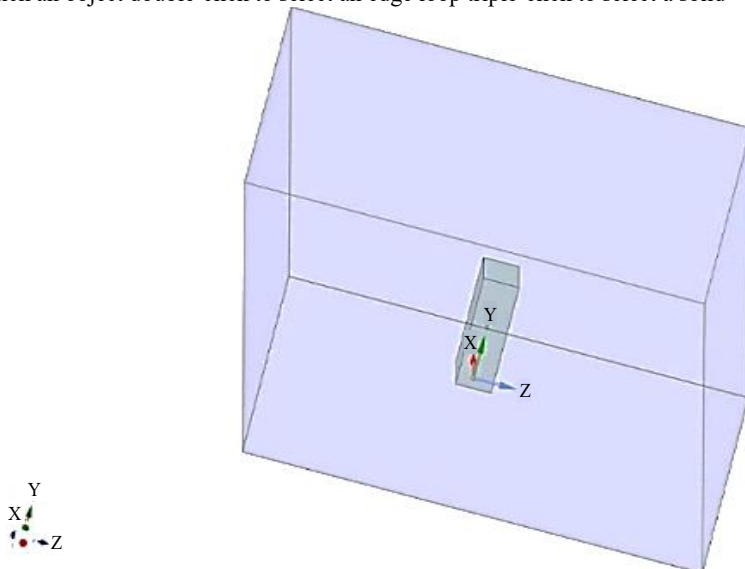


Figure 1. Building geometry ($50\text{ m} \times 50\text{ m} \times 150\text{ m}$) modeled in ANSYS (SpaceClaim).

Click an object double-click to select an edge loop triple-click to select a solid

Ansys
2024 R1

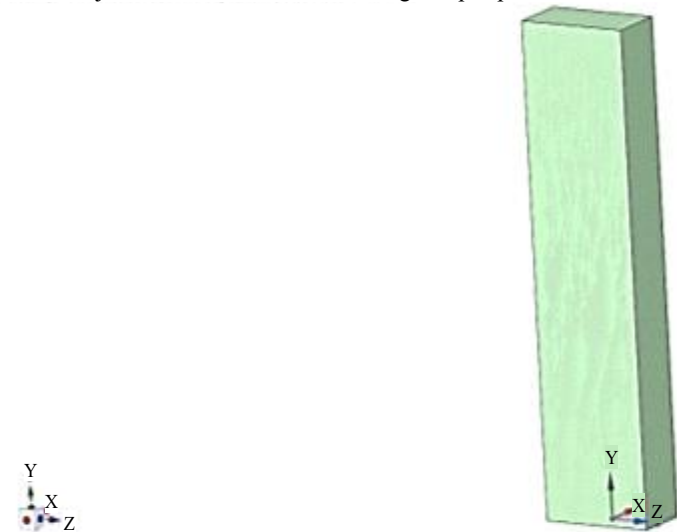


Figure 2. Building geometry ($18\text{ m} \times 12\text{ m} \times 51\text{ m}$) modeled in ANSYS (SpaceClaim).

Click an object double-click to select an edge loop triple-click to select a solid

Ansys
2024 R1

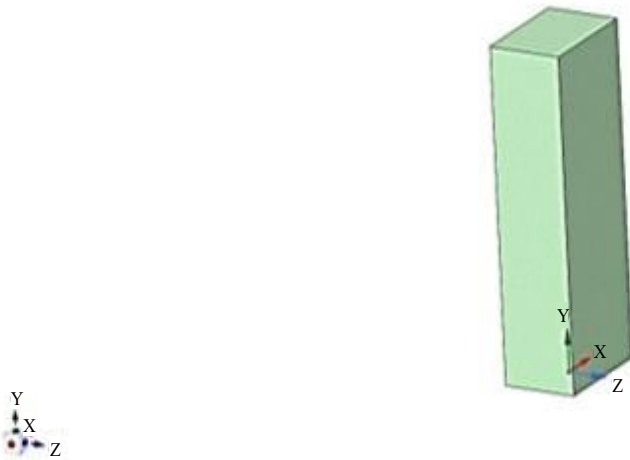


Figure 3. Building geometry (24 m × 12 m × 96 m) modeled in ANSYS.

