WIND PRESSURE AND VELOCITY PATTERN AROUND ‘N’ PLAN SHAPE TALL BUILDING-A CASE STUDY

A. Mukherjee¹ and A. K. Bairagi²*
¹Department of Civil Engineering, Techno India University, Saltlake, Kolkata-700091, India
²Kolkata Metropolitan Water and Sanitation Authority, Kolkata- 700 006, West Bengal, India

Received: 10 April 2017; Accepted: 29 August 2017

ABSTRACT

The principal aim of this case study includes the determining of wind pressure co-efficient and wind velocity analysis of N-plan shape tall building using $k$-$\varepsilon$ method. The wind angles varying from 0° to 180° at 30° interval scale of 1:300 and terrain category 2. A square plan with same area compared with the N-shape model also validated with different codes AS/NZS: 1170.2 (2002), ASCE/SEI 7-10 (2010), BS: 6399-2 (1997), EN: 1991-1-4 (2005) and IS 875 (part 3) (1987). The wind velocity and force coefficients around this model at different faces are also discussed here.

Keywords: CFD simulation; N-shape; pressure coefficients; force coefficients; velocity profile; wind load.

1. INTRODUCTION

The analytical results of N plan shape tall building are investigated in a series of fluid flow (CFX) analysis to study the wind pressure co-efficient. The model can be rotated from 0° to 180° at 30° interval. Therefore, attacking wind angles are 0°, 30°, 60°, 90°, 120°, 150° and 180°. This study also validates with wind pressure of the same plan area of an isolated square plan shape, tall building using CFD simulation and compared it with AS/NZS: 1170.2 (2002) [1], ASCE/SEI 7-10 (2010) [2], BS: 6399-2 (1997) [3], EN: 1991-1-4 (2005) [4] and IS 875 (part 3) [5]. Zhou et al. [6] investigated the along wind load on tall building using different international codes, and suggested that the scatter occurring in the calculation of wind load is mainly due to the variation in definition of wind load characteristics. Gomes et al. [7] follow the wind effects on and around irregular-plan buildings is presented for the L and U configurations. The experiments in a (1.25×1.00) m² close circuit wind tunnel. The velocity in the wind tunnel was approximately 10 m/s. Mendis et al. [8] investigated wind...
loading on tall building of a typical model of a (40×40×300) m building of 1:400 scale. Kim et al. [9] studied on dynamic wind loading on ‘H’ shaped buildings at wind angles 0° and 90°. Simulation of natural wind was achieved using triangular spires and floor roughness elements in a boundary layer wind tunnel. Amin and Ahuja [10] researched on wind-induced pressures on buildings of L-shape and T-shape models and noticed the pressure distribution. Chakraborty et al. [11] highlighted the pressure developed on ‘+’ shape tall building using CFD simulation and validated with wind tunnel test. The pressure variation on different faces of the model for 0°, 30°, and 45° wind incidence angle are highlighted in this study. Ahlawat and Ahuja [12, 13] Researched on the effects of wind loads on ‘Y’ plane shaped and ‘T’ plan shape, tall building used by wind tunnel to investigate the wind loads generated on the building in isolated as well as interference condition. This paper highlight the pressure and force coefficient at the face of “N” plan shape building model and also evaluated the wind velocity around the building from inlet towards the outlet of the domain. The turbulence developed at the corner of the inclined limb is also carried out in this study.

2. NUMERICAL ANALYSIS

ANSYS CFX is widely used software package for the wind analysis. With the help of ANSYS CFX-solver theory guide [14] the equations (1)-(7) are fixed. In this study the $k$-$\epsilon$ method is used and the equations contain many immeasurable and unknown terms. $k$ is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of $(L^2T^2)$; for example, $m^2/s^2$. $\epsilon$ is the turbulent eddy dissipation and has dimensions of per unit time $(L^2T^3)$; for example, $m^2/s^3$.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0$$

(1)

The momentum equation will be

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{\text{eff}}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_M$$

(2)

where, $S_M =$ sum of body forces; $\mu_{\text{eff}} =$ effective viscosity accounting for turbulence; $p'$= modified pressure as defined by:

$$p + \frac{2}{3}\rho k + \frac{2}{3}\mu_{\text{eff}} \frac{\partial U_k}{\partial k}$$

