Chapter 17
Wind Load Analysis of a Tall Structure with Sharp and Corner Cut Edges

Abhipsita Rej and Amlan Kumar Bairagi

Abstract The massive wind load is always harmful to tall building structure, especially the structure which has sharp edges. The separation of wind near the sharp edge creates local effects around the neighbor faces of the building and also creates enormous damage on that region. The present study illuminated the pressure variation around the sharp edge setback and corner cut setback tall building models. The models are inducted inside the computational domain and simulated with different wind angles. One model has a sharp edge corners and other models has a corner cut at the bottom part of setback zone and rounder corner at top part of setback zone. The considerable amount of pressure, drag and lift variation has been detected for across- and along-wind consideration. The tremendous amount of pressure fluctuation observed at corner cut region. The local pressure recognized on the wall due to sharp edges for the sharp edge model. However, the local pressure is minimized by the use of rounded edges at the top portion of the setback. The drag and lift coefficient decrease with the decrease of sharpness of edges. The study tries to catch the minimum local pressure due to corner cut edges of the tall building model.

Keywords Sharp and corner cut edge · Set back tall building · Pressure variation

17.1 Introduction

Wind effects are the challenges that designers have to deal with in super-tall building design. In association with high slenderness, low natural frequencies, low inherent damping level and high wind speed at upper lever, super-tall buildings are more susceptible to wind. Wind pressures on building surfaces result in both steady and

A. Rej (✉)
Department of Civil Engineering, Techno India University, Salt Lake, Kolkata, India
e-mail: abhipsit07@gmail.com

A. K. Bairagi
Department of Civil Engineering, Indian Institute of Engineering Science and Technology, Shibpur, Howrah, India
e-mail: bairagiak@gmail.com

© Springer Nature Singapore Pte Ltd. 2020

M. Vinyas et al. (eds.), *Advances in Structures, Systems and Materials*,
Lecture Notes on Multidisciplinary Industrial Engineering,
https://doi.org/10.1007/978-981-15-3254-2_17

unsteady loading, while air motion over and around buildings transports heat, mass and momentum. In this study, the method of corners cutting and setback condition is effective for the reduction of wind-induced vibrations in an across-wind, direction for high-rise buildings with a regular square plane. Wind forces acting on high-rise buildings with corner cuts in along-wind direction were measured by wind tunnel tests. During this study, Kawai [1] gave the concept of the corner modifications promote the instability at low speed. Tanaka et al. [2] determined the aerodynamic forces and wind pressures acting on the square-plan tall building models with various configurations like corner cut, setbacks and helical. Elshaer et al. [3] examined the aerodynamic shape optimization (ASO) example is to reduce the drag force acting on a tall building by changing the shape of its corners. Roy and Bairagi [4] illustrated the pressure variation and velocity around the multiple shape setback tall building. Mooneghi and Kargarmoakhar [5] reviewed the past/recent work on various aerodynamic mitigation techniques developed for reducing wind loads on buildings by modifying their shapes and/or adding simple architectural elements. Bairagi and Dalui [6, 7] conferred the pressure fluctuation at top roof of setback model and also highlighted the power spectral density (PSD) at neighbor faces of the setback and top roof. Alminhana et al. [8] modified corner cuts, reduce significantly the aerodynamic forces on the building structures and improve flow conditions near the building locations. The current research is based on the analytical method by computational fluid dynamics (CFD) where an isolate setback tall building is set inside the domain to estimate the dynamic behavior of setback corner cut tall building due to wind excitation. Horizontal pressure coefficient, drag and lift coefficients are observed for along different wind directions.

17.2 Description of Meshing and Model

In this study, the domain has been created using the concept of Frank et al. [9] according to which the inlet and the lateral and top boundary of the domain should be apart for 5 times of the overall height (5H) from the furthest point of the model on that side. The outlet of the domain should be at a distance of 15 times the overall height (15H) from the furthest point of the model on that side. The domain details are shown (see Fig. 17.1a). The meshing has been done to calculate the solution while the run is ongoing, in such a manner that the mesh can automatically be refined in locations that depends on specified adoption criteria, where solution variables are hanging most rapidly so that it can resolve the flow features in these regions. To avoid unusual flows inside the domain, tetrahedron meshing has been used and it is inflated near the boundary. By the use of this mesh, the regions near the boundaries can be made so inflated that the flows with unusual characteristics can be avoided. The meshing details are shown (see Fig. 17.1b).

