CFD Analysis for wind flow characteristics of varying cross section tall building using ANSYS

Ritu Raj (✉ rituraj@dtu.ac.in)
Delhi Technological University  https://orcid.org/0000-0002-9727-5048

Ark Rukhaiyar
Delhi College of Engineering: Delhi Technological University

Bhagya Jayant
Delhi College of Engineering: Delhi Technological University

Kunal Dahiya
Delhi Technological University

Research Article

Keywords: Tall building, CFD, ANSYS, varying cross-section, pressure contours, streamlines

Posted Date: March 25th, 2022

DOI: https://doi.org/10.21203/rs.3.rs-1471297/v1

License: This work is licensed under a Creative Commons Attribution 4.0 International License.
Read Full License
CFD Analysis for wind flow characteristics of varying cross section tall building using ANSYS

Ark Rukhaiyar\textsuperscript{1a}, Bhagya Jayant\textsuperscript{1a}, Kunal Dahiya \textsuperscript{1a} Ritu Raj \textsuperscript{2,*}

\textsuperscript{1} Student, Department of Civil Engineering, Delhi Technological University, Delhi, India
\textsuperscript{2} Assistant Professor, Department of Civil Engineering, Delhi Technological University, Delhi, India

\textsuperscript{a}) arkrukhaiyar\_2k18ce025@dtu.ac.in, bhagjayant\_2k18ce032@dtu.ac.in, kunaldahiya\_2k18ce052@dtu.ac.in
\textsuperscript{*}) Corresponding Author: rituraj@dtu.ac.in

Abstract. Tall buildings are being developed at rapid rates owing to the need for optimization of space for increasing urban crawl. Tall buildings having simple cross – sections are designed using IS 875 (Part 3) – 2015, however there is a lack of set of codal provisions for complex cross – section shapes. This study has been undertaken to interpret the wind flow characteristics of an isolated tall building (Model A) utilizing CFD (Computational Fluid Dynamics) and \(k-\varepsilon\) turbulence model in ANSYS: CFX (Analysis of Computational Fluid Dynamics program). Model A is 192 m in height having two types of cross sections – plus and square; each of height 96 m. Wind of intensity 10 m/s is incident on it at four incidence wind angles 0°, 30°, 60° and 90°. Pressure contours obtained on simulation indicate that for incident wind angles 0° and 90°, all lateral faces of the Model A (proposed design model) show similar pressure impact. Whereas, for incident wind angles 30° and 60°, wind ward faces (A and C) show decreased pressure impact with respect to 0° case. Vertical streamlines depict a recirculation zone at one – third the height of Model A for incident wind angle, 30°. Whereas, horizontal streamlines indicate the occurrence of vortex formation in the wake region of the building for incident wind angles, 0° and 90°.

Keywords. Tall building, CFD, ANSYS, varying cross – section, pressure contours, streamlines.

1. INTRODUCTION

Designing of tall buildings has found its significance with emerging manhattanization. In regards to the vertical mode of expansion, several revisions have been made on acceptable heights of tall buildings over the time across the world. Dynamic excitations such as winds and earthquakes are key factors responsible for structural failure. In tall buildings, lateral stiffness against wind loads predominates over the structural strength (Connor, 2003). Earlier, symmetrical shapes were necessary for building optimized, serviceable, and stable G+11 designs (Harman, 2017). However, (Moon, 2018) proved that tapered (twisted and irregular) forms in tall buildings produced superior structural performance in terms of resisting static and dynamic responses, saves limited natural resources to be used as building material for a more sustainable and architecturally effective structure satisfying different design requirements. For wind attack angles of 0°, 30° and 60°, V – inverted and V – bracing systems gave comparable values with minimal sway (Raj, 2020). Numerical simulations on ‘E’ (B. Bhattacharyya and S. K. Dalui, 2018), ‘Y’ (P. Sanyal and S. K. Dalui, 2020), ‘H’ (R. Raj, T. Rana, T. Anchalia, and U. Kholo, 2020), ‘+’ (R. Paul and S. K. Dalui, 2020), octagonal (S. Hajra and S. K. Dalui, 2016) and stepped shapes (A. K. Bairagi and S. K. Dalui., 2021) were also performed by various researchers. It was found that mesh size affected the CFD simulations, pressure distribution depended on the height of the model, increase in velocity at the roof top is observed and drag force depended on exposure to wind. However, wind analysis using standard codes of practice provided information for simple shapes like square, rectangle, triangle etc. but lacked in providing exhaustive information on pressure distribution for complex shape designing at skew angles of attack.

