International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

Wind Load Response Improvement of Multistorey Tall Building by Aerodynamic Modification

MS. Rutuja Babar
Sinhgad college of Engineering
Solapur, India

Prof. S. B. Tanawade
Sinhgad college of Engineering
Solapur, India

Abstract - The rapid increase in the construction of tall buildings has made wind load effects one of the most critical factors in structural design. Excessive wind-induced vibrations can lead to structural instability, occupant discomfort, and increased construction costs. This study focuses on the improvement of wind load response in tall buildings through various aerodynamic modifications. Different building configurations such as setback structures, corner cuttings, chamfered edges, tapered forms, and openings are considered to evaluate their effectiveness in reducing wind effects. The analysis is carried out using structural modelling and wind load analysis software to compare parameters such as storey displacement, storey drift, base shear, and lateral deflection under wind loading conditions. The results indicate that aerodynamic modifications significantly reduce wind pressure and vortex shedding effects, thereby improving the overall stability and performance of tall buildings. Among the studied configurations, modified shapes with tapered and chamfered profiles showed better resistance to lateral wind forces compared to regular building forms. The study concludes that aerodynamic shaping is an efficient and economical method for enhancing the wind performance of tall buildings without major increases in structural weight. These modifications not only improve structural safety but also enhance occupant comfort and serviceability. The findings of this study can be useful for the planning and design of sustainable and wind-resistant high-rise structures in urban areas. Among the considered configurations, tapered and chamfered buildings demonstrate superior performance in controlling lateral displacement and storey drift compared to conventional rectangular buildings. Buildings with corner modifications and openings also show noticeable reduction in wind pressure and vibration effects. The study highlights that aerodynamic optimization can effectively improve structural efficiency without significantly increasing construction materials or cost. The findings of this research emphasize the importance of integrating aerodynamic considerations during the planning and design stage of tall buildings. The use of aerodynamic modifications not only improves structural safety and occupant comfort but also contributes to economical and sustainable high-rise construction. This study can serve as a useful reference for structural engineers, architects, and researchers involved in the design of wind resistant tall buildings in urban environments.

Keywords - aerodynamic modifications,structural modelling

INTRODUCTION

Tall buildings are designed not only to resist gravity loads but also to withstand lateral loads caused by wind, earthquakes, and other environmental forces. Among these, wind load plays a significant role in the design of high-rise structures due to the increase in building height and slenderness. Wind effects on tall buildings can generally be classified into three categories: static effects, dynamic effects, and aerodynamic effects. Static effects refer to the response of a structure under steady wind forces without considering time variation. Dynamic effects consider the behavior of the structure with respect to time, accounting for vibrations and oscillations induced by fluctuating wind forces. When a building becomes highly flexible, its interaction with wind alters the flow pattern around the structure, leading to aerodynamic effects. The response of tall buildings under wind loading depends on several factors such as structural stiffness, mass distribution, damping characteristics, and geometrical configuration. Excessive wind-induced motion can affect structural safety, serviceability, and occupant comfort. Therefore, it is essential to minimize these effects through proper design approaches. One effective method is the use of aerodynamic modifications, which improve the wind performance of buildings by altering their shape and reducing wind pressure and vortex shedding. These modifications are generally classified into macro modifications, such as tapering, setbacks, and twisting, and micro modifications, such as corner chamfering, openings, and rounded edges. In addition to aerodynamic improvements, the selection of efficient structural systems also plays an important role in controlling wind response. Structural systems with higher stiffness and better energy dissipation capacity help reduce lateral displacement and vibrations. Hence, a combination of aerodynamic shaping and efficient structural design provides an effective solution for enhancing the stability and performance of tall buildings subjected to wind loads.