The analytical model has been simulated by computational fluid dynamics (CFD) method. Two setbacks tall building model has same length: breadth ratio (1:1) and length: height ratio (1:4) has been adopted in this study. The models are square-plan

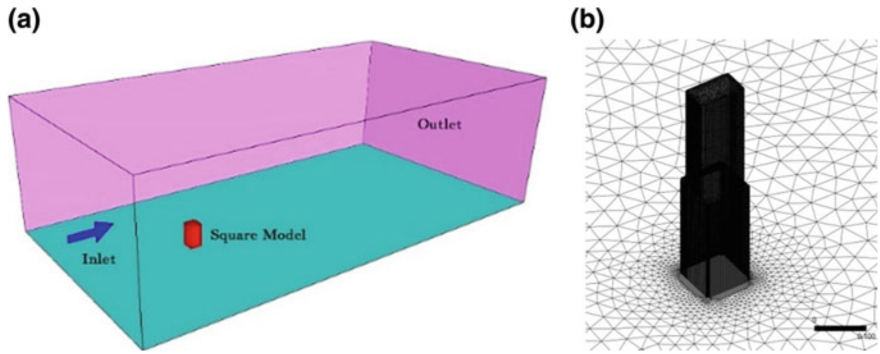


Fig. 17.1 **a** Model inside the domain and **b** meshing of corner cut model

shape and setback at half of the model height. The sharp edge model, namely M1 has setback level H/2 and setback distance 0.2L from both the opposite edges. Similar pattern model M2 has been considered but has a corner cut at the bottom part and the curve cut at the top part. The corner cut size has been demarcated by D1 and curved portion is R1. The attacking wind angle is 0° for along-wind direction and 90° for across-wind condition and an intermediate wind angle 45° also considered to observe the deviation of flow pattern around the model. The detail description of model M1 and M2 are shown (see Fig. 17.2a–b).

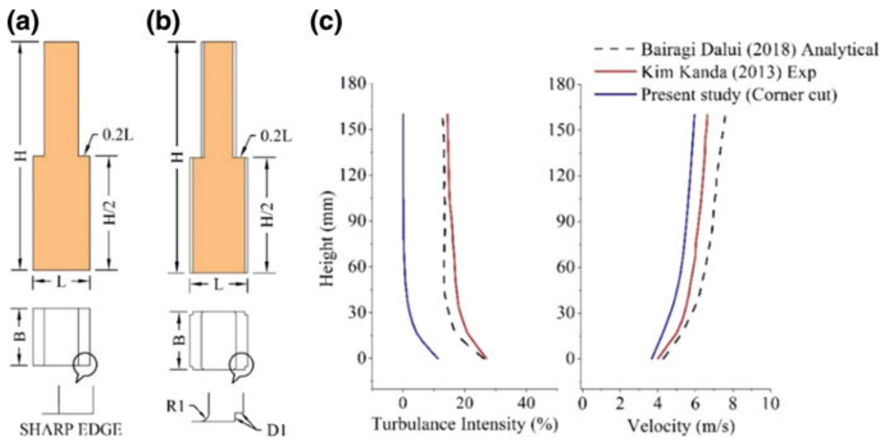


Fig. 17.2 Dimension of **a** model M1, **b** model M2 and **c** profile of velocity and turbulence inside the domain

17.3 Boundary Condition of the Domain

This study is conceived and analyzed by creating virtual domain and simulate the wind flow pattern around the model. The simulated flow conditions have been generated as per the criteria of terrain category two taken from IS 875 (part 3) [10]. The magnitude of inlet velocity has been considered normal to the boundary. Following the requirements of directional constraints, the flow direction was considered parallel to the normal of the surface of the boundary. The inlet and outlet are specified in the virtual domain. The sidewalls of this virtual domain are specified to be as free-slip wall in this study. The walls of both of the models are assumed no-slip walls with no surface roughness. In this study, power law has been used which is shown in Eq. (17.1), to calculate the velocity profile of the atmospheric boundary level in this CFD-based study.

