Research Article
Wind Excited Action around Tall Building Having Different Corner Configurations

Rahul Kumar Meena, Ritu Raj, and S. Anbukumar
Department of Civil Engineering, Delhi Technological University, Delhi, India
Correspondence should be addressed to Ritu Raj; rituraj@dtu.ac.in
Received 18 September 2021; Revised 25 January 2022; Accepted 29 January 2022; Published 25 February 2022

Aeroelastic instabilities are common in square or rectangular plan shape structures due to the development of powerful vortices by the rolling motion of the separated shear layers. Windward corner modifications such as corner cut, recession, rounded, and slotted help to reduce instabilities. Codal recommendations about common shapes like square, rectangle, triangle, and circular shapes are available in different international standards, but they did not provide the detailed analysis results for the different regular shapes with corner configurations. Although analysis was performed on various plan shapes of tall buildings using the computational fluid dynamics technique through k-ε turbulence model, a very small number of studies were performed on particular shapes with different corners having same plan area and height. The mean pressure coefficients of Model-A and Model-B were compared with various international standards and wind tunnel data, respectively, for validation, showing a nearly equivalent consistency; however, international standards consist of coefficients at 0° and 90° wind angles only. Wind effects on building shapes having different corners change the characteristics of the separated shear layer and reduce the downstream wake which helps to reduce drag and lift forces simultaneously. Recent study shows that the windward pressure distribution pattern is almost independent of building size and height. Therefore incorporation of corners in building helps to reduce the forces caused by extreme wind. A very large amount of numerical simulation data about wind pressure is generated, which can be used by the designer while designing such a building for wind load. Comparison have been made between buildings having different corners under same wind speed, and Model-D (round corner) performed very well against the wind.

1. Introduction
Because of the ever-increasing population, cities’ growth in the horizontal direction has reached a saturation point. As a result, tall structures are springing up all over the planet. Tall buildings are particularly vulnerable to wind loads, necessitating effective wind load design. Irwin [1] studied the bluff body aerodynamics in wind engineering. Bluff bodies subjected to strong large vortices in their wakes, wind turbulence, are another factor that has a significant impact on vortex shedding. The rectangular building is very common in plan shape, and it is generally symmetrical about both axes. This plan-shaped building is very common for residential as well as corporate buildings [2–5]. Effects of wind load for common plan shape are available in various international standards [6–10]. Wind tunnel testing and CFD technologies are used to study the impact of wind on tall buildings. Wind tunnel studies and/or CFD simulations are required to get wind driven forces in complex plan shaped structures.

Wind effects on structure are varying as per the shape of building; to evaluate the effect of wind on such structures, wind tunnel testing has been performed by many researchers; for example, Sheng et al. [11] investigated the fluctuating properties of global and local wind loads; Sun et al. [12] conducted wind tunnel test on 1040-meter building model; Irwin et al. [13] examined the effect of corners; Raj and Ahuja [14] performed an experiment on rigid models with varied cross-sectional shapes and equal
Numerical study was performed using computational fluid dynamics. Most of the studies in the past were done using the k-\(\varepsilon\) model. This model has well-established prediction capability and has been proven to be stable and numerically reliable. This model gives an accurate balance of reliability for general-purpose simulations. It is very less expensive and is mostly used to simulate the turbulent flow characteristics. k-\(\varepsilon\) turbulence is a two-equation model and provides the solution by using two transport equations, that is, turbulent kinetic energy (k) and turbulent dissipation rate (\(\varepsilon\)). It has positive advantage of not including any geometry-related parameters in the modelling. The turbulent kinetic energy and the turbulence dissipation rate are two variables introduced into the system of equations for the model.

(i) Basic equations. The basic equations used to study the fluid flow problems are Navier-Stokes and continuity equations.