LITERATURE REVIEW

Peter Irwin," Effects of Geometry on the Wind Response of Super-tall Towers", ASCE (structure 2008):

Peter Irwin works on tall buildings under wind loading. He mentioned that as buildings goes higher and higher wind becomes dominating factor for structural design for strength as well as serviceability criterias. According to him across wind forces are more important than the along wind forces. We resist the forces by the making building stiffer by adding extra members but the index citation is product of wind building interaction. Therefore, building shape becomes key factor. And he also added that the dominating form of that excitation is vortex shedding most of the times. Vortex shedding can be eliminated or reduced by modifying the shape

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

of building is known as aerodynamic modification. Aerodynamic modification only disturbs the vortex shedding formation and helps to reduce or eliminate the vortex shedding in a very effective way. Aerodynamic modifications such as tapering, change in cross section area, inserting voids or openings, providing spoilers, chamfering the corners, etc. can be used in tall buildings. If shape of building is decided with concern of aerodynamic modification, then it will help to save cost. Vortex shedding has a frequency and if buildings natural frequency matches the vortex shedding frequency, then resonance will happen. Resonance brings maximum deflection and it can be hampering the building badly. To avoid that resonance frequencies must be separated for safe margin or eliminate the vortex shedding. In case of softening the corner by 10% of width of building. Which will give satisfactory results. In case of varying cross section shape circular shape cross section on rectangular shape cross section
can be use and get satisfactory results. Spoilers are provided at outer side of building which disturbs the formation of the vortex shedding. Also, openings are used to disturb the formation of vortex shedding by allowing the wind through the building itself.

Yukio Tamura, Hideyuki Tanaka, Kazuo Ohtake,Masayoshi Nakai, Yongchul Kim,” Aerodynamic Characteristics of Tall Building Models with Various Unconventional Configurations”, ASCE (2010 structures congress):
Yukio Tamura1, Hideyuki Tanaka2, Kazuo Ohtake3, Masayoshi Nakai4, Yongchul Kim5 carryout the testing in wind tunnel on different aerodynamically modified building keeping rectangular building as a reference. Generally tall buildings configuration kept symmetric for avoid complicacy but due to advancement of analysis technique based on computer aids the freestyle in configuring is increased.31 tall building models analyzed in boundary layer wind tunnel with aerodynamically modified shapes such as square plan, rectangular plan, elliptic plan, with corner cut, with corner chamfered, tilted, tapered, inverse tapered, with setbacks, helical, openings, etc. All the models are having 400m ht. and 2500sq.m base area. For rectangular model side ratio was1:2. For the corner modification models, Chamfered and Corner cut, the modification length was 10% of the building width, B. The top floor of the Tilted model was shifted by 2B from the base floor, and for the Shaking models, the floors at 0.25H and 0.75H were shifted by 0.5B to the left and right side, respectively, from the middle floor with a smoothly curved surface. For the Tapered models, five structural shapes were employed. The 2-Tapered model had only two tapered surfaces and the 4-Tapered model had
four tapered surfaces.
The area ratio between top and base floor was 1:6. The Inverse 4-Tapered model had the inverse structural shape of the 4-Tapered model, and for the Drum model, the middle floor area was 300% larger than the top and base floor areas. The Setback model had a 3-step setback with an area ratio between top and base floor of 1:6, as for the 4-Tapered model. helical models with identical floor areas were also investigated. The sectional shapes of the Helical models were square, rectangular and elliptical, and the twist angle was used as a prefix of the model’s name. For example, the 180o Helical Square model means the twist angle was 180o and the sectional shape was square. To identify the effect of twist angle on aerodynamic force characteristics, models with twist angles of 90o, 270o, and 360oand square sectional shape were also used. For the Void models, openings were provided at the top center and top-corner of the surface, labeled Cross void and Oblique void, respectively. Various opening sizes of 2/24H， 5/24H, and 11/24Hmade it possible to clarify the effect of opening size on the aerodynamic force characteristics. A 3-Circle model consisting of three circular cylinders was also classified as a Void model. The Combination models had the combined structural and sectional shapes described above, and the Cross-section change & 180o Helical had various structural and sectional shapes, whose section changed successively from circular to elliptical to circular to elliptical to circular, having a twist angle of 180o .The structural properties of the test models (except the Void models) were assumed to be the same, having identical heights and volumes. Wind analysis of the all models carried out in wind tunnel for wind coefficient in across wind and along wind. For the measurement of aerodynamic force wind tunnels working section is kept as 1.8m high and 2m wide. Fluctuating wind forces were measured
with a 6-component load cell located at the bottom of the models, and the models for aerodynamic force measurements were made of balsa or chemical wood as stiff as possible (length scale 1/1000). The measurements were made at wind direction intervals of 5º. Wind pressure measurements were made for 9 models, and were determined considering the results of the aerodynamic force measurements. The selected models were Square, Circle, Chamfered, Corner cut, 4-Tapered, Setback, 90o Helical Square, 180o Helical Square, and Cross void (l/H=5/24). The coordinate system and flow conditions were the same as those for the aerodynamic force measurement, except that the wind speed at the model height was 11.8m/s. The wind pressure measurements were made from 0oto 355oat wind direction intervals of 5o. It has seen that wind force coefficients for the Setback model were the smallest among the models tested, the response analyses show that the acceleration of the Setback model was larger than that of the square model, thus showing worse habitability than the square model. helical models showed better results than the square model based on both safety and habitability criteria.
Islam Abohela, Neveen Hamza, Steven Dudek, Validating CFD Simulation Results:
Wind flow around a surface mounted cube in a turbulent channel flow, PLEA2012 -28th Conference: Abdollah Baghaei Daemei a, ,Elham Mehrinejad Khotbehsara b, Erfan Malekian Nobarani c
, Payam Bahrami d believes in case of modern tall buildings wind is dominating load. So, wind load must be considered very carefully. In their work they have introduced the theoretical framework and simultaneously express basic aerodynamic studies. Then tried to reduce drag coefficient performance of aerodynamic modification such as chamfered, rounded, a…

