IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 3 Ver. IV (May- Jun. 2014), PP 67-72
www.iosrjournals.org

"Wind Effects on Isolated Buildings with Different Sizes through CFD Simulation"

Er.Ankit Mahajan, Er. Puneet Sharma, Er. Ismit Pal Singh
[1](Civil Department, Chandigarh University,India)
[2](Civil Department, Chandigarh University,India)
[3](Civil Department, Chandigarh University,India)

Abstract: *Calculation of wind load is necessary for design of tall buildings, usually wind load is calculated with the help of wind codes or wind tunnel testing. As with technological advancement it is now possible to simulate wind tunnel environment in computers with the help of software like ANSYS which is quite cheaper as compared to costly wind tunnel experiments.*

In this study we carried out the Computational Fluid Dynamics (CFD) experiment of wind tunnel flow. Validation of the CFD Simulation of wind tunnel simulation through the comparison of the available results of experimental work carried out. In this study the three different models have been considered. Setup 1 consists of the isolated building of size 50X50X300mm, Setup 2 consists of the isolated building of size 75X75X300 mm and compares it with step 1. Setup 3 consists of the isolated building of size 100X100X300 mm and compare with step1 and step 2. Wind incidence angle is varied from 0^0 to 90^0 with 15^0 variation.

After computational study for setup 1 it is observed that for 0^0 wind incidence angle for isolated building model the values of the coefficients from CFD is close to those specified in ASCE 7-02 and AS/NZS 1170.2. On comparison with IS 875 (part 3) it is observed that for front face and side face of the building the values obtained are in close agreement. But for back face there is large variation when compared.

After comparison of all three setup it is observed that for face A and face B there is not any significant variation between all three setups. For face C setup 3 gives most severe suction at 60^0 as compared to the other two whereas for face D setup 2 gives most severe suction value at 45^0.

Keywords: *wind load, wind tunnel, ANSYS 13.1, fluent,meshes*

I. INTRODUCTION

I Wind effects on buildings and structures can be accurately obtained by direct measurements on small-scale models tested in a wind tunnel capable of representatively simulating the characteristics of natural wind at the site. The advantage of model experiments over full scale measurements resides chiefly in the possibility of establishing exactly the properties of basic materials, the boundary conditions of the model and the air flow characteristics. Further, wind tunnel experiments are convenient because they are not disturbed by effects likely to present in full scale tests.

II. The success of model analysis depends on correct simulation of the phenomenon and quantities participating in the process, exact construction of the problem to be solved and on the modeling and measurement techniques used. For complete similarity all the relevant numbers as Reynolds, Froudes, Newton, Strouhal, Cauchy or Hookes number should be equal for both model and prototype. But in all the cases complete similarity is either not possible or not required. There are evidences that failure to make complete similarity does not compromise the reliability of wind tunnel tests**.**

III. Basically there are two types of models rigid and aeroelastic. Rigid models are used in the studies involving phenomenon induced by the constant wind component which are not influenced by model response as coefficients of drags and Lifts, aerodynamic coefficients of other forces and moments. Rigid models are used to determine local suction on leading edges and static wind loading on tall buildings in a built environment. These models satisfy only the geometric similarity requirement. The structure mass stiffness and damping are not modeled. Aeroelastic models are used to study the quantities which are influenced by the model response as acceleration, natural frequency, amplitude of vibrations etc. In these models mass, stiffness, natural frequency and damping parameters are modeled and all the relevant model laws are satisfied.

IV. Flow characteristics generally modeled in a boundary layer wind tunnel are vertical distribution of mean wind speed, intensity of longitudinal turbulence, and integral scale of longitudinal turbulence. For tall buildings entire boundary layer is simulated, while for low buildings only the surface layer simulation may be sufficient.

II. TURBULENCE INTENSITY

The turbulence intensity, also often referred to as turbulence level, is defined as:

$$I = \frac{u'}{U}$$

Where u' is the root-mean-square of the turbulent velocity fluctuations and U is the mean velocity (Reynolds averaged).

