

CFD Simulation for Wind Load on Octagonal Tall Buildings

Dr S.K. Verma ^{#1}, A.K Roy ^{*2}, Sunil Lather ^{*3}, Manoj Sood ^{*4}

¹Associate Professor, Department of Civil Engineering, PEC university of Technology, Chandigarh, India

² Civil Engineering Department, NIT Hamirpur, Himachal Pradesh, India

Abstract— Lateral loads i.e. wind load, seismic load, govern the design of tall buildings. Wind code IS 875 (PART-3rd) provides the pressure coefficients (C_p) only at 0° and 90° wind incidence angles and only for few standard cases. Therefore, experimental studies become important and wind tunnel experiment is carried out to measure wind loads on the models of tall buildings at different wind incident angles. CFD offers a very powerful alternative to predict the wind related phenomena on buildings or different kind of structures. A software package, FLUENT-14 (ANSYS 14.0) is used for CFD analysis. CFD analysis is carried out considering the same parameters as used in the experimental study. The buildings chosen in present study is octagonal in plan. Mean area weighted average wind pressures on the faces of the building models are measured in order to study the effect of varying wind incident angles namely 0° , 15° and 30° on wind pressure distribution.

Keywords: Tall building; computational fluid dynamics (CFD); pressure coefficients; wind tunnel testing; wind incidence angle

I. INTRODUCTION

Structural design of tall buildings is dictated by wind loads. Generally wind loads are determined by referring to relevant wind loading codes and relevant literature. Some of the relevant wind load codes are: [AS/NZS: 1170.2-2002, ASCE: 7-02-2002, IS: 875-(Part-3)-1987, NBC-(Part-4)-1995]. However, the information available in such codes regarding wind pressure or force coefficients even for a simple plan shape such as square one is for limited wind directions only. For building shapes other than rectangular plan shapes very limited information with regards to wind loads is available. This necessitates the need for extensive wind tunnel testing for buildings of peculiar shapes. But wind tunnel tests being very expensive and time consuming; Computational Fluid Dynamics (CFD) simulation has become a good alternative. Dalui 2008 has carried out experimental investigation on octagonal shaped tall buildings, Stathopoulos 1996 and Blocken 2008 used CFD for assessing the wind loads. In present study a numerical modeling employing CFD has been used and the results obtained are compared with those obtained from experimental study

II. EXPERIMENTAL PROGRAM

The experimental study has been conducted in the Closed Circuit Wind Tunnel at the Department of Civil Engineering, Indian Institute of Technology Roorkee, Roorkee, India. It is

a closed circuit, continuous flow, closed test section type (i.e. closed jet type) wind tunnel with single fan (65 HP). Cross-section of the test section is $1.3 \text{ m} \times 0.85 \text{ m}$ size and the length of the test section is 8.2 m. In the present study models are made at a length scale of 1:300. Table 1 gives the comparison of dimensions in case of prototype and the models.

TABLE 1
PROTOTYPE AND MODEL DIMENSIONS

Parameter	Prototype (m)	Model (mm)	Scale
Length	30	100	1:300
Width	30	100	
Total Height	150	500	

Models (Fig. 1) have been prepared using 6 mm thick perspex sheets for the experimental study in the wind tunnel. Pressure tapings are installed on the surfaces of building models for measuring the surface pressure and mean pressure coefficient values are calculated for each face at different wind incidence angles.

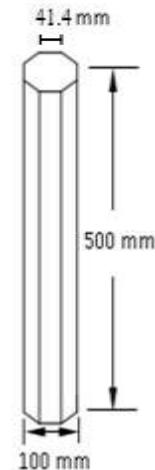


Fig. 1 Model used for experimental study

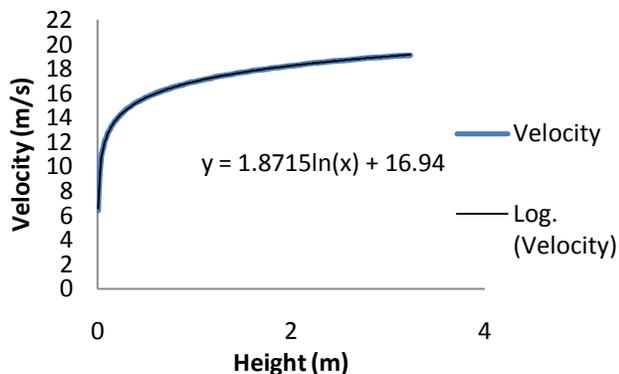


Fig 2 Velocity profile used in CFD Simulation

III. CFD SIMULATION OF WIND TUNNEL STUDY

CFD simulation for Wind essentially consists of following:

A. Simulation of Atmospheric Boundary Layer Flow
 These profiles should be representative of the roughness characteristics of that part of the upstream terrain that is not included in computational domain (i.e., the terrain upstream of inlet plane).