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

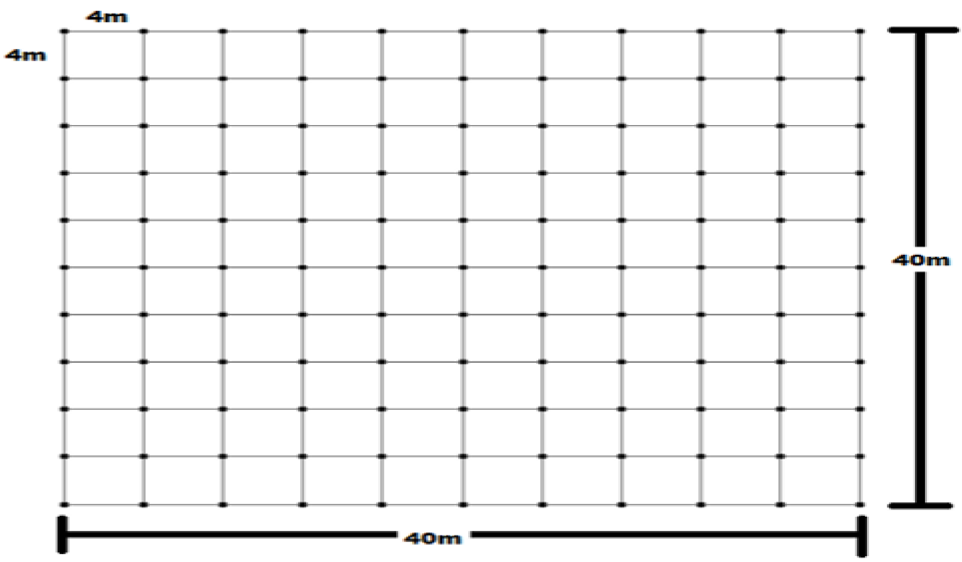
The proportions of the models are B/H = 1/6. Wind drag coefficient were found out. The results show that aerodynamic modification of rounded-corners, tapered are capable to cause a reduction in the drag coefficient of the building by 66% and 24%, respectively than the simple triangular building. Moreover, the technique of aerodynamic modification is approximately 74% more efficient to meet wind effect than aerodynamic form techniques. Authors concluded that the results lead us to the conclusion that setback aerodynamic modification and tapered aerodynamic modification should be used to design a tall building with a triangular section because they show 20% to 25% reduction in drag coeffient.

METHODOLOGY

Modeling:

Modeling is done on Ansys design modeler. Total 17 models were created throughout this dissertation work.