$$V/V_z = (Z/Z_h)^{\alpha} \tag{17.1}$$

where V represents the speed of wind in horizontal direction at a certain elevation. V_Z represents the speed of the wind at a reference elevation Z_h. The value of V_z is taken as 10 m/s. The value of Z_h is taken as 1.0 m. α represents the coefficient of surface roughness 0.133 for terrain category 2.

17.4 Acceptance of Present Study

The numerical simulation used in the computational domain also validated with different researchers used by experimentally and analytically. The experimental study on sharp edge setback tall building was presented by Kim and Kanda [11] and conducted by Eiffel-type wind tunnel (1.8 m × 1.8 m × 12.5 m) at the University of Tokyo. The adopted factors are 1:400 scale, height of model 160 m, breadth of model 40 m and the power law exponent was 0.13, and the turbulence intensity was 15%. Wind speed at the tunnel was 6.5 m/s with 1.2% blockage ratio. Same size and same factors are considered and presented by Bairagi and Dalui [12, 13]. The present study also considered the same conditions for corner cut model and presented the profile of velocity and turbulence (see Fig. 17.2c).

17.5 Results

Two unconventional square setback tall building model, namely M1 and M2 have been simulated in a computational domain for 0°, 45° and 90° attacking wind. The pressure variation, drag and lift coefficients are observed and presented in this section.

17.5.1 Pressure Variation Around the Models

The pressure contours at different faces of both the models are conducted by using the formulas as stated in Eq. 17.2 and presented in Table 17.1.

$$C_p = C_{p(\mathrm{cal})}/\left(0.5\,\rho V_z^2\right) \tag{17.2}$$

where C_p is the pressure coefficient, $C_{p(\mathrm{cal})}$ is the pressure coefficient from the analytical data, ρ is the density of fluid at 25 °C and V_z is the velocity of wind at the particulate height. For along-wind condition, the windward face has positive pressure for both the models and suction at the sidewalls and leeward face. An interesting point comes out from the local pressure zone. The local pressure zone observed in the wall and to roof of setback part of model M1, but model M2 has minimum amount of wall and negligible amount of top roof. Therefore, it may be said that the local pressure depends upon the sharpness of the edge. The top setback part of the M1 model has a sharp edge, but the model M2 has curved edges. According to this condition, the flow pattern around the sharp edge of the model M1 has high amount of turbulence and create tremendous suction. On the other hand, the model M2 has curved edges at the top setback part, so the wind passed smoothly on that particular edge.

To understand the proper pressure fluctuation, horizontal pressure variation carried out in this study. A pressure belt considered at height 0.475H from the base of both the model M1 and M2. The initial pressure tapping point considered at the corner of face *A* and assumed a pressure belt around the model. The pressure variation of different azimuth is presented in (see Fig. 17.3). For 0° wind angle, the initial point at the corner of face A has 3.82% difference of suction with respect to M1 model. The high amount of suction detected at the corner cut region of M2 model as shown in Fig. 17.3a. In this zone, the pressure difference is 163% less than the M1 model. According to this analysis, it is clear that the large amount of suction developed at the corner cut region. At the same time, for 45° wind has less pressure fluctuation at the corner cut zone compare with M1 and M2 model. The similar amount of pressure difference observed for 90° wind angle.