Click an object double-click to select an edge loop triple-click to select a solid

Ansys
2024 R1

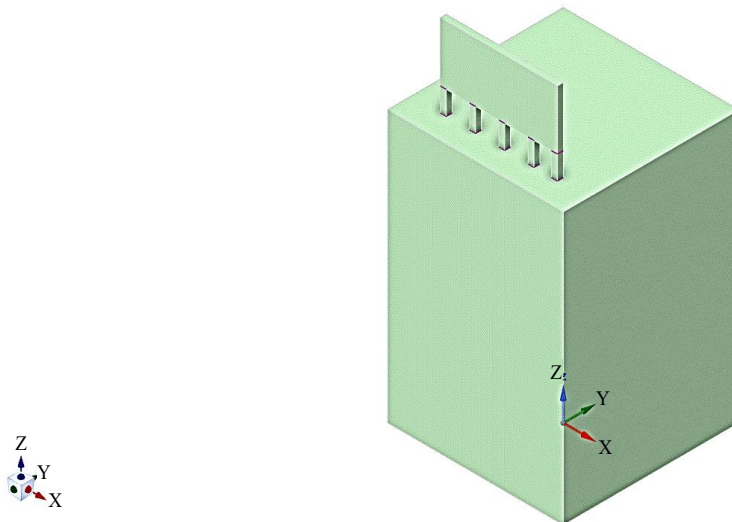


Figure 4. Signboard hoarding structure modeled in SpaceClaim (ANSYS).

IS 875 (Part 3) 2015 Wind Load Calculations

Wind pressure calculations were performed in accordance with IS 875 (Part 3):2015 using the appropriate analytical method for each structural configuration [12]. Graphical comparisons between codal and CFD-derived wind pressures are presented later in Table 1.

Design Wind Speed and Pressure

The design wind speed at height z was calculated using:

$$V_z = V_b \times k_1 \times k_2 \times k_3 \times k_4 \quad (1)$$

Where,

V_b = Basic wind speed,

k_1 = Risk coefficient,

k_2 = Terrain, height, and structure size factor;

k_3 = Topography factor,

k_4 = Importance factor.

The corresponding design wind pressure was obtained as:

$$p_z = 0.6 \times V_z^2 \quad (2)$$

Where,

p_z = Wind pressure

Direct Wind Pressure Method

For the 50 m × 50 m × 150 m building and signboard hoarding, the wind pressure was calculated directly using the design wind pressure p_z . This method assumes a uniform pressure distribution and is typically used for rigid and moderately tall structures [10]. The CFD pressure contours used for comparison with this method are shown in Figures 7–9 [9].

Force Coefficient Method

For the 18 × 12 × 51 m rectangular building, the wind force was evaluated using the force coefficient method prescribed in IS 875:

$$F = C_f A p_z \quad (3)$$

Where,

C_f = force coefficient based on the building shape and aspect ratio;

A = projected area normal to wind direction.

p_z = Wind pressure.

The equivalent wind pressure was obtained by distributing the calculated force over an exposed surface area. The comparison between the codal and CFD-derived windward pressures for this case is illustrated in Figure 6 [8, 18].

Gust Factor Approach

For the 24 × 12 × 96 m tall rectangular building, the gust factor approach was adopted to account for the dynamic amplification effects due to turbulence:

$$F = G C_f A p_z \quad (4)$$

Where,

G = Gust Factor Approach for turbulence intensity, size reduction factor, and structural response characteristics.

C_f = force coefficient based on the building shape and aspect ratio;

A = projected area normal to wind direction.

p_z = Wind pressure.

This methodology is particularly appropriate for tall and flexible structures, where dynamic wind effects are dominant. The gust factor pressure distribution values obtained from IS 875(Part 3): 2015 and comparison with the CFD are presented in Figure 7.

CFD Modeling and Simulation

Computational Domain and Geometry

Three-dimensional (3D) CFD models were developed for each structural configuration considered in this study. The structures were positioned within adequately sized computational domains extending upstream, downstream, and laterally to minimize blockage effects and ensure realistic flow development [8]. The computational domains and corresponding geometries for the tall building, vertical structural element, and signboard hoarding are illustrated in Figures 1–4, respectively.