Several studies on interpretation of aerodynamic properties on tall buildings concludes that improving the cross – sectional shapes leads to reduced wind – induced vibrations. (Ding, F., & Kareem, A., 2020) used aerodynamic shape tailoring or implementing a dynamic façade to counter the effects of changing wind environment. Their findings offer enormous potential in using AI agents and modern morphing technologies to mitigate wind-induced effects, thus controlling their impact on the performance and safety of civil infrastructures. (Karimimoshaver, M., Hajivaliei, H., Shokri, M., Khalesro, S., Aram, F., & Shamshirband, S., 2020) proposed a model using visual analysis approach to
locate tall buildings based on technical and visual criteria from Fuzzy visibility results. (Fuka, V., Xie, Z. T., Castro, I. P., Hayden, P., Carpentieri, M., & Robins, A. G., 2018) compared the scalar dispersion in an array of uniform height buildings with flow and dispersion in an array with another building three times its height and found that the former showed more sensitivity to small changes in wind direction when the approaching flow faced the longest face of the building. This implied that great care needs to be taken in arranging the wind-tunnel model so as to produce a symmetric flow field. It also implies that long time averaging is necessary in LES to converge to nearly symmetric mean fields. This paper aims to fill the research gap of less investigation on sensitivity analysis for determining wind flow characteristic of varying cross-section tall building (G+64) using ANSYS by adopting a Computational Fluid Dynamics (CFD) approach.

For analysis of wind flow characteristics of tall buildings, wind speed and wind direction are governing factors. (Nicola Longarini, Luigi Cabras, Marco Zucca, Suvash Chapain, Aly Mousaad Aly, 2017), in their study of the impact of wind loads on slender buildings, investigated the strength and design criteria and found the potential of damping enhancement to lessen the design loads and vibrations thereby providing optimal balance among resilience, serviceability, and sustainability requirements. Other methods like gust factor methods and wind tunnel methods are also used to find out the shear and deflection of tall buildings and have conforming results. It was observed that the gust factor method or dynamic-wind method by IS 875 – 2015 provided more factor of safety, however, the wind tunnel gave more accurate results complying closer with actual site conditions (Er. Deshmukh, Pranil, Shiyekar, M.R., 2020).

Wind tunnel testing has high acceptance when designing tall buildings. However, a shift to computational fluid dynamics analysis has been observed for an in-depth study at the early design stage in areas with lower to moderate turbulence. While both methods are moderate to highly accurate, wind tunnel testing takes months to analyse, whereas CFD analysis is faster and cost-effective. It provides direct reporting in comparison to abstract reporting in wind tunnel testing. The wind tunnel is however preferred in studies dealing with higher turbulent winds by resolving unsteady gusts and eddies directly. Another advantage of CFD analysis is it capability to eliminate scaling and probes that may influence the measurements. It provides high resolution and mapping of results in a highly visual and informative way.

Computational fluid dynamics is the analysis of complex situations involving fluid-liquid, fluid-solid, and/or fluid–gas interaction. A numerical computational process is used for the analysis of solid–air interaction in this study where pressure coefficients and streamlines are obtained in ANSYS: CFX mode. It generally involves formulating the flow problem, modelling it, establishing initial and boundary conditions, generating the mesh, establishing simulation strategy, inputting parameters and files, performing the simulation, and monitoring the simulation for completion.

In this study, Computational Fluid Dynamics technique is utilized to analyse G+64 tall building as shown in Fig. 1 having two types of cross-sections – square and plus, dimensions of which are as shown in Fig. 2. The results are interpreted in terms of the pressure coefficient values (from pressure contours) and velocity streamlines obtained for proposed design model (Model A). The model is validated with the help of a reference isolated model, Model X which is also a 192 m height model having only “square” as its cross-section throughout as shown in Fig. 1. Since, IS 875 (Part III) – 2015 gives a range of deflection for pressure coefficient for simple cross-section shapes, therefore, if Model X is in agreement with the code, Model A will also conform consequently.

