Research Article

Wind Effects on Rectangular and Triaxial Symmetrical Tall Building Having Equal Area and Height

Asthा Verma, 1 Rahul Kumar Meena, 2 Hrishikesh Dubey, 2 Ritu Raj, 2 and S. Anbukumar 2

1Department of Civil Engineering, Govind Ballabh Pant University of Agriculture and Technology, Pantnagar, India
2Department of Civil Engineering, Delhi Technological University, New Delhi, India

Correspondence should be addressed to Ritu Raj; rituraj@dtu.ac.in

Received 25 February 2022; Revised 11 May 2022; Accepted 23 May 2022; Published 20 July 2022

Copyright © 2022 Astha Verma et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The study aims to investigate wind effects on the equal area and the same height building model having the plan cross-sectional shape in the form of regular and irregular shape buildings. The ratio of modification in the cross-sectional shape is kept same for regular and irregular shaped models because limited studies are available for such type of comparison. The responses of the structure to resist the wind load is increased by applying these modifications in the corner configuration such as corner cut, chamfer, and fillet in the plan shape of the structure. The numerical study is performed using ANSYS CFX and wind incidence angle varies in the range of 0° to 180° at the interval of 15°. The results of velocity and pressure distribution are presented in the form of pressure contours and wind force coefficients for all four-building models which are presented in the form of a graph. The comparison of numerical simulation results on two models are compared with the different international standards and with the experimental results. The regular shape with a corner cut and a “Y”-shaped structure with a fillet corner model performed the best among all models for resisting overall wind force. The results of pressure distribution of all four models are presented in the form of pressure contours for 0° and 90° wind angles.

1. Introduction

The tall structure is the most appropriate solution for the current situation since land availability in the conventional shape is approaching at saturation. The population is also booming and there is a scarcity of space available for individual residences consequently, accommodating such a large population within the available land is feasible with tall buildings. Various international standards [1–10] are already discussed different parameters to design tall buildings. There is significant constraint on the design of tall buildings as the majority of international standards are silent on the modification of the shape of plan cross-sectional area buildings. Tavakoli et al. [11] discussed the important aspects of high rise buildings. Dadkhah et al. [12] investigated the performance of the steel structure during an earthquake. Kamgar et al. [14] investigated the soil-structure interaction for a different storey outrigger braced building resting on two different types of soil. Dadkhah et al. [15] performed nonliner dynamic analyses using incremental dynamic analysis, which is most accurate and time-consuming, and also need a high computational cost. Dadkhah et al. [16] presented the optimization of a tuned mass damper to improve the seismic performance of a six storey steel structure based on the ductility damage index. Kamgar and Samani [17] conducted the numerical study for evaluating the effect of length to the height ratio on the behavior of a concrete frame retrofitted with steel infill plates and observed that the value of absorbed energy by two diagonal viscous dampers is higher than that of the chevron viscous damper. Numerous studies have been conducted in the subject of wind engineering, the majority of which focus on modifying the shape or height of the structure but the current study focuses on equal area structures of regular and

during an earthquake. Kamgar et al. [14] investigated the soil-structure interaction for a different storey outrigger braced building resting on two different types of soil. Dadkhah et al. [15] performed nonliner dynamic analyses using incremental dynamic analysis, which is most accurate and time-consuming, and also need a high computational cost. Dadkhah et al. [16] presented the optimization of a tuned mass damper to improve the seismic performance of a six storey steel structure based on the ductility damage index. Kamgar and Samani [17] conducted the numerical study for evaluating the effect of length to the height ratio on the behavior of a concrete frame retrofitted with steel infill plates and observed that the value of absorbed energy by two diagonal viscous dampers is higher than that of the chevron viscous damper. Numerous studies have been conducted in the subject of wind engineering, the majority of which focus on modifying the shape or height of the structure but the current study focuses on equal area structures of regular and
irregular shapes. When the regular shape of the rectangular building model is contrasted to the irregular shape of the “Y”-shaped building model, the designer can choose the shape of the building when constructing such shapes.