.1 Steps to model a rectangular building: 1) First step was to select a plane (XZ)on which we were going to draw base and then go for sketching. 2) In step just draw a rectangle of size 40mX40m by using rectangular draw tool as well as dimension to give proper dimensions. 3) In this step 2D sketch was extruded to 3D by using extrude tool with providing the height about as 180m (on Y axis) and then clicked on generate. 4) The enclosure of 340mX220mX240m were provided 5) Then symmetry was provided to the model to reduce the time of simulation by 1 symmetric plane longitudinally to the direction of flow of wind. 6) Named selection were assigned in this step as inlet, outlet and sides etc. This named selection very useful at the time of assigning boundary conditions. 7) At last, subtract boolean was added to make building as bluff body



Rectangular building layout

G+40 Steps to model 5% and 10% chamfered buildings: 1) This Chamfered building is modeled by only chamfering the rectangular model by corresponding percentage.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

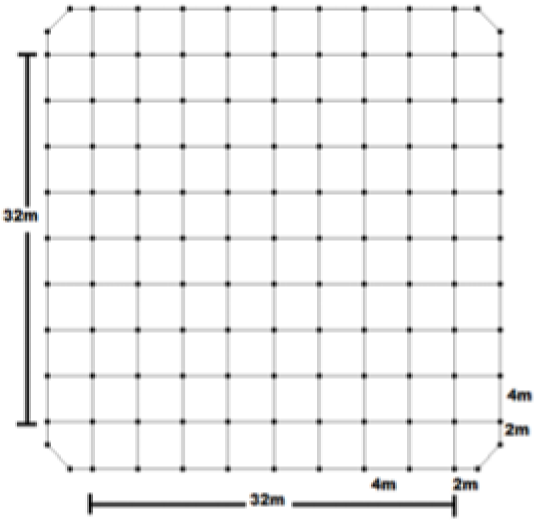


Figure 4.2: 5% Chamfered building layout

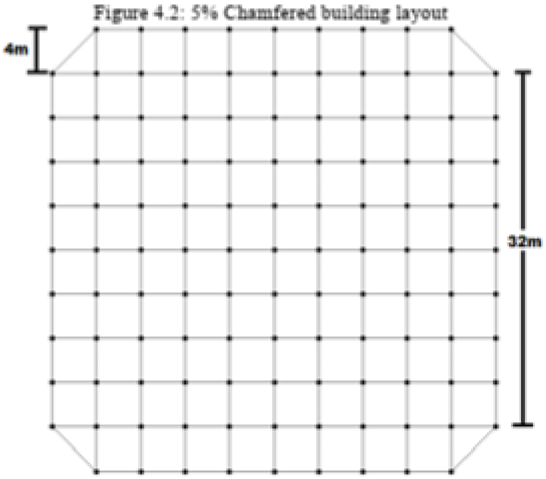


Figure 4.3: 10% Chamfered building layout

G+40 Steps to model 5% and 10% corner cut buildings: 1) This Corner cut building is modeled by only corner cutting the rectangular model by corresponding percentage.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

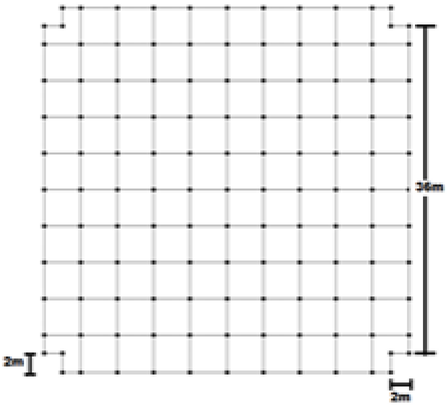


Figure 4.4: 5% Corner cut building layout

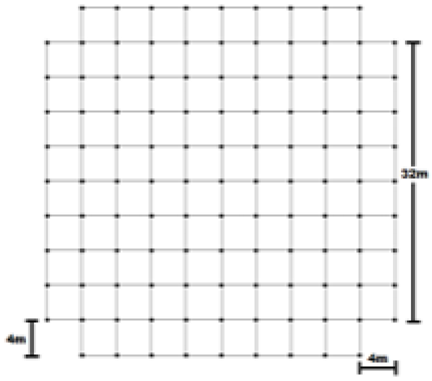
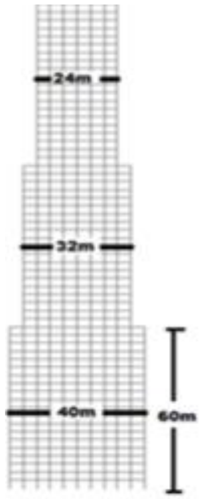
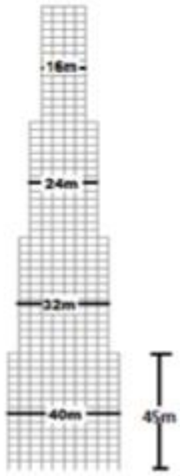


