

Comparative CFD Analysis of Major and Minor Aerodynamic Modifications for Tall Building Shapes under Wind Loading

Tanmay Rajiv Jadhav^{1*}, Yuwaraj M. Ghugal²

Abstract

Aerodynamic performance of tall buildings is significant to structural safety and serviceability under wind. The major and minor aerodynamic changes of the tall building shape are compared with the computational fluid dynamics (CFD) simulation. The base model was a rectangle of 75 m × 75 m × 300 m. Various aerodynamic modifications of shapes were examined: circular, elliptical, tapered, twisted, rounded-corner, and chamfered-corner shapes. Simulations were carried out in the ANSYS Fluent software, with the same boundary conditions. The velocity profile, pressure contours, streamlines, and the formation of wake were considered as the criteria for flow behavior. It was discovered that this sharp-cornered rectangle model had quite a lot of flow separation and a large wake area. Significant modifications (elliptical, tapered) have relatively smooth flow and minimized wake. There are also good gains in flow separation observed in the smaller changes, like rounded corners. Moderate improvement over baseline is seen at chamfered corners. The results indicate that major and minor geometric changes have a significant influence on aerodynamic behavior. Major modifications mainly affect the global flow structure. Minor modifications improve the local flow characteristics near edges and corners. Among the investigated configurations, elliptical and tapered shapes demonstrated improved aerodynamic performance under identical CFD conditions. Elliptical and tapered configurations showed the smallest wake regions and lowest normalized drag coefficients among all investigated cases. The findings may assist in preliminary aerodynamic design and shape optimization of tall buildings.

Keywords: CFD, tall buildings, aerodynamic modifications, wind load, flow separation, wake region

INTRODUCTION

Increasing urbanization has resulted in a trend towards taller and narrower buildings, especially in crowded cities with scarce land resources. These buildings provide significant economic benefits and added functionality but are vulnerable to aerodynamic effects because of their slender shape. Wind loads impact not only the structural performance but also the serviceability (comfort and façade performance) of tall buildings. Therefore, it is important to understand and optimize the aerodynamic performance of new tall building designs [1–4].

*Author for Correspondence

Tanmay Rajiv Jadhav
E-mail: jtanmy404@gmail.com

¹Student, Department of Applied Mechanics, Government College of Engineering, Chhatrapati Sambhajinagar, Maharashtra, India

²Professor, Department of Applied Mechanics, Government College of Engineering, Chhatrapati Sambhajinagar, Maharashtra, India

Received Date: May 26, 2026

Accepted Date: June 11, 2026

Published Date: July 02, 2026

Citation: Tanmay Rajiv Jadhav, Yuwaraj M. Ghugal. Comparative CFD Analysis of Major and Minor Aerodynamic Modifications for Tall Building Shapes under Wind Loading. Journal of Offshore Structure and Technology. 2026; 13(2): 9–35p.

The aerodynamic behavior of buildings is typically addressed through wind tunnel testing. Although these approaches provide accurate predictions, they can be costly and time-consuming, particularly if many design variations are required.

Recently, Computational fluid dynamics (CFD) approaches have proven to be an efficient and viable alternative that allows for the visualization of airflow patterns, surface pressures, and wakes around complex structures. The increasing availability of computational tools has allowed researchers to explore a wide range of building shapes and aerodynamic modifications with greater flexibility [5–8].

One of the widely adopted approaches to reduce wind-induced effects is modifying the building geometry. These modifications can generally be classified into major (global) changes, such as tapering, twisting, setbacks, and variations in cross-sectional shape, and minor (local) changes, such as corner rounding, chamfering, and recessing. Structural modifications affect the global flow interaction with the building, and minor modifications mostly affect the local flow separation and turbulence around the edges. Both major and minor modifications have been reported to influence aerodynamic performance; however, their methods of application and viability are different. However, the existing literature lacks comparative studies to guide the improvement of aerodynamic performance through CFD studies, making it difficult to choose a suitable aerodynamic modification [9–12].

Previous studies have shown that streamlined designs, such as circular and elliptical buildings, can reduce flow separation and wake effects compared to conventional rectangular buildings. Similarly, corner tapering and twisting have been demonstrated to mitigate vortex shedding and enhance the aerodynamic performance. However, minor changes, such as rounded or chamfered corners, reduced the pressure loads and delayed flow separation while maintaining the basic building shapes. While past research has generally examined major and minor changes separately, fewer studies have directly compared the two prevailing conditions. Another important aspect discussed in the literature is the effect of flow separation and vortex structures on aerodynamic performance. Sharp edges promote early flow separation and vortex formation. This creates large areas of suction and wakes on the downwind side. However, the smoothness of the aerodynamics favors smooth flow, which works against the turbulence of the flow and promotes stability. However, it remains unclear how these geometric variations affect these effects when tested using the same numerical simulations [13–15].

