



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 13, Issue 5, May 2024

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423

 9940 572 462

 6381 907 438

 ijirset@gmail.com

 www.ijirset.com

Wind Load Analysis and Wind Resistant Design of Structures

Lokesh Dabarase, Dr. N. S. Raman

P.G. Student, Department of Civil Engineering, Guru Nanak Institute of Engineering and Technology, Nagpur,
Maharashtra, India

Associate Professor, Department of Civil Engineering, Guru Nanak Institute of Engineering and Technology, Nagpur,
Maharashtra, India

ABSTRACT Buildings are defined as structures utilized by the people as shelter for living, working or storage. As now a days there is shortage of land for building more buildings at a faster growth in both residential and industrial areas the vertical construction is given due importance because of which Tall Buildings are being built on a large scale. Wind in general has two main effects on the Tall buildings: Firstly it exerts forces and moments on the structure, Secondly it distributes the air in and around the building mainly termed as Wind Pressure. Sometimes because of unpredictable nature of wind it takes so devastating form during some Wind Storms that it can upset the internal ventilation system when it Passes into the building. For these reasons the study of air -flow is becoming integral with the planning a building and its environment. In the last few years, many wind studies have been carried out on a particular shape and particular height of building. The analysis of wind load in tall building has been suggested as below.

KEYWORDS: Wind pressure, High rise Structure, L shape geometry, Residential Building, Stress.

I. INTRODUCTION

The evolution of computational fluid dynamics (CFD) has indeed revolutionized the field of wind engineering. Initially met with skepticism due to computational constraints and modeling errors, CFD has now become an indispensable tool in understanding complex fluid flows, including those encountered in wind engineering.

The transition from reliance on full-scale and wind tunnel analyses to incorporating CFD into research and design processes has brought several advantages. First and foremost, CFD allows for a more detailed and comprehensive understanding of wind flow phenomena, including turbulent flows, recirculation zones, and stagnation points. This level of insight is crucial for optimizing the design and performance of structures subject to wind loads, such as buildings, bridges, and other infrastructure.

Moreover, the utilization of CFD reduces the reliance on expensive wind tunnel facilities and extensive data recording processes. While wind tunnels remain valuable for validating CFD results and studying certain flow aspects, CFD offers a cost-effective alternative for preliminary analyses and design iterations.

The continual advancements in computer hardware and software have further enhanced the applicability and accuracy of CFD in wind engineering. With increasingly powerful computers and refined modeling techniques, CFD simulations can now capture complex flow behaviors with greater fidelity, providing engineers with valuable insights into wind-induced loads and structural responses.

Overall, the integration of CFD into the field of wind engineering signifies a paradigm shift in how fluid flow problems are analyzed and addressed. By leveraging the computational capabilities of modern technology, engineers can now tackle intricate wind flow scenarios more efficiently and effectively, ultimately leading to safer and more resilient built environments.

II. METHODOLOGY

OBJECTIVES:

- To Study the structural performance of RC high rise multi storey Building.

- To carry out dynamic analysis of Rectangular and C shape of building as per IS:875 part III using ETABS 2016 software.
- To evaluate various responses like Time period, Base shear, Storey displacement and storey drift of the tall buildings on different height and their comparison.
- To study the comparison of different height and different shape of the building on the basis of their results.

The mathematical model used to simulate the airflow around the building employs an incompressible finite mass model. This model is implemented using a commercial computational fluid dynamics (CFD) code, specifically Abaqus CFD, available in the Fulton School of Engineering eSpace High-Performance Computing Lab. While a brief overview of the numerical models applied will be provided, for more detailed derivations and programming techniques, reference materials such as those by Joel H. Ferziger (2002) and C.A.J Fletcher (1991) can be consulted.

To streamline the simulation process and minimize computational resources, a hollow square cylinder is employed to represent the deflection resistance of the diagrid structure. This cylinder model incorporates descending mass and stiffness from the bottom to the top, mirroring the behavior of the actual diagrid structure. By adopting this approach, the number of elements in the simulation is reduced, and the internal influence of the structure is avoided, leading to more efficient and accurate results.

Furthermore, to capture the interaction between the fluid flow and the structure, a Fluid-Structure Interaction (FSI) approach is utilized. This ensures that the dynamic response of the structure to the fluid flow is properly accounted for in the simulation.

For the wind load conditions, a basic wind velocity of 51m/s is selected, representing the maximum wind load condition considered during the design of the structure. This wind velocity serves as a critical parameter for assessing the structural performance under extreme wind conditions.