Mesh Generation

The computational domains were discretized using unstructured and hybrid mesh systems for the numerical analysis. Local mesh refinement was applied near the structural surface, edges, and corners

to accurately capture high pressure gradients, flow separation, and vortex formation [8, 21, 23]. A gradual transition from fine near-wall cells to coarser far-field elements was maintained to ensure numerical stability and computational efficiency [8, 9]. The mesh arrangement for the tall building is shown in Figure 5, while the mesh details for the vertical structural element and signboard hoarding are presented in the corresponding Figures.

Boundary Conditions

The boundary conditions were defined uniformly for all numerical simulations to ensure consistency and comparability of the results.

- *Velocity inlet*: Prescribed wind velocity corresponding to the design wind speed
- *Pressure outlet*: Zero-gauge pressure condition
- *Structural surfaces*: No-slip wall condition
- *Ground surface*: No-slip wall condition

Data Extraction and Post-Processing

The static pressure contours of the windward and leeward faces of all structural configurations were obtained from the CFD simulation [22, 24]. The corresponding Figures show representative pressure distributions for the signboard hoarding, vertical structural element, and tall building. Figure 8 shows the specific changes in the front-face pressure on the signboard hoarding [10, 8].

Evaluation by Comparison

The spatial pressure distribution and maximum windward pressure percentage deviation were compared between wind pressure values obtained using IS 875 (Part 3) 2015 and CFD-derived results [28, 29]. This comparative framework would enable the assessment of the accuracy of codal provisions and show how the localized aerodynamic phenomena can be represented using CFD [7, 9].

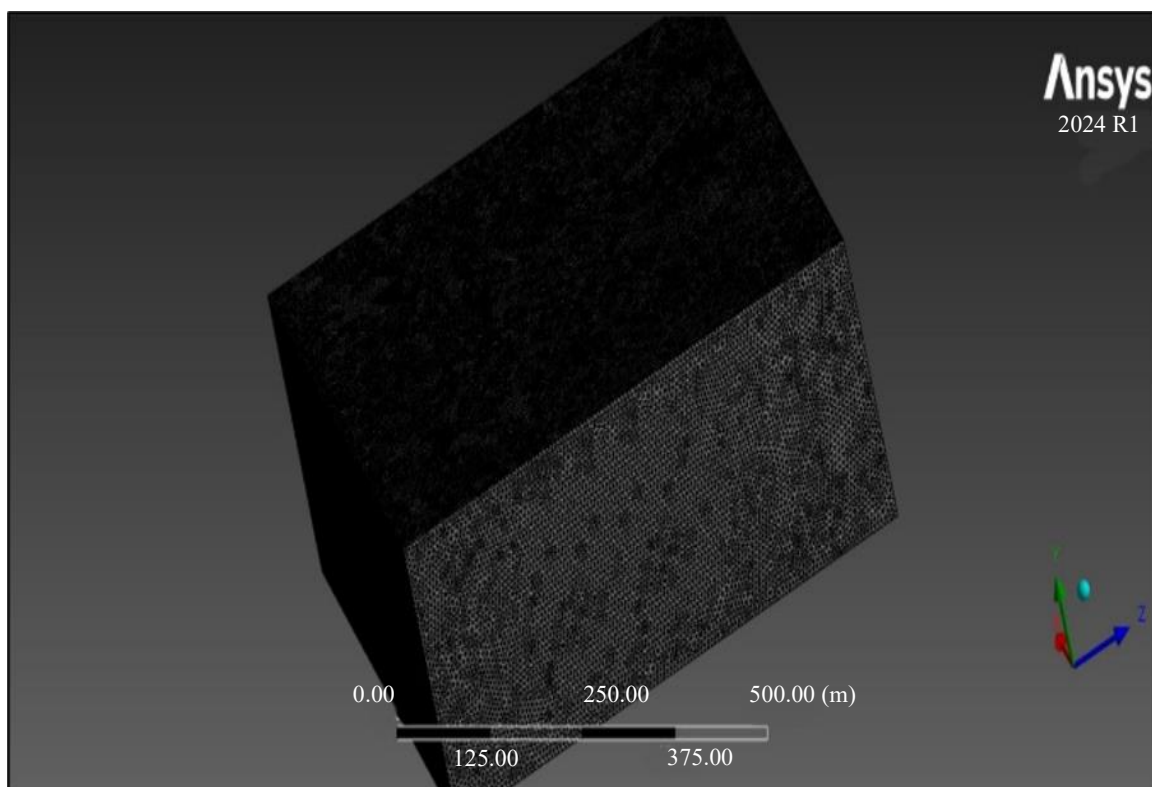


Figure 5. Meshing of the computational domain showing the very fine wall cells and gradual transition towards far-field regions.