2. NUMERICAL MODEL DEVELOPMENT

2.1 Geometric Modelling

In the present study, Model A is designed in AutoCAD: 3D Modeling as shown in Fig. 1. The model A is a G+64 building, 192 m in height with two types of cross-sections – square and plus having height of 96 m each as shown in Fig. 2, and is compared with a reference isolated model X, which is as depicted in Fig. 3a and Fig. 3b.
The procedure for analysis using ANSYS is divided into 5 steps – Geometry, Mesh, Setup, Solution and Result. The domain is set in accordance to Indian Standards such that the boundaries of the domain are at a distance of 5h from the windward face (A), lateral faces (C & D), and roof of the model and at a distance of 15h from the leeward face (B) to avoid flow re-circulation (where let h be the height of the model). Meshing for the domain is done using the tetrahedral elements having element size of 0.2 m. Meshing inflation with 15 layers is provided at the boundaries of the Model A and Model X for allowing smooth flow and is represented in Fig. 4.

2.2 Basic Equations

The study aims to interpret the fluid-structure interaction in terms of various contours and streamlines obtained as a result and validate the observations to conform with IS: 875 (Part III) - 2015. It utilizes the research in effect of wind on low-rise buildings and turbulence study using standard k - ε, standard k - ω and SST k – ω models.

The following equations are used in this present study.

2.2.1 Standard k - ε model

K – ε model is a two – equation model of computational fluid dynamics used to replicate flow characteristics for turbulent flow conditions. This model assumes the turbulent viscosity of wind considered as 10 m/s in this study; to be isotropic. Turbulence is less pronounced in this model in comparison to SST k – ω model.
For turbulent kinetic energy,

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_i)}{\partial x_i} = \frac{\partial }{\partial x_j} \left[ \mu_t \frac{\partial k}{\partial x_j} \right] + 2\mu_t E_{ij}E_{ij} - \rho \varepsilon
\]  

(1)

For dissipation,

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial }{\partial x_j} \left[ \mu_t \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t E_{ij}E_{ij} - C_{2\varepsilon} \rho \varepsilon^2
\]

(2)

### 2.2.2 Standard k – ω model

It is a two – equation model, generally used for flows having low reynold’s number used to interpret turbulent flow conditions. For open channel flow problems, k – ω gives best result.

For turbulent kinetic energy,

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial }{\partial x_j} \left[ \Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k
\]

(3)

Specific dissipation rate,

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega u_i)}{\partial x_i} = \frac{\partial }{\partial x_j} \left[ \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega
\]

(4)

### 2.2.3 SST k – ω model

SST acronym for Shear Stress Transport, is used to obtain better flow separation prediction for adverse pressure gradient conditions.

For turbulent kinetic energy

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial }{\partial x_j} \left[ \Gamma_k \frac{\partial k}{\partial x_j} \right] + G'_k - Y'_k + S_k
\]

(5)

Specific dissipation rate,

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega u_i)}{\partial x_i} = \frac{\partial }{\partial x_j} \left[ \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_k
\]

(6)

### 2.3 CFD Pre – simulations

Power law is used for interpreting the results in ANSYS in set – up and solution steps. The inputs for the set – up step (Fig. 5) is as below.

- \( \alpha = 0.147 \)
- \( Z_{ref} = 1 \text{ m} \)
- \( U_{ref} = 10 \text{ m/s} \)
Power law is used for calculation of pressure (Equation 7).

\[ P = U_{ref} \left( \frac{Z^a}{Z_{ref}^a} \right) \]  

(7)

The program is run for about an hour before the results are evaluated. The variable coefficient of pressure is then found out (Equation 8).

\[ C_p = \frac{(P - P_{ref})}{\left( \frac{1}{2} \rho_u V_{ref} \right)} \]  

(8)

\( V_{ref} \) is assumed to be 10 m/s

Thus, value of pressure density comes out to be

\( p_a = 1.225 \text{ kg/m}^3 \)

The wind load effect on Model A is analysed for 17 faces. The faces are named as shown in Fig. 6.
Fig. 6. Representation of front and side faces of Model A

Windward side is that side which is facing the on-coming wind. The side resisting this on-coming wind is called as the Leeward side. Classification of faces is done majorly based on four directions into faces A, B, C and D as the “front faces” as shown in the fig. 6. The remaining faces are classified into subscripts of 1 and 2 of front face as “side faces”. As explained in the Fig. 7, each direction or front face will have 2 side faces. The side face to the left of front face is subscripted to 1 (A₁/B₁/C₁/D₁) while the one to the right is subscripted to 2 (A₂/B₂/C₂/D₂).