(3)

$$\left(\frac{2}{3}\mu_{\text{eff}} \frac{\partial U_k}{\partial k}\right)$$

is involves the divergence of the velocity. It is neglected in CFX. Therefore, this assumption is strictly correct only for incompressible fluids. The $k$-$\epsilon$ model is based on the eddy viscosity concept, so that:
\[ \mu_{\text{eff}} = \mu + \mu_t \]  

(4)

where \( \mu_t \) is the turbulent viscosity. The \( k-\varepsilon \) model assumes that the turbulent viscosity is linked to the turbulence kinetic energy and dissipation via the relation:

\[ \mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \]  

(5)

The values of \( k \) and \( \varepsilon \) come directly from the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate:

\[
\begin{align*}
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho k U_j) &= \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] + \nu \frac{\partial^2 k}{\partial x_i \partial x_i} + P_k + P_b - \rho \varepsilon - Y_M + S_k \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \varepsilon U_j) &= \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_1 \frac{\varepsilon}{k - C_3 \varepsilon} P_b + S_\varepsilon
\end{align*}
\]

(6)

where, \( C_1 = \max \left[ 0.43 \left( \frac{\eta}{\varepsilon} \right) \right] \); \( \eta = S \frac{k}{\varepsilon} \); \( S = \sqrt{2S_{ij} S_{ij}} \); \( P_k = \) the generation of turbulence kinetic energy due to the mean velocity gradients; \( P_b = \) the generation of turbulence kinetic energy due to buoyancy; \( Y_M = \) the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. \( k-\varepsilon \) turbulence model constant \( C_\mu = 0.09 \), \( C_{1\varepsilon} = 1.44 \), \( C_2 = 1.92 \), \( \sigma_\varepsilon = 1.2 \), Turbulence model constant for the \( k \) equation \( \sigma_k = 1.0 \).

3. DOMAIN AND BOUNDARY DETAILS

The experiment was conducted in ANSYS software using a numerical study by creating a domain and wind flow at a velocity 10m/s. The experimental flow was simulated similar to that of terrain category 2 as per Indian standard for wind load IS: 875 (part 3) - 1987 on a geometric scale of 1:300. Frank et al. (15), Revuz et al. (16) recommended that inlet, top and two side faces of domain are 5H and outlet is 15H from the edge of building model face. Where ‘H’ is the height of building. Fig. 1 shows the domain used for analytical purpose. Side wall and top of the domain considered as free slip and the domain floor and building walls are no slip. The velocity profile of the atmospheric boundary layer in the CFD is calculated by the following power law as discussed in equation (8).

\[ \frac{V_Z}{V_R} = \left( \frac{Z}{Z_R} \right)^{\alpha} \]  

(8)

where \( V_Z \) is the design wind velocity at height \( Z \), \( V_R \) is the speed at a reference height \( Z_R \), usually taken as the standard meteorological height of 10 m, here \( Z_R \) is 1.0m for this case. \( Z \)
is the wind speed at an elevation $Z$ from ground, $\alpha$ is the constant for a particular site and terrain which varies with the ground roughness which is 0.133 for terrain category 2.

Figure 1. Details of the domain used in numerical study.

4. OVERVIEW OF MODEL

The ‘N’ plan shape prismatic bluff body has been studied using CFD simulation. The vertical limbs are 250mm $\times$ 50mm and the outer to outer distance of limbs are 250mm. The inclined limb is connected with top and bottom part of the vertical limb having width 50mm. The height of the model is 500mm. Attacking wind angles are from 0°, 30°, 60°, 90°, 120°, 150° and 180° which are at 30° interval. The face names are like Face A, Face B, Face C, Face D, Face E, Face F, Face G, Face H, Face I and Face J and isometric view of N shape model as shown in Fig. 2 (a)-(b). This study is primarily motivated on analysing wind pressure on N-shape unsymmetrical tall building and external pressure co-efficient ($C_p$) due to wind at different wind angles. This study is particularly important to assess the impact of wind on isolated irregular shaped structures within a particular area with different boundary conditions influenced by wind pressure up to speed 10m/s, which utilize public domain within the vicinity of the proposed structure. The study carried out for 1:300 scale and terrain category: 2 for $k$-$\varepsilon$ numerical simulation approach using tetrahedral meshing. Mesh adaptation in a CFX process in which, once or more during a run, the mesh is selectively refined in areas that depend on the adoption criteria specified. A numerical simulation also have been done with same plan area of N-shape body with a prototype square plan shape building having dimension 190.97mm $\times$ 190.97mm and height 500mm. The face names are Face A, Face B, Face C and Face D are shown in Fig. 2 (c). This square model simulated under same domain with the same boundary conditions and scale with same terrain category. The meshing of N-shape and square models are shown below Fig. 3 (a)-(b).
WIND PRESSURE AND VELOCITY PATTERN AROUND ‘N’ PLAN