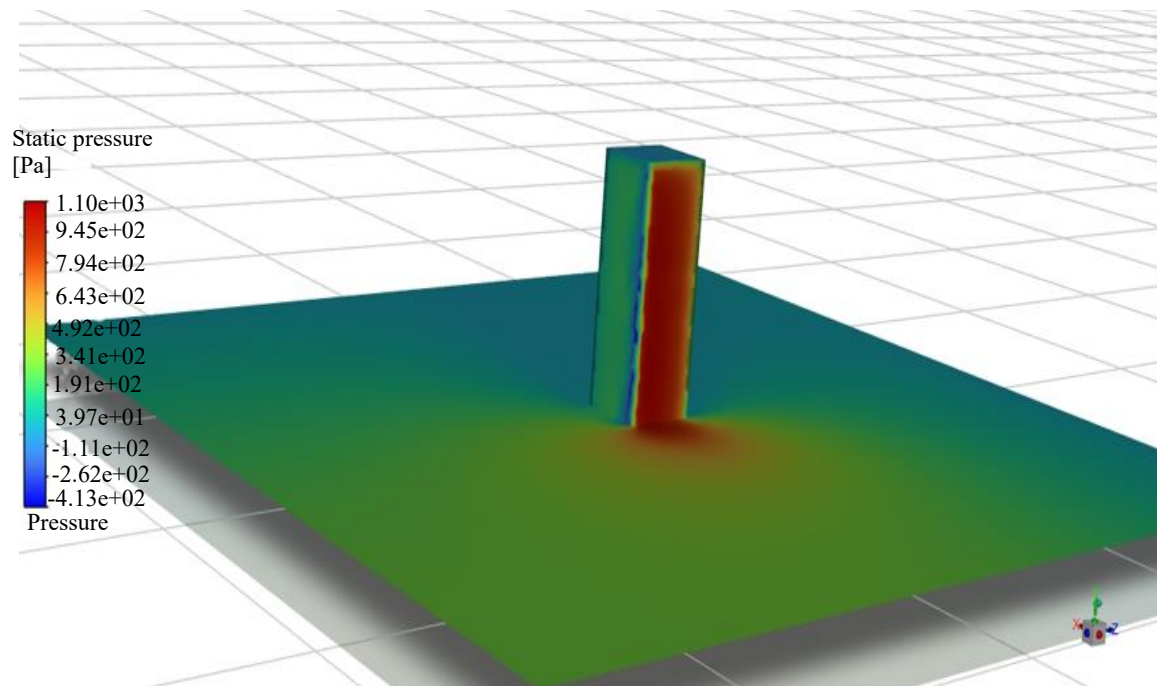


Figure 6. Static pressure distribution CFD-based on a 50 m × 50 m × 150 m building.