Also, for wind analysis in ANSYS further cases for each face will be considered for wind angle incident at 0°, 30°, 60°, and 90° on the x–y plane.

3. VALIDATION

For validating the study, an isolated reference model named Model X was designed and analyzed. Model X was designed such that the entire model was formed with a single square cross-section having dimensions 40 m x 40 m throughout the height of 192 m (fig. 2).

Coefficient of pressure, $C_p$ values for each face (A, B, C, D) classified similarly as discussed above; was got after analysis using ANSYS: CFX mode and is as listed in Table 1 below. It is compared with the acceptable values as given in IS: 875 (Part III) – 2015.

<table>
<thead>
<tr>
<th>Coefficient of Pressure, $C_p$</th>
<th>Faces of Model X</th>
</tr>
</thead>
<tbody>
<tr>
<td>According to IS: 875 (Part III) – 2015</td>
<td>A</td>
</tr>
<tr>
<td>+0.8</td>
<td>-0.25</td>
</tr>
<tr>
<td>Model X</td>
<td>+0.659769</td>
</tr>
</tbody>
</table>

From Table 1, it may be observed that $C_p$ values for faces A, B, C, and D varied by 17.528%, 11.162%, 18.558% and 18.55875% respectively.
4. RESULTS AND DISCUSSION

4.1 Pressure contours

Contours are used to indicate the change in constant pressure values over a surface. A contour plot with color bands which have discrete coloured regions while the display of a variable on a locator (such as a boundary) shows a finer range of colour detail by default. The following observations of pressure contours for all faces are listed and classified into different cases as below. The cases are classified into 1, 2, 3, and 4 on the basis of incident wind angle – 0°, 30°, 60°, and 90°. Fig. 8 reiterates the classification of faces of Model A. The face which is left to the front face is subscripted to 1 and the other on the right is subscripted to 2.

The following pressure contours are evaluated using ANSYS for Model A and Model X. Note that for side faces, left contours are for ‘1’ subscripted side face, while right contours are for ‘2’.

CASE 1 – Incident wind angle is 0°

![Pressure contours for Model A and Model X](image-url)
**CASE 2** – Incident wind angle is 30°

- Face A front
- Face B front
- Face C front
- Face D front
- Top view (Roof)

**CASE 3** – Incident wind angle is 60°

- Face A front
- Face B front
- Face C front
- Face D front
- Top view (Roof)
CASE 4 – Incident wind angle is 90°
Model X – From Model X, pressure contours were obtained for wind incident angle 0°.

Face A front          Face B front          Face C front          Face D front          Top view (Roof)

The following conclusions are drawn from the pressure contours obtained:

- Variation in $C_p$ range is highest in case 3 for incident wind angle 60°. Therefore, most wind effect on Model A is observed in this case. Whereas, least variation in $C_p$ range is observed for incident wind angle 90°. Therefore, least wind effect on Model A is observed in this case.
- For wind direction <= 30° on the horizontal plane, $C_p$ values of faces C & D were almost similar and $C_p$ variation across these faces were similar too. Therefore, the sides of the Model A experience similar pressure impact.
- For wind direction >= 30° on the horizontal plane, contours of plus cross section were symmetrical (except for Face A and its side faces where slight disturbances were observed).
- Also $C_p$ range for Model X is comparable to Model A. Value of $C_p$ for roof faces is constant over its entire area, although slight variation was observed with change in wind direction.
- For 0° incident wind angle on Model A, value of $C_p$ lies in the range $\epsilon [-0.86084, 0.666623]$. Maximum negative and positive coefficients of pressure of -0.86084 acted on the face B$_2$ and 0.666623 acted on the face D$_1$ respectively. Therefore, faces B$_2$ & D$_1$ are the critical faces in this case.
- For 30° incident wind angle on Model A, value of $C_p$ lies in the range $\epsilon [-1.10667, 1.14674]$. Maximum negative and positive coefficients of pressure of -1.10667 acted on the face B$_2$ and 1.14674 acted on the face C$_2$ respectively. Therefore, faces B$_2$ & C$_2$ are the critical faces in this case. The range of $C_p$ has increased in this case in comparison to 0° case.
- For 60° incident wind angle on Model A, value of $C_p$ lies in the range $\epsilon [-1.03526, 1.1391]$. Maximum negative and positive coefficients of pressure of -1.03526 acted on the face D$_1$ and 1.1391 acted on the face A$_1$ respectively. Therefore, faces D$_1$ & A$_1$ are the critical faces in this case. The range of $C_p$ has increased in this case in comparison to 0° case.
- For 90° incident wind angle on Model A, value of $C_p$ lies in the range $\epsilon [-0.85415, 0.642927]$. Maximum negative and positive coefficients of pressure of -0.85415 acted on the face D$_1$ and 0.642927 acted on the face B$_2$. Therefore, faces D$_1$ & B$_2$ are the critical faces in this case. The range of $C_p$ in this case is similar to 0° case.
- For Model X, value of $C_p$ lies in the range $\epsilon [-0.734214, 0.623015]$. Maximum negative and positive coefficients of pressure of -0.734214 acted on the face D and 0.623015 acted on the face A respectively. The range of $C_p$ for Model X is similar to Model A (0° case).