In this context, the present study aims to provide a systematic comparative analysis of major and minor aerodynamic modifications applied to tall building shapes using CFD. This study used a reference rectangular building model and examined several modified building shapes (circular, elliptical, tapered, twisted, rounded-corner, and chamfered corner) under identical computational conditions. This study assessed the flow behavior in terms of velocity, pressure distribution, and wake patterns, allowing for a direct comparison between cases [16–18].

By analyzing global and local geometric modifications within a consistent framework, this study aims to provide insights into their comparative effects on aerodynamic performance. This research will inform better design practices for high-rise buildings, especially in the early design phase, where the shape is a key factor influencing performance.

LITERATURE REVIEW

Several aerodynamic modifications have been studied in detail to enhance the wind performance of tall buildings. Various modifications of geometry, such as tapering, setbacks, corner recessions, twisting, rounded corners, chamfered edges, and openings, are frequently used to minimize wind-induced forces, manage vortex shedding, and enhance structural stability in strong winds. Aerodynamic optimization has been found to have significant effects on the drag force, pressure distribution, wake structure, and structural response of high-rise buildings in previous investigations [19].

The aerodynamic performance of the modified tall building configurations has been studied by various researchers using experimental and numerical methods. An experimental study of tapering on tall buildings in wind found that tapering is effective in suppressing the responses across the wind by breaking the organized vortex shedding and avoiding resonance effects. Wind tunnel tests on chamfered

and recessed corner treatments found that they resulted in substantial decreases in crosswind response and structural stresses. The various aerodynamic modifications, including tapering, setbacks, rounded corners, chamfered corners, slots, and openings, were studied, and it was found that passive aerodynamic treatments are among the most effective aerodynamic measures to reduce wind-induced responses in slender, tall structures [20].

The aerodynamic performance analysis of tall structures has gained increasing popularity as a tool of CFD owing to its flexibility and low cost in terms of experimental investigations. A GA-NN aerodynamic optimization approach that showed that optimized corner configuration changes can effectively reduce drag force and vortex-induced vibration. Aerodynamic mitigation and shape optimization methods emphasized the increasing use of CFD-based optimization techniques for drag reduction and aerodynamic improvement in high-rise buildings. Similarly, both CFD and experimental studies conducted on irregular C-shaped buildings showed good comparison between CFD and experimental results using the standard $k-\epsilon$ turbulence model [21].

Aerodynamic optimization and comparative analyses of modified building geometries have been the subject of several studies. Setback tall buildings concluded that setback arrangements have a significant impact on the pressure distribution, force coefficients, and torsional behavior. Conducted a comparative study between the regular and irregular configurations of tall buildings with modifications in the corner chamfering and truncation using CFD found that the wind pressure coefficient was significantly reduced. Conducted a study on multi-parameter corner recession modifications and concluded that the drag coefficient (C_d) can be lowered by almost 45% by employing corner treatments, while vortex shedding is significantly suppressed. An aerodynamic optimization design model of spiral supertall buildings with multi-objectives was developed and proved the strong effect of twist angle and taper ratio on the aerodynamic efficiency and vibration performance of supertall buildings [22].

In addition, machine learning and artificial intelligence-based optimization techniques have become significant in aerodynamic shape optimization studies. Using CFD simulation, Artificial Neural Networks (ANN), and Genetic Algorithms (GA), significant reductions in aerodynamic load and wake formation were achieved for tall buildings. The applications of machine learning in aerodynamic optimization found that the use of surrogate models and deep learning techniques can significantly reduce the computational cost while retaining acceptable prediction accuracy. Provided a detailed overview of wind engineering applications to tall buildings and discussed the growing trend of applying CFD, optimization, and machine learning techniques in current wind engineering design practices [23].

Recent investigations have also highlighted the advantages of wind performance and structural stability of the aerodynamic modification of tall structures. The importance of aerodynamic shape optimization in minimizing drag force and enhancing serviceability performance. Conducted a CFD and wind tunnel study of the across-wind response of rectangular tall buildings and concluded that the vortex shedding effect is more prominent for slender buildings. In the investigation, both micro- and macro-aerodynamic modifications were analyzed using ETABS analysis and concluded that corner treatments, fins, and setbacks significantly improve wind performance by reducing displacement, drift, and base shear. High-rise buildings with orifices were analyzed using CFD simulations and demonstrated that geometric openings reduce windward pressure and leeward suction, thereby improving aerodynamic efficiency and structural safety [24].