Investigation of wind effects is possible through two techniques like wind tunnel and CFD, the various amount of research is already conducted through the wind tunnel, such as Kwok [18] studied the effect of wind on the shape of a building by applying different measures to control the wind effect and found that the building with the chamfer corner is best among all to resist the along and across wind response. Jozwiak et al. [19] presented the aerodynamic interference effect on pressure distribution on the building model and observed that there is a significant difference in pressure distribution with and without modeling of the boundary layer. Pal et al. [20] presented the experimental investigation performed in an open circuit boundary layer wind tunnel on the “fish plan shape” and observed that pressure on the windward face in the case of a fish shape is less than the square shape when the wind incidence angle is 0°. Nagar et al. [21] performed an experiment in BLWT on the “H” shaped model and found that pressure increases until the wind incidence angle reach up to 60° wind, and it was found that the interference factor for the square and the “H” shaped building is less than one. Pal et al. [22] carried out a comparative study of wind-induced mutual interference effects on the twin square and the fish plan shaped model, which is having equal volume, and the experiment is performed in the boundary layer wind tunnel at the length scale of 1 : 300. This is observed from the experimental study that the drag and lift force for the fish shaped model is lesser than that of the square shaped model in the case of 0° and 180° wind. Nagar et al. [23] investigated the wind-induced interference effect between two plus shaped high rise buildings on mean and RMS pressure coefficients and concluded that these interference effects on local wind pressure are significantly higher on the windward side face near the recessed corner. Pal et al. [24] performed an experimental investigation on a square and a remodeled triangle shaped building at the length scale of 1 : 300. This is found that the front-to-front interference case of the remodeled triangle shaped model performed best in resisting along and across the wind. The maximum suction of the testing model at the FBI condition is decreased by nearly 40% to that of the isolated case of the SPS model at the same condition. Amin and Ahuja [25] presented a review on an aerodynamic modification such as corner cut, chamfering, rounding, horizontal slots, vertical slots, dropping of corners, and many more to the shape of the tall building. The major outcome from this review is that the opening along the cross-wind direction at the top significantly reduces the wind excitation of the building, and the tapering of tall buildings has significant effects on wind force in across wind direction than that of the along wind response. Blackmore [26] estimated the wind effects on and around the building model, wind forces are dynamic and fluctuate continuously in both magnitude and position. Most of the international standards considered static loads and having limitations. Bhattacharyya and Dalui [27] presented the study on the “E” plan shaped building, which has symmetry about both the axis and wind response is measured from 0° to 330° at the interval of 30°. The result obtained from the experimental study and numerical study were compared. Various amounts of research work is performed using the wind tunnel test, however, this study is performed by using the k-ε turbulence model and most of the studies in the present times are performed using the k-ε turbulence model to obtained the solution using the numerical simulation.