4.2 Streamlines

Wind flow behaves as aero elastic and is highly complex when passed through permeable claddings or nets and the pressure coefficients for impermeable and permeable nets were found to be in proportion to the aerodynamic resistance of permeable nets (Agarwal, Ashish & Irtaza, Hassan, 2021). However, cladding failure is of much more significance.
in low-rise building designing, in comparison to the designing for sustained wind loads in high-rise buildings (Mehta, 1984).

The wind in the present study is taken to be a fluid body. Streamline is a path of imaginary particles suspended in the fluid and carried along with it. The vertical and horizontal streamlines are evaluated and represented in the following sub-sections.

4.2.1 Vertical streamlines

**Case 1 – Incident wind angle is 0°**

**Case 2 – Incident wind angle is 30°**

**Case 3 – Incident wind angle is 60°**

**Case 4 – Incident wind angle is 90°**

The following conclusions are drawn from the vertical streamlines obtained.

- For 0° incident wind angle on Model A, it is observed that wind direction reverses in the wake region, creating a separation zone. Increased velocity is observed at the top most point of the building or Model A from where the streamlines change direction. It is observed that the streamlines are symmetrical and therefore will have minimum wind effect on structure.
- For 30° incident wind angle on Model A, leeward recirculation zone or vortex formation is observed at some distance from the building or model A. Flow reattachment is observed at the leeward side of the building or Model A. Increased velocity is observed at the top – most point of the building or Model A from where the streamlines change direction.
- For 60° incident wind angle on Model A, flow separation is observed at the wake region. Fewer flow lines are observed. Increased velocity is observed at the top – most point of building or Model A from where the streamlines change direction.
- For 90° incident wind angle on Model A, formation of re-circulation zone at the leeward side of the building. Flow separation in the wake region, especially in the leeward side of the plus cross – section of the building is observed. Fewer flow lines are observed. Increased velocity is observed at the top – most point of building or Model A from where the streamlines change direction.
4.2.2 Horizontal streamlines

**Case 1** – Incident wind angle is 0°
**Case 2** – Incident wind angle is 30°

For Model X, horizontal streamlines were obtained for wind flow at four different heights (incident wind angle 0°).

**Case 1** – Wind flow at 192 m  
**Case 2** – Wind flow at 144 m
Case 3 – Wind flow at 96 m

The following conclusions are drawn from the horizontal streamlines obtained.

- For 0° incident wind angle on Model A, two vortices or recirculation zones are formed symmetrical on both its sides near faces $C_1$ & $B_2$ and $B_1$ & $D_2$.
- For 30° incident wind angle on Model A, symmetrical streamlines are observed indicating minimum wind effect on the structure.
- For 60° incident wind angle on Model A, vortex or recirculation zone is formed near face $D$ (leeward face). This region is not good for providing openings for windows, etc.
- For 90° incident wind angle on Model A, vortex or recirculation zone formation is observed near the faces $C_1$ & $D_2$. This region is not good for providing openings for windows, etc.
- For Model X, symmetrical horizontal streamlines are observed with minimal turbulence and no recirculation zone nearby, indicating ideal cross-section shape for tall buildings. It is safely designed and

4.3. EXTERNAL PRESSURE COEFFICIENT

The positive value of $C_p$ indicates that the face is in the windward region, whereas the negative value indicates that the face is in the leeward region. The following bar charts are plotted using the results got after analysis in ANSYS for Model A classified similarly as above on the basis of incident wind angles.