In wind engineering models, the onset flow can be approximated as a horizontally homogeneous turbulent boundary layer. In this model, the flow is primarily driven by a shear stress at the top boundary, which represents the interaction between the atmosphere and the building or terrain.

To simulate such a flow using computational fluid dynamics (CFD) and turbulence models like the k-ε model, appropriate inlet profiles and boundary conditions need to be established. The k-ε turbulence model is one of the most commonly used turbulence models for simulating turbulent flows in engineering applications.

For the inlet profiles, the mean velocity profile and turbulence intensity profile need to be specified. These profiles are typically derived from empirical or experimental data, such as wind tunnel measurements or atmospheric boundary layer profiles. The mean velocity profile describes how the velocity varies with height, while the turbulence intensity profile describes the fluctuations in velocity due to turbulence.

Boundary conditions are set at the inlet and other boundaries of the computational domain to represent the flow characteristics. At the top boundary, a shear stress boundary condition is applied to represent the driving force of the flow. Other boundary conditions, such as no-slip conditions at solid surfaces and far-field boundary conditions at the outer boundaries of the domain, are also specified to ensure that the flow behavior is accurately captured throughout the simulation.

By establishing appropriate inlet profiles and boundary conditions, the k-ε turbulence model can effectively simulate the turbulent boundary layer flow in wind engineering models, providing valuable insights into the aerodynamic behavior of structures and their interaction with the surrounding flow field.

To forestall this bizarre spike in the worth of k the creation might be reevaluated into a structure that utilizes the shear stresses at the north and south cell faces

Since τ_{xz} is a constant across the boundary layer (and using Eqn. (5.4.5) for μT),

$$P_k = \frac{2\rho^2 u_*^4}{2\rho u_* k z} = \frac{\rho u_*^3}{k z} = \rho \epsilon \quad (5.4.13)$$

and so the boundary layer can be seen to be in equilibrium.

Table 1 Wind Profile of Turbulence Model

Wind velocity (m/s)	Turbulent kinetic energy (m ² /s ²)	Dissipation rate near ground	Dissipation rate
40	35.65	23.59	1.05
51	57.82	48.9	2.18
63	88.23	92.18	4.11
72	115.23	137.6	6.13
87	168.25	242.75	10.81

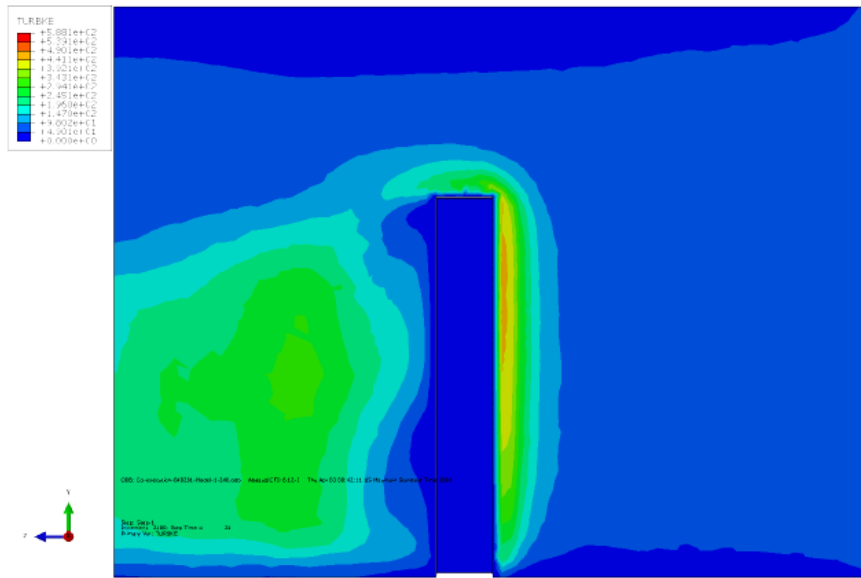


Figure 1 Turbulent Kinetic Energy Distribution (Center Cut View)