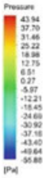
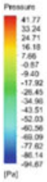
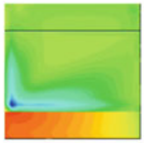
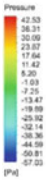
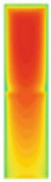
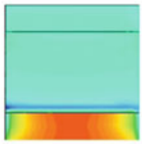
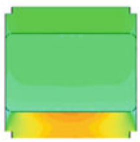
17.5.2 Drag and Lift Coefficient

The drag and lift coefficient for both the models are calculated by using the equation as stated in Eqs. 17.3–17.4

$$F_d = 0.5\,\rho v_z^2\,C_d A \tag{17.3}$$

$$F_l = 0.5\,\rho v_z^2\,C_l A \tag{17.4}$$

Table 17.1 Comparison of pressure coefficient between M1 and M2 model for 0°, 45° and 90° wind angles

Wind angle	Model	Legend	Face A	Face B	Face C	Face D	Roof
0°	M1						
	M2						
45°	M1						
	M2						
90°	M1						
	M2						

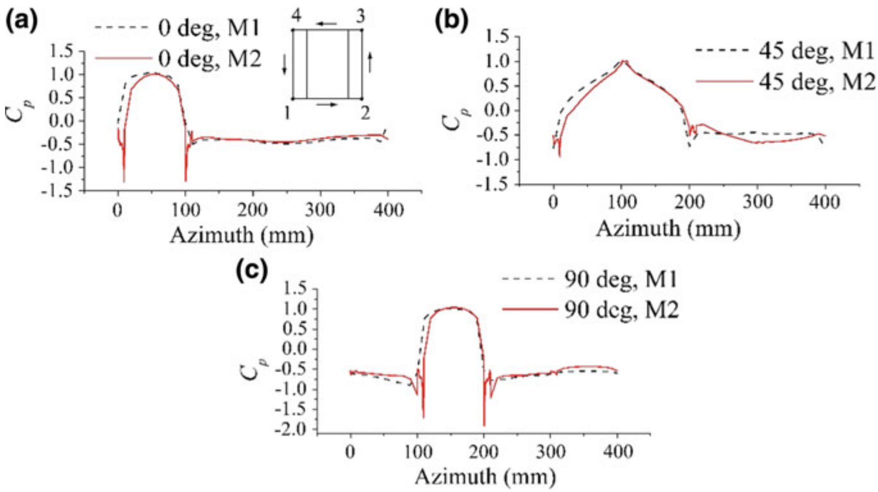


Fig. 17.3 Azimuth of pressure coefficient at 0.475H for **a** 0° wind, **b** 45° wind and **c** 90° wind angle

where F_d and F_l are drag and lift force, A is the attacking wind area of the model, C_d and C_l are the drag and lift coefficient. The comparative study of drag and lift force coefficient between M1 and M2 model showcased under (see Fig. 17.4). The maximum drag coefficient for model M1 is 1.06 for 0° wind and gradually decreasing with the increase of the wind angle. For 45° and 90° wind angle has 0.56 and −0.03, respectively, for M1 model as shown in (see Fig. 17.4). On the other hand, model M2 has 0.34, 0.23 and zero for the wind angle 0°, 45° and 90°. The lift coefficient values also vice versa, and the values also increasing with the increase of the wind angle. For 0°, the wind has minimum value and 90° has maximum value 1.17 for the M1 model as shown in (see Fig. 17.4). However, the M2 has a negative force coefficient for both the 0° and 45° wind angles. From the graphical representation of the drag

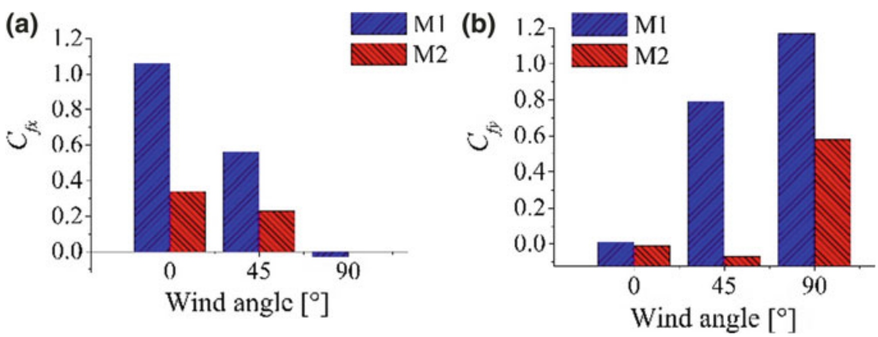


Fig. 17.4 **a** Drag and **b** lift coefficient on M1 and M2 model for 0°, 45° and 90° wind angles

and lift coefficient, it may be said that the corner cut model experienced less drag and lift forces compared with the sharp edge model.