5. RESULTS AND DISCUSSIONS

5.1 Isolated square building
Wind load on isolated square plan shape tall building due to 0° and 90° wind angles are studied to validate the pressure coefficients at different face of the building with AS/NZS: 1170.2 (2002), ASCE/SEI 7-10 (2010), BS: 6399-2 (1997), EN: 1991-1-4 (2005) and IS 875 (part 3). The square plan shape tall building used in this study have height (H) =500mm, Width (W) =190.97 mm, Length (L) =190.97mm. So the aspect ratio (H/L = 2.61) and (W/L=1). The same aspect ratio of the square plan shape tall building has been considered from codes AS/NZS: 1170.2 (2002), ASCE/SEI 7-10 (2010), BS: 6399-2 (1997), EN: 1991-1-4 (2005) and IS 875 (part 3) and validate the CFD model. The table: 1 shows the result of the comparative study between CFD simulation and different codes. Pressure contours of all the faces and streamline of isolated square building also analysed here for the wind angle 0° and 90° as shown in Fig. 4 and 5. The pressure for windward face A having positive pressures and at the same time the leeward face C having completely suction. The side faces
B and D experienced with negative pressure. The streamline of isolated square plan shape tall building shows the flow generated around the model for 0° wind incidence angle. The velocities around the edges are increased due to separation of flow.

Table 1: Comparative study between CFD simulation and codal provision for square plan shape tall building

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0°</td>
<td>90°</td>
<td>0°</td>
<td>90°</td>
<td>0°</td>
<td>90°</td>
</tr>
<tr>
<td>Windward</td>
<td>0.80</td>
<td>0.80</td>
<td>0.80</td>
<td>0.80</td>
<td>0.80</td>
<td>0.80</td>
</tr>
<tr>
<td>Leeward</td>
<td>-0.49</td>
<td>-0.49</td>
<td>-0.50</td>
<td>-0.50</td>
<td>-0.50</td>
<td>-0.63</td>
</tr>
<tr>
<td>Side face</td>
<td>-0.78</td>
<td>-0.78</td>
<td>-0.65</td>
<td>-0.70</td>
<td>-0.80</td>
<td>-0.80</td>
</tr>
</tbody>
</table>

Figure 4. Contours of different faces for isolated tall building for 0° wind angle

Figure 5. Streamline of isolated square plan shaped tall building at 0° wind angle (a) Plan, (b) Elevation
5.2 N-plan shaped tall building

The pressure contour of faces for the N plan shape tall building at 0° and 90° wind angles are shown below the table: 2. Here the windward face A has been experienced with maximum positive pressure for 0° wind angle and local effect developed on this face for 90° wind angle. The positive pressure developed at face A for 0° wind angle only, whereas suction developed at all other faces for that particular angle. On the other hand positive pressure developed at windward face B, C, D and E for 90° wind, but other faces have suction.

Table 2: Pressure contours on different faces of N plan shape tall building at 0° and 90° wind angle

<table>
<thead>
<tr>
<th>Face Name</th>
<th>0° wind angle</th>
<th>90° wind angle</th>
<th>Face Name</th>
<th>0° wind angle</th>
<th>90° wind angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Face A</td>
<td></td>
<td></td>
<td>Face F</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Face B</td>
<td></td>
<td></td>
<td>Face G</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Face C</td>
<td></td>
<td></td>
<td>Face H</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

![Pressure contours images]
Table 3 below are presented the plan and elevation of streamlining of N shape model for 0° and 90° wind angle. The high turbulence developed for 0° wind at the corner between face C-D and H-I, but for 90° wind no turbulence observed at that particular corner, but the suction at corner C-D has comparatively higher than the corner H-I.