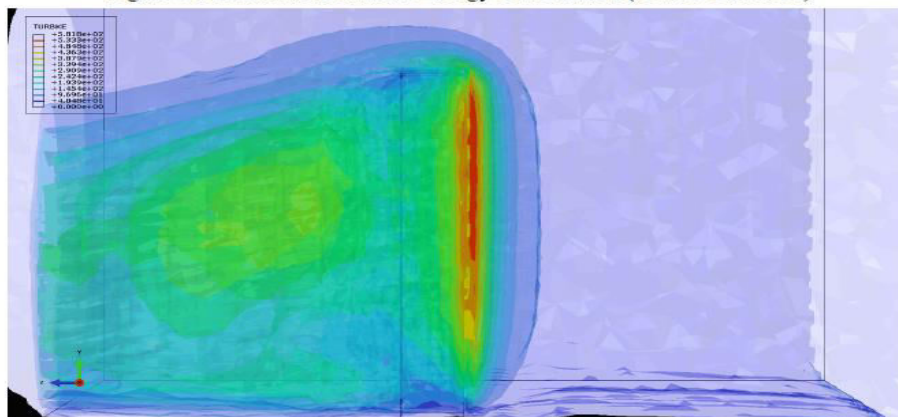


Figure 2 Turbulent Kinetic Energy Isolate Faces

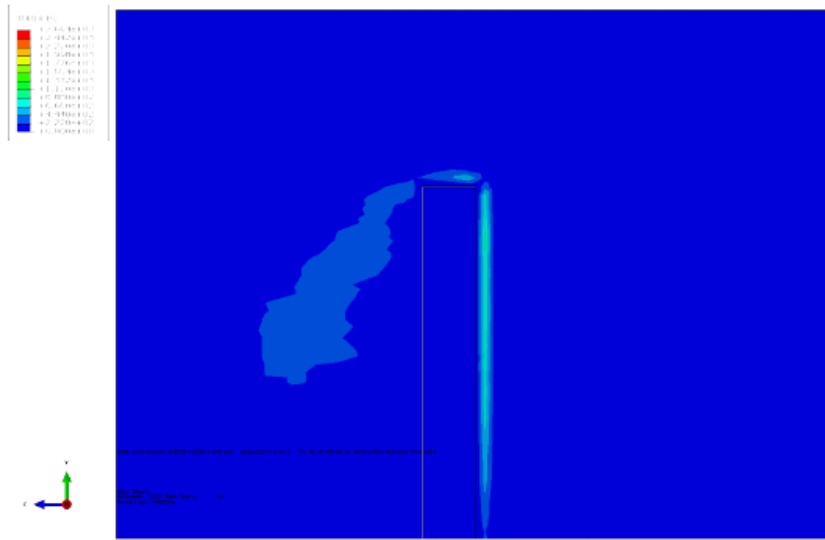


Figure 3 Dissipation Rate (Side Wall Cut View)

4 Numerical Analysis Tactics

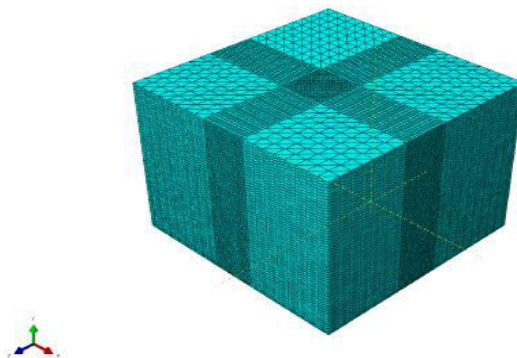


Figure 4 Mesh of the Entire Domain 3D view ($\theta=0^\circ$)

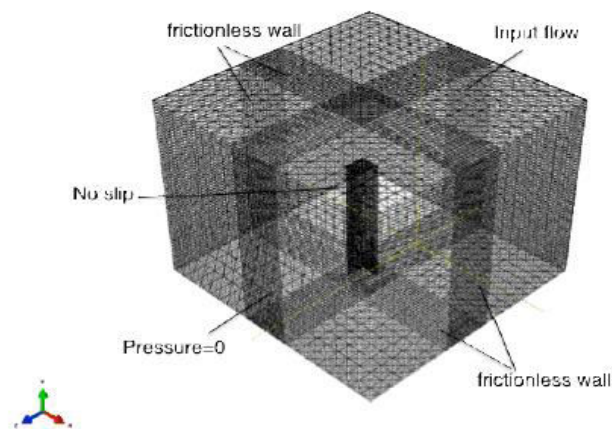


Figure 5 Mesh of the Entire Domain Transparent View ($\theta=0^\circ$)