Numerical simulation is the better option for the estimation of wind loads because the wind tunnel test is time taking and costly, and it also needs heavy machinery and resources to investigate the wind effects. Various studies are also available and presented on the various aspect of CFD simulation, such as Tominaga and Stathopoulos [28] investigated the turbulent scalar flux by assuming the gradient diffusion hypothesis using Reynolds averaged Navier stock equation in CFD. Blocken [29] discussed the important aspect of CFD for defining urban physics and gave ten important tips and tricks towards the accurate and reliable simulations. Behera et al. [30] performed an experiment in the boundary layer wind tunnel to investigate the wind interference effect on tall buildings with varying plan ratios, and observed that the interference zone extends over a larger area as the building plan ratio increases. Tamura et al. [31] performed a series of wind tunnel tests to determine the aerodynamic performance and pedestrian level wind characteristics of different super tall buildings having square, rectangular, and elliptic plan with corner cuts. Mittal et al. [32] performed the numerical simulation on tall buildings and discussed the effect of the shorter building placed in front of a taller building. Blocken et al. [33] conducted the CFD simulation of the atmospheric boundary layer and discussed the effect of using wall function. The accuracy of CFD simulation can be seriously compromised when wall function roughness modification based on the experimental data for sand grain roughened pipes and channels are applied at the bottom of the computational domain. Numerical studies are already conducted on the different types of the plan shape structure, such as Tian et al. [34] on a rectangular shape, Gaur and Raj [35] on corner modification on a square and a plus shaped building, Raj et al. [36], Keerthana and Harikrishna [37] on the “H” plan, Kumar and Raj [38] on an octagonal plan shape, Raj et al. [39] obtained response of a square and a plus shaped building by varying wind loads, Gaur et al. [40] studied the interference effect using CFD, Mukherjee et al. [41], Sanyal and Dalui [42-43], Goyal et al. [46] on the “Y” shape, Meena et al. [47], Kumar and Raj [48] on the “L” shape, Raj and Ahuja [49], on the “+” shape, Sanyal and Dalui [50] on the rectangular shape having some modification, Meena et al. [51] investigated the wind effects on the regular shaped structure  having a rectangular plan with various corner modifications, Mallick et al. [52] on the “C” shape, Amin and Ahuja [53] on the side ratio of a rectangular building, Amin and Ahuja [54] obtained response of a tall building against the wind load. The considerable outcomes after modifying the shapes of the structure are as the Strouhal number is not sensitive to the aspect ratio, flow reattachment of the separated shear layer in the corner cut model present suction into corners, which is further
caused in the reduction of drag, interference position of the building is highly effecting principal building pressure distribution, the length/width (side ratio) makes changes into the upwash, downwash, and the stagnation zone on upstream of flow, shielding effects were the main factors which is controlling the spacing between the buildings, highest wind velocity is obtained at the edges of the windward side and minimum at the leeward side, maximum suction is observed at the corner of the building, it is important to give due consideration for suction on the corner region, by increasing the $L/W$ ratio horizontal force coefficient and overturning moment coefficient increases by the significant amount, the model with the rounded corner is best to reduce wind loads and overturning moments, in comparison of the chamfer corner, rounded corner are efficient in reducing the wind load, deflection is observed due to the presence of vortices in the wake region of buildings, the accuracy of the $k$-$\varepsilon$ turbulence model is more than than the SST model.

The current study will examine the wind pressure on tall structures of both regular and irregular shapes. The majority of earlier research has concentrated on the wind effects of either regular or irregular shaped buildings, however, the current study examines the results of these two types of buildings by altering the corners. This study is particularly noteworthy because the cross-sectional area and height of the building model are kept constant to minimize wind loads. Numerous corner modifications, such as chamfer, fillet, and corner cut, are also employed and the ratio of cuts is maintained, allowing the designer to select the most efficient structure for the situation. Additionally, two models are validated and the numerical simulation results are compared to experimental results and a number of relevant standards. Furthermore, the result of velocity streamlines is illustrated graphically in the form of contours, pressure distribution, and wind force coefficient for wind incidence angles ranging from 0° to 180° at 15°-degree intervals.

2. Methodology

The present study concentrates on the investigation of wind effects and compares the results of the regular and irregular shaped models. The numerical simulation results are validated before the starting of the work and the results of numerical simulation are found identical with experimental results.

2.1. Method of Study. The present study is performed using the numerical simulation tool ANSYS CFX and investigated the wind effects on tall buildings having an equal plan area and same height. Most of the available studies are either for regular shape or for irregular shape, but these studies present the comparison among such shapes having both regular and irregular shape. The geometry is selected and applied in the CFX-Pre setup. After the geometry, the meshing is performed and flow physics are defined in the solution and simulation is performed and the results are presented in different graphical forms.

2.2. Numerical Simulation. Numerical simulation is performed using ANSYS CFX and boundary conditions used into this simulation are kept as the boundary condition used by Raj [55] in the experiment performed in the boundary layer wind tunnel at IIT Roorkee, India. The numerical simulation performed into this study is performed using the $k$-$\varepsilon$ turbulence model [56].

(i) $k$-$\varepsilon$ turbulence model

This method is generally used to solve a complex fluid problem and it is a two-equation model. The solution is generated after solving two different equations during the entire numerical simulation. This model performs nearly equivalent to the experimental problem and the $k$-$\varepsilon$ model also uses the scalable wall-function to increase the efficiency of the solver and this model performs more robustness in the case of the fine mesh.