(ii) Navier-Stokes equations as follows:

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + F_i.
\]  

(iii) continuity equation is as follows:

\[
\frac{\partial p}{\partial t} + \frac{\partial f}{\partial x_j} = 0.
\]  

The standard k-\(\varepsilon\) model uses the following transport equations for the turbulence kinetic energy and turbulence dissipation:

\[
\rho \frac{\partial k}{\partial t} + \rho u_i \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu}{\sigma_k} \right) \frac{\partial u_i}{\partial x_j} - \rho \varepsilon + \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j}.
\]  

Momentum equation is as follows:

\[
\frac{\partial (\rho U_i)}{\partial t} = - \frac{\partial (\rho U_i U_j)}{\partial x_j} - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_{\text{eff}} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + S_{Mi}.
\]  

2.1. Building Model. In the present study, the building is modelled at a 1:200 length scale. CFD simulations were used to explore the effect of wind on the modified corners on rectangular, corner cut, chamfered, and fillet shapes of almost identical plan area and equal height of 750 mm. The variation of pressure coefficient on building surface around the whole building models is discussed in this study, for the building models shown in Figure 1. Slight difference in the
proportions for all the building models is kept to maintain the equal plan area. Four building models are used to understand the wind-induced responses on various building faces with varying skew angles of wind stream varying from 0° to 90° at 15° intervals.

2.2. Boundary Condition. The effect of wind on building models is studied using CFD simulation. CFD works by dividing a space into a grid containing a large number of cells. The grid of the cells surrounded by boundaries that simulate the surfaces, opened and enclosed space, pressure of the boundaries, and air movements within the cell is then set to a starting condition, which is closed to the real environment of the wind tunnel experiment. The inlet, top, and sidewall boundaries are considered 5 H from the model and outlet boundary is placed at 15 H behind the model for simulation to develop the flow properly as recommended by Frank et al. [41] and Revuz et al. [42]. To define a problem that results in a unique solution, it is required to specify the information on the dependent variables at the domain boundaries, as the governing equation is differential, and their solution requires integration. The boundary condition is the mathematical equivalent of the constant of integration, the value of which is required to gain a unique solution. The domain is represented in Figure 2, and the sidewall and top are kept as free slip in the CFX-Pre setup. Ground and building model surfaces in this study are considered as no-slip wall in CFX-Pre setup. No-slip is defined as follows: the velocity of the air at wall boundary is the same as the air velocity at inlet. Free slip is defined as follows: velocity components parallel to wall have finite value (computed by the solver), but the velocities normal to the wall and shear stress are both set to zero.

At the inlet, the free stream velocity is set to 10 m/s. The domain is designed according to Revuz et al. [42] (Figure 3). The velocity profile and turbulence intensity profile are graphically depicted in Figure 4 and compared with previous wind tunnel data by Raj (2015).

2.3. Meshing. Meshing is the most important part of numerical simulation. Finer mesh might not give you reliable results, but a bad mesh will surely affect the results. Meshing is of different sizes and provides capturing the flow characteristics in critical zones. Meshing divides the complex geometry into elements that can be used to discretize a domain since meshing typically consumes time. Meshing is represented in Figure 3, domain meshing is provided as tetrahedron meshing as it consumes fewer resources in computing the result, and building meshing is of different sizes so that solution can capture a clear picture of flow behaviour near the building surfaces. The edge meshing is also provided and is finer than the other meshing sizes to capture the flow characteristics. Inflation is provided to simulate the flow easily around the building model’s geometry.

Figure 1: Plan and isometric view of (a) rectangular, (b) corner cut, (c) chamfered, and (d) fillet shapes of tall building models.
2.4. Velocity and Turbulent Intensity Profile. The boundary condition used in the numerical simulation is similar to the boundary condition used by Raj [43]. To obtain a good relationship between numerical simulation and wind tunnel experiment, the boundary condition should be the same as those used in experiments, especially for inflow boundary conditions.

The variation in the wind with height is generally expressed in logarithmic form:

\[ U_z = \frac{u_0}{k} \ln \left( \frac{Z}{Z_o} \right), \]  

where \( U_z \) is the wind speed at height \( z \), \( u_0 \) is friction velocity, \( Z_o \) is the characteristic height of surface roughness, and \( k \) is a Karman constant with a value of 0.4.

