Journal of Material Science and Mechanical Engineering (JMSME)
p-ISSN: 2393-9095; e-ISSN: 2393-9109; Volume 6, Issue 3; April-June, 2019 pp. 212-213

http://www.krishisanskriti.org/Publication.html

NUMERICAL SIMULATION OF FLOW AROUND VARIOUS SHAPE STRUCTURES: A REVIEW

Abhishek Verma[1a] and Dr. N.K. Singh[2]

[1]M.Tech Research Scholar, Department of Mechanical Engineering, NIT Kurukshetra
[2]Associate Professor, Department of Mechanical Engineering, NIT Kurukshetra
E-mail: [a]abhishekverma6054@gmail.com

Abstract—*Structures experience various forces exerted over them in routine. Thus they are designed in order to bear them all in certain limits modified with some factor of safety. Wind forces are one of the most important of such forces especially in high rise buildings and tall structures like towers, etc. The wind forces have been tracked in the vicinity of the structure at the faces treated as walls for the fluid-structure interaction. The study involved comparison of the drag force magnitude by various angle of attack by wind and the vortices for various structural shapes feasible for construction. The observation of vortices in the nearby areas and the magnitude of forces has been compared with various other studies and has been found in good accordance with each other. This helps in knowing the building shape and construction method to be adopted for constructing various building shape. CFD is used as a simulation software it predicts drag and lift, pressure distributions etc.*

Keywords*: CFD- Computational Fluid Dynamics; Ansys Fluent; Aerodynamics Drag and Lift.*

Introduction

CFD (Computational Fluid Dynamics)is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering and fluid flows, and engine and combustion analysis. Aerodynamics is the study of the properties of moving air and the interaction between the air and solid bodies moving through it.

The effects of wind loads of various

Buildings and structures has been considered for design purposes since late in the 19th century to 1950 during these years the studies in this field have not been considered seriously.

Structure and its various components have to be designed in such a way so that they can withstand wind load.

Calculation of wind load is important in the design of structures, forces and moments that are acting on object can be calculated with the help of motion of air. While designing the tall buildings wind effects are the major task to deal. With the advancement in the structure design modern tall buildings are made higher and higher. A drag force is the resistance force caused by the motion of a body through a fluid, such as water or air. A drag force acts opposite to the direction of the oncoming flow velocity that is along wind is drag.*Lift* is the component of this *force* that is perpendicular to the oncoming flow direction. When the wind flows structure experiences aerodynamic forces which includes drag which is along wind and lift force perpendicular to direction of wind.In design of most of the tall building lift force dominates as compared to the drag force.

So an appropriate selection of the building shape is important in order to minimize the wind load.

Literature Review

Benny Gabriel Jebaraj et al [1] performed a CFD analysis and calculated aerodynamic force coefficients like lift, drag, coefficient of lift and drag at various angle of attack by wind on an octagonal shape building and its results are compared with the wind tunnel experiment.

[2]When the wind flows structure experience various aerodynamic forces along wind is drag, across wind is lift across wind is perpendicular to the direction of wind. In design of most of the tall building lift dominates as compared to the drag force.

Dong- Hyeog Yoon et al [3] undergone a parametric study to study the characteristics of flow past a inclinedsquare cylinder Cuong Nguyen et al [4]

CFD analysis is performed on different buildings with rectangular cross section and the dynamic torsional behaviour Of tall buildings under different wind loads is studied. The numerical results are being compared with the experimental results Gera et al [5] performed CFD analysis on two dimensional square cylinder and studied the unsteady flow considering the flow as laminar.

The effects of Reynolds number on lift, drag, Strouhal number is studied.

Ahmad et al[6] performed CFD simulation of wind loads like drag and lift on the roofs of several low rise buildings with full scale Reynolds number, and simulation of turbulence boundary layer properties have also been done for various angle of attacks by wind. The study shows the importance of simulation of wind loads while designing.

Olawore and Odesola [7] Pressure distribution, Strouhal number drag coefficient, lift coefficient are calculated for various Reynolds number around the cylinders with rectangular cross section Thus the conclusion is vortex shedding phenomenon should also be considered while designing.

Ankit Mahajan et al [8] compared the result of CFD simulation with the wind tunnel experiment. Testing of three different square cylinders were done with angle of incidence from 0° to 90° at a step of 15°. It was observed that the value for side face and front face are close.

Kavya et al [9] study about the two dimensional unsteady flow past a square cylinder at various angle of incidence and determined that the eddy size depends on angle of incidence and size of eddy increases as the angle of incidence increases.

Md. Jomir Hossain et al [10] performed experiment for calculating wind load on cylinder with octagonal cross section.

The study was done on both single and group of cylinders .Firstly experiment was done on a single cylinder at various angle of attack. Calculation of pressure coefficient, drag and lift is done at constant wind velocity of 13.2 m/sec

Conclusion

CFD analysis is well suited for complex structures as the various load like drag and lift can be easily calculated which helps in knowing the building shape and construction method to be adopted for constructing various building shape. Calculation of wind load is important in the design of structures, while designing the tall buildings wind effects are the major task to deal. The coefficient of drag in aoctagonal cylinder is less as compared to that of cylinder with square cross section at same angle of attack by wind. So an appropriate selection of the building shape is important in order to minimize the wind load.

References

1)Benny Gabriel Jebaraj and Freeda Christy C**, "**CFD Simulation of flow around octagonal shaped structures"

2).Taranath, B., Structural Analysis, and Design of Tall Buildings, McGraw-Hill Book, 2011

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)Cuong K. Nguyen, Tuan D. Ngo, Priyan A. Mendis, John C.K. Cheung, "Dynamic torsional behaviour of tall building under wind loads using CFD approach", The fourth International Symposium on Computational Wind Engineering (CWE2006), Yokohama, 2006, pp 405 – 408

5)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

6). 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

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

8)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

9)Kavya H.P., BanjaraKotresha, KishanNaik, CFD Analysis of 2D unsteady flow past a square cylinder at an angle of incidence, International Journal of Advanced Research in Mechanical and Production Engineering and Development, Vol.1 (2), 2014, pp 117-125

10) Md. Jomir Hossain, Md. Quamrul Islam and Mohammad Ali, "Lift and drag on cylinder of octagonal cross section in a turbulent stream", Journal of Engineering Science & Technology Review, Vol. 6 Issue 3, 2013, pp 105-110.