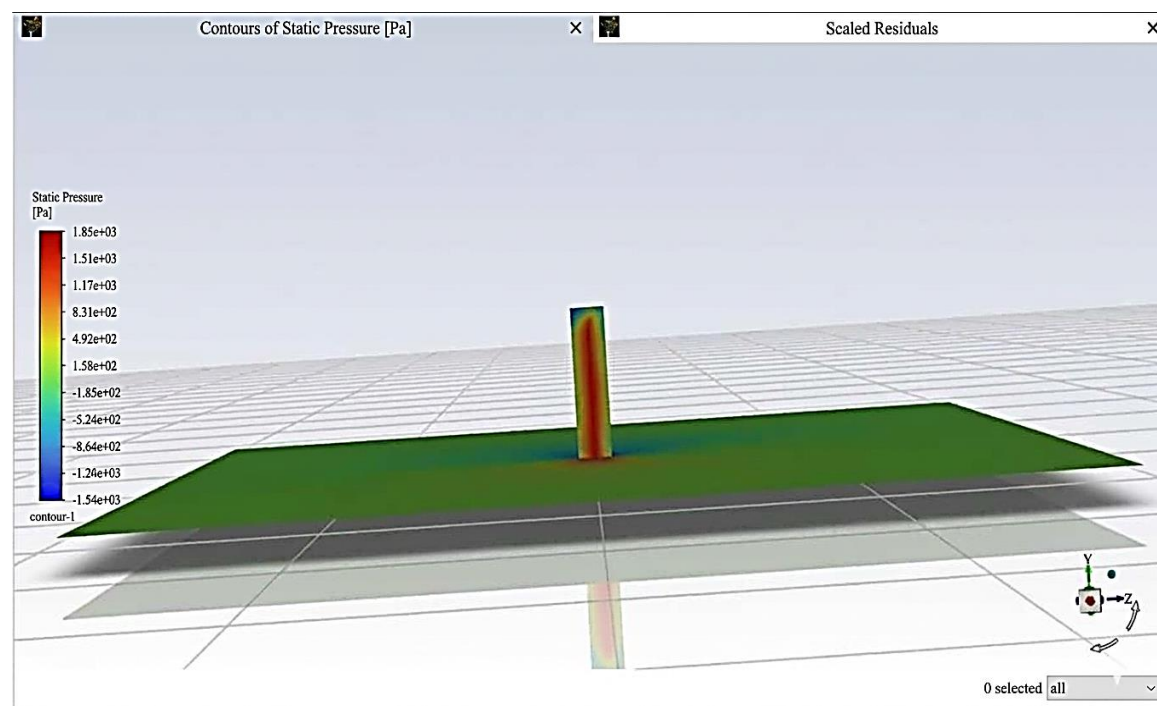


Figure 7. Computational fluid dynamics-based static pressure distribution for the 24 m × 12 m × 96 m building.

RESULTS

This section presents the results of numerical calculations of wind flows with the assistance of CFD and analyzes them against the values of wind pressure calculated according to IS 875 (Part 3) 2015 [26].

Wind Pressure Characteristics from CFD Simulations

The wind pressure characteristics were computed using CFD. The computations showed that all the structural types exhibited consistent aerodynamic performance.