The boundary layer wind profile is governed by the power low equation:

\[ U_z = U_o \left( \frac{z}{Z_o} \right)^n, \]  

where \( U_z \) is the wind speed at height \( Z \), \( U_o \) is the mean wind speed at a reference height \( Z_o \), and "n" is a parameter that varies with ground roughness.

Generally, turbulence intensity for the smooth terrain is less than the turbulence intensity for rough terrain. Turbulence intensity is a nondimensional quantity derived from the variance and for the mean wind speed.

\[ I_z = \frac{\sigma_z}{U_z} \]  

where \( I_z \) is the turbulence intensity at height \( z \), \( \sigma_z \) is the standard deviation of the wind speed at height \( z \), and \( U_z \) is the mean wind speed at reference height.

2.5. Validation. CFD results are based upon a number of factors. Therefore, it is necessary to validate the accuracy of the numerical results. For this purpose, building the various numerical results of Model-A (rectangular) and Model-B (corner cut) is compared with various international standards and experimental results.

The external pressure coefficients, \( C_p \) (face average value), for the different faces of Model-A are tabulated in Table 1 and compared between numerical results and various international standards. Results in Figure 5 and Table 1 are obtained using the \( k-\varepsilon \) turbulence models, and results are nearly the same with various international standards and experimental results.

The external pressure coefficient "\( C_p \)" is calculated using the following equation:
\[ C_p = \frac{p - p_o}{1/2 \rho U_Z^2} \]  

where \( p \) is the pressure derived from the needed point, \( p_o \) is the reference height static pressure, \( \rho \) is the air density \((1.225 \text{ kg/m}^3)\), and \( U_Z \) is mean wind velocity at the building reference height.

3. Results and Discussions

3.1. Flow Pattern. The wind pressure distribution against a wall of the high-rise building is affected by various wind characteristics such as vortex, separation, stagnation, recirculation, and reattachments. Wind flow for Model-A is presented in Figure 6, while streamlines for rectangular...
model having corner cut are represented in Figure 7. The streamlines in plan for Model-C are depicted in Figure 8 and for Model-D wind flow patterns in plan are illustrated in Figure 9. The size of downstream wake depends on the wind incidence angle and is maximum for Model-A at 30°.

Pressure distribution and vortex generation are based on the geometric configuration of the building. Suction is observed in the rear side of the building with respect to the wind flow direction. Velocity at the edge is increased due to the separation of flow. The reattachment of flow is observed more on side faces at 0° and 15° wind. Streamlines patterns are unaffected by wind speed and are mostly determined by the geometry of the building and upwind conditions.

3.2. Pressure Distribution. The pressure distribution is expressed in the form of the pressure contours for various building shapes in Figures 10–13. For the case of building Model-A (Figure 10), maximum pressure on the centre of the face is 1.05, on windward face-B, at 90°, and the minimum (−0.7) is on face-B at 0°. Pressure distribution is maximum on the centre of the face, and then it is decreasing towards the edges of the faces because of the increase in the flow velocity. Contour lines become dense towards the edges of the face, indicating the changes in the pressure distribution. Wind pressure follows the same pattern for face-A at 0° and face-B at 90° because wind flow is in direct contact with both faces. For side face-B, at 0°, suction is increasing as wind flow passes the building model from windward to leeward edge.