Table 3: Streamline for N plan shape tall building at 0° and 90° wind angle

<table>
<thead>
<tr>
<th>Streamline position</th>
<th>0° wind angle</th>
<th>90° wind angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plan</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pressure</td>
<td></td>
<td></td>
</tr>
<tr>
<td>45.492</td>
<td>45.492</td>
<td></td>
</tr>
<tr>
<td>39.095</td>
<td>39.095</td>
<td></td>
</tr>
<tr>
<td>32.697</td>
<td>32.697</td>
<td></td>
</tr>
<tr>
<td>26.296</td>
<td>26.296</td>
<td></td>
</tr>
<tr>
<td>19.901</td>
<td>19.901</td>
<td></td>
</tr>
<tr>
<td>13.503</td>
<td>13.503</td>
<td></td>
</tr>
<tr>
<td>7.105</td>
<td>7.105</td>
<td></td>
</tr>
<tr>
<td>0.707</td>
<td>0.707</td>
<td></td>
</tr>
<tr>
<td>-5.691</td>
<td>-5.691</td>
<td></td>
</tr>
<tr>
<td>-12.086</td>
<td>-12.086</td>
<td></td>
</tr>
<tr>
<td>-18.468</td>
<td>-18.468</td>
<td></td>
</tr>
<tr>
<td>-24.864</td>
<td>-24.864</td>
<td></td>
</tr>
<tr>
<td>-31.262</td>
<td>-31.262</td>
<td></td>
</tr>
<tr>
<td>-37.680</td>
<td>-37.680</td>
<td></td>
</tr>
<tr>
<td>-44.078</td>
<td>-44.078</td>
<td></td>
</tr>
<tr>
<td>-50.476</td>
<td>-50.476</td>
<td></td>
</tr>
<tr>
<td>-56.874</td>
<td>-56.874</td>
<td></td>
</tr>
<tr>
<td>Elevation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pressure</td>
<td></td>
<td></td>
</tr>
<tr>
<td>45.492</td>
<td>45.492</td>
<td></td>
</tr>
<tr>
<td>39.095</td>
<td>39.095</td>
<td></td>
</tr>
<tr>
<td>32.697</td>
<td>32.697</td>
<td></td>
</tr>
<tr>
<td>26.296</td>
<td>26.296</td>
<td></td>
</tr>
<tr>
<td>19.901</td>
<td>19.901</td>
<td></td>
</tr>
<tr>
<td>13.503</td>
<td>13.503</td>
<td></td>
</tr>
<tr>
<td>7.105</td>
<td>7.105</td>
<td></td>
</tr>
<tr>
<td>0.707</td>
<td>0.707</td>
<td></td>
</tr>
<tr>
<td>-5.691</td>
<td>-5.691</td>
<td></td>
</tr>
<tr>
<td>-12.086</td>
<td>-12.086</td>
<td></td>
</tr>
<tr>
<td>-18.468</td>
<td>-18.468</td>
<td></td>
</tr>
<tr>
<td>-24.864</td>
<td>-24.864</td>
<td></td>
</tr>
<tr>
<td>-31.262</td>
<td>-31.262</td>
<td></td>
</tr>
<tr>
<td>-37.680</td>
<td>-37.680</td>
<td></td>
</tr>
<tr>
<td>-44.078</td>
<td>-44.078</td>
<td></td>
</tr>
<tr>
<td>-50.476</td>
<td>-50.476</td>
<td></td>
</tr>
<tr>
<td>-56.874</td>
<td>-56.874</td>
<td></td>
</tr>
</tbody>
</table>

5.3 Pressure coefficients
External pressure coefficient (C_{pe}) of computational model can be found using the formula
as stated in IS 875 (Part-3): 1987 which is discussed under equation (9).

\[ C_{pe} = \frac{\text{Pressure at any point (pa)}}{0.6V_z^2} \]  \hspace{1cm} (9)

where \( V_z \) is the design wind speed in m/s which is considered as 10 m/s for this experiment.

The pressure at any point on the face has been collected for CFD analysis and \( C_{pe} \) values of that particular point are calculated. The pressure coefficients of different faces around the building peripheries and at the vertical height of the particular faces are measured and discussed under the following articles.