If the turbulent energy, k, is known then u' can be computed as:

$$u' = \sqrt{\frac{1}{3}\left(u'_x + u'_y + u'_z\right)} = \sqrt{\frac{2}{3}k} \quad (1)$$

U can be computed from the three mean velocity components U_x, U_y and U_z as:

$$U = \sqrt{(U_x^2 + U_y^2 + U_y^2)} \quad (2)$$

2.1 Estimating the Turbulence Intensity

When setting boundary conditions for a CFD simulation it is often necessary to estimate the turbulence intensity on the inlets. To do this accurately it is good to have some form of measurements or previous experience to base the estimate on. Here are a few examples of common estimations of the incoming turbulence intensity:

1. High-turbulence case: High-speed flow inside complex geometries like heat-exchangers and flow inside rotating machinery (turbines and compressors). Typically the turbulence intensity is between 5% and 20%
2. Medium-turbulence case: Flow in not-so-complex devices like large pipes, ventilation flows etc. or low speed flows (low Reynolds number). Typically the turbulence intensity is between 1% and 5%.
3. Low-turbulence case: Flow originating from a fluid that stands still, like external flow across cars, submarines and aircrafts. Very high-quality wind-tunnels can also reach really low turbulence levels. Typically the turbulence intensity is very low, well below 1%.

2.2 Wind Tunnel Used

The wind tunnel used from which the present study is verified is an closed circuit type, continuous flow with closed test section wind tunnel using fan (65 HP) section of 1.3 m (width) x 0.85 m (height) cross-section with 8.25m length. Line diagram of the wind tunnel is given in Fig- 3.1. The reference wind velocity was maintained as 15 m/sec.

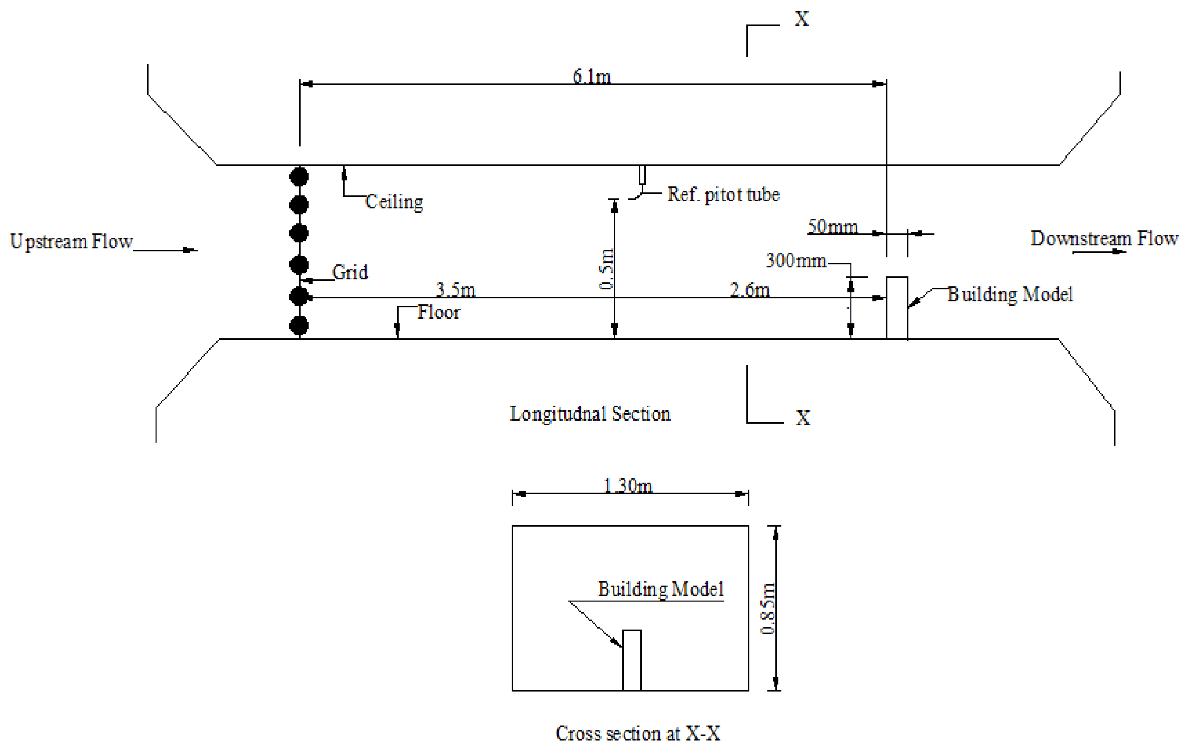

Fig-2.1 Line Diagram of Wind Tunnel used in the experimental work

2.3 Flow Characteristics At Wind Tunnel

The study is done on building models, the prototype of which is assumed to be situated in open terrain with well scattered obstructions having height generally between 1.5 to 10 m. As per IS: 875 (part3) – 1987 code, this type of terrain is categorized as terrain category 2 with a power law index of 1/7. Theoretically, variation of wind speed with height is represented by an equation given as:

