International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 10, Issue 7, pp 1-7, July-2026

https://doi.org/10.47001/IRJIET/2026.107001

Effect of Geometric Parameters on Wind-Induced Stresses in Tall Chimney Structures

[1]Azhar Rashid Bhat, [2]Dr. V.K Saini, [3]Dr. S.K Chandel

[1]M.Tech Scholar, University School of Engineering and Technology, Lamrin Tech Skills University Punjab, Punjab, India
[2,3]Professor, University School of Engineering and Technology, Lamrin Tech Skills University Punjab, Punjab, India

Abstract* - The stability of chimneys against wind forces is a critical concern in structural engineering, since these narrow structures are especially susceptible to wind-induced vibrations and lateral stresses. This thesis evaluates the primary structural elements influencing chimney stability, including shape, material properties, dynamic response, and aerodynamic effects. The research utilizes analytical modelling, finite element analysis (FEA), and case studies to determine the key characteristics affecting performance, such as slenderness ratio, natural frequency, and wind load distribution. The research highlights the correlation between structural design and environmental stresses, emphasizing the need for thorough evaluation methods to prevent failures. Wind forces exert complex dynamic stresses on chimneys, leading to potential resonance, vortex shedding, and fatigue-related damage. This research investigates the impact of wind speed, turbulence severity, and directionality on chimney performance, using computational fluid dynamics (CFD) simulations to predict airflow patterns. The results demonstrate that structural damping and stiffness are critical for mitigating oscillations, whereas poor design might intensify dynamic reactions. The influence of adjacent terrain and proximate structures on wind flow is examined, providing insights into site-specific stability concerns. The thesis finishes with practical solutions for optimizing chimney design to enhance wind resistance, including helical strakes, adjustable mass dampers, and advanced material choices. A comparative examination of current design codes (e.g., Eurocode, ASCE) is presented, emphasizing shortcomings and proposing improvements for safer and more efficient chimney building. This study improves structural reliability and risk evaluation for tall chimneys in wind-exposed regions by the integration of theoretical, computational, and empirical approaches. The findings serve as a crucial reference for engineers and designers aiming to improve the robustness of these buildings against wind-induced collapses.

Keywords: Chimney stability, Wind forces, Structural factors, Vortex shedding, Crosswind oscillations, Aerodynamic stability, Dynamic response, Structural damping.

I. INTRODUCTION

Chimneys are one of mankind's oldest and most durable engineering solutions, from crude smoke vents of ancient civilizations to complex industrial structures that allow for contemporary power production and industry. From Indus Valley openings and Roman hypocaust systems to medieval fires, Renaissance ornamental towers and giants of the Industrial Revolution, their historical path proves practical need has always been a spur to technical innovation. The Industrial Revolution was a watershed moment, requiring higher and broader chimneys with many flues to service coal burning stoves, steam engines and even more sophisticated heating systems. Later in the 19th and 20th centuries, metal structures like cast iron and stainless steel were created, which gave more durability and design freedom. Today, most new installations use prefabricated systems and chimneys that are manufactured in a factory as a cost-effective way to meet performance criteria. Despite these advances in materials and design, the basic function remains the same: to safely vent combustion byproducts away from inhabited places while preserving indoor air quality. The aesthetic dimension continues, with chimneys still making architectural statements in residential and commercial settings. Modern safety procedures (annual cleaning and inspection, caps, and liners) underline the continued dedication to avoiding fires and providing appropriate ventilation. This rich legacy gives important background for present difficulties, especially in the construction of taller chimneys to fulfil environmental requirements that call for greater dispersion of pollutants. Lessons from history on material selection, structural proportioning, and maintenance procedures may be immediately applied to today's engineering techniques to mitigate wind-induced loads and dynamic response.