Figure4.5: 10% Corner cut building layout

Steps to model 2 stage and 3stage setback buildings: 1) First step was to select a plane (XZ) on which we were going to draw base and then go for sketching. 2) In step just draw a rectangle of size 40mX40m by using rectangular draw tool as well as dimension to give proper dimensions. 3) In this step 2D sketch was extruded to 3D by using extrude tool with providing the height (on Y axis) about as 60m for 2 stage and 45m for the 3stage then clicked on generate. 4) Then before heading to second stage model had exported as .STEP file and then imported for making the use of same plane many times for different heights. 5) The enclosure of 340mX220mX240m were provided 6) Then symmetry was provided to the model to reduce the time of simulation by 1 symmetric plane longitudinally to the direction of flow of wind. 7) Named selection were assigned in this step as inlet, outlet and sides etc. This named selection very useful at the time of assigning boundary conditions. 8) At last, subtract boolean was added to make building as bluff body.

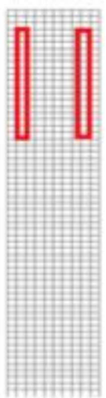


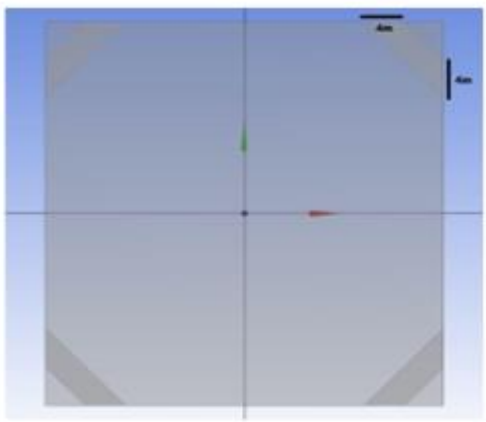
Elevation of 2stage setback building layout



G+40 Elevation of 3stage setback building layout

Steps to model inclined openings buildings: 1) First step was to select a plane (XY)on which we were going to draw base and then go for sketching. 2) In step just draw a rectangle of size 40mX180m by using rectangular draw tool as well as dimension to give proper dimensions. 3) Then the rectangles of same size (4mX3m) as that openings were drown. 4) In this step 2D sketch was extruded to 3D by using extrude tool with providing the height about as 40m (on Z axis) and then clicked on generate. 5) The enclosure of 340mX220mX240m were provided 6) Then symmetry was provided to the model to reduce the time of simulation by 1 symmetric plane longitudinally to the direction of flow of wind. 7) Named selection were assigned in this step as inlet, outlet and sides etc. This named selection very useful at the time of assigning boundary conditions. 8) At last, subtract boolean was added to make building as bluff body.





l: Elevation of building with openings Figure 3.9: Layout building with openings

Steps to model chamfering and setback combined buildings:

1) First step was to select a plane (XZ) on which we were going to draw base and then go for sketching. 2) In step just draw a rectangle of size 40mX40m by using rectangular draw tool as well as dimension to give proper dimensions 3) Then using modified tool this 2D drawing was chamfered 5% and 10% (i.e., in this case for 5% it is 2m and for 10% it is 4m) in respective models. 4) In this step 2D sketch was extruded to 3D by using extrude tool with providing the height (on Y axis) about as 60m for 2 stage and 45m for the 3stage then clicked on generate. 5) Then before heading to second stage model had exported as .STEP file and then imported for making the use of same plane many times for different heights. 6) The enclosure of 340mX220mX240m were provided 7) Then symmetry was provided to the model to reduce the time of simulation by 1 symmetric plane longitudinally to the direction of flow of wind. 8) Named selection were assigned in this step as inlet, outlet and sides etc. This named selection very useful at the time of assigning boundary conditions. 9) At last, subtract boolean was added to make building as bluff body. 4.1.7 Steps to model corner cut and setback combined buildings: 1) First step was to select a plane (XZ) on which we were going to draw base and then go for sketching. 2) In step just draw a rectangle of size 40mX40m by using rectangular draw tool as well as dimension to give proper dimensions Methodology 3) Then using modified tool this 2D drawing was corner cut 5% and 10% (i.e., in this case for 5% it is 2m and for 10% it is 4m) in respective models. 4) In this step 2D sketch was extruded to 3D by using extrude tool with providing