$$V_z = Vo\left(\frac{Z}{Zo}\right)^n$$

V_z = Wind speed at height z

V_0 = Mean wind speed at a reference height Z_0

n = A constant dependent on ground roughness (0.159).

The velocity profile from the wind tunnel and used in CFD is as shown in Fig. 3.2. The turbulence is also recorded with the help of Pitot tube in combination with Baraton pressure transducer; Fig. 3.3 shows the turbulent intensity of flow in the wind tunnel with maximum value of about 2.5% near the floor. In order to find the power law index n, the velocity profile is plotted on log-log scale as shown in Fig 3.4. The graph is plotted between the log of V_z/V_0 on the X - axis and log of Z/Z_0 on Y- axis. The n is found to be about 0.159.

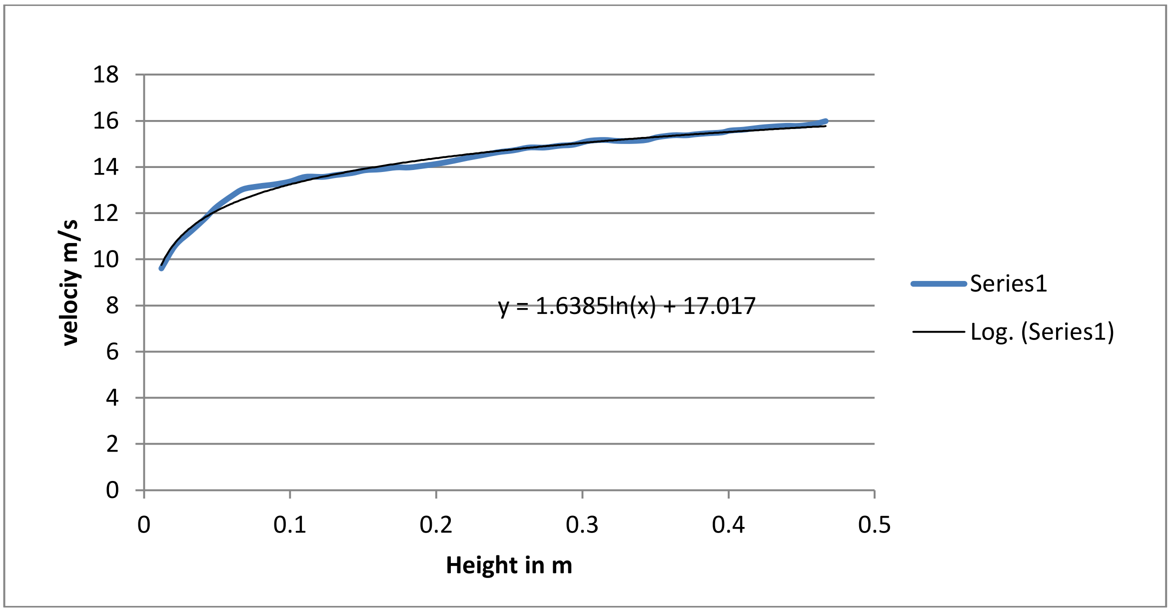

Fig 2.2 Velocity profile used in CFD simulation

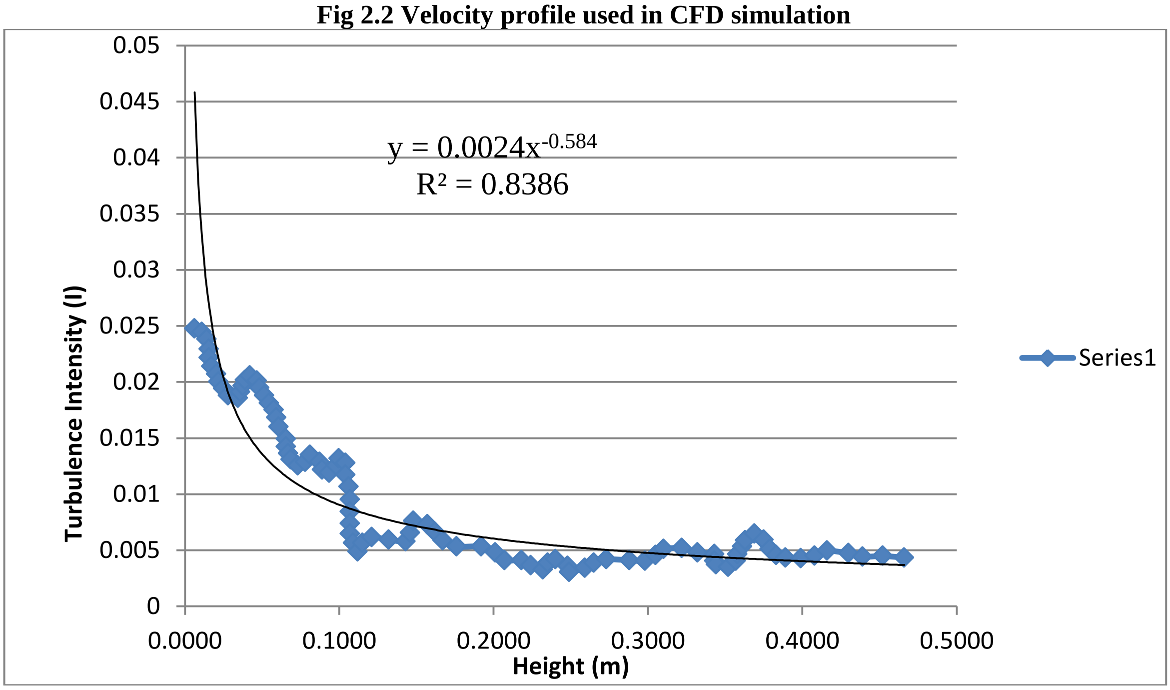

Fig 2.3 Turbulence intensity used in CFD simulation

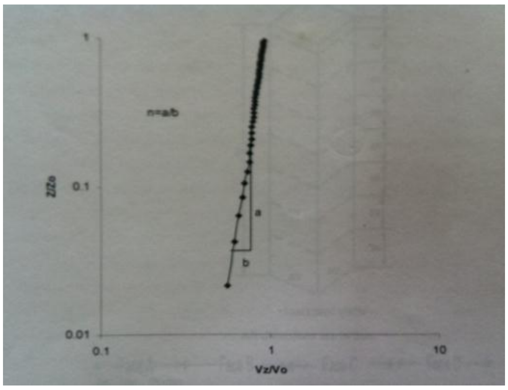

Fig. 2.4 Velocity profile in wind tunnel on log-log scale

2.4 Details Of Wind Tunnel Models

The present research work aims at studying the interference effect of neighboring building on the static forces and moments on a tall building. Such forces and moments are not basically affected by the response of the building. Thus the rigid models are used for the study. The simulation of rigid model with sharp edges is straight forward. It is achieved by making geometrically similar model.

The geometric scale of model of a building is chosen to maintain the equality of ratios of overall building dimensions to the inherent lengths of the generated model wind. These inherent lengths may be roughness length of terrain, the boundary layer depth and the integral scale of the longitudinal component turbulence. Some other factors include the size of the test section and the ranges of parameters to be studied. In selecting the model scale it is important to avoid the influence of the wind tunnel walls and an excessive blockage of the test section. Corrections are generally applied if the blockage by the model of the building and its immediate surroundings exceeds about 5% to 10%. Typical geometrical scales used in studies of wind, effects on large buildings are about 1:300 to 1:600, while for models of small buildings larger scales in the range of 1:100 may be used. In experimental study 1:600 scale was used.

III Analytical Programme And Data Analysis

3.1 General

Details of the wind tunnel and model used for experimental work are presented in the chapter-3 and the basic of wind tunnel flow simulation and requirements of atmospheric boundary layer is given in Chapter-4. In this chapter we present the details of the model used for the analytical study through CFD simulation of wind tunnel flow.