Figure 11 represents the wind pressure distribution for building Model-B; maximum positive pressure is 1.05 on face-D at 90°, and maximum negative pressure is 0.8 on face-J at 0°. Pressure distribution on face-A at 0° and face-D at 90° is the windward face for the respective case and follows the same pressure distribution pattern. Pressure distribution on the windward face is almost independent of the width and depth of the face.

| International code | Wind angle | Windward side | Sidewall | Leeward side |
|--------------------|------------|---------------|----------|-------------|
| CFD results        | 0°         | 0.78          | −0.64    | −0.30        |
|                    | 90°        | 0.70          | −0.61    | −0.45        |
| IS 875 (part 3)    | 0°         | 0.7           | −0.7     | −0.4         |
|                    | 90°        | 0.8           | −0.5     | −0.1         |
| ASCE/SEI 7-16      | 0°         | 0.8           | −0.7     | −0.5         |
|                    | 90°        | 0.8           | −0.7     | −0.5         |
| AS/NZS 1170.2.2011 | 0°         | 0.8           | −0.65    | −0.5         |
|                    | 90°        | 0.8           | −0.65    | −0.5         |
| EN 1991-1-4        | 0°         | 0.8           | −0.5     | −0.7         |
|                    | 90°        | 0.8           | −0.5     | −0.7         |
| BS 6399-2          | 0°         | 0.8           | −0.5     | −0.7         |
|                    | 90°        | 0.8           | −0.5     | −0.7         |
| GB 50009-2001      | 0°         | 0.8           | −0.5     | −0.7         |
|                    | 90°        | 0.8           | −0.5     | −0.7         |
| NSCP 2015          | 0°         | 0.8           | −0.5     | −0.7         |
|                    | 90°        | 0.8           | −0.5     | −0.7         |
| ES/ISO 4354: 2012  | 0°         | 0.8           | −0.65    | −0.7         |
|                    | 90°        | 0.8           | −0.65    | −0.7         |

Figure 6: Flow lines for building Model-A at various wind incidence angles. (a) 0° wind. (b) 15° wind. (c) 30° wind. (d) 45° wind. (e) 60° wind. (f) 75° wind. (g) 90° wind. (h) Model-A.
Figure 7: Flow lines for building Model-B at various wind incidence angles. (a) 0° wind. (b) 15° wind. (c) 30° wind. (d) 45° wind. (e) 60° wind. (f) 75° wind. (g) 90° wind. (h) Model-B.

Figure 8: Flow lines for building Model-C at various wind incidence angles. (a) 0° wind. (b) 15° wind. (c) 30° wind. (d) 45° wind. (e) 60° wind. (f) 75° wind. (g) 90° wind. (h) Model-C.

Figure 9: Flow lines for building Model-D at various wind incidence angles. (a) 0° wind. (b) 15° wind. (c) 30° wind. (d) 45° wind. (e) 60° wind. (f) 75° wind. (g) 90° wind. (h) Model-D.
Wind pressure distribution for building Model-C (Figure 12) at 90° on face-C (windward) has maximum pressure (1.07) at the centre of the face and minimum pressure (−2.9) on face-B at 90°. In Figure 10, the maximum pressure is 1.05 on face-C (windward) at 90°, and the minimum pressure is −0.8 on face-G (side face) at 0° wind and it decreases in negative from windward face to leeward face. Pressure generally increases with height because of increasing velocity in the approach flow as wind speed increases with height.
Figure 11: Distribution of mean wind pressure coefficient on different faces of Model-B.

Figure 12: Distribution of mean wind pressure coefficient on different faces of Model-C.
3.3. Vertical $C_p$

Variation in $C_p$ along the vertical centreline for 0° to 90° wind incidence angles at an interval of 15° of each face is in Figure 14 for rectangular building Model-A. In Figure 14, face-C and face-D are subjected to negative pressure for all wind incidence angles. Face-A and face-B are subjected to maximum positive pressure at 0° and 90° wind incidence angles, respectively. Negative pressure zone is observed on face-A at 60°, 75°, and 90° wind incidence angles. Face-B has been subjected to negative pressure at 0°, 15°, and 30° wind incidence angles. In Figure 14, the vertical centreline pressure distribution on face-A is decreasing continuously from 0° to 90°, is maximum at 0° wind, and is minimum at 90° for Model-A.