It is very less expensive and mostly used to simulate turbulent flow characteristics. The solution is solved by two transport equations, i.e., turbulent kinetic energy ($k$) and the turbulent dissipation rate ($\varepsilon$). It has the positive advantage of not including any geometry-related parameters in the modeling. Turbulent kinetic energy and the turbulence dissipation rate are two variables introduced into the system of equations for the model. The values for turbulence are calculated according to turbulence eddy dissipation using

(ii) Basic equations

The basic equation used to study fluid flow problems using Navier-Stokes and continuity equation.

(iii) Navier stokes equation is written as

$$\frac{\partial (\rho u_i)}{\partial t} = -\frac{\partial (\rho u_i u_j)}{\partial x_j} + \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + F_i.$$  \hspace{1cm} (1)

(iv) Continuity equation is written as

$$\frac{\partial e}{\partial t} + \frac{\partial}{\partial x_i} (e u_i) = 0.$$  \hspace{1cm} (2)

The standard $k$-$\varepsilon$ model uses the following transport equations for the turbulence kinetic energy and turbulence dissipation.

$$\rho \frac{\partial k}{\partial t} + \rho \vec{u}_i \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \rho \varepsilon + \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial e}{\partial x_j} \right].$$  \hspace{1cm} (3)

Momentum equation is written as

$$\frac{\partial (\rho U_i)}{\partial t} = \frac{\partial (\rho U_i U_j)}{\partial x_j} - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + S_{M_i}.$$  \hspace{1cm} (4)
2.2.1. Advantages

(1) It performs best for the boundary layer separation fluid problem.
(2) It is the most use turbulence model for validation purposes.
(3) Generally, it is used by the industrialist to solve the complex fluid problem.
(4) It provided the solution based on boundary conditions.

2.2.2. Disadvantage

(1) It does not perform well in the case where flow is changing instantaneously.
(2) It is unable to generate the accurate solution for the rotating fluid problem.
(3) It is notable to generate the accurate solution for a low Reynold number.

2.3. Geometrical Configuration of Model

The regular shape of a rectangular building Figure 1 represents the building having no corner modification, corner cut model and irregular shaped model of Y-shape is represented in Figure 2 are considered in this study. The plan cross-sectional area and height are kept the same for the study so that the results can be compared on an equal volume building model. Both sorts of building models have the same modification ratio.

2.4. Meshing and Domain

Meshing contains important flow features, which are dependent upon the flow parameters, such as grid refinement inside the wall boundary layer. The meshing applied after the geometry and the name selection for each part of geometry are required before meshing to understand the CFX-Pre about the geometry. The name selection is required to define flow physics. It is better to provide name selection before the meshing so that the surface mesh exactly matches with nodes on the two sides of the boundary, which allows a more accurate fluid solution.
Nameselectionalsohelpstheprogramtocontrolthe inflation, i.e., automatically select for the wall and inflation automatically provided during the auto mesh generation. The meshing is represented in Figure 3 for domain, building, and inflation.

The mesh generation steps are automatically worked into the program; however, this can be controlled by varying the element size, type of mesh to generate, and where and how the mesh should be refined. Tetra dominant meshing, which is patch independent, is suitable for the CAD model having many surface patches and if the geometry has small edges then this is suitable [58]. Hex meshing is used both in general sweep and thin sweep; it is also recommended for the model which has the clean cad geometry. Mapped and free meshing is adopted where the fluid problem needs a different type of meshing like structured (mapped) and unstructured (free); however, this is suitable for the problem where the sweeping method does not work without extensive geometry decomposition.

Inflation is provided to capture the flow properly at the interface and the same can be provided by various methods available in the CFD model. Smooth transition is the default option and it uses the local tetrahedron elements size to compute each local height and total height so that the rate of volume change is smooth. Each triangle that is being inflated
will have an initial height that is computed with respect to its area averaged at the nodes. This means that for the uniform mesh, the initial height will be roughly same, while for a varying mesh, the initial height will vary. An increment in the value of the growth rate that controls the reduction in the total height of inflation layers.

The numerical simulation predicts the result based on boundary conditions, as the finite element method work well with boundary conditions. Inlet wind speed is provided as the power law where the reference height in the simulation is considered as 1 m, while the reference velocity is defined as 10 m/s. The wall of the domain is considered as a free slip wall, while the ground of the domain is a rough wall [43].