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

the height (on Y axis) about as 60m for 2 stage and 45m for the 3stage then clicked on generate. 5) Then before heading to second stage model had exported as .STEP file and then imported for making the use of same plane many times for different heights. 6) The enclosure of 340mX220mX240m were provided 7) Then symmetry was provided to the model to reduce the time of simulation by 1 symmetric plane longitudinally to the direction of flow of wind. 8) Named selection were assigned in this step as inlet, outlet and sides etc. This named selection very useful at the time of assigning boundary conditions. 9) At last, subtract boolean was added to make building as bluff body.

Meshing:

• Meshing is done in Ansys Mesher. • Tetrahedral mesh is assigned for the whole model. • Mesh is of size 4m. Quality of mesh: Mesh quality is checked by mesh matrix in mesher by 3 methods 1) Element quality 2) Skewness 3) Jacobian ratio Element Quality: In case of Element quality the element quality 1 is ideal however for practical purpose, the value close to 1 is acceptable.

Quality

C = *[volume [□ (Edgelength])] 2 3

C=124.707

Building Model	Element Quality
Rectangular	0.84
2-Stage setback	0.84
3-Stage setback	0.84
2-Opening	0.84
4-Opening	0.84

4.2.1 **Skewness ratio:**
A Skewness ratio is the measure of skewness of actual element with the ideal.

$$Skewness = \frac{optimal\_cellsize - cellsize}{optimal\_cellsize}$$

Skewness ratio: A Skewness ratio is the measure of skewness of actual element with the ideal. Skewness = _ optimal cellsize cellsize − optimal cellsize

In case of skewness ratio as per Ansys user manual skewness ratio 1 is ideally worst but for practical model max value of skewness ratio should not be greater than 0.9

Model	Max. Skewness Ratio
Rectangular	0.84
2-Stage setback	0.83
3-Stage setback	0.83
2-Opening	0.83
4-Opening	0.84

Jacobian ratio: • In Jacobian ratio software select nodes at certain locations, mostly it selects nodes at corners or at curved portions. • At each location determinant of jacobian matrix is computed and it is called as Rj. • Rj at given location represents the magnitude of mapping function between element natural co ordinates and real space.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

Figure 4.10: Mapping function

In case of jacobian ratio as per Ansys user manual Jacobian ratio 1 is ideally best.

Model	Jacobian Ratio
Rectangular	1
2-Stage setback	1
3-Stage setback	1
2-Opening	1
4-Opening	1

Fluent Setup:

Process of solver: Turbulent flow is decomposed by Reynolds decomposition of Navier-Stokes equation into average part and fluctuation part and it is solved by applying following equations 1) Conservation of mass 2) Conservation of momentum 3) Conservation of energy 4) Ideal gas law 5) & 6) Equations from Turbulence model K-epsilon or K-omega turbulence models can be used to find that turbulent or fluctuating part. Turbulence model helps to find out fluctuating term and add two different independent equations to solve the unknowns. All the unknowns are find out by solving set equations by using fluent solver. Turbulance model: 1) K-omega Model

Methodology

2) K-epsilon model These are two models we can use at a time of solving in Ansys fluent. K=Turbulence Kinetic energy ; Omega = Specific dissipation rate ; epsilon= Turbulent dissipation ➢ Turbulence model helps to find out fluctuating terms. ➢ When use one of the turbulence model then we gets two more equations, equation of ‘k’ and equation of ‘epsilon’ or ‘omega’ Boundary conditions : 1)Inlet : set initial velocity as 39 m/s(positive x face) 2)Interior solid: internal part of the domain 3)Wall solid : stationary walls with no slip shear condition(building walls) 4)sides : stationary walls with no slip shear condition(sides of domain) 5) Outlet : Pressure outlet(negative x face) Calculation setup : • Calculation were set for 45 time steps each time step was 0.2sec. • Number of iterations were set to 5 per time step, finally count reach to 225 as end of the 45th time step. • Results were set to record at the end of every time step. • Solution is calculated for 9 sec of flow. • Transient mode of solving is used which gives time based solution. Monitor Points : Total 2 points were monitored continuously throughout time in calculation scope. All the points were monitored for pressure.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