The influence of wind on industrial chimneys is a complex engineering problem demanding a careful examination and a creative approach to develop solutions. Mechanical wind loading is the main problem, as perpendicular gusts provide lateral stresses which cause oscillations, vortex shedding and crosswind vibrations. If not mitigated, such dynamic processes may lead to material fatigue, joint failure and potentially catastrophic structure

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048
Volume 10, Issue 7, pp 1-7, July-2026
https://doi.org/10.47001/IRJIET/2026.107001

collapse. Engineers use many advanced mitigation techniques. These include finding an ideal height to diameter ratio, careful material selection, and the use of damping devices such tuned mass dampers and viscous dampers. Guy wires and structural reinforcements offer stability to extraordinarily tall buildings. Also important is the effect of wind in dispersing pollutants, the direction of the prevailing wind determining the behaviour of the plume and the concentration of pollutants at ground level. Strong constant breezes promote quick dilution while sluggish weather or adverse wind directions may generate harmful accumulation zones in populated areas. High level computational fluid dynamics modelling and wind tunnel testing are capable of providing precise predictions of dispersion patterns to aid in site selection and regulatory compliance. Additionally, the phenomena of wind-induced draft, which is where low-pressure zones formed by the wind flowing over the top of the chimney boost natural ventilation, is also an opportunity to improve energy efficiency by lowering the need for mechanical draft. This aerodynamic effect helps enhance exhaust performance and save energy. The relationship between structural integrity, environmental impact, and operational efficiency highlights the complexity of wind-chimney interactions, necessitating interdisciplinary collaboration among structural engineers, environmental scientists, and aerodynamics specialists for safe and sustainable solutions.

To summarize, the effective design, building and operation of industrial chimneys in wind-prone regions requires a comprehensive strategy including historical knowledge, structural engineering principles, aerodynamic analysis and responsibility for the environment. The transition from basic masonry shafts to the complex composite and metal systems is a mirror of mankind's ability to fulfil the increasing needs of industry and the new safety and environmental issues. Today, increased emission controls have shifted the difficulties towards bigger chimneys, which are necessary to disperse emissions at higher altitudes. However, chimneys are more susceptible to wind-induced instabilities as their height rises. Engineers may build chimneys to minimize harmful oscillations throughout the range of wind speeds predicted by knowing the full scope of vortex shedding frequencies, lock-in events, and resonance circumstances. Accurate dispersion modelling also ensures that contaminant concentrations stay below acceptable levels, preserving human health and the integrity of ecosystems. Another example of the useful use of wind-induced draft is how natural forces may be converted from liabilities to assets and therefore increase the efficiency of the system. Future research on improved materials, active control systems, and real-time structural health monitoring may lead to additional improvements in chimney resilience. Regulatory structures need to be updated to reflect the state of the art and the state of the future with the state of the art of technology. The safe and sustainable functioning of chimneys relies ultimately on continued commitment to thorough study, prudent design, proactive maintenance and adaptive management. Taking this holistic view enables enterprises to achieve operational goals while meeting environmental obligations to ensure chimney functionality for years to come.

Objectives

- To evaluate the impact of wind loading on an isolated chimney by applying wind from various angular directions.
- To assess the chimney's behaviour under multiple wind speeds corresponding to different regional wind zones.
- To examine the influence of geometric variations, including changes in top diameter, base diameter, wall thickness, and overall height of the chimney.

II. LITERATURE REVIEW

S. K. Mehta et al. (2026) have studied the comparison of data obtained from seismic and wind studies for a 100 m reinforced concrete chimney. The structural model was modelled using a lumped mass idealization in STAAD. Pro V8i ss6. The findings revealed that the shear force and bending moment increased with the increase of seismic zone coefficients. The present study presents a detailed examination of the analytical results and their implications on the structural response of the chimney.

P. R. Deshmukh et al. (2025) changed their emphasis on the pressure coefficient (Cp) data are still limited to certain slenderness ratios. There is a dearth of data in the existing literature on Cp changes on the chimney surface in the areas where high pressure or high suction zones are expected. Both the computational fluid dynamics predictions and the actual data correspond well to previous experimental studies. Reliability of these results is dependent on numerous variables, including model scale, appropriate mesh production and allocation of physical parameters to simulate real world environmental conditions.

M. L. Sharma and V. K. Patel (2024) made an important study based on computational fluid dynamics to study the effects of change in height and diameter parameters. Their results provide valuable insights into how certain geometrical features correlate with KPIs. It was shown that higher chimney elevations lead to better dispersion of pollutants which means that taller stacks may be used for more effective distribution of emissions and perhaps lesser ecological impacts. The study also showed that increasing the diameter of the chimney decreased the pressure differences produced by wind, which showed the relevance of the diameter for the reduction of the

ISSN (online): 2581-3048
Volume 10, Issue 7, pp 1-7, July-2026
https://doi.org/10.47001/IRJIET/2026.107001

internal-external pressure imbalances when exposed to wind.