The faces of the model are considered as no slip. Wall function defines that if a coarse mesh near the wall than it assumes that the logarithmic low applies as wall function. At the same time, the fine mesh near the wall and various turbulence models account for a low Reynold number. Wall functions are useable to low Reynold number problems. In general, if the fine mesh is defined near the wall then the gradient changes are considered automatically by the solver. The domain is represented in Figure 4 in both the top and isometric view. A domain feature is considered after the various recondition provided by the past studied [59–62] numerical model. No-slip walls is the wall where fluid immediately next to the wall assumes the velocity of the wall as zero; free slip wall is the wall where the shear stress at the wall is zero and the velocity of the fluid near the wall is not retarded by the wall friction effects.

2.5. Grid Sensitivity. In this study a grid sensitivity study is performed on model-A and tabulated in Table 1, which is having no corner modification into the rectangular shape model. A grid sensitivity study is essential to CFD programming as it helps to achieve the proper meshing size, which predicts the most accurate result for the numerical simulation. For the present study GS-3 is adopted for all the wind incidence angle.

2.6. Mean Wind Speed and Turbulence Intensity Profile

2.6.1. Mean Wind Speed. Wind flow over the tall building involves complex flow patterns, although wind flow is generally separated from the surface, where the flow is mostly recirculating. Flow patterns are unsteady around the buildings model, therefore creating turbulence. The development of turbulences near the ground level is observed due to the presence of other obstacles. Such effects are also considered while calculating the turbulence intensity and wind flow up to the gradient height and after that it becomes constant.

Mean wind speed defined in the CFX-Pre is represented in Figure 5, the variation in the form of a power law between height and mean wind speed is as follows:

\[
\frac{U(z)}{U(Z_{ref})} = \left(\frac{Z}{Z_{ref}}\right)^n,
\]

where \(U(z)\) = mean velocity at height \(Z\); \(U(Z_{ref})\) = mean velocity at reference height \(Z_{ref}\); and \(n\) = power law exponent, a measure of ground roughness, varies between 0.13 and 0.15 in open terrain.

The variation in the mean wind speed with height is generally expressed in an alternate form in a logarithmic form.

\[
\frac{U(z)}{U_*} = \left(\frac{1}{k}\right) \ln\left(\frac{Z}{Z_o}\right).
\]

Also, a simplification of this logarithmic equation is as follows:

\[
\frac{U(z)}{U(Z_{ref})} = \frac{\ln(Z/Z_o)}{\ln(Z_{ref}/Z_o)}
\]

where \(U_*\) = shear velocity; \(k\) = Von Karman constant, 0.4; \(\ln\) = natural log function; and \(Z_o\) = effective roughness length, another measure of ground roughness, varies from 0.01 to 0.05 meters in open terrain.

2.6.2. Turbulent Intensity. Turbulence intensity is a nondimensional quantity derived from the variance and for the mean wind speed.

\[
I_z = \frac{\sigma_z}{U(z)}
\]

where \(I_z\) = turbulence intensity at height \(z\); \(\sigma_z\) = standard deviation of the wind speed at height \(z\); and \(U(z)\) = mean wind speed at a reference height.
2.7. Validation. Validation is a prerequisite part of numerical simulation and two different models are considered in this study to validate with available international standards and an experimental study. Such buildings model are selected and external $C_p$ compared with the available experimental results. This is also clearly depicted in Figure 6 that the result obtained in this numerical simulation show nearly an identical result for the pressure coefficient.

The results plotted for the normal and corner cut rectangular building model are representing a very close match with the experimental results. Therefore, the result obtained in this study are very much accurate for designing the model, which are considered in this entire study. The rectangular shape model is also simulated for the wind load at 0° and 90° wind and a pressure coefficient is compared with available international standards mentioned at Table 2.

The external pressure coefficient "$C_p$" is calculated using (9).

$$C_p = \frac{p - po}{(1/2)\rho U_H^2},$$  \hspace{1cm} (9)

where $p =$ pressure derived from the external lines plotted on the model; $po =$ static pressure at the reference height; $\rho =$ air density ($1.225 \text{ kg/m}^3$); $U_H =$ mean wind velocity at the building reference height.