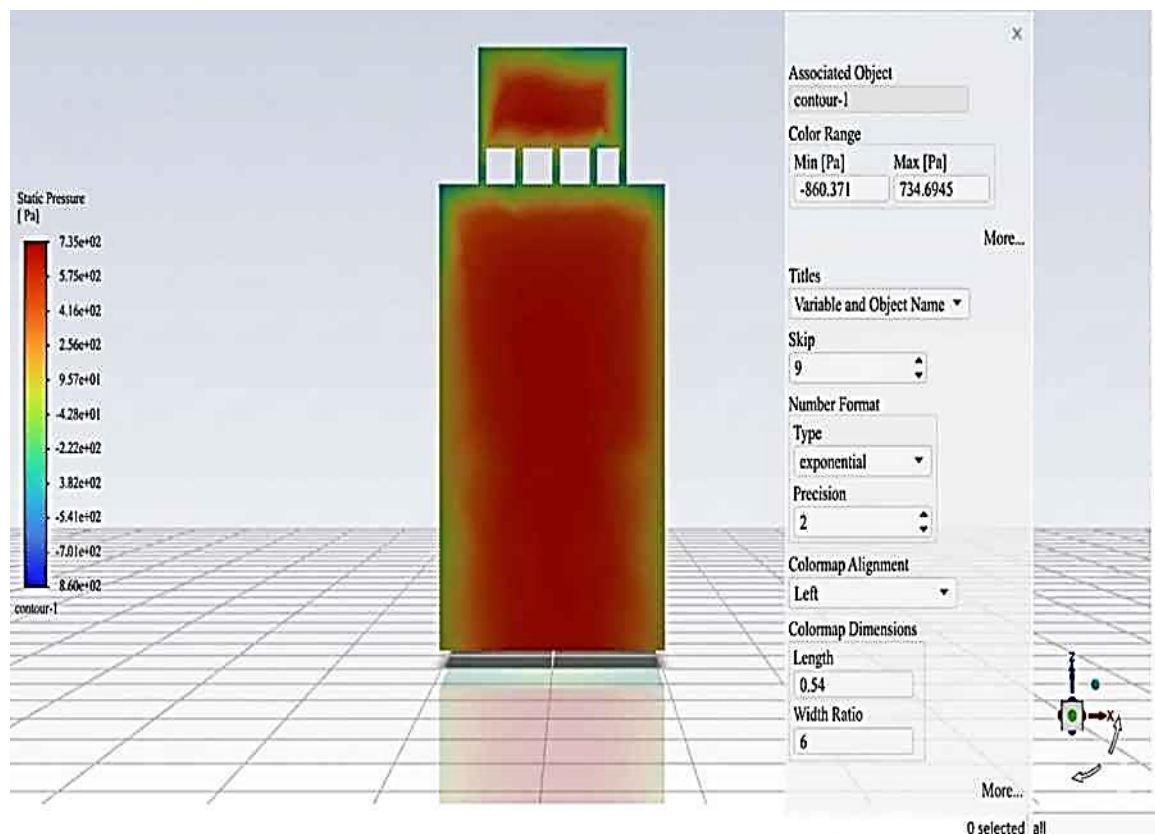


Figure 8. Computational fluid dynamics-based static pressure distribution around the signboard hoarding structure CFD, ANSYS Fluent).

In the case of a high-rise structure of $50\text{ m} \times 50\text{ m} \times 150\text{ m}$ in height, the windward façade has high compressive pressure [which is caused by the lack of movement of air], especially when the exposed sides are subjected to oblique wind directions. However, the pressure level is low on the leeward and side faces owing to the development of boundary layer separation and downstream wakes [8, 9].

In the case of an $18\text{ m} \times 12\text{ m} \times 51\text{ m}$ rectangular building with a significant spatial difference in pressure, a large variation in pressure on the windward face was observed. Increased pressure concentrations are observed along the corners and edges, and powerful suction is generated on the leeward surface [9, 12]. These results suggest that the force coefficient approach, which estimates uniform pressure distribution, might not accurately describe localized aerodynamic behavior.

In the signboard hoarding system, the windward panel experiences nearly uniform compressive pressure. In contrast, sharp pressure drops were observed near the boundary edges and on the rear surface. These high-gradient regions are critical for structural safety, particularly for fastening and support mechanisms [8, 9].

IS 875 (Part 3) 2015 and CFD Wind Pressure Value Comparison

Table 1 presents a quantitative comparison of wind pressures derived from IS 875 (Part 3) 2015 and CFD simulations, considering the maximum windward pressure for each configuration [8].

For all cases, the results show a strong correlation between IS 875 (Part 3) 2015 and CFD wind pressure, with percentage differences remaining below 4%. Excellent agreement between the numerical and analytical methods was demonstrated by the smallest deviations found for the gust factor and signboard hoarding cases.

Table 1. Comparison of wind pressure obtained from IS 875 (Part 3) 2015 and computational fluid dynamics (CFD).

Structure type	Dimensions (m)	IS 875 method used	IS code wind pressure	CFD wind pressure	Percentage difference
Tall Building	50 × 50 × 150	Direct Pressure method	$1.11 \frac{KN}{m^2}$	$1.15 \frac{KN}{m^2}$	3.6%
Rectangular building	18 × 12 × 51	Force coefficient method	$1.90 \frac{KN}{m^2}$	$1.85 \frac{KN}{m^2}$	2.6%
Tall rectangular building	24 × 12 × 96	Gust factor approach	$2.087 \frac{KN}{m^2}$	$2.09 \frac{KN}{m^2}$	0.14%
Signboard hoarding	—	Direct pressure method	736.62 Pa	734.69 Pa	0.26%

Analysis of The Findings

The CFD results show significant spatial variation in pressure distribution, particularly in areas impacted by flow separation and wake formation, even though IS 875 (part 3) 2015 offers accurate estimates of global wind pressure [10, 12]. The design of cladding system connections and secondary structural elements depends on these localized effects, which are not clearly captured by simplified codal methods [18, 8].