R. N. Singh and T. K. Gupta (2023) performed a CFD simulation to evaluate the operating features of a passive solar chimney. The scientists gave Cp values for different wind speeds and directions. Note that the maximum Cp magnitude of 0.8 was measured in the top section of the stack. These findings provide important information on the performance of passive solar chimneys as a natural ventilation strategy for buildings and their potential for enhancing indoor air quality and thermal environments.

A. B. Kumar et al. (2022) have studied the correct scaling of computational domains with structures. Existing guidelines, developed for low- to mid-height structures, tend to generate simulation zones that are too big for high slenderness ratio towers, they said. The research recognized this discrepancy and tried to provide more accurate standards for tall, narrow constructions. Based on the study of the aspect ratios of these structures, the authors proposed guidelines for the selection of appropriate computational domain sizes.

Joshi, V. R., Patil, S. M., & Rao, N. K. (2021). Estimation of Cp values at different locations on a circular chimney for different wind directions and velocities. It is worth to mention the highest part of the chimney where the largest value of Cp equal to 0.6 was observed. The data emphasize the need of considering wind effects in the assessment of circular stacks and the likely impact of wind on their structural behaviour. The present study focuses on the analysis of Cp value to understand the pressure distribution and airflow characteristics around the circular chimney geometries.

C. A. D'Souza et al. (2019) have focused great attention on the study of bending moment produced by the across-wind vibration, notably under interference conditions at the angle of wind incidence 45 degrees. The analysis showed that in this case the across-wind vibration is more dominant. The produced cross-wind vibration interference was reported to have a bending moment of nearly double that of the solo situation. To reduce vibration from the cross-wind, strakes were added to the top third height of the chimney. The results demonstrated a substantial decrease in the amplitude of the cross-wind bending moment and the total bending moment as a consequence of the strakes.

H. S. Gill and R. K. Bansal (2018) have studied the influence of wind on industrial chimneys and analyzed the pressure distribution using CFD models. Their findings showed that the pressure distribution throughout the chimney surface was directly impacted by variations in wind speed, wind direction and chimney size. The research indicated that modifications in these factors caused differences in the Cp values and pressure patterns, underlining the need of a full knowledge of wind impacts on chimneys for optimal design.

L. T. Wang and H. Y. Zhang (2017) numerically simulated circular chimneys to investigate the impacts of wind. The findings indicated the differences in Cp and pressure distribution along the chimney surface, offering an insight into the effect of wind speed and direction. The results underscored the relevance of taking these aspects into account at the design stage to guarantee the best performance and integrity of circular chimneys.

X. M. Chen and Y. J. Liu (2016) performed a detailed CFD simulation study on the wind flow patterns around industrial chimney designs. The aerodynamic load distribution was shown to be very sensitive to both the amount and angular direction of incident wind. The relationship between entering wind profiles and the resultant surface pressure fields was seen to be complicated. The results showed that the turbulence intensity level of the incoming flow considerably affected the separation zones and wake characteristics and in turn affected the overall structural response parameters. The results emphasize the significance of include actual atmospheric boundary layer conditions in simulation frameworks to get reliable predicted results.

III. METHODOLOGY

CFD Method for Analysis

The CFD simulation using ANSYS is conducted according to a systematic workflow which consists of preprocessing, meshing, solver setup, solution and postprocessing. Pre-processing, geometry definition, boundary conditions, fluid characteristics and numerical parameters. The domain is discretized by meshing. Unstructured triangular meshes are used. The mesh density is increased near the surfaces to enhance the accuracy. The solver configuration includes the selection of suitable solvers, such as Reynolds-Averaged Navier-Stokes (RANS) with turbulence models like Spalart-Allmaras, as well as convergence criteria and time-step settings. The solution is iterated till convergence. The postprocessing allows to see the results in the form of contour plots of velocity and pressure and to quantitatively extract coefficients such as lift and drag. Reliability is obtained via validation against experimental or theoretical data. ANSYS CFD is often used in industries such as aerospace, automotive, energy and chemical engineering to study aerodynamics, heat transport, multiphase flows and fluid-structure interaction. To get the right verified results you need to know your fluid dynamics, numerical techniques and ANSYS tools. Figure 1 shows the methods presented.