Table 4.4: Monitor points

Point	x	y	z
Pz	0	90.8	21
Nz	0	90.8	-21

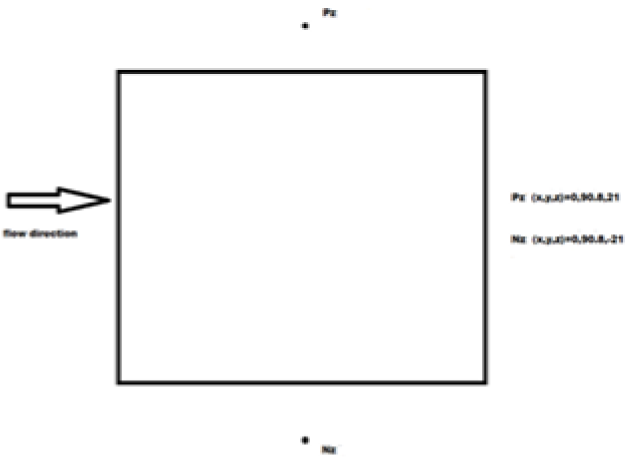


Figure4.11: Location of monitor points

Judging convergence of the solution:

• Convergence testing required practice of simulation work and accurate practical judging methodologies because practical solutions will not always give ideal indications. •

We can judge convergence by following patterns:

1) Residuals pattern

2) Flow Pattern

Residual pattern:

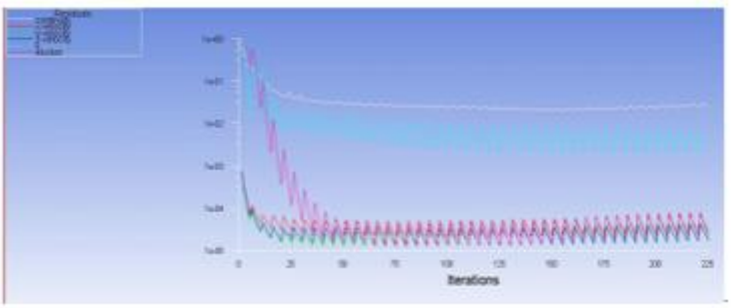


Figure 4.12: Residual pattern

Above graph is error vs iteration or time for each individual equation that has been solved by solver for the given model, meshing and boundary conditions. This plot is useful to judge the convergence in practical scenario. Above plot shows that error of maximum equation is less than 10E-4 which is ideally accepted, but 2 equations still shows the error greater than 10E-2 which is not greater than what is ideally accepted but for practical scenario every equation may not show the ideal convergence, so we have to find out the indications of convergence and it can be pointed out by guidance of experienced CFD experts. Some of such indications as follows: 1) The plot of the equation's error may make up and down pattern. 2) In that up and down pattern plot every new peak may lower than the previous.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

Flow Pattern: Flow pattern that we get should make sense.

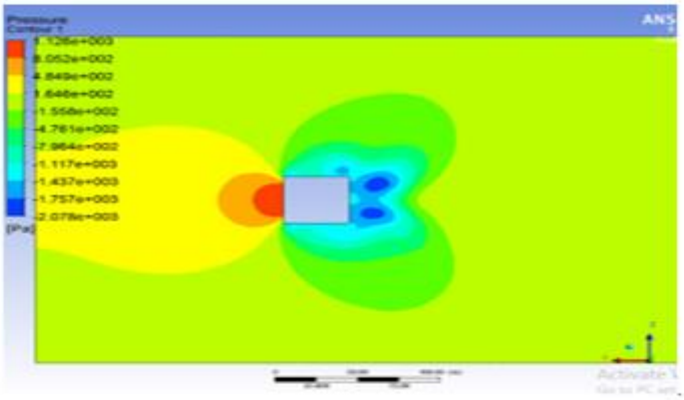


Figure 4.13: Flow pattern contour

Wind flow pattern is also the indication of the convergence. The flow pattern should make sense, that means it must show the it's common or regular behavior around the bluff body for example when wind obstructed by any object or bluff body it makes negative pressure at the body's opposite side of wind facing side. If it is there then the situation that simulated may converged.