B. Validation of Atmospheric Boundary Layer Flow Simulation
 The velocity profile graph of wind tunnel is used and mathematical equation is generated which are used in user defined functions (UDF's). UDF are obtained through C-programming which are generally defined at inlet face of model in Fluent to simulate atmospheric boundary layer. For generating our UDF the velocity profile obtained in wind tunnel is used which is shown in Fig 2.

C. Detail of meshing of the geometry of experimental setup
 Fine meshing is used on the building face, near the bottom area around building. The quality of mesh is 0.95-1.0 for model with 0° and between 0.55 to 0.95 for 75° wind incident angle. The quality of mesh can be checked in ICEM CFD and if this value is above 0.5 (range is 0.0-1.0) the quality of mesh is referred as good. To get first cell height we need some calculations as suggested by CFD package. If Reynolds number is more than 5×10^5 flow over a surface will be turbulent. For most of wind flow problems flow is turbulent.

V. RESULTS AND DISCUSSION
 The wind pressure measurements are made by varying the wind incidence angle from 0° to 30° at an interval of 15° i.e. at 0°, 15°, 30° for octagonal building.

Wind flow striking at right angle to Face A of all the buildings is taken as 0° wind incidence angle and the flow of other wind incidence angles (i.e. 15°, 30°) is the

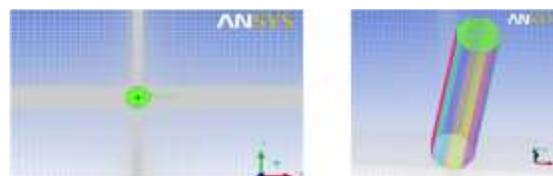


Fig 3 Top view and side view of mesh of Octagonal Building Model

D. Domain Considered for CFD Simulation of Experimental Models

The building dimensions were same as the buildings used for the wind tunnel study as explained earlier i.e. 100mm × 100mm × 500 mm. The wind tunnel cross section (i.e. domain dimensions) taken for the CFD simulation was considered using optimum domain recommendations by Roy A.K.(2012).

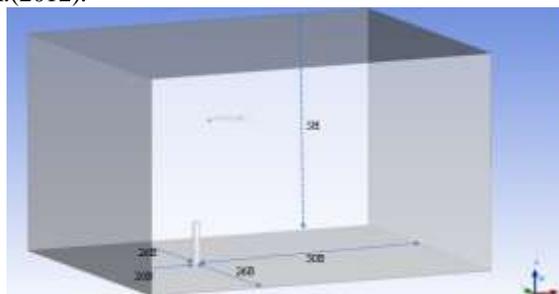


Fig 4. Computational domain of the building models used for CFD simulation (Roy . 2012)

IV. SOLVER SETTINGS

Normally the solutions are steady-state solutions. Second-order differencing was used for the pressure, momentum and turbulence equations and the “coupled” pressure-velocity coupling approach due to its robustness for steady-state, single-phase flow problems. The residuals fell below the commonly applied criteria of falling to 10^{-4} of their initial values after several hundred iterations. However, this was not the only test for convergence the drag, lift and side forces and the moments acting on the building were monitored during the simulation and only when they achieved stationary values were the simulations deemed to have converged.

wind deflection in clockwise direction with reference to 0° wind incidence flow at A is taken into account in all three building cases. Fig. 5 shows the direction of wind flow for the building models

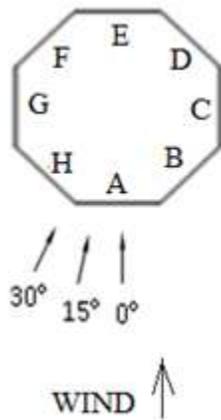


Fig. 5 Direction of incidence angle (wind flow) for Building Models

The results are calculated as area weighted average of pressure coefficient C_p on each face. The reference velocity is taken as 15 m/s at the height of 500 mm from the base of the model.

A. Uniform Octagonal Plan Shape Building

The building model with uniform octagonal plan shape was tested in the wind tunnel by S.K. Dalui (2008) to measure the wind pressure distribution on its surfaces under varying wind incidence angles. Table 1 depicts the variation of mean wind pressure coefficients on different faces of uniform octagonal building at different wind incidence angles.