For plus-shaped building Model-B, the distribution of pressure coefficients on the vertical centreline is represented in Figure 15 for 0° to 90° wind incidence angles at an interval of 15°. In Figure 15, face-E, face-F, face-G, face-H, face-I, and face-J are subjected to negative pressure on all wind incidence angles. Face-B and face-C are subjected to negative pressure at 90° wind incidence angles and subjected to...
positive pressure at 0° to 75° wind incidence angles. Face-D is subjected to negative pressure at 0° and 15° wind incidence angles. At the rest of the angles, this is subjected to positive pressure. Face-K and face-L are under the effect of positive pressure at 0° wind incidence angle for the rest of the angle, and they are subjected to negative pressure.

Model-C vertical centreline pressure distribution is presented in Figure 16; for face-A, maximum $C_p$ is at 0° wind and minimum $C_p$ is at 75° wind incidence angle. Face-D, face-E, face-F, face-G, and face-H are under the effect of negative pressure, that is, suction for 0° to 90° wind incidence angles at an interval of 15° each. Face-E and face-F have a maximum magnitude of pressure at 0° wind and minimum magnitude of pressure at 75° because of recirculation of flow. Minimum $C_p$ is at 0° wind incidence angle because flow separation takes place at these recessed corners.

Figure 15: Pressure variation along the vertical centreline for all faces of Model-B at various wind incidence angles.
For recesses/chamfer Model-D, vertical centreline pressure distribution pattern is shown in Figure 17, face-A, under the direct exposure to wind at 0° wind, so the maximum vertical centreline pressure is at 0° wind and the maximum on face-C is at 90° wind incidence angle. Face-A is under the reattachment of flow from 45° to 90° wind incidence angle. Face-D, face-E, face-F, face-G, and face-H are under the effect of negative pressure, that is, suction for 0° to 90° wind incidence angles at an interval of 15° each. Recessed corner face-B has maximum vertical centreline pressure at 0° and minimum at 90° wind incidence angle, as well as recessed corner face-H.

3.4. Drag and Lift Force. Wind load has long been a subject of research for wind engineers. Wind-produced responses on structures are difficult to quantify due to the complexity of structural geometry and field category. In modern design practice, wind loads can be calculated using wind tunnel tests and CFD technique. In this study, force coefficients in X-direction ($C_{fx}$) and Y-direction ($C_{fy}$) are obtained using equations (9) and (10). Therefore, values of $F_x$ and $F_y$ are obtained using ANSYS simulation.

Figure 18 depicts the variation of force coefficient in X-direction ($C_{fx}$) and Y-direction ($C_{fy}$) with respect to varying wind incidence angles. Maximum $C_{fx}$ and $C_{fy}$ are at 45° wind incidence angle for Model-C and Model-D.

$$C_{fx} = \frac{F_x}{(0.5 \rho U_h^2 A_p)}$$

$$C_{fy} = \frac{F_y}{(0.5 \rho U_h^2 A_p)}$$

Figure 19 shows the variation of moment coefficient in X-direction ($C_{mx}$) and Y-direction ($C_{my}$) with respect to varying wind incidence angles. $C_{mx}$ is maximum for Model-C and Model-D at 30° wind incidence angle. $C_{my}$ is maximum at 45° wind incidence angle for Model-C and Model-D.

$$C_{mx} = \frac{M_x}{(0.5 \rho U_h^2 A_p 1/2 H)}$$

$$C_{my} = \frac{M_y}{(0.5 \rho U_h^2 A_p 1/2 H)}$$

The moment coefficient in the X-direction and Y-direction is obtained using equations (11) and (12); here, $\rho$ is the density of air and $U_h$ is the reference velocity at the building height, $A_p$ is the area projected in the direction of...
the wind, and \( H \) is the height of the building model. \( C_{mx} \) is the moment coefficient of the building model in the \( X \)-direction, and \( C_{my} \) is the moment coefficient of the building model in the \( Y \)-direction.