CONCLUSIONS

The maximum pressures of the windward side, which were calculated using CFD, were in good agreement with IS 875 (Part 3):2015, with discrepancies of approximately 4%. Therefore, the codal direct pressure method coefficient of force and gust factor is applicable and suitable for the normal estimation of global wind loads in structural design.

However, CFD entails the simulation of so-called direct aerodynamic processes, which are indirectly or crudely represented in the code, such as the separation of flows, wake generation, corner vortices, and very localized suction minimum. The peak windward pressures are the same for both the gust factor and CFD techniques in tall and slender buildings; however, with CFD, significant leeward suction pressures and side-face pressures that are the primary concerns in cladding design, serviceability, and detailing requirements can be observed. In signboard hoardings, CFD is used to show that the pressure is highly concentrated at the edges and supports, and numerical analysis is required to design connections and anchorage.

Overall, IS 875 (Part 3):2015 can be deemed sufficient for the preliminary and design-level estimation of global wind actions, and CFD can be viewed as a useful auxiliary tool for the additional evaluation and estimation of the local effect, optimization, and performance-driven design. An integrated codal process and established CFD analysis can make wind-sensitive structural systems much more accurate, stronger, and safer.

REFERENCES

1. Bureau of Indian Standards. IS 875 (Part 3):2015. Design Loads (Other than Earthquake) for Buildings and Structures—Part 3: Wind Loads. New Delhi: Bureau of Indian Standards; 2015.
2. ANSYS Inc. ANSYS Fluent User's Guide. Canonsburg (PA): ANSYS Inc.; 2023.
3. Bhandari NM, Krishna P, Kumar K. An Explanatory Handbook on Proposed IS 875 (Part 3): Wind Loads on Buildings and Structures. IITK-GSDMA-Wind06-V3.0. Kanpur: National Information Centre of Earthquake Engineering, Indian Institute of Technology Kanpur; 2011.
4. Daniels SJ, Xie ZT. Overview of large-eddy simulation for wind loading on slender structures. *Proc Inst Civ Eng Eng Comput Mech*. 2022;175(2):41–71. doi:10.1680/jencm.18.00028.
5. Köse M, Akyürek EF, Afshari F. Design, experimental and CFD analysis of a solar-air heating system with recycled thermal storage units for indoor applications. *Int J Numer Methods Heat Fluid Flow*. 2026;36:1590–1614. doi:10.1108/HFF-11-2025-0840.