3.2 Parameters Considered

In the current study we have considered case of two building models with dimensions 50mm×50mm×300mm and spacing between interfering and principal building is taken as 50mm (SET-2). Wind incidence angle (β) is varied from 0^o to 90^o with variation of 15^o as shown in fig-3.1

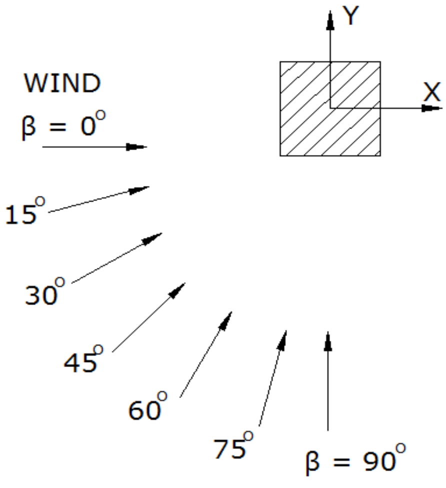

Fig-3.1 Models considered for analytical study

As explained earlier that the no turbulence model is perfect so one have to try and choose best suited model for the study under consideration. Realizable k-epsilon model is most commonly used model but literature suggests that the k-epsilon model gives low suction values at the rear face of the building. In this study we have considered realizable k-epsilon and SST (Shear stress transport 4 equation model).

3.3 Pressure Model Analysis

In this analytical study the pressure model analysis was carried out. A pressure test measures local pressures at discrete points on a model. The point pressure measurements are distributed to ensure sufficient coverage to provide information on entire surfaces of the building. The results are generally used for the design of windows, cladding, curtain walls, etc.

In case of wind tunnel experiment model is made of 6 mm thick Perspex sheet. Each "pressure tap" on the rigid model is connected to a pressure transducer through PVC tubes. The pressure measurements are carried out using a high-speed pressure scanning system. But for the CFD simulation of this study first we created the geometry of the wind tunnel section and model used for the experimental work and generated its mesh as discussed in Chapter-4.

Pressure Coefficient (Cp)

The pressure exerted by wind at a point on the structure will be different from the pressure of wind far upstream of the structure, called 'free stream pressure'. The ratio of the difference of the pressure at a point on the structure and static pressure of the incident wind to the design wind pressure is termed as pressure coefficient.

$$C_p = \frac{p - p_f}{\frac{1}{2}\rho U_f^{\,2}}$$

Where

C_p = pressure coefficient or shape factor
p = relative pressure
p_f = reference pressure -subscript implies the free stream conditions
ρ = density of air
U_f = velocity in the approach flow at the height of building

IV. Conclusion

Most of the conclusions are discussed in the result and discussion chapter. Therefore, the broad conclusions arrived at and the future scopes of research are given below.

Computational Study

i. Computational Fluid Dynamics (CFD) offers a very powerful alternative to predict the wind related phenomena on buildings or different kind of structures.
ii. Flow separation and visualization can be possible accurately by the CFD simulation.
iii. The pressure values, flow streamline, velocity vector and numbers of related parameter variables etc. through the model surface can be determined with the help of CFD analysis.
iv. The accuracy of results depend also on exactly the modelling according to the scale, proper meshing of the model geometry and defining the physical property values exactly as the realistic environment conditions.
v. Many complicated and complex model can be examined with less effort by the help of CFD analysis and the designing criteria of the structure system or any fluid flow related system can be considered approximately.
vi. A part from wind tunnel study, full scale model of physical problem can be modelled and analysed with perfection by this numerical simulation.

Major Contribution of The Research Work

1. CFD results are close to the corresponding experimentally obtained values and hence can be used as a good alternative of wind tunnel testing.
2. The designers should be very careful in picking up the appropriate values of pressure coefficient as in CFD it is observed that values on front face is more and on back face is less with minor differences as compared to wind tunnel data.

Recommendations For Future Research

Even after carrying out research work in this thesis, there is still further scope of work which can be carried out in the area of wind effects on tall buildings with CFD as detailed below.