3.5. Face Average \( C_{p,mean} \) Coefficient of pressure is a non-dimensional ratio of wind-induced pressure on a building to the velocity pressure of the wind speed at the reference height. Pressure coefficient depends on the shape of the
Figure 19: Variation of moment coefficient at X-axis and Y-axis at various wind incidence angles.

Figure 20: Continued.
building wind incidence angle and the profile of the wind velocity. Figure 20 presents mean pressure coefficients for all building models at various wind incidence angles of 0° to 90° at an interval of 15°. Model-A, face-A, has maximum positive mean pressure of 0.78 and then it starts to decrease till 45° wind to 0.35. Model-A is attacked by various wind skew angles, that is, beyond 45°, \( C_{p,\text{mean}} \) changes on face-A from positive to negative. Model-A, face-B, has negative \( C_{p,\text{mean}} \) at 0° and 15° wind attack; after 15° it increases with respect to wind incidence angles, that is, changing with respect to wind incidence angle. In case of face-C, \( C_{p,\text{mean}} \) is maximum for 0° wind attack and maximum for 45° wind. For face-C, \( C_{p,\text{mean}} \) is increasing in negative with wind incidence angle. For face-D, maximum \( C_{p,\text{mean}} \) is at 0° wind attack and minimum \( C_{p,\text{mean}} \) is at 90° wind attack. \( C_{p,\text{mean}} \) of Model-B, Model-C, and Model-D are represented in Figure 20.

4. Conclusions

This paper presents the wind loads effect on the various plan-shaped tall buildings by varying the corner shapes. Numerical simulation is performed using ANSYS CFX, \( k-\varepsilon \) turbulence model at 1:200 length scale. Face average \( C_p \) is compared and found in a range with various international standards and experimental studies for modelling in the numerical simulation. A clear view of streamlines has been presented at different wind incidence angles. The notable outcomes of the current study are summarized as follows:

(i) The validation study demonstrated very prominent results in line with various international standards and wind tunnel results; however, several upgradations are required in codal provision as the wind data is not updated with time.

(ii) Model-A without any corner configuration has maximum \( C_p \) at face-A for 0° wind incidence angle; after incorporation of the various corners in building model, it has been observed that Model-B is subjected to maximum windward pressure at 0°, that is, 20% greater wind impact than Model-A, and Model-C and Model-D have chamfer and fillet corners, respectively; however, less effect of wind on windward side is observed in case of Model-D. Furthermore, it is clear that incorporating the round corner helps to reduce the effect of wind flow on building.
(iii) In case of 15° wind incidence angle, windward face-A experiences similar wind response for Model-B and Model-C; however, Model-C with cross corner helps to reduce overall wind forces on building as compared to Model-B having cut corner configuration.

(iv) Drag force coefficients are compared between all the buildings models having various corner configurations; it was found that Model-C and Model-D experienced the same drag forces with little variation. Minimum drag forces in X-axis were experienced by Model-A at 75° wind incidence angle.

(v) Although lift force coefficient is calculated for all models and it has been observed that maximum lift force is measured on Model-C at 45° wind incidence angle, it was also concluded that the building having corner configuration has experienced greater lift force as compared to the building without corner configurations. Lift and drag forces are majorly affected by providing the corners on buildings.

(vi) Moment of X and Y axes is plotted with respect to drag and lift forces and found identical to base forces plotted against X and Y axes, respectively. It can be useful while calculating the overall response of buildings having different corner configuration. Building with normal rectangular shape experienced lowest CMY as compared to other building models having different corners.

Data Availability

All data, models, and code generated or used during the study appear in the submitted article. Data are available in the form of contour plots and graphs.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

Acknowledgments

The authors would like to express their sincere gratitude to Delhi Technological University, Delhi, India, for providing research facilities and funding to conduct the research work.