6. Zheng Y, Li Z. Wind resource simulation using coupled WRF-CFD model: a case study in Liaoning, China. *Proc Inst Civ Eng Energy*. 2025;179(1):23–43. doi:10.1680/jener.23.00004.
7. Wijesooriya K, Mohotti D, Amin A, Chauhan K. Wind loads on a super-tall slender structure: a validation of an uncoupled fluid-structure interaction (FSI) analysis. *J Build Eng*. 2021;35:102028. doi:10.1016/j.jobe.2020.102028.
8. Colak L, Balci MN. Parametric design of cladding system for tall buildings using computational fluid dynamics approach. *J Mech Eng Sci*. 2023;17(4):9663–9678. doi:10.15282/jmes.17.4.2023.1.0765.
9. Wijesooriya K, Mohotti D, Lee CK, Mendis P. A technical review of computational fluid dynamics (CFD) applications on wind design of tall buildings and structures: past, present and future. *J Build Eng*. 2023;74:106828. doi:10.1016/j.jobe.2023.106828.
10. Thordal MS, Bennetsen JC, Koss HHH. Review for practical application of CFD for the determination of wind load on high-rise buildings. *J Wind Eng Ind Aerodyn*. 2019;186:155–168. doi:10.1016/j.jweia.2018.12.019.
11. Shen B, Yuan Y, Zhou H, Zhang Z, Zhou T, Zheng X, et al. Comparative assessment of large eddy simulation and detached eddy simulation in predicting aerodynamic and flow characteristics around a tall building. *Phys Fluids*. 2025;37. doi:10.1063/5.0289342.
12. Meena RK, Raj R, Anbukumar S, Khan MI, Khatib JM. Fluid dynamic assessment of tall buildings with a variety of complicated geometries. *Buildings*. 2024;14:4081. doi:10.3390/buildings14124081.
13. Huang S, Li QS, Xu S. Numerical evaluation of wind effects on a tall steel building by CFD. *J Constr Steel Res*. 2007;63:612–627. doi:10.1016/j.jcsr.2006.06.033.
14. Cao Y, Liu X, Zhou D, Ren H. Investigation of local severe suction on the side walls of a high-rise building by standard, spectral and conditional POD. *Build Environ*. 2022;217:109047. doi:10.1016/j.buildenv.2022.109047.
15. Sahani R. A study on wind load analysis of high rise building using CFD. *Commun Appl Nonlinear Anal*. 2025;32:689–727. doi:10.52783/cana.v32.5099.
16. Chandra N, Kumar Singh P, Kadian V, Rai S. Evaluation of wind effects on high-rise buildings using ANSYS CFX. *J Eng Res Appl*. 2023;2:1–11. doi:10.55953/jera.2023.2101.
17. Khaled MF, Aly AM. Assessing aerodynamic loads on low-rise buildings considering Reynolds number and turbulence effects: a review. *Adv Aerodyn*. 2022;4:1–33. doi:10.1186/s42774-022-00114-0.
18. Lu C, Zhang Z, Ji C, Zheng X, Alam MM. Three-dimensional wake transition and hydrodynamics of a circular cylinder near a plane wall. *Phys Fluids*. 2025;37. doi:10.1063/5.0285934.
19. Suthar N, Goyal PK. Comparison of response of building against wind load as per wind codes [IS 875 (Part 3)-1987] and [IS 875 (Part 3)-2015]. *IOP Conf Ser Earth Environ Sci*. 2021;796:012007. doi:10.1088/1755-1315/796/1/012007.
20. Jayasundara HMAD, Koliyabandara SMNH, Wijesundara KK. Wind loads on tall buildings: A comparative study of the international wind codes and numerical simulation. *Engineer*. 2018;51(3):31. doi:10.4038/engineer.v51i3.7304.
21. Dokhaei B, Abdelaziz K, Shafei B, Sarkar P, Hobeck J, Alipour A. Aeroelastic boundary layer tests of a 1:76 model of tall building and effects of adjacent building interference. *J Wind Eng Ind Aerodyn*. 2025;257:106006. doi:10.1016/j.jweia.2025.106006.
22. Mohammadjani C, Zisis I. Exploring the impact of vertically separated flows on wind loads of multi-level structures. *Front Phys*. 2023;11. doi:10.3389/fphy.2023.1225817.
23. Verma DSK, Roy AK, Lather S, Sood M. CFD simulation for wind load on octagonal tall buildings. *Int J Eng Trends Technol*. 2015;24:211–216. doi:10.14445/22315381/IJETT-V24P239.
24. Metwally O, Ibrahim HA, Elawady A, Zisis I, Chowdhury AG. Wind load impact on tall building facades: damage observations during severe wind events and wind tunnel testing. *Front Built Environ*. 2025;10. doi:10.3389/fbuil.2024.1514523.
25. Aly A, Gol-Zaroudi H. Peak pressures on low-rise buildings: CFD with LES versus full scale and wind tunnel measurements. *Wind Struct*. 2020;30(1):99–114. doi:10.12989/was.2020.30.1.099.

26. Lu B, Li QS, Han XL. Assessment of computational fluid dynamic as a design tool for estimation of wind loads on unconventional skyscrapers in urban environment. *Struct Des Tall Spec Build.* 2024;33. doi:10.1002/tal.2165.
27. W S, K. High-rise building wind analysis using computational fluid dynamics and dynamic analysis using ETABS program. *Int J Emerg Trends Eng Res.* 2020;8:3994–4012. doi:10.30534/ijeter/2020/172872020.
28. Hou F, Jafari M. Investigation approaches to quantify wind-induced load and response of tall buildings: a review. *Sustain Cities Soc.* 2020;62:102376. doi:10.1016/j.scs.2020.102376.
29. Kumar A, Meena RK, Sharma D, Raj R. Structural parameters of Wrench-C shape building using CFD. *Int J Constr Manag.* 2024;24:664-672. doi:10.1080/15623599.2023.2271218.