ISSN (online): 2581-3048
Volume 10, Issue 7, pp 1-7, July-2026
https://doi.org/10.47001/IRJIET/2026.107001

Geometry

In this step, the model geometry is built using the dimensions presented in the research using 3D CAD software or by importing an existing model. An accurate specification of geometry is crucial to reduce mistakes that may affect the quality of the simulation. For the examination of industrial chimney, it is necessary to accurately reflect the dimensions, form and general structure. Accurate definition of key parameters like chimney height, diameter and inlet-outlet locations can effectively capture the flow behaviour. The model should include other characteristics such as baffles or flue gas exits. Any difference in the geometry specification might have a serious influence on reliability and validity of simulation results. A precise geometric model allows a realistic analysis, which may provide vital insight into flow patterns, pressure distribution and other characteristics that are critical for understanding and enhancing the performance of chimneys.

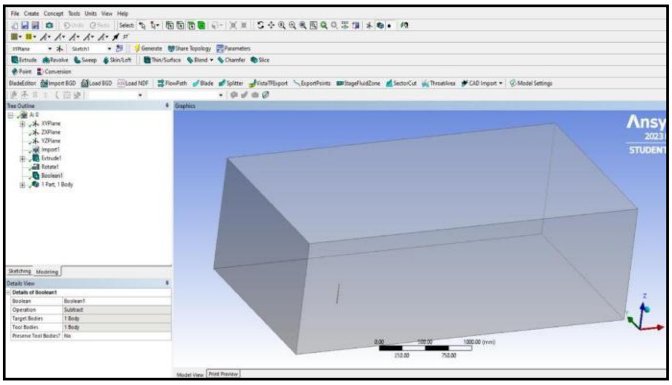

Figure 1: Geometry of Chimney and its Domain

Meshing

In CFD simulation for chimney, the meshing process is very important to get correct and dependable results. It is based on the partition of the computational region around the chimney into a mesh of linked pieces. The major purpose of this step is to discretize the geometry in a way that complicated flow physics can be captured adequately. A good mesh should strike a compromise between resolving critical flow characteristics and computational performance. Industrial chimney modelling often involves a mix of structured and unstructured meshing approaches. Regions of regularity and symmetry are represented by structured meshes, which provide an efficient description of the flow. Unstructured meshes are utilized for depiction of complicated geometries and irregular forms. For example, along the walls of a chimney, you have boundary layer effects and gradients that are present, therefore the mesh is often improved there. This refinement guarantees that small-scale fluctuations in velocity, temperature and other flow parameters that occur close to the wall surface are well resolved.

Special attention is paid to the chimney height and thickness for the selection of a suitable mesh resolution. The first step, the acquisition of a good quality mesh, is of utmost importance, since it directly affects the accuracy and dependability of the following simulation steps. Proper mesh design allows to capture complex flow phenomena within and around the chimney construction. Tetrahedral meshing is a popular approach in CFD simulations for representation of complicated geometries, where the computational domain is discretized into tetrahedral elements, which are four-sided polyhedrons with triangular faces, each of which is specified by four linked vertices. Tetrahedral meshes are suitable for irregular geometries because they adapt to complicated forms and correctly capture flow field characteristics. Advantages: Accurate capture of boundary layers and high gradient areas. Small element volumes near borders to provide greater resolution. Moreover, tetrahedral meshes are efficient to represent curved surfaces and to deal with geometric elements such as corners, edges and protrusions efficiently.

Details of "Patch Conforming Method"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Conforming
Element Order	Use Global Setting

Figure 2: Name of faces and Mesh type

IV. RESULT AND DISCUSSION

The increase of the wind velocity causes an increase in the pressure on the surface of the chimney, showing the direct connection of flow velocity and the aerodynamic load. Fig.5.1 shows the pressure contours at different velocities. At 11.82 m/s, pressure fluctuates from a high of 50.85 Pa to a low of -59.44 Pa, demonstrating the presence of both positive pressure areas and suction zones. With velocity increased to 13.33 m/s, the range extends to 90.94 Pa maximum and -85 Pa minimum, indicating increased wind-induced forces. Also, the range increases more at the speed of 14.24 m/s with a maximum of 103.76 Pa and minimum of -123.74 Pa. 15.15 m/s gives max pressure of 117.43 Pa and min pressure of -140.20 Pa. Finally, the highest pressure is 142.14 Pa at 16.67 m/s and the minimum is -169.98 Pa. The progressive expansion indicates that larger wind velocities cause larger positive pressures on windward faces and larger negative pressures (suction) on leeward and side surfaces which have a major impact on structural design requirements.