Case 1 – Incident wind angle is 0°

**Fig. 7.** Bar chart depicting variation in $C_p$ for incident wind angle 0°
Case 2 – Incident wind angle is 30°

Fig. 8. Bar chart depicting variation in $C_p$ for incident wind angle 30°

Case 3 – Incident wind angle is 60°

Fig. 9. Bar chart depicting variation in $C_p$ for incident wind angle 60°
Case 4 – Incident wind angle is 90°

From the above bar charts, the following conclusions are drawn.

- For Case 1 – Incident wind angle is 0°, the highest negative value of \( C_p \) is -0.9 (rounded – off) on face \( B_2 \) and the highest positive value of \( C_p \) is 0.67 (rounded – off) on face \( D_1 \). Independent groups of side faces (\( A_1 & A_2, B_1 & B_2, C_1 & C_2, D_1 & D_2 \)) have similar pressure impact. Faces A, B, C, D have almost equal value of coefficient of pressure, \( C_p \approx 0.6 \).

- For Case 2 – Incident wind angle is 30°, the highest negative value of \( C_p \) is -1.1 (rounded – off) at face \( B_2 \) and the highest positive value of \( C_p \) is 1.15 (rounded – off) at face \( C_2 \). \( C_p \) values for side faces \( A_1 & A_2 \) and \( C_1 & C_2 \) have high variation in comparison to side faces \( B_1 & B_2 \) and \( D_1 & D_2 \) which show low variation in values. \( C_p \) (faces \( B & D \)) \( \approx -0.6 \). With respect to 0° case, values of \( C_p \) for faces A & C decreased and for faces B & D increased very slightly (negligible).

- For Case 3 – Incident wind angle is 60°, the highest negative value of \( C_p \) is -1.03526 at face \( D_1 \) and the highest positive value of \( C_p \) is 1.139 at face \( A_1 \). \( C_p \) values for side faces \( B_1 & B_2 \) have equal pressure impact with \( C_p \approx 0.5 \). Whereas, \( C_1 & C_2 \) show high variation in comparison to side faces \( A_1 & A_2 \) and \( D_1 & D_2 \) which show medium variation in \( C_p \) values. \( C_p \) (faces \( B & D \)) \( \approx -0.57 \). \( C_p \) values of faces A & C obtained in this case is equal to \( C_p \) values of C & A respectively obtained in Case 2 (for incident wind angle 30°). With respect to 0° case, values of \( C_p \) for faces A & C decreased and for faces B & D increased very slightly (negligible).

- For Case 4 – Incident wind angle is 90°, the highest negative value of \( C_p \) is -0.854 at face \( D_1 \) and the highest positive value of \( C_p \) is 0.6304 at face \( A_1 \). Independent side faces (\( A_1 & A_2, B_1 & B_2, C_1 & C_2, D_1 & D_2 \)) have similar pressure impact. Faces A, B, C, D have almost equal value of coefficient of pressure, \( C_p \approx 0.6 \).

5. CONCLUSIONS

Wind is a phenomenon of great complexity because of the many flow situations arising from the interaction of wind with structures. Wind is composed of a multitude of eddies of varying sizes and rotational characteristics carried along in a general stream of air moving relative to earth’s surface. These eddies give wind its gutsy and turbulent character. The gustiness of strong winds in the lower levels of the atmosphere largely arises from interaction with surface features. The average wind speed over a time period of the order of ten minutes or more, tends to increase with height, while the gutsiness tends to decrease with height.

Complex nature of wind is due to its turbulent and gutsy nature that gives rise to many flow conditions upon interaction with tall buildings. Therefore, the study of wind flow characteristics on tall buildings is necessary for investigating the behavior these slender flexible structures. The following conclusions may be drawn from this paper:
International standards are used for analysing wind effects on tall buildings having simple shapes as cross-section only. Therefore, in this study complex shapes (plus) for varying cross-section tall building (Model A) has been analysed by adopting CFD analysis and k-ε turbulence model in ANSYS: CFX mode for interpreting the wind flow characteristics.

As discussed in the validation section, the $C_p$ values obtained for Model X were in agreement with those given by IS: 875 (Part III) – 2015. Therefore, Model A is also validated by consequence.