### 3. Result and Discussion

CFD postprocessed results enables the visualization and analysis of wind effects in variety of graphical representations. The pressure distribution along the face is a function of the shape and size and the same is illustrated using contours. The wind response is also represented graphically in the form of the wind force coefficient.

#### 3.1. Velocity Contours. Velocity contours are represented for model-A in Figures 7 and 8. The main cause of the pressure difference is the velocity field around and along with the building model. The velocity at downstream is mentioned at Figure 7, and the size of the wake is increasing with respect to the frontal exposed area, which changes according to wind incidence angle.

Velocity along the height of model-A is depicted in Figure 8 and the wind velocity distribution pattern is illustrating the variation of the wind at the downstream. The suction observed in the case of 15° and 75° are of the same type. The suction on the roof of the building is also observed and demonstrated in Figure 8.

#### 3.2. Wind Forces. The wind-generated effect in the form of wind force is evaluated after the numerical simulation performed on the building model. The value of $F_x$ and $F_y$ are obtained using CFD postprocessed result while the frontal exposed area is calculated at each wind incidence angle. Drag force is calculated as per equations (16) and (17). Along wind force, i.e., the drag for model-A, is obtained a maximum of −4.72 is between 60° and 120°, while the minimum 0.0 is noted when the wind is striking to the model at 0° and 90°. The corner cut model of rectangular shape, i.e., model-B, has the highest magnitude of drag is −4.39 in the case of 30° and 150° wind and the smallest drag is 0.04 in the case of 0° and 180° wind. The maximum amount of drag is reduced for model-B with respect to model-A.

$$F_x = \left(0.5\rho U_H^2 \cdot A_p\right)C_{f_x},$$  \hspace{1cm} (10)

$$F_y = \left(0.5\rho U_H^2 \cdot A_p\right)C_{f_y}.$$

An irregular "Y" shaped model having corner modifications such as model-C and D are having chamfer and fillet, respectively, the drag is measured using numerical...
simulation and graphically plotted in Figure 9. The drag is also measured for model-C (chamfer-Y shape) and the maximum magnitude is $-9.44$ in the case of $150^\circ$ wind. Model-D (fillet–Y shape) is investigated for along wind forces and the maximum drag of $-7.92$ is noticed in the case of $150^\circ$ wind, while the lowest drag is $-0.04$ in the case of $180^\circ$ wind is noticed for model-D. 

The across wind, i.e., lift force in the $y$-direction, is calculated and graphically plotted in Figure 10 for model-A, B, C, and D. The regular shaped model of the rectangular shape (model-A & B) is having some different type of corner modification so that lift force can be reduced on the tall building by making some modification in the corners. The maximum lift of $-13.73$ is noted in the case of $90^\circ$ wind for model-A, while the smallest of $-5.23$ is spotted in the case of $0^\circ$ and $180^\circ$ wind. Model-B (corner cut rectangle) is having the highest lift of $-11.28$ in the case of $90^\circ$ wind, while the lowest lift of $-4.73$ is detected in the case of $15^\circ$ and $165^\circ$ wind.

The irregular “Y” shaped model-C and D are numerically investigated for the across wind response and lift force for model-D (fillet-Y) is having the maximum lift force of

<table>
<thead>
<tr>
<th>International code</th>
<th>Wind angle (°)</th>
<th>Windward side</th>
<th>Side wall</th>
<th>Leeward side</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD results</td>
<td>0</td>
<td>0.78</td>
<td>-0.64</td>
<td>-0.30</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.70</td>
<td>-0.61</td>
<td>-0.45</td>
</tr>
<tr>
<td>IS 875 (part 3)</td>
<td>0</td>
<td>0.7</td>
<td>-0.7</td>
<td>-0.4</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.1</td>
</tr>
<tr>
<td>ASCE/SEI 7-16</td>
<td>0</td>
<td>0.8</td>
<td>-0.7</td>
<td>-0.5</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.7</td>
<td>-0.5</td>
</tr>
<tr>
<td>AS/NZS 1170.2.2011</td>
<td>0</td>
<td>0.8</td>
<td>-0.65</td>
<td>-0.5</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.65</td>
<td>-0.5</td>
</tr>
<tr>
<td>EN 1991-1-4</td>
<td>0</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td>BS 6399-2</td>
<td>0</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td>GB 50009:2001</td>
<td>0</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td>NSCP 2015</td>
<td>0</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.5</td>
<td>-0.7</td>
</tr>
<tr>
<td>ES/ISO 4354: 2012</td>
<td>0</td>
<td>0.8</td>
<td>-0.65</td>
<td>-0.7</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td>0.8</td>
<td>-0.65</td>
<td>-0.7</td>
</tr>
</tbody>
</table>
11.92 in 60° wind, while the minimum of −9.08 for 0° wind 135° wind. The along wind, lift force is found greatest of −10.42 in the case of model-D (fillet-Y) for 60° wind, while the smallest of −7.6 is noticed in the case of 0° wind. Among the Y-shaped model, the lowest lift force is observed for the model having a fillet corner into each limb of the “Y” shape [63].