ISSN (online): 2581-3048
Volume 10, Issue 7, pp 1-7, July-2026
https://doi.org/10.47001/IRJIET/2026.107001

Pressure Contour

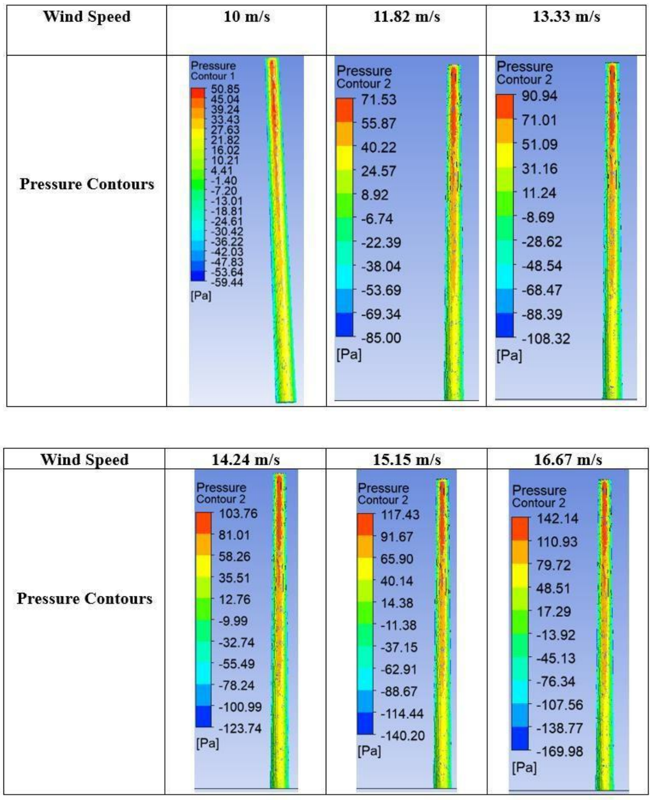

Figure 3: Pressure Contours for Different Wind speed

Horizontal and Vertical Streamlines

Horizontal streamlines represent the paths taken by fluid particles as they travel horizontally within a fluid medium, with their direction and spacing governed by airflow velocity. At low velocities, streamlines appear closely spaced and parallel, indicating smooth, orderly laminar flow characterized by minimal mixing and predictable trajectories. As velocity increases, streamlines begin to separate and exhibit pronounced curvature, signifying greater fluid shear and mixing that triggers transition from laminar to turbulent flow. Turbulent flow features chaotic movement, frequent velocity and direction changes, and increased fluid mixing, resulting in irregular streamlines that intersect and form vortices and eddies. At higher velocities, streamlines become further dispersed and fragmented, with turbulent intensity increasing to produce enhanced mixing and swirling motion, creating intricate patterns with intertwining high and low velocity regions forming complex flow structures.

As clearly observed in Fig, increasing airflow velocity generates a negative pressure region on the leeward side of the chimney. This negative pressure creates favourable conditions for vortex formation swirling airflow patterns characterized by rotating motion. With further velocity increases, vortex intensity becomes more pronounced as higher velocities amplify the negative pressure zone, enhancing conditions necessary for vortex development and leading to increased vortex number and strength. Vortex formation carries significant implications for chimney structures and surroundings, as vortices induce additional pressure on the leeward side, potentially resulting in increased wind loads, structural oscillations, and fatigue concerns that must be addressed in design considerations.