Previous studies on the aerodynamic modification of tall buildings can be broadly categorized into major geometric modifications, minor corner modifications, optimization-based approaches, and CFD validation studies. Although numerous investigations have demonstrated the effectiveness of aerodynamic modifications in reducing wind-induced effects, many studies have primarily focused either on aerodynamic behavior through CFD analysis or on structural response using codal wind loading procedures independently. Few studies have comparatively evaluated multiple major and minor

aerodynamic modifications under identical CFD conditions within a unified computational framework. Therefore, a systematic comparative CFD-based assessment is necessary to better understand the relative aerodynamic effectiveness of different geometric modifications for tall buildings [25].

Research Gap

The literature has been reviewed extensively, and it has been found that many efforts have been made to improve and understand the aerodynamic efficiency of skyscrapers under wind loading. A range of techniques have been investigated, from major geometric changes such as tapers, setbacks, twists, and openings to minor modifications such as chamfered, rounded, and recessed corners. It has been shown that these methods can reduce wind-induced responses by changing the flow separation, vortex shedding, and pressure distribution. However, despite these developments, existing research has several major limitations.

First, many studies have focused on large and small aerodynamic changes separately, without a direct comparison between them under the same conditions. This makes it difficult to clearly identify which type of alteration is more effective or how they compare with each other when applied to the same building configuration.

Second, the results are often inconsistent and cannot be directly compared because many studies use different modeling techniques, simulation parameters, or experimental designs. There is no common framework in which several aerodynamic shapes are evaluated under the same CFD setup, boundary conditions, and evaluation parameters.

Third, the across-wind effects due to vortex shedding have been recognized as important and often control the design of slender tall buildings. However, few studies have simultaneously considered key aerodynamic parameters, such as drag force, lift force, pressure coefficient distribution, and wake characteristics, in a single comparative study. Additionally, there is a lack of studies on setback configurations relative to other major and minor modifications under a consolidated CFD framework.

Moreover, although recent studies have proposed advanced techniques such as machine learning, surrogate modeling, and optimization algorithms, their application has been mostly limited to specific cases rather than being part of a wider comparative evaluation of different aerodynamic modifications. Hence, there is a clear need for a detailed and systematic comparison using CFD for both significant and minor aerodynamic modifications under controlled conditions. This approach would enable a better understanding of their relative performances and provide a means to develop practical performance-based design standards for tall buildings under wind loading conditions.

A systematic comparative study under the same CFD conditions is required to obtain a clear understanding of the relative aerodynamic effects of both global and local geometry modifications.

Objectives

The aims of the present study are:

- To design various tall building geometries with major and minor aerodynamic modifications.
- To simulate the behavior of the airflow around the chosen configurations using CFD.
- To compare the velocity distribution, pressure characteristics, flow separation, and wake behavior under identical computational conditions.
- To evaluate the relative aerodynamic effectiveness of various geometric modifications.

METHODOLOGY

Geometry Modeling

The basic layout was selected as a typical tall building model with dimensions of 75 m × 75 m × 300 m. This is an example of a typical tall building with a square plan and a high aspect ratio.

To study the effect of aerodynamic modifications, different building geometries were developed and categorized as follows:

- *Base model*: Rectangular (square plan) building
- *Major modifications*: Circular, elliptical, tapered, and twisted shapes
- *Minor modifications*: Rounded corners and chamfered corners

All models were created using CAD tools and exported. SAT format for further processing (Figure 1).

Computational Domain

A 3D computational domain was created to simulate the atmospheric wind flow around the building (Table 1). This domain size ensures proper development of flow and minimizes boundary effects.

Meshing

Meshing was performed using ANSYS Fluent Meshing.

- *Mesh type*: Unstructured tetrahedral mesh
- *Refinement*: Finer mesh near building surfaces; Coarser mesh away from the structure
- *Inflation layers*: Applied near walls to capture boundary layer effects
- A mesh sensitivity approach was followed by increasing the mesh density until the results showed stable trends, as shown in Figure 2.

Table 1. Computational domain dimensions.