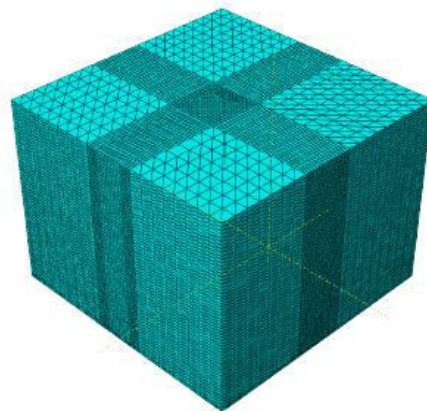


Figure 6 Mesh of the Entire Domain Transparent View ($\theta=45^\circ$)

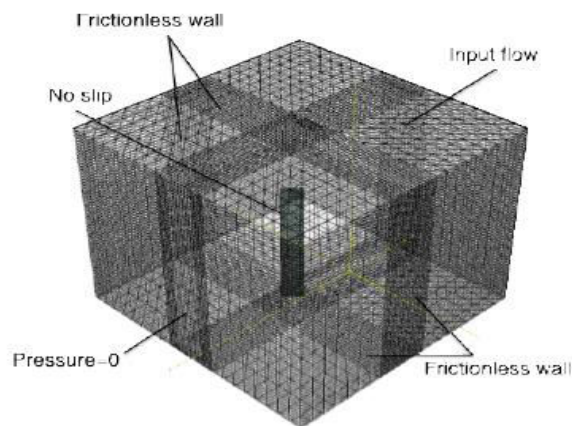


Figure 7 Mesh of the Entire Domain Transparent View ($\theta=45^\circ$)

This study demonstrates the methodology for investigating the coupled fluid-structure interaction (FSI) of wind flow around a cantilever building. The staggered FSI solution approach was employed, utilizing Abaqus/Standard as the structural solver and Abaqus/CFD as the incompressible Navier-Stokes solver. The case study showcases the effectiveness of Abaqus' multiphysics capabilities in analyzing the structural, fluid, and strongly coupled dynamics of wind flow interacting with an isotropic simulated building, with nearly identical fluid-solid density ratios.

To ensure the stability of the strongly coupled FSI problem, a fixed step algorithm within a staggered solution paradigm was utilized, eliminating the need for iterative strategies.

The boundary conditions are defined as follows:

- The input flow, detailed in the previous section, is directed along the z-axis.
- Zero pressure is imposed on the boundary opposite the input flow boundary.
- The surface of the structure is assumed to be smooth, subject to a no-slip condition, meaning there is no penetration or friction velocity on the surface.
- Other sides are subjected to a frictionless wall condition, ensuring sufficient distance from the structure to allow turbulence vortices to develop without influence from the boundary. This condition is verified by checking all fluid fields and results to ensure convergence.

III. CONCLUSION

The study you described represents a comprehensive analysis of wind load effects on a high-rise square-plan building, particularly focusing on comparing the results obtained from Computational Fluid Dynamics (CFD) simulations with the wind code ASCE 7-10. Here's a summary of the key points and methodologies used in the study:

1. **Diagrid Building Design** : The high-rise square-plan building is designed using a stiffness-based methodology, assuming it will be built in Arizona.
 2. **Wind Load Analysis** : Wind loads ranging from 40m/s to 87m/s are applied to the structure to investigate the relationship between wind loads and structural response.
 3. **CFD Analysis with Abaqus** : The 3D wind load analysis using the CFD module in Abaqus provides detailed wind loads that are more complex than those described in ASCE 7-10.
 4. **Assumptions and Methodologies** : Several assumptions and methodologies are employed to facilitate the numerical analysis, including:
 - Finite volume testing domain with independent results from mesh size, time step length, and distance from structure to boundaries.
 - Horizontally homogeneous boundary layer with constant ground aerodynamic roughness.
 - Use of HHTBL to modify input flow, assuming frictionless ground in the model.
 - Application of a linear isotropic hollow slender beam instead of detailed structure.
 - No-slip condition at the surface of the building without considering the influence of openings and claddings.
 - Application of the RNG k- ϵ turbulence model in Abaqus.
 - Assumption of small damping in the structure.
 - Focus on time-averaged displacements for comparison with wind codes, as top displacements are crucial in serviceability design for high-rise buildings.
 5. **Dragging Coefficient Calculation** : Dragging coefficient for this specific case is calculated using Timoshenko beam theory, but it's noted that it's not necessary for the CFD analysis.
 6. **Considerations in Analysis** : Dynamic responses of the structure are not initially considered to avoid numerical errors during the early time steps when the fluid is transitioning into a steady state.
 7. **Conclusion** : The study concludes that CFD modeling for wind load analysis on blunt body structures like high-rise buildings is a valid prediction and an effective methodology.
- Overall, this work provides valuable insights into the complex interactions between wind and high-rise buildings, showcasing the capabilities of CFD simulations in predicting wind loads and their effects on structures..