CONCULSION

General

The present study investigated the effect of various aerodynamic modifications on the wind response of tall buildings using ANSYS Fluent. A rectangular building was taken as the reference model and compared with different modified configurations such as chamfered corners, corner cuts, setbacks, openings, and combined aerodynamic modifications. From the simulation results, it was observed that the rectangular building showed significant pressure oscillations due to vortex shedding. These oscillations were caused by sharp corners that created strong flow separation and alternating low- and high-pressure zones around the building. Such vortex shedding can be dangerous when its frequency matches the natural frequency of the structure, leading to resonance and excessive structural displacement. The introduction of aerodynamic modifications significantly improved the wind performance of the buildings. Chamfered corners and corner-cut configurations effectively reduced or eliminated pressure oscillations by providing a smoother path for airflow around the structure. Similarly, setback configurations disrupted the continuous vertical formation of vortices and minimized vortex shedding effects. The study also showed that openings provided through the building were less effective in controlling vortex shedding compared to other aerodynamic modifications. Pressure oscillations were still observed in buildings with openings, indicating that the airflow disturbance created by openings alone was insufficient to eliminate vortex formation. Among all the configurations, combined aerodynamic modifications such as chamfering with setbacks and corner cuts with setbacks produced the best overall performance. These hybrid configurations successfully disturbed vortex formation and eliminated pressure oscillations, thereby improving structural safety and stability under wind loading. Overall, the study concluded that aerodynamic modifications are highly effective in reducing wind-induced effects on tall buildings. Properly designed aerodynamic building forms can minimize vortex shedding, reduce dynamic wind response, improve serviceability, and enhance the overall safety and performance of high-rise structures.

Conclusion 1. The rectangular building showed significant pressure oscillations due to vortex shedding caused by sharp corners and direct wind obstruction.

2. Pressure oscillations in the rectangular building varied approximately between 600 Pa and 2400 Pa, indicating strong vortex formation around the structure.

3. Vortex shedding may lead to resonance when its frequency matches the natural frequency of the building, causing excessive displacement and structural instability.

4. Aerodynamic modifications such as chamfering, corner cuts, and setbacks effectively reduced or eliminated vortex shedding by disturbing airflow around the building.

5. Buildings with 5% and 10% chamfering showed absence of pressure oscillations due to smoother wind flow around modified corners.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

6. Corner-cut configurations also successfully eliminated vortex shedding by reducing sharp edge effects and providing a smoother path for airflow.

7. Two-stage and three-stage setback buildings performed effectively by interrupting continuous vortex formation along the building height.

8. Buildings with openings were comparatively less effective because pressure oscillations were still observed in these configurations.

9. Combined aerodynamic modifications such as chamfering with setbacks and corner cuts with setbacks provided the best overall performance against wind effects.

10. The combined modifications eliminated pressure oscillations completely and improved structural safety and stability under wind loading.

11. The study confirms that aerodynamic modifications are highly effective in reducing wind-induced vibrations and improving the wind resistance of tall buildings.

12. Therefore, aerodynamic shaping can be considered an economical and efficient method for enhancing the performance, safety, and serviceability of tall structures subjected to wind loads.

EFERENCES

[1]. Peter Irwin," Effects of Geometry on the Wind Response of Super-tall Towers", ASCE (structure 2008)
[2]. Yukio Tamura, Hideyuki Tanaka, Kazuo Ohtake,Masayoshi Nakai, Yongchul Kim," Aerodynamic Characteristics of Tall Building Models with Various Unconventional Configurations", ASCE (2010 structures congress)
[3]. U. Weerasuriya, "Computational Fluid Dynamic (CFD) Simulation of Flow around Tall Buildings", The institution of engineers(ENGINEER - Vol. XXXXVI, No. 03
[4]. Islam Abohela, Neveen Hamza, Steven Dudek, Validating CFD Simulation Results: Wind flow around a surface mounted cube in a turbulent channel flow, PLEA2012 - 28th Conference
[5]. IS:16700-2017 Criteria for structural safety of tall concrete buildings.
[6]. National building code of India 2016
[7]. National fire protection association.
[8]. 8. IS:875 part 1 –Code of practice for design loads (other than earthquake) for buildings and structures
[9]. 9. IS:875 part 2 –Code of practice for design loads (other than earthquake) for buildings and structures.