Wind flow characteristics were determined for all faces of Model A in terms of pressure contours and vertical and horizontal streamlines for four incident wind angles 0°, 30°, 60°, and 90°. Change in pressure behavior is observed at different faces of Model A due to change in incident wind angle. It is observed that for wind incident angle 0° and 90°, faces A, B, C and D have approximately equal pressure impact ($C_p \approx 0.6$). However, $C_p$ values for faces A and C decreased for incident wind angle 30° and 60°.

Changing cross-sections have significant influence in the stability of the structure. From the study, it was observed that recirculation zone was formed at the leeward side of Model A at one-third of its height for incident wind angle 30°. Windward faces showed positive values of coefficient of pressure. It was also observed that $C_p$ values for faces A & C got interchanged for incident wind angles - 30° and 60°.

Use of CFD for wind analysis in tall buildings is better than other popular methods like wind tunnel testing as it is less time-consuming and cost-effective.

**FUTURE SCOPE OF THE PAPER**

The future scope of the paper is to vary the heights of the cross-sections in the proposed model of this paper (Model A) and check for the dynamic response of wind on it. Also, the results generated would be inputted in ETABs to compare the predicted results with available international standards.

**AVAILABILITY OF DATA AND MATERIALS**

All data, models and results generated or used during the study appear in the submitted paper.

**COMPETING INTERESTS**

Authors has no competing interests.

**FUNDING**

No funding received for this study.

**AUTHORS CONTRIBUTION**

Conceptualization, Kunal Dahiya and Ritu Raj; methodology, Ritu Raj, Ark Rukhaiyar, and Bhagya Jayant; software, Kunal Dahiya and Ark Rukhaiyar; validation, Ritu Raj, formal analysis, Kunal Dahiya Saroj; investigation, Ark Rukhaiyar; Bhagya Jayant and Kunal Dahiya; resources, Ritu Raj; data curation, Ritu Raj; writing—original draft preparation, Bhagya Jayant; writing—review and editing, Bhagya Jayant and Ritu Raj; visualization Ritu Raj; supervision, Ritu Raj; project administration, Ritu Raj; funding acquisition, Ritu Raj.

**ACKNOWLEDGEMENT**

Authors would like to convey the exceptional thanks and appreciation to our Institute DTU, Delhi for providing resources to the authors to submit the paper within the allowable time.
REFERENCES


APPENDIX

Nomenclature of terms used in study of flow characteristics.

\( \rho \) – density of air
\( F \) – body force per unit volume
\( \mu \) – dynamic viscosity of air
\( \mu_t \) – eddy or turbulent viscosity
\( \epsilon \) – dissipation rate of \( k \)
\( k \) – von karman’s constant
\( T_{ij} \) – turbulent stress tensor
\( Z \) – height above the ground
\( v_z \) – wind speed at any height
\( P \) – pressure at the point
\( U_H \) – wind speed at reference height
\( C_M \) – base moment coefficient
\( k \) – turbulent kinetic energy
CFD – Computational Fluid Dynamics
atm – atmospheric
\( u \) – filtered scale velocity field
\( v_0 \) – mean wind speed at reference height
\( P_0 \) – static pressure at reference height
FC – correction function of eddy viscosity
\( u,v,w \) fluctuating wind in different directions
\( C_{1\epsilon}, C_{2\epsilon}, C_{k} \) – turbulence model constants
\( i \) – general values of \( u, v, w \) component at point
\( \sigma_{fy} \) – standard deviation of life coefficient
variation at time \( t \)
\( x, y, z \) system of rectangular Cartesian coordinates
\( P \) = buoyant production term
\( P^* \) – modified pressure
\( t \) – time step
\( P \) – pressure
\( z_0 \) – reference height
\( n \) – power law index
\( K \) – energy of turbulent fluctuations
\( P_k \) – turbulence due to mean velocity
\( P_{kb} \) – turbulence due to buoyancy
PSD – Power Spectral Density
\( S(n) \) – PSD for life coefficient
\( A_p \) – area projected
\( \beta, \beta^*, \sigma, \sigma^* \) – closure coefficients
\( \sigma_e \) – turbulence Schmidt number
\( K^2 \) – fluctuation energy of velocity components normal to wall
\( Re \) – Reynolds number
\( u_i u_j \) – Reynolds stress
SST – Shear Stress Transport
\( C_f \) – force coefficient
\( C_p \) – pressure coefficient