17.6 Conclusion

The numerical simulation conducted in a computational domain to determine the pressure coefficient of different faces also drag and lift coefficients for different angles. After a number of iterations, the following conclusions are made.

As both the models are setback, so the top part of setback model has a different local pressure zone. The sharp edge model has high suction at this local pressure zone, but the corner cut model has curved zone at the top part of setback section, therefore, the wind passes smoothly at that corner. Due to this, the local pressure zone also decreases.

- High amount of pressure difference (163%) observed at the corner cut region at 0.475H distance from the base of the model. The pressure drop is due to the corner cut of the model.
- The drag coefficients are gradually changed from 1.06 to −0.03 and lift coefficient from 0.01 to 1.17 for sharp edge corner model. However, for corner cut model does not satisfy this rule.
- The drag and lift coefficients create a huge amount of variation on corner cut model compare to the sharp edge model.

References

1. Kawai, H.: Effect of corner modifications on aeroelastic instabilities of tall buildings. J. Wind Eng. Ind. Aerodyn. **74–76**, 719–729 (1998)
2. Tanaka, H., Tamura, Y., Ohtake, K., Nakai, M., Kim, Y.C.: Experimental investigation of aerodynamic forces and wind pressures acting on tall buildings with various unconventional configurations. J. Wind Eng. Ind. Aerodyn. **107–108**, 171–191 (2012)
3. Elshaer, A., Bitsuamlak, G., Damatty, A.E.: Aerodynamic shape optimization for corners of tall buildings using CFD. In: 14th International Conference on Wind Engineering, Porto Alegre, Brazil (2015)
4. Roy, K., Bairagi, A.K.: Wind pressure and velocity around stepped unsymmetrical plan shape tall building using CFD simulation—A case study. Asian J. Civ. Eng. (BHRC) **17**(8), 1055–1075 (2016)
5. Mooneghi, M.A., Kargarmoakhar, R.: Aerodynamic mitigation and shape optimization of buildings: Review. J. Building Eng. (2016)
6. Bairagi, A.K., Dalui, S.K.: Comparison of aerodynamic coefficients of setback tall buildings due to wind load. Asian J. Civ. Eng. Building Hous. **19**(2), 205–221 (2018)
7. Bairagi, A.K., Dalui, S.K.: Aerodynamic effects on setback tall building using CFD simulation. In: 2nd International Conference on Advances in Dynamics, Vibration and Control (ICADVC-2018). pp. 381–388. NIT Durgapur (2018)

8. Alminhana, G.W., Braun, A.L., Loredo-Souza, A.M.: A numerical study on the aerodynamic performance of building cross sections using corner modifications. Lat. Am. J. Solids Struct. **15**(7), e88 (2018)
9. Franke, J., Hirsch, C., Jensen, A., Krüs, H., Schatzmann, M., Westbury, P., Miles, S., Wisse, J., Wright, N. G.: Recommendation on the use of CFD in wind engineering. COST Action C14. Impact of Wind and Storm on City Life and Built Environment. Von Karman Institute for Fluid Dynamics (2004)
10. IS: 875 (Part-3), Indian standard code of practice for the design loads (other than earthquake) for buildings and structures (part-3. wind loads). Bureau of Indian Standards, New Delhi, India (2015)
11. Kim, Y.C., Kanda, J.: Wind pressures on tapered and set-back tall buildings. J. Fluids Struct. **39**, 306–321 (2013)
12. Bairagi, A.K., Dalui, S.K.: Aerodynamic effects on setback tall building using CFD simulation. Int. J. Mech. Prod. Eng. Res. Dev. 413–420 (2018)
13. Bairagi, A.K., Dalui, S.K.: Comparison of pressure coefficient between square and setback tall building due to wind load. In: SEC18: Proceedings of the 11th Structural Engineering Convention—2018, Jadavpur University, Kolkata, India (2018)