5.3 Horizontal \( C_{pe} \) around the building

Horizontal external pressure coefficient (\( C_{pe} \)) also carried out in this study around the perimeter of the N-plan shape tall building at level H/4=125mm, H/2=250mm and 3H/4=375mm from the base of the building model for 0° and 90° wind angles. Horizontal line around the perimeter of the building has been drawn from the edge point, namely “a” and passes through the a-b-c-d-e-f-g-h-i-j-a. This line is drawn at H/4, H/2 and 3H/4 level as shown below Fig. 6 (a)-(b). Horizontal \( C_{pe} \) at H/4, H/2 and 3H/4 levels for 0° and 90° wind angles are discussed under Figs. 7-8. For 0° wind angle, the positive pressure is maximum in between point a to b for all the levels and the maximum positive pressure increasing towards the higher level. In this region, the maximum positive pressure according to the level of perimeter are \( C_{pe} = 1.2 \) for 3H/4 level, \( C_{pe} = 1.08 \) for H/2 and \( C_{pe} = 0.87 \) for H/4 level. Pressure between points b-c-d-e-f-g-h-i-j-a is negative in nature at all the level like H/4, H/2 and 3H/4. Minimum suction (-0.20) developed at f-g zone at H/4 level, which is the leeward face of the building. For 90° wind, the face B, C, D and E behave like windward face shown in Fig. 8. Therefore the pressure between points b-c-d-e-f are positive. On the other hand, pressures at rest of the faces are negative. Maximum \( C_{pe} \) (1.13) value of positive pressure is located at 3H/4 level in face B and E. Again, maximum negative pressure (-1.55) developed at 3H/4 level, i.e. point b and f. Therefore a remarkable parameter developed here, that the wind pressure are decreasing towards the outer side of the face i.e. face A. Furthermore the pressure on the opposite side, i.e. face C has positive pressure.

![Image of horizontal \( C_{pe} \) for 0° and 90° wind angle at level H/4, H/2 and 3H/4](image-url)
5.4 Vertical $C_{pe}$ for different faces

The graphs shown in Fig. 9 (a)-(j) represents the vertical face pressure coefficient at different faces for $0^\circ$, $30^\circ$, $60^\circ$, $90^\circ$, $120^\circ$, $150^\circ$ and $180^\circ$. The vertical line has been drawn in the middle of the faces which are parallel to the Z axis and from base to top of the building and the pressure coefficients measure of the different wind angles. For face A, angles $0^\circ$, $30^\circ$ and $60^\circ$ are creating positive pressure, which is gradually increasing from the base to the height of the building where $0^\circ$ angle produced maximum pressure and $60^\circ$ have minimum positive pressure. Other angles are negative pressure, where $90^\circ$ angle has maximum negative pressure due to the leeward face A as shown in Fig. 9 (a). Face B represents the windward face for $90^\circ$ angle, but the maximum positive pressure is given by $60^\circ$ wind angle, because wind at this angle doesn’t affected by any side wash, whereas for $90^\circ$ angle positive pressure on face B has been disturbed by turbulence developed between face C and D. Wind angle $120^\circ$ produced minimum positive pressure and other angles like $30^\circ$ and $150^\circ$ has
minimum suction. Maximum suction developed by 0° and 90° angles as the face B is leeward for those two angles as shown in Fig. 9 (b). Face C and D as shown in Fig. 9 (c)-(d) also have positive pressure for 60°, 90° and 120° angles and have maximum suction due to 0° and 180°. Face E have a maximum $C_{pe}$ for 120° and decreasing by 90° to 60°. Maximum suction developed for 0° and 180° angle as shown in Fig. 9 (e). From Fig. 9 (f), the $C_{pe}$ at face F have a similar pattern of pressure as compared with face A. Here wind angle 180°, 150° and 120° has positive pressure, since the face A has positive pressure for 0°, 30° and 60°. Fig. 9 (g)-(j), presents the negative pressure for all the wind angles.
WIND PRESSURE AND VELOCITY PATTERN AROUND ‘N’ PLAN …

[g] 0.0 0.1 0.2 0.3 0.4 0.5 0.6
Building height (m)

(h) 0.0 0.1 0.2 0.3 0.4 0.5 0.6
Building height (m)

(i) 0.0 0.1 0.2 0.3 0.4 0.5 0.6
Building height (m)
A. Mukherjee and A. K. Bairagi

Figure 9. Vertical pressure coefficients of N plan shape tall building at different faces (a) Face A, (b) Face B, (c) Face C, (d) Face D, (e) Face E, (f) Face F, (g) Face G, (h) Face H, (i) Face I, (j) Face J

5.5 Force co-efficient

The force components along direction of wind flow are called drag force and force components in other normal directions are called lift force. The value of drag and lift force are collected from the CFD analysis for N shape model again analytical result of force are estimated from the equation (10). Finally the calculated results of CFD and analytical are developed force coefficient for drag and lift force. The expose area of N shape building is different for the different wind angles. Simiu E. and Scanlan R.H (17) discussed the drag coefficient equation:

\[ F_D(t) = \frac{1}{2} \rho v^2(t) B^2 C_d \]  

