

BEHAVIOR OF TALL MASONRY CHIMNEYS UNDER WIND LOADINGS USING CFD TECHNIQUE

DEEPAK SINGH¹, PUKHRAJ SAHU²

¹Research scholar, Department of Civil Engineering, Govt. Engineering College, Jagdalpur.

²Assistant Professor, Department of Civil Engineering, Govt. Engineering College, Jagdalpur.

Abstract: The design of building structures significantly affects its aerodynamic characteristics. The current research studies various researches conducted to determine the effect of building design on drag force and shear. The study conducted by various researchers includes experimental and numerical methods to determine aerodynamic characteristics. The research findings would provide useful information to engineers which could assist them in improving designs of building structures. This study proposes a technique based on Computational Fluid Dynamics(CFD) to calculate wind induced response of tall masonry chimneys using Finite Volume Method (FVM) by ANSYS.

Key Words: Aerodynamics, Building design, Drag force, Shear, Wind induced response, CFD, FVM.

1. INTRODUCTION

Aerodynamics is a field of dynamics that studies how air moves, especially when it collides with a solid object. The calculation of forces and moments acting on an item is made possible by understanding the motion of air around the object (also known as a flow field). The issues that designers face while designing super-tall buildings include wind influences. Super-tall buildings are more vulnerable to winds due to their slenderness, low natural frequencies, low inherent dampening level, and high wind speed at upper levels. With breakthroughs in structural design and high-strength materials, modern tall structures are becoming more taller. Every increase in height, however, introduces a new challenge. Building weight and damping are reduced, and slenderness is increased, thanks to efficient structural systems, high-strength materials, and increased height. Understandably, an appropriate building shape and architectural alterations are equally essential and successful design options for reducing wind produced motion by changing the flow pattern around the building. Drag forces are referred to as along wind or just wind [1]. Structures are subjected to aerodynamic forces caused by the wind, including the drag (along wind) force operating in the direction of the mean wind flow.

2. LITERATURE REVIEW

Cuong Nguyen et al [2] investigated the dynamic torsional behaviour of tall buildings under wind loads using a CFD analysis on different aspect ratios of rectangular tall buildings. The numerical outcomes are comparable to the experimental outcomes.

In the laminar flow regime, Hyeog Yoon et al [3] conducted a parametric investigation to elucidate the properties of flow past a square cylinder tilted with respect to the main flow. A topological classification of the linked flow patterns is attempted, yielding three unique patterns in total. The effects of Reynolds number and angle of incidence on flow-induced forces on the square cylinder were investigated in depth. For the ranges of the two parameters considered, contour diagrams of force and moment coefficients, Strouhal number, rms of lift-coefficient fluctuation, as well as a flow-pattern diagram, are proposed.

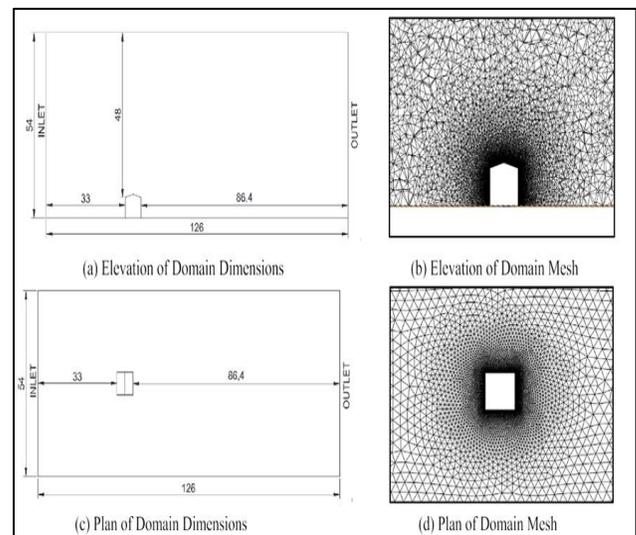


Fig. 1 Gable roof domain dimensions and mesh

3. CONCEPT OF CFD

The simulation of liquids in structures utilising numerical and demonstrative (scientific physical issue plan) approaches is known as computational fluid dynamics (CFD) (discretization strategies, solvers, numerical parameters, and matrix ages, and so forth.).

4. LIMITATION OF CFD TOOL

By solving a sequence of mathematical equations, CFD analyses the air velocity, temperature, pollutant concentrations, and degree of turbulence surrounding a building in order to predict air flow in and around that building. As a result, it's critical to start with mathematical modelling to estimate the essential computational inputs. Unlike other types of building simulations, such as energy simulations, CFD requires users to be familiar with mathematical modelling and numerical methodologies. The CFD modelling approach also employs physical models like as turbulence. However, in outdoor environment studies, computational domains might be rather vast, and boundary requirements aren't always obvious. This could lead to major computational errors in simulation results. Non-experts or architects have a difficult time performing CFD simulations.

4. CONCLUSION

Various researchers carry out numerical and experimental testing. Various studies have revealed that the incidence angle and geometric design of a building have a major impact on the structure's aerodynamic characteristics. The turbulence model employed in CFD analysis has a considerable impact on the drag and lift coefficients, pressure and velocity profile. The findings demonstrate that numerous elements impacting drag force and base shear on buildings should be considered in building designs.

REFERENCES

1. Taranath, B., Structural Analysis, and Design of Tall Buildings, McGraw-Hill Book, 2011
2. Cuong K. Nguyen, Tuan D. Ngo, Priyan A. Mendis, John C.K. Cheung, "Dynamic torsional behavior of tall building under wind loads using CFD approach", The fourth International Symposium on Computational Wind Engineering (CWE2006), Yokohama, 2006, pp 405 - 408

3. Dong-Hyeog Yoon, Kyung-Soo Yang and Choon-Bum Choi, "Numerical Study of Flow patterns past an inclined square cylinder", American Institute of Physics, 2010

4. Gera. B, Pavan K, Sharma, Singh R.K, "CFD analysis of 2D unsteady flow around a square cylinder", International Journal of Applied Engineering Research, Vol. 1 Issue 3, 2010, p 602

5. Ahmad. S, Muzzammil. M, Zaheer. I, "Numerical Prediction of wind load on low buildings", International Journal of Engineering, Science and Technology, Vol. 3, No. 5, 2011, pp. 59-72

6. Olawore A.S., Odesola I.F, "2D Flow around a rectangular cylinder: A computational study", International Journal of Science and Technology, Vol 2, No 1, 2013, pp 1 - 26

7. Ankit Mahajan, Puneet Sharma, Ismit Pal Singh, "Wind Effects on Isolated Buildings with different sizes through CFD simulation", Journal of Mechanical and Civil Engineering, Volume 11, Issue 3 Ver. IV, 2014, pp 67 - 72