3.3. Pressure Contours. Pressure distribution is represented in the form of contours for each surface of the building model. Pressure distribution for rectangular building model-A is represented in Figure 11 for 0° wind, while for 90° wind, the pressure distribution pattern along the height of the
model is represented in Figure 12. The pressure on the windward face in both the case is having nearly the same pressure distribution pattern. The maximum pressure is observed in the case of 0° wind of 0.95 in the central part of face-A, while face-B in 90° wind is having the maximum of around 1.05, which is more than face-A for 0° wind because of the face
width is increased and that is why the pressure exposure area is increased. For 0° wind, pressure along face-B starts to increase from upstream to downstream of wind, this is because of reattachment of flow. Face-B and face-D is having almost the same pressure distribution pattern, this is because of the symmetric faces of a building model. Leeward face, i.e., face-C, is having suction (negative pressure) and the maximum is on the bottom part of the building, while suction is increased at face-C from base to top of the model.

Figure 12 represents the pressure distribution variation of model-A. Pressure is presented in the form of contours, which is varying as per the shape and size of the particular face. In the case of 90° wind angle, face-A and face-C are having almost the identical pressure distribution pattern. This is increased from the ground of the model to the top of the building model. Also, pressure in the negative is increased as the wind moves from upstream to downstream. Pressure along face-D (Leeward) is having highest negative pressure in the top one-third part of the building model.

Figure 13 is pictorially representing [63] the pressure distribution in the form of contours for the “+” shaped building model, while present studies numerically investigate the wind effects on model-B. Model-B (rectangular model having corner cuts) pressure contours for 0° wind is represented in Figure 14, while for 90° wind the pressure contours for model-B are represented in Figure 15. The nature of pressure distribution on windward face-A is identical and the maximum pressure of 1.04 is spotted on face-A, while the face-A of the “+” shaped building is having maximum of 1.06. This is clearly depicted that each face of model-B is having a pressure distribution pattern and is the same as that with the experimental study performed on the “+” shaped model [63]. The pressure distribution variation is not much dependent on the size of the face, while the height more or less controls the pressure patterns.

Model-C (Y-Chamfer) is investigated numerically for wind effects and the pressure distribution pattern for 0° wind is in Figure 16, and for 90° wind, pressure contours are represented in Figure 17. The maximum positive pressure of 0.55 is on face-D, while the maximum negative of −1.3 is on face-E. The pressure variation along the shape and size is
clearly demonstrating the magnitude of pressure as contours are showing with the label, and such labels are showing the increment or decrement in pressure. The actual phenomenon along model-C is valuable for the designer to investigate the wind effects on such types of corner configurations.
Pressure distribution on model-D is represented graphically in Figure 18 for 0° wind, while pressure contours in Figure 19 is for 90° wind. The positive pressure is observed on face-A, B, C, D, M, and N, while the negative pressure is noticed on face-E, F, G, H, I, J, K, L, and O in the case of 0° wind. The maximum positive pressure of 0.64 is observed on face-D.
Figure 16: Pressure contours at 0° wind for model-C.
Figure 17: Pressure contours at 90° wind for model-C.
Figure 18: Pressure contours at 0° wind for model-D.
**Figure 19:** Pressure contours at 90° wind for model-D.
face-D, while the maximum suction of 1.4 is spotted on face-E in the case of 0° wind. Pressure contours for 90° wind for model-D (Y-fillet) is represented graphically Figure 19. The positive pressure is spotted on face-C, D, and E only in the case of 90° wind, while other faces are under the effect of negative pressure. The maximum positive pressure of 0.75 is observed on face-C and D, while the negative pressure of 1.4 is noticed on face-G for model-D (Y-fillet). This is also noticed from pressure contours that symmetrical faces are having same pressure distribution patterns.