(10)

where,

- \( F_D(t) \) = the time varying drag on a body
- \( \rho \) = density of the fluid
- \( v(t) \) = speed of the object relative to the fluid varying with time
- \( B \) = typical body dimension
- \( C_d \) = drag co-efficient

Here force coefficients are discussed and plotted in two different graphs along X and Y direction as \( C_{fx} \) and \( C_{fy} \) respectively, for angles 0°, 30°, 60°, 90°, 120°, 150° and 180°. Fig. 10 (a) present the force coefficients at different wind incidence angles. Where 0° have maximum \( C_{fx} \) (1.04) and 180° has minimum \( C_{fx} \) (-1.03) but for 90° angle have \( C_{fx} \) zero. Fig. 10 (b) highlight the force coefficient along Y axis. The maximum \( C_{fy} \) (1.05) developed for 90° angle and minimum \( C_{fy} \) (0.00) developed for 0° and 180°.
Figure 10. Force coefficients of N plan shape tall building at different wind angles (a) for $C_{fx}$, (b) for $C_{fy}$

5.6 Velocity profile
Velocity profile is the vertical gradient of mean horizontal wind velocity, which is increased with the increase of height. The experiment is carried out at terrain category 2. Velocity profile has been drawn from the square plan shape model and validated with S.P 64 (S&T): 2001[18]. The velocity profile diagram of square plan shape building also shown in Fig. 11. The velocity profile along the X axis has been drawn at certain interval from inlet to outlet for N shape model. The points are namely 1-10 where point 6 and 7 are placed near the corner of the inclined limb to get the proper turbulence near the corner regions. The vertical lines are drawn up to full domain height are shown below Fig. 12(a). The velocity profile from the top of the building (i.e, point 1, 2, 3, 4 and 5) to the domain height as shown in Fig. 12(b) are also calculated in this study.
A. Mukherjee and A. K. Bairagi

Figure 11. Velocity profile at inlet of isolated square plan shape tall building

![Figure 11](image)

Figure 12. (a) Plan view showing the different velocity measuring points from inlet to model and model to outlet. (b) Plan view showing the velocity measuring points at the top of the N plan shape tall building

![Figure 12](image)

Fig. 13(a) focus the graphical representation of velocity profile from inlet towards outlet. Point 1 has been considered at the inlet of the domain and the points 2, 3 and 4 are 1500mm, 2100mm and 2300mm from the inlet point 1 respectively and they are reaching to the model. Point 5 is placed on the face of A. There for the pattern of velocity profile obey the velocity profile as discussed in S.P 64 (S&T): 2001. The arrangement of the velocity profile changed from the base up to the 1m of the domain, which is 2H from base; here H is the height of the building. Point 6 and 7 are attaining the minimum velocity at the corner point of the inclined limb at the level of 0.3m i.e. 0.06H. At the outlet portion of the building, the velocity at 0.06H level is increased from point 8, 9 and 10. The velocity above 1m height (i.e. 2H) at every point is same. Therefore, it is clear that the velocity near the building is decreasing with respect to the inlet of domain up to the 2H height. And above 2H, the velocity doesn’t change for all the velocity tapping points. Whenever the velocity tapping points are shifted from the base to the building top, the velocity at the exact tapping points are slightly fluctuating. But lowest velocity was observed at point 5 which is the canter point of the model. The velocity above 1m is same for all the velocity tapping points.
Figure 13. (a) Velocity profile at the different velocity measuring points from inlet to model and model to outlet. (b) Velocity profile at the top of the N plan shape tall building

6. CONCLUSION

After a number of iterations and burdensome calculation of the N shape model due to wind load on 0° to 180° at 30° intervals are concluded here.

- The large amount of turbulence is developed at the corner of inclined limbs of N shape bluff body due to the 0° wind action, but for 90° angle the turbulence is not detected at that location.
- Horizontal pressure coefficients at windward face for 0° wind angle are increased proportionally with respect to the height of the building. For 90° wind, the positive $C_{pe}$ is the maximum at the corner of face B and E.
- Vertical pressure coefficient at face B is maximum for 60° angle, although the face B behaves windward face for 90°. Because no disturbance was observed in face B for 60° wind, but for 90° angle flow has been interrupted by a wake at corner i.e. intersecting zone of face C and D.
- The same effect has been observed in face E. Here maximum positive pressure developed by wind 120°, whereas this face is performed as the windward face for 90° wind angle.

Velocity fluctuations at different velocity tapping points are observed between 2H of building and minimum velocity develop at 0.06H between the corner of connected face C and D.
REFERENCES