References

[1] P. A. Irwin, “Bluff body aerodynamics in wind engineering,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 96, no. 6-7, pp. 701–712, 2008.
[2] G. Solari, Wind Actions and Effects on Structures, Springer, New York, NY, USA, 2019.
[3] E. Simiu and D. Yeo, Wind Effects on Structures, John Wiley & Sons, Hoboken, NJ, USA, 4TH edition, 2019.
[4] B. S. Taranath, Wind and Earthquake Resistant Buildings, Routledge, England, UK, 2004.
[5] C. C. B. Ted Stathopoulos, Wind Effects on Buildings and Design of Wind-Sensitive Structures, Springer, New York, NY, USA.
[6] IS-875 (Part-3), Indian Standard Design Load (Other Than Earthquake) for Building and Structures-Code of Practice, BIS, Old Delhi, India, 2015.
[7] AS/Nzs11702, Structural Design Actions - Part 2, Wind actions. Standards Australia/Standards New Zealand, Sydney, Australia, 2011.
[8] Asce716, Minimum Design Loads and Associated Criteria for Buildings and Other Structures, Structural Engineering Institute of the American Society of Civil Engineering, Reston, 2017.
[9] GB 50009-2001, National Standard of the Peoples’ Republic of China, Ministry of Construction (MOC) and the General Administration of Quality Supervision, Inspection and Quarantine (GAQSIQ) of the People’s Republic of China, Beijing, China, 2002.
[10] H. Kong Building Department, Code of Practice on Wind Effects in Hong Kong 2019, Building Department Hong Kong, Hong Kong First Issue, 2019.
[11] R. Sheng, L. Perret, I. Calmet, F. Demouge, and J. Guilhot, “Wind tunnel study of wind effects on a high-rise building at a scale of 1:300,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 174, pp. 391–403, 2018.
[12] X. Sun, H. Liu, N. Su, and Y. Wu, “Investigation on wind tunnel tests of the Kilometer skyscraper,” Engineering Structures, vol. 148, pp. 340–356, 2017.
[13] P. Irwin, J. Kilpatrick, and A. Fiske, “Friend or foe, wind at height,” CIBWU 2008, 8th World Congr.- Tall Green Typology a Sustain. Urban Futur. Congr. Proc., pp. 336–342, 2008.
[14] R. Raj and A. K. Ahuja, “Wind loads on cross shape tall buildings,” Journal of Academia Industrial Resear, vol. 2, no. 2, pp. 111–113, 2013.
[15] B. Bhattacharyya and S. K. Dalui, “Experimental and numerical study of wind-pressure distribution on irregular-plan-shaped building,” Journal of Structural Engineering, vol. 146, no. 7, Article ID 04020137, 2020.
[16] S. K. Nagar, R. Raj, and N. Dev, “Experimental study of wind-induced pressures on tall buildings of different shapes,” Wind Struct. An Int. J., vol. 31, no. 5, pp. 441–453, 2020.
[17] E. K. Bandi, Y. Tamura, A. Yoshida, Y. Chul Kim, and Q. Yang, “Experimental investigation on aerodynamic characteristics of various triangular-section high-rise buildings,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 122, pp. 60–68, 2013.
[18] R. Raj, S. Jha, S. Singh, and S. Choudhary, “Response analysis of plus shaped tall building with different bracing systems under wind load,” International Journal of Advanced Research in Engineering & Technology, vol. 11, no. 3, pp. 371–380, 2020.
[19] L. Carassale, A. Freda, and M. Marré-Brunenghi, “Experimental investigation on the aerodynamic behavior of square cylinders with rounded corners,” Journal of Fluids and Structures, vol. 44, pp. 195–204, 2014.
[20] H. Tanaka, Y. Tamura, K. Ohtake, M. Nakai, and Y. Chul Kim, “Experimental investigation of aerodynamic forces and wind pressures acting on tall buildings with various unconventional configurations,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 107-108, no. 108, pp. 179–191, 2012.
[21] J. He and C. C. S. Song, “A numerical study of wind flow around the TTU building and the roof corner vortex,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 67-68, no. 68, pp. 547–558, 1997.
[22] S. Pal, R. Raj, and S. Anbukumar, “Comparative study of wind induced mutual interference effects on square and fish-plan shape tall buildings,” Sadhanā, vol. 46, no. 2, 2021.
[23] S. Pal and R. Raj, “Evaluation of wind induced interference effects on shape remodeled tall buildings,” *Arabian Journal for Science and Engineering*, vol. 46, no. 11, pp. 11425–11445, Article ID 0123456789, 2021.