4. Conclusion

The present numerical study is performed to investigate the wind effects on the corner effects of the tall building having various corners (corner cut, chamfer, and fillet) in the corner of the cross-sectional plan area of the building. The present study is performed using ANSYS CFX by utilizing the $k$-$\varepsilon$ turbulence model. The wind incidence angle varies in the range of 0° to 180° at the interval of 15° each. All four building models have the same corner modification ratio, as well as the same area and height. Validation studies show the closer agreement of the numerical simulation result with the experimental studies and different international standards. Various results of the pressure distribution wind force coefficient are calculated and presented. The notable outcomes from the present are as follows:

(i) Validation investigations indicate a strong degree of congruence between numerical results and experimental results and a number of international standards. Validation is carried out on two distinct models of rectangles, one with a regular shape and one with a corner cut.

(ii) The shape of the building is having the significant effect in determining the wind load on the tall building, wind forces are varying with the wind incidence angle. A regular plan shape model with no corner modification and a model with corner modification exhibit identical along the wind response, but the overall lift is less for the model having corner modification.

(iii) The maximum wind forces at 30°, 90°, and 150° wind angles were observed for an irregular “Y” shaped building with modified corners. These types of structures are well-suited to withstand such high-intensity forces, if appropriate openings are provided in the cladding unit.

(iv) Among the regular shape models, a model with corner cuts is optimal; however, an irregular “Y” shaped model with fillet corners into each limb is optimal for resisting overall wind force. To avoid excessive wind loads, the base shear of all four buildings is also investigated.

(v) The majority of available international standards are silent on providing data for irregular shape models; however, this study investigates wind load information for such buildings.

(vi) The designer can choose the best model from one of these four models, since this study compares the wind effects on equal area building models with the same ratio of modifications and the results are presented in the form of pressure contours for the regular and irregular models.

4.1. Future Scope. The present study analyses and compares the result of the simple type of corner and corner cut on a regular shaped building model, while for irregular shaped modifications of chamfer and fillet shape is provided. Different types of modification can be applied and checked for stability against the wind load.

Nomenclature

\( \rho \): Density of air

\( F \): Body force per unit volume

\( u_i \): The filtered scale velocity field

\( \mu \): Dynamic viscosity of air

\( t \): Time step

\( \mu_t \): Eddy or turbulent viscosity

\( x, y, z \): System of rectangular cartesian coordinates

\( \varepsilon \): Dissipation rate of \( k \)

\( p \): Pressure

\( k \): von Karman’s constant

\( u, v, w \): Fluctuating wind

\( \tau_{ij} \): Turbulent stress tensor

\( z_o \): Reference height

\( z \): Height above the ground

\( n \): Power law index

\( v_z \): Mean wind speed at any height

\( v_o \): Mean wind speed at reference height

\( P \): Pressure at the point

\( P_0 \): Static pressure at reference height

\( U_H \): Mean wind speed at reference height

\( C_F \): Force coefficient

\( C_M \): Base moment coefficient

\( A_p \): Area projected

\( H \): Height of the building model

\( \beta, \beta^*, \sigma^*, \sigma \): Closure coefficients

\( k \): Turbulent kinetic energy

\( i \): General values of \( u, v, w \) component at point

BLWT: Boundary Layer Wind Tunnel

TTU: Texas Tech University

SST: Sher Stress Transport

GC: Grid Convergence

ABL: Atmospheric boundary layer flow

CFD: Computational fluid dynamics

WT: Wind Tunnel

atm: Atmospheric

Data Availability

All data, models, and code generated or used during the study appear in the submitted article. Data are available from...
the corresponding author upon request 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 the Delhi Technological University, Delhi, India, for providing funding to conduct the research work and for the institutional fellowship for the second author under Institutional fellowship for the Ph.D.

**References**