Face A: It can be observed from Table 1 that at 0° wind incidence angle, as the wind strike at right angle to the Face A this face experiences maximum positive pressure. It can also be observed from this table that positive pressure decreases with the increase of wind incidence angle. At the wind incidence angle of 30° , almost half mean positive pressure coefficient is noticed as compared to the 0° wind incidence angle at this face because of increase in suction with the increase in wind incidence angle.

TABLE 1
FACE AVERAGE C_p (MEAN) CFD COMPUTED VALUES ON DIFFERENT FACES FOR VARIOUS WIND INCIDENCE ANGLES FOR OCTAGONAL BUILDING MODEL

Wind Incidence Angle	C_p (mean)value for Building Face							
	Face A	Face B	Face C	Face D	Face E	Face F	Face G	Face H
0°	0.835	0.287	-0.572	-0.694	-0.638	-0.674	-0.572	0.286
15°	0.776	-0.38	-0.58	-0.705	-0.681	-0.668	-0.601	0.34
30°	0.473	-0.447	-0.671	-0.63	-0.634	-0.627	-0.514	0.681
Max +ve	0.835	0.287	-0.572	-0.63	-0.634	-0.627	-0.514	0.681
Max -ve	0.473	-0.447	-0.671	-0.705	-0.681	-0.674	-0.601	0.286

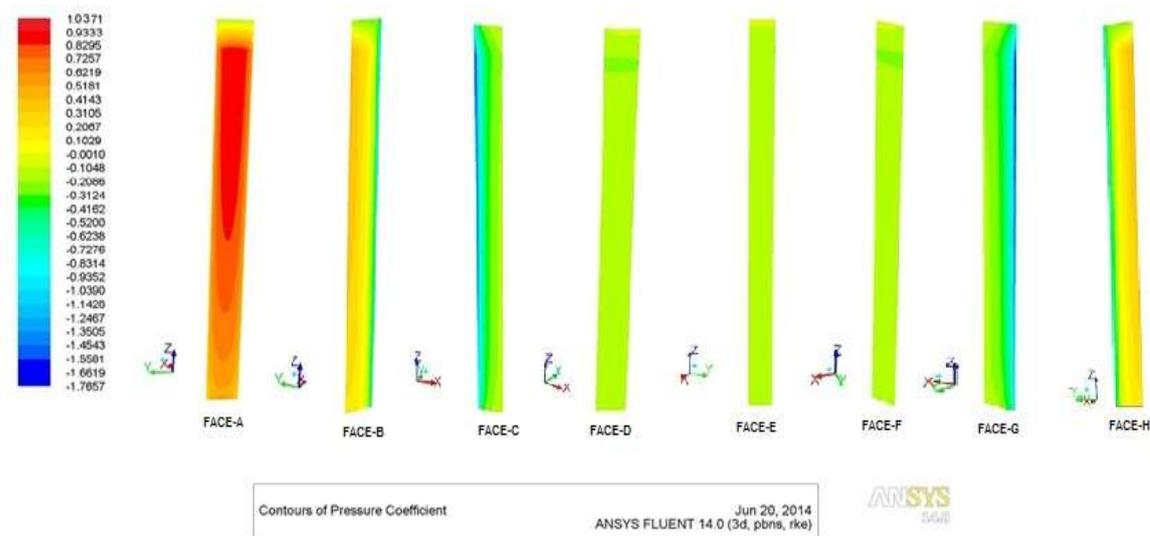


Fig. 6 Contours of pressure coefficient at 0° wind incidence for Face A to H

From Fig. 6 it can be observed that positive pressure on wind ward face increases from bottom to top edge of the face due to increase in wind velocity with height. Further it can also be seen that maximum positive pressure is observed along the centre line of the wind ward face and this pressure decreases towards the edges of the face.

Face B: From Table 1 at 0° wind incidence angle Face B experience positive pressure but its value is much less as compared to that at Face A. Suction goes on increasing from windward side to leeward side. For 15° and 30° wind incidence angles this face experiences suction. Variation in

suction from 15° wind incidence angle to 30° wind incidence angle variation is very less i.e. from -0.38 to -0.447 respectively.

Face C: It can be observed from Table 1 that at all the wind incidence angles i.e. 0°, 15° and 30° this face experience suction and suction value keep on increasing with increase in wind incidence angle. This face is sideward face at 0° wind incidence angle and it become inclined leeward at 15° and 30° wind incidence angles.