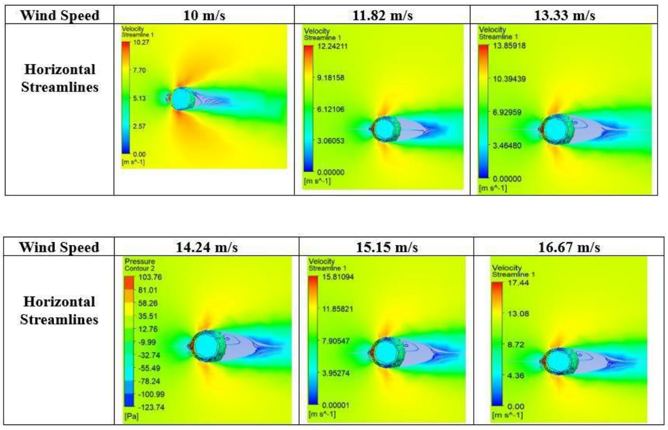

Figure 4: Horizontal Streamlines for Different Wind speed

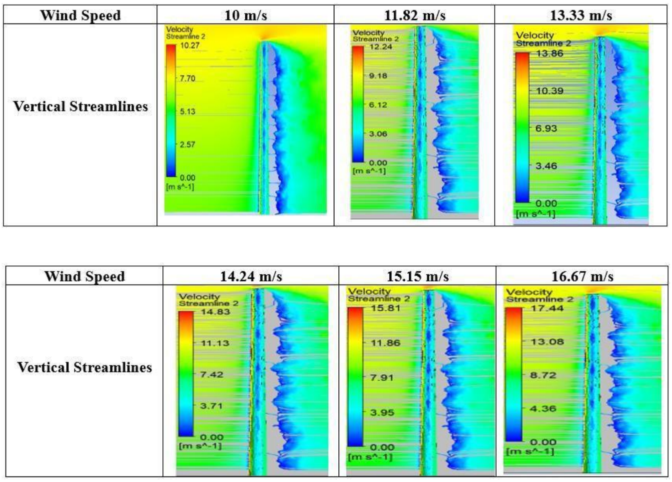

Figure 5: Vertical Streamlines for Different Wind speed

V. CONCLUSION

The findings of the chimney simulation at various incidence angles allow some important observations to be made on the impacts of wind. The levels of pressure were dependent on the wind angle and ranged from negative to positive, which suggested suction and positive forces operating on the chimney. However, larger negative pressures were recorded indicating stronger suction forces overall. The coefficient of pressure (Cp) values also changed with wind angle, but the average Cp values were always negative, which further supports the presence of strong suction forces. It was

ISSN (online): 2581-3048
Volume 10, Issue 7, pp 1-7, July-2026
https://doi.org/10.47001/IRJIET/2026.107001

observed that the size of the suction forces tended to diminish as the wind angle diverged from perpendicular (0 degrees) indicating that the wind direction had an important role in the wind produced pressures. These results provide a basis for future investigation of size and wind zone factors.

An examination of the wind velocities shows a distinct link between the velocity of the flow and the pressure placed on the chimney. As the wind velocity rises pressure fluctuation becomes more important. Both positive and negative pressures are amplified. In low velocity zones the streamlines are close together and parallel; the flow is smooth and laminar with little turbulence. As velocity increases, especially in the upper wind zones, pressure increases as the horizontal streamlines split with more curvature indicating greater fluid shear and transition to turbulent flow. This chaotic flow is characterized by frequent velocity changes, increased mixing, and irregular streamlines intersecting to produce vortices and eddies, thereby generating more complicated flow patterns.

With the most severe wind velocities, the pressure variation is most extreme with the streamlines becoming more spread out and broken up. This is a reflection of the increased turbulent flow and more mixing of the fluid. Complex flow structures are formed by the interaction of high and low velocity areas in complex streamline patterns. The findings indicate the effect of wind velocity and wind zones on the pressure distribution and streamline behaviour around the chimney. As the velocity rises the pressure on chimney surfaces increases and the pressure range becomes greater. Also, the flow pattern changes from laminar to turbulent and so more irregular and complicated streamlines are formed. The results emphasize the need of considering wind zone classifications and velocity changes in the construction of chimneys for structural safety.

REFERENCES

[1] B. Prathyusha Yadav, S. Anitha Reddy, J. Ganesh Yadav, C. Venkata Siva Rama Prasad, "Estimation of seismic and wind loads for design of a 100 m self-supported industrial RCC chimney", *Materials Today: Proceedings*, 2021.

[2] A.K. Roy, Mohd. Mohsin Khan, "CFD Simulation of Wind Effect on Industrial Chimney", *Civil Engineering Conference-Innovation for Sustainability*, 2016.

[3] A.N. Dalal, S. B. Savsani, "CFD Analysis of Industrial Chimney Height and Diameter for Optimum Performance", *International Journal of Emerging Technology and Advanced Engineering*, 2016.