[24] S. Pal, R. Raj, and S. Anbukumar, “Bilateral interference of wind loads induced on duplicate building models of various shapes,” *Latin American Journal of Solids and Structures*, vol. 18, no. 5, 2021.

[25] S. K. Nagar, R. Raj, and N. Dev, “Proximity effects between two plus-plan shaped high-rise buildings on mean and RMS pressure coefficients,” *Scientia Iranica*, 2021.

[26] B. Bhattacharyya and S. K. Dalui, “Investigation of mean wind pressures on “E” plan shaped tall building,” *Wind Structures An International Journal*, vol. 26, no. 2, pp. 99–114, 2018.

[27] T. Uchida and Y. Ohya, “A numerical study of stably stratified flows over a two-dimensional hill - Part I. Free-slip condition on the ground,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 67-68, no. 68, pp. 493–506, 1997.

[28] D.-h. Yu and A. Kareem, “Numerical simulation of flow around rectangular prism,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 67-68, no. 68, pp. 195–208, 1997.

[29] P. Sanyal and S. K. Dalui, “Effects of courtyard and opening on a rectangular plan shaped tall building under wind load,” *International Journal of Advanced Structural Engineering*, vol. 10, no. 2, pp. 169–188, 2018.

[30] R. Raj, T. Rana, T. Anchalia, and U. Khola, “Numerical study of wind excited action on H Plan-shaped tall building,” *International Journal on Emerging Technologies*, vol. 11, no. 3, pp. 591–605, 2020.

[31] R. Paul and S. K. Dalui, “Prognosis of wind-tempted mean pressure coefficients of cross-shaped tall buildings using artificial neural network,” *Periodica Polytechnica: Civil Engineering*, vol. 65, no. 1, pp. 134–149, 2021.

[32] A. Okajima, D. Yi, A. Sakuda, and T. Nakano, “Numerical study of blockage effects on aerodynamic characteristics of an oscillating rectangular cylinder,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 67-68, no. 68, pp. 91–102, 1997.

[33] A. K. Bairagi and S. K. Dalui, “Estimation of wind load on stepped tall building using CFD simulation,” *Journal of Building Engineering*, vol. 33, Article ID 101843, 2021.

[34] P. Sanyal and S. K. Dalui, “Comparison of aerodynamic coefficients of various types of Y-plan-shaped tall buildings,” *Asian Journal of Civil Engineering*, vol. 21, no. 7, pp. 1109–1127, 2020.

[35] J. Franke, A. Hellsten, H. Schlunzen, and B. Carissimo, *Guideline for the CFD Simulation of Flows in the Urban Environment: COST Action 732 Quality Assurance and Improvement of Microscale Meteorological Models*, COST Office, Brussels, Belgium, 2007.

[36] J. Revuz, D. M. Hargreaves, and J. S. Owen, “On the domain size for the steady-state CFD modelling of a tall building,” *Wind and Structures*, vol. 15, no. 4, pp. 313–329, 2012.

[37] R. Raj, T. Rana, T. Anchalia, and U. Khola, “Proximity effects between two plus-plan shaped high-rise buildings on mean and RMS pressure coefficients,” *Scientia Iranica*, 2021.

[38] S. Hajra and S. K. Dalui, “Numerical investigation of interference effect on octagonal plan shaped tall buildings,” *Journal of Building Engineering*, vol. 33, Article ID 101843, 2021.

[39] P. Sanyal and S. K. Dalui, “Comparison of aerodynamic coefficients of various types of Y-plan-shaped tall buildings,” *Asian Journal of Civil Engineering*, vol. 21, no. 7, pp. 1109–1127, 2020.