1. Building models with different shape, plan area and front area.

2. Use of CFD simulation with the wind tunnel model and studying number of parameters on different building models.
3. The effect of increasing the terrain roughness on wind load for tall building should also be studied at different wind incidence angles.

References

[1]. Ansys tutorial manuals
[2]. American National standard ANSI: A58, 1982
[3]. Blocken B., Stathopoulos T. and Carmeliet J. (2007b), "CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow", Journal of Wind Engineering and Industrial Aerodynamics, 95, pp. 941-962
[4]. Blocken B., Stathopoulos T., Saathoff P. and Wang X. (2008), "Numerical evaluation of pollutant dispersion in the built environment: Comparisons between models and experiments", Journal of Wind Engineering and Industrial Aerodynamics, 96, pp. 1817-1831.
[5]. Beneke D.L., Kwol K.C.S., Aerodynamic effect of wind induced torsion on tall buildings,
[6]. Journal of Wind Engineering and Industrial Aerodynamics, 50 (1993) 271-280
[7]. Davenport,A.G.(1960). "Rationale for Determining Design Wind Velocities." J. Struct. Engrg., ASCE, 86(ST5), 39-67.
[8]. Davenport,A.G.(1961a). "The Application of Statistical Concepts to the Wind Loading of Structures." Proc. ICE London, 19, 449-4
[9]. Gumbel, E,J. (1958). Statistics of Extremes, Coloumbia University Press, New York
[10]. Gu M., Quan Y., Across-wind loads of typical tall buildings, Journal of Wind Engineering and Industrial Aerodynamics 92 (2004) 1147–1165
[11]. Gu Z., Yang L., Li Y., Chen B., Aeroelastic modeling of wind loading on a cable-net supported glass wall, Acta Mech Sin (2010)
[12]. Ilgin H.E., Gunel M.H., The role of aerodynamic modifications in the form of tall buildings against wind excitations, METU Journal of Faculty of Architecture, Vol 2, 2007, pp 17-25
[13]. Isyumov,N.(1999). "Overview of Wind Action on Tall Building and Structures." *Wind Engineering into the 21st Century, Larsen, Larose & Livesey (eds-1999) Balkema*, 1527
[14]. Joseph, M.G. and Radhakrishnan, R (1978). A response spectrum of dynamic effects of wind on tall structures. Symposium Earthquake Engineering, University of Roorkee
[15]. Jain, P.K. (1971). Maximum wind pressure over India. Indian Journal of Met. & Geophysics, Vol.22.
[16]. Kareem,A.(2003). "A Tribute to Jack E. Cermak - Wind Effects on Structures: A Reflection on the Past and Outlook for the Future." *Proc. XI Int. Conf. on Wind* Engrg.,(Invited Speakers: INV.M2).,1-28
[17]. Kim Y., Kanda J., Characteristics of aerodynamic forces and pressures on square plan buildings with height variations, Journal of Wind Engineering and Industrial Aerodynamics 98 (2010) 449–465
[18]. Kim Y., Kanda J., Tamura Y., Wind-induced coupled motion of tall buildings with varying square plan with height, Journal of Wind Engineering and Industrial Aerodynamics, 99 (2011) 638–650
[19]. Lin N., Letchford C., Tamura Y., Liang B., Nakamura O., Characteristics of wind forces acting on tall buildings, Journal of Wind Engineering and Industrial Aerodynamics 93 (2005) 217–242
[20]. Mani, Anna and Mooley, D.A. (1984). Wind Energy Data for India. Allied Publishers, New Delhi
[21]. Mendis P., Ngo T., Haritos N., Hira A., Samali B., Cheung J., Wind Loading on Tall Buildings, EJSE Special Issue: Loading on Structures (2007)
[22]. Merrick, R. , Bitsuamlak, G.T., 2009, Shape effects on the wind-induced response of high-rise buildings, Journal of Wind Engineering, 6(2), 1-18
[23]. Ning Lina, Chris Letchforda, Yukio Tamurab, Bo Liangc and Osamu Nakamurad (Local wind forces acting on rectangular prisms. Proceedings of 14th N ational Symposium on Wind Engineering, 4 -6 December 1996, Japan Association for Wind Engineering, Tokyo, pp. 263 -268