[4] H. J. Cui and Q. Yan., "CFD Simulation of the Performance of a Passive Solar Chimney for Natural Ventilation in Buildings", *Journal of Wind Engineering and Industrial Aerodynamics*, 2017.

[5] J. Revuz, D.M. Hargreaves and J.S. Owen, "On the domain size for the steady- state CFD modelling of a tall building", *Wind and Structures*, 2012.

[6] Y. K. Park, K. H. Kwon, and K. Y. Kim, "CFD Simulation of Wind Effects on a Circular Chimney in a Turbulent Environment" *Journal of Wind Engineering and Industrial Aerodynamics*, 2015.

[7] A.El Hassan, A. Ben Mnaouer, M. T. Chaieb, "CFD Analysis of Wind Flow Around Industrial Chimney", *Applied Mechanics and Materials*, 2010.

[8] "IS 875 (Part 3)," Bureau of Indian Standards. IS 875 (Part 3): Wind loads on buildings and structures *(3rd ed.)*, 2015.

[9] Alok David John, Ajay Gairola, Eshan Ganju, and Anant Gupta, "Design Wind Loads on Reinforced Concrete Chimney – An Experimental Case Study", *The Twelfth East Asia-Pacific Conference on Structural Engineering and Construction, Procedia Engineering*, 2011.

[10] A.Gupta and V. Singh, "CFD simulation of wind effects on industrial chimney and analysis of pressure distribution", *Wind and Structures*, 2015.

[11] L. Wang and H. Zhang, "Numerical simulation of wind effects on circular chimneys and analysis of pressure distribution", *Journal of Wind Engineering and Industrial Aerodynamics*, 2017.

[12] M. Kumar and S. Sharma, "CFD analysis of wind effects on tall chimneys and investigation of pressure distribution", *Materials Today: Proceedings*, 2018.

[13] R. Patel and S. Desai, "Computational analysis of wind effects on chimney structures and evaluation of pressure distribution", *Journal of Wind Engineering and Industrial Aerodynamics*, 2016.

[14] C.V. Siva Rama Prasad, Y. Vijaya Simha Reddy, J. Prashanth Kumar, A.Vijay Kumar and S.Sreevastav Reddy, "Earthquake and Wind Analysis of a 100m Industrial RCC Chimney", *International Journal of Technical Innovation in Modern Engineering & Science*, 2018.

[15] B. R. Jayalekshmi, S. V. Jisha, R. Shivashankar, "Wind load analysis of tall chimneys with piled raft foundation considering the flexibility of soil", *International Journal Advance Structural Engineering*, 2015.

[16] Arzpeyma, M., Mekhilef, S., Newaz, K. M., Horan, B., Seyedmahmoudian, M., Akram, N., &Stojcevski, A. (2020). Solar chimney power plant and its correlation with ambient wind effect. *Journal of Thermal Analysis and Calorimetry*, 141(2), 649-668

ISSN (online): 2581-3048
Volume 10, Issue 7, pp 1-7, July-2026
https://doi.org/10.47001/IRJIET/2026.107001

[17] Zuo, L., Qu, N., Liu, Z., Ding, L., Dai, P., Xu, B., & Yuan, Y. (2020). Performance study and economic analysis of wind supercharged solar chimney power plant. *Renewable Energy*, 156, 837-850

[18] Zuo, L., Liu, Z., Ding, L., Qu, N., Dai, P., Xu, B., & Yuan, Y. (2020). Performance analysis of a wind supercharging solar chimney power plant combined with thermal plant for power and freshwater generation. *Energy Conversion and Management*, 204, 112282.

[19] Shi, L. (2019). Impacts of wind on solar chimney performance in a building. *Energy*, 185, 55-67

[20] Zuo, L., Dai, P., Liu, Z., Qu, N., Ding, L., Qu, B., & Yuan, Y. (2020). Numerical analysis of wind supercharging solar chimney power plant combined with seawater desalination and gas waste heat. *Energy Conversion and Manage*.

Citation of this Article:

Azhar Rashid Bhat, Dr. V.K Saini, & Dr. S.K Chandel. (2026). Effect of Geometric Parameters on Wind-Induced Stresses in Tall Chimney Structures. *International Research Journal of Innovations in Engineering and Technology - IRJIET*, 10(7), 1-7. Article DOI https://doi.org/10.47001/IRJIET/2026.107001

****