International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 05, May-2022

Behavior of Tall Masonry Chimneys under Wind Loadings using CFD Technique

Deepak Singh
Department of Civil Engineering
Govt. Engineering College Jagdalpur

Mr. Pukhraj Sahu
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. Wind loads have a significant influence on the structural design of a power station chimney. Pressure coefficients (Cp) for chimneys are only available for a few aspect ratios, according to the literature. Furthermore, the data does not account for differences in wind pressure coefficients on the surface of the chimney where significant pressure or suction is expected. As a result, wind tunnel tests are required to determine the wind loads on chimney models. In order to achieve the goal of serviceability, interference must be considered in the design of high-rise structures. When one rises above ground level, it accounts for the change in mean wind velocity and turbulence parameters. The computation of the wind load necessitates extensive wind tunnel testing. CFD is a very strong tool for predicting wind-related events on chimneys and other structures.**

Key Words: Aerodynamics, Building design, Drag force, Shear, Wind induced response, Pressure coefficients, Wind tunnel test, 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.

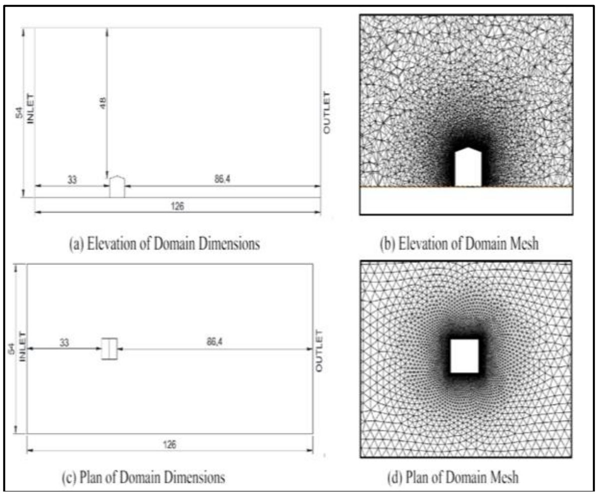

Fig. 1 Gable roof domain dimensions and mesh

3.WIND TUNNEL TEST

Wind tunnels have been utilised extensively for both industrial and scientific applications over the last 50 years.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 05, May-2022

Some are huge enough to hold and test small aircraft, while others are small enough to fit flow generators used in the calibration of small sensors, and they vary greatly in size and geometry. Wind tunnels have been used to test aerodynamic theories and simplify aircraft design, as well as to construct new aeroplanes, wind turbines, and other equipment that require complex interactions with airflow.

4.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.).

5. 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.

6. 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, "umerical 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 nsteady 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 ind 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 ylinder: 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