Face D: It can be observed from Table 1 that at all the wind incidence angles i.e. 0°, 15° and 30° this face experience suction. For this inclined leeward face maximum suction occurs at 15°.

Face E: It is observed that Table 1 that negative pressure (suction) occurs on leeward face, parallel side faces and

inclined leeward faces. Positive pressure decreases from wind ward edge to leeward edge. For Face E at all the wind incidence angles i.e. 0°, 15° and 30° this face experience suction. For this leeward face maximum suction of -0.681 occurs at 15° wind incidence angle. For 0° and 30° wind incidence angle suction is almost same at both the incidence angles.

Face F: For 0° wind incidence angle this face experience the same suction as face D faces. For other angles its value are different from face D. It can be observed from Table 1 that at all the wind incidence angles i.e. 0°, 15° and 30° this face experience suction. For this inclined leeward face maximum suction occurs at 0° and minimum suction occurs at 30° wind incidence angle.

Face G: It can be observed from Table 1 that at all the wind incidence angles i.e. 0°, 15° and 30° this face experience suction and suction value keep on decreasing with increase in

wind incidence angle. This face is sideward face at 0° wind incidence angle and it become inclined windward at 15° and 30° wind incidence angles. Minimum suction value i.e. - 0.672 occurs at 30° wind incidence angle.

Face H: At 0° wind incidence angle face H experience positive pressure but its value is much less than that at face A. Suction goes on increasing from windward side to leeward side. With increase in wind incidence angle this face becomes from inclined windward to windward face. Hence with increase in wind incidence angle (i.e. from 0° to 15°, 15° to 30°) pressure coefficient values for this face goes on increasing. Minimum and maximum mean C_p values for this face are 0.286 and 0.681 at 0° and 30° respectively.

Velocity vector field: Velocity vector field as obtained for octagonal tall building is shown in Fig 7. It shows vortices are formed behind face D

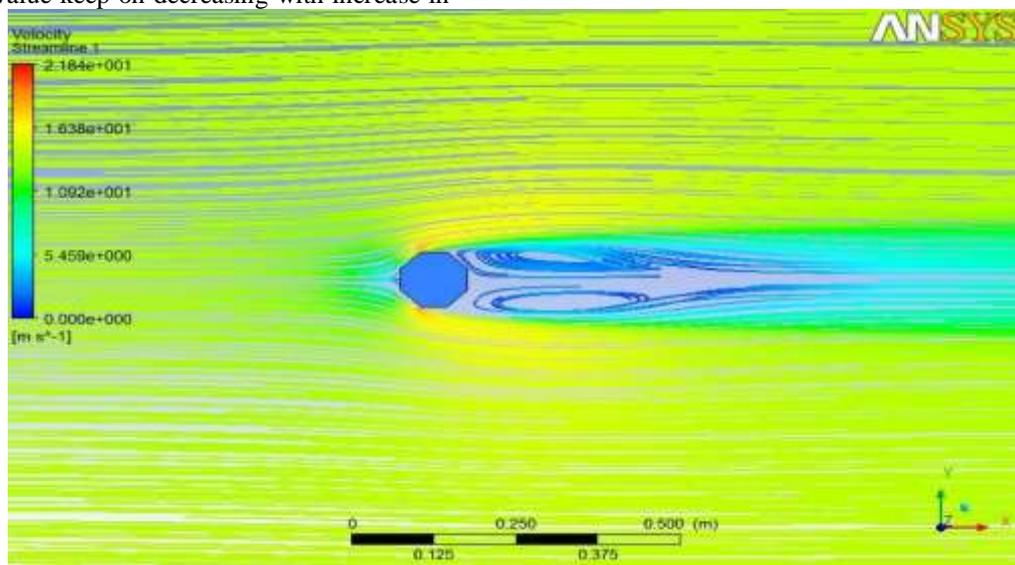


Fig. 7 Velocity vector field for octagonal building at 0° wind incidence angle

VI. CONCLUSION

From the study carried out the following broad conclusions are arrived. Computational Fluid Dynamics (CFD) offers to be a suitable alternative to predict the wind related phenomena on buildings and other different kind of structures. Visualisation of boundary layer separation can be observed accurately with the help of CFD simulation.

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.

The accuracy of results depends upon meshing of the geometry model and defining the physical property values exactly as the realistic environment conditions.

The study of boundary layer separation and wake formation can be studied easily with the help of CFD simulations and mitigation measures to counter the effects due to vortex shedding can be applied accordingly.