REFERENCES

- [1] M. Gu, Y. Quan(2004),” Across-wind loads of typical tall buildings”, Wind Energy and Industrial Aerodynamics , doi:10.1016/j.jweia.2004.06.004
- [2] Guoqing Huang, Xinzhong Chen(2007),” Wind load effects and equivalent static wind loads of tall buildings”, Engineering Structure , doi:10.1016/j.engstruct.2007.01.011
- [3] Michael Kasperski (2009),” Specification of the design wind load”, Journal of Wind Engineering and Industrial Aerodynamic, doi:10.1016/j.jweia.2009.05.002
- [4] Aly Mousaad Aly and Srinivasa Abburu(2011),” Dynamics and Control of High-Rise Buildings under Multidirectional Wind Loads”, Smart Material Research, doi:10.1155/2011/549621
- [5] K. Vishnu Haritha, Dr.I. Yamini Srivalli(2013),” Effect of Wind on Tall Building Frames”, International Journal of Research in Civil Engineering, ISSN Online: 2347-2855, Print: 2347-8284
- [6] F. Cluni , M. Gioffrè, V. Gusella (2013),” Dynamic response of tall buildings to wind loads by reduced order equivalent shear-beam models”, Journal of Wind Engineering and Industrial Aerodynamics, <http://dx.doi.org/10.1016/j.jweia.2013.09.012>
- [7] Ki-PyoYou, Jang-YoulYou andYoung-MoonKim(2014),” LQG Control of Along-Wind Response of a Tall Building with an ATMD”, Mathematical Problems in Engineering, <http://dx.doi.org/10.1155/2014/206786>
- [8] Vikram.M.B, Chandradhara G. P , Keerthi Gowda B.S.(2014),” A Study on Effect of Wind on the Static and Dynamic Analysis”, International Journal of Emerging Trends in Engineering and Development, ISSN 2249-6149

- [9] Umakant Arya, Aslam Hussain, Waseem Khan (2014),” Wind Analysis of Building Frames on Sloping Ground”, International Journal of Scientific and Research Publications, ISSN 2250-3153
- [10] Aly Mousaad Aly and Srinivasa Abburu (2015),” On the Design of High-Rise Buildings for Multihazard”, Shock and Vibration, <http://dx.doi.org/10.1155/2015/148681>
- [11] Djamal Hissein Didane, Nurhayati Rosly, Mohd Fadhli Zulkafli, and Syariful Syafiq Shamsudin(2017),” Evaluation of Wind Energy Potential as a Power Generation Source in Chad”, International journal of rotating machinery, <https://doi.org/10.1155/2017/3121875>
- [12] Nicola Longarini, Luigi Cabras, Marco Zucca, Suvash Chapain, Aly Mousaad Aly(2017),” structural Improvements for Tall Buildings under Wind Loads: Comparative Study”, Shock and Vibration, <https://doi.org/10.1155/2017/2031248>
- [13] Rabi Akhtar, Shree Prakash, Mirza Aamir Baig(2017),” Study of compmparison between Static and Dynamic Analysis Subjected to Wind and Earthquake Load”, International Research Journal of Engineering and Technology, eISSN: 2395-0056 , p-ISSN: 2395-0072
- [14] Huili Xue, Kun Lin, Yin Luo, Hongjun Liu (2017),” Time-Varying Wind Load Identification Based on MinimumVariance Unbiased Estimation”, Shock and Vibration, <https://doi.org/10.1155/2017/9301876>
- [15] Ashish Sadh, Ankit Pal (2018), “A Literature Study of Wind Analysis on High Rise Building”,International Journal of Advanced Engineering Research and Science ,ISSN: 2349-6495(P) | 2456-1908(O), <https://dx.doi.org/10.22161/ijaers.5.11.36>
- [16] A. Joseph Andrew, N Parthasarathi, S Selvi Rajan, Prakash(2018),” Dynamic response of tall buildings to wind loads by reduced order equivalent shear-beam models”, Materials Today, Materials Today: Proceeding: 5 (2018) 8923-8932
- [17] Fatih Topalog lu and Hu seyin Pehlivan(2018),” Analysis of Wind Data, Calculation of Energy Yield Potential, and Micrositing Application with WAsP”, Advance in Meteorology, <https://doi.org/10.1155/2018/2716868>



INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  ijirset@gmail.com



www.ijirset.com

Scan to save the contact details