REFERENCES

- [1] ANSYS Ltd., 2010. Ansys Fluent solver, Release 14.0.0: Theory. Canonsburg.
- [2] Ansys tutorial manuals, Release 14.0.0.
- [3] ASCE 7-95 Standard, minimum design load for buildings and other structures, New York, 1995.
- [4] Australian standard AS: 1170 Part 2, 1975
- [5] B. Blocken, and J. Carmeliet, Pedestrian wind conditions at outdoor platforms in a high-rise apartment building :generic sub-configuration validation, wind comfort assessment and uncertainty issues. Wind and Structures. 11(1) , Pages 51-70. 2008.

- [6] B. Blocken, J. Carmeliet, , T. Stathopoulos, "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*, Volume 95, Issues 9–11, Pages 941-962. 2007a.
- [7] B. Blocken, J. Carmeliet, T. Stathopoulos, "CFD simulation of the atmospheric boundary layer: wall function problems", *Atmospheric Environment*, 41, 238–252. 2007b.
- [8] B.J. Vickery, "Wind loads on low rise buildings", drc, seminar, Darwin, Australia, 30 march, 1976.
- [9] B.L. Sharma, & U.N .Sehgal, Study of extreme winds over North India. *Indian Journal of met. and geophysics*. Vol. 19. 1968.
- [10] Code of practice for design loads (other than Earthquake forces) for buildings and structures IS 875-1987 part 3 wind load Bureau Indian Standard New Delhi.
- [11] E. Jack. Cermak., wind tunnel studies of building and structures. ASCE publications. 1998.
- [12] Explanatory Handbook On Indian Standard Code Of Practice For Design Loads (Other Than Earthquake) For Buildings And Structures Part 3 Wind Loads [Is 875 (Part 3): 1987]
- [13] H.C. Shellard, The estimation of design wind speeds. Proc. of conference on Wind Effects on Buildings & Structure, NPL, Middlesex. 1963.
- [14] Hideyuki Tanaka, Yukio Tamura, Kazuo Ohtake, Masayoshi Nakai, Yong Chul Kim. "Experimental investigation of aerodynamic forces and wind pressures acting on tall buildings with various unconventional configurations". *J. Wind Eng. Ind. Aerodyn.* 107–108 (2012) 179–191, 2012
- [15] <http://www.grc.nasa.gov/www/wind/valid/tutorial/spatconv.html>
- [16] Kim Young-Moon, You Ki-Pyo, Nag-HoKo. Across-wind responses of an aeroelastic tapered tall building. *Journal of Wind Engineering and Industrial Aerodynamics* 96 (2008) 1307–1319. 2008.
- [17] Kwok., "Effect of building shape on wind induced response of tall buildings," *Proceedings, 7th Int. Conf. on wind engineering*, Aachen, West Germany, 1987.
- [18] M. Sanayei, L. Edgers, J.L. Alonge, and P. Kirshen, , "Effects of increased wind loads on a tall building," *civil engineering practice*, vol. 19, no. 1, 2004.
- [19] P. Mendis, T. Ngo, N. Haritos, A. Hira "Wind loading on tall buildings." 2007.
- [20] P.J. Richards, R.P. Hoxey, Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. *Journal of Wind Engineering and Industrial Aerodynamics* 46&47, 145-153. 1993.
- [21] Peterka and Cermak, "Aerodynamics of bluff bodies", *Journal fluid mechanics*, vol.5, 1975.
- [22] Revuz ., On the domain size for the steady-state CFD modelling of a tall building. *Wind and structures*, Vol. 15, No. 4 (2012) 313-329. 2012
- [23] S.K. Dalui " Wind effects on tall buildings with peculiar shapes " , PhD. Thesis, Dept. of Civil Engineering, I.I.T. Roorkee, Roorkee, India. (2008).
- [24] S.K. Verma "Wind effects on structurally coupled buildings", PhD. Thesis, Dept. of Civil Engineering, I.I.T. Roorkee, Roorkee, India. (2009).
- [25] S.K. Verma, A.K. Ahuja and A.D. Pandey "Effects of wind incidence angle on wind pressure distribution on square plan tall buildings" *J. Acad. Indus. Res.* Vol. 1(12) May 2013 pp 747-752
- [26] T. Stathopoulos and B.A . Baskaran. "Computer simulation of wind environmental conditions around buildings", *Engineering Structures*, 18, pp. 876-885. 1996
- [27] T. Stathopoulos, "Computational wind engineering: Past achievements and future challenges", *Journal of Wind Engineering and Industrial Aerodynamics*, 67&68, pp. 509-532, 1997.