Parameter	Value
Upstream	5H (1500 m)
Downstream	15H (4500 m)
Side clearance	5H (1500 m)
Top clearance	5H (1500 m)

Where H = building height

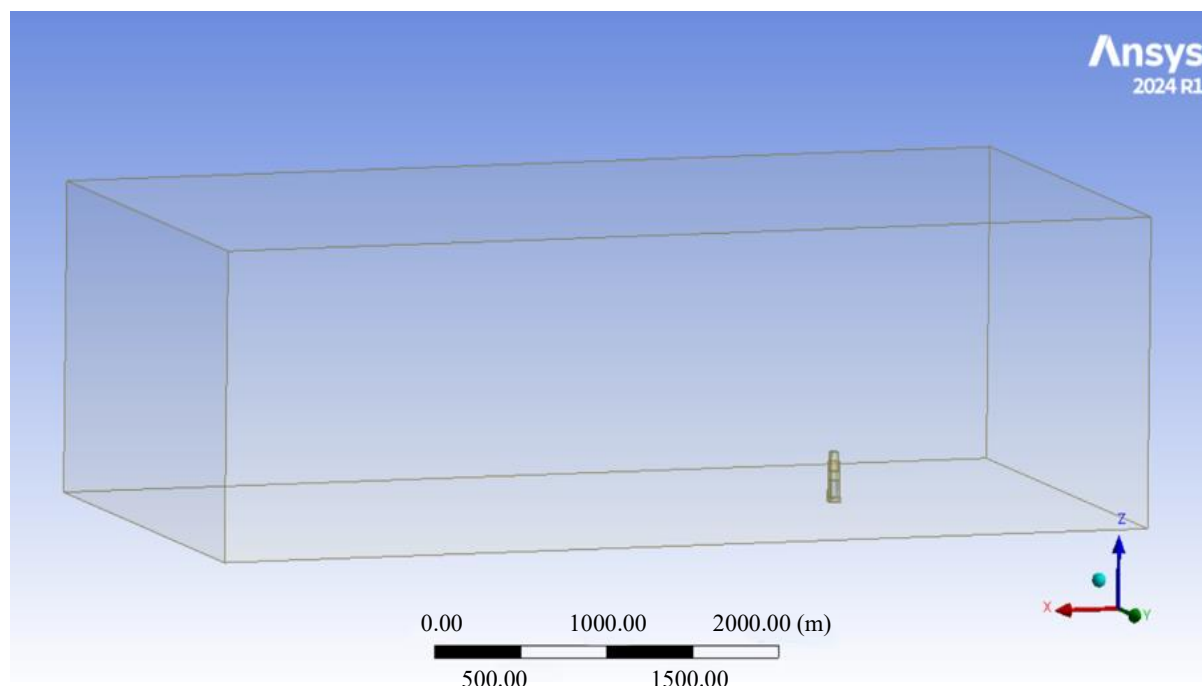


Figure 1. Computational domain size of the building.

Boundary Conditions

The following boundary conditions were applied:

- *Inlet*: Velocity inlet
- *Outlet*: Pressure outlet (0-gauge pressure)
- *Walls (building surface)*: No-slip condition
- *Ground*: No-slip wall
- *Side and Top boundaries*: Symmetry or slip wall

The aerodynamic differences between the configurations are clearly observed at a high wind velocity of 55 m/s, which was selected for the comparative analysis of the configurations and corresponds to an extreme wind condition.

The flow conditions corresponded to a high Reynolds number external flow regime, which justified the use of the standard k - ϵ turbulence model to capture the turbulent behavior.

Solver Settings

The simulations were performed using ANSYS Fluent with the following settings:

- *Solver type*: Pressure-based, steady-state
- *Turbulence model*: Standard k - ϵ model
- *Discretization–Pressure*: Second-order; momentum: second-order upwind
- *Pressure-velocity coupling*: Simple scheme

The adopted CFD approach is in line with previous studies on the external flow analysis of tall buildings using the standard k - ϵ model, which has been widely validated in the literature. The standard k - ϵ turbulence model was chosen because it has been extensively validated for external flow analysis around bluff bodies and tall buildings under high Reynolds number conditions. This model has been successfully used in comparative aerodynamic assessments in previous wind engineering studies.

The results of the adopted CFD methodology and the observed aerodynamic trends were qualitatively compared with previously published wind engineering studies of streamlined and modified tall building geometries. The decrease in wake formation and flow separation for elliptical, tapered, and rounded configurations was observed to be in good agreement with those reported in previous experimental and numerical studies.

Grid Independence Study

A grid independence study was conducted by refining the mesh until an insignificant variation in the flow characteristics was obtained. The mesh was progressively refined until no significant changes occurred in the velocity distribution and wake behavior. This ensured the mesh size independence of the obtained results (Table 2).

The transition from medium to fine mesh did not reveal any significant change in the flow characteristics, indicating mesh-independent behavior. We used a medium mesh configuration for all the simulations to balance the accuracy of the computation with the time of the solution.

Table 2. Grid Independence study.

Mesh level	Approx. elements	Observation
Coarse	Lower mesh density	Slight flow fluctuation
Medium	Moderate mesh density	Stable wake pattern
Fine	High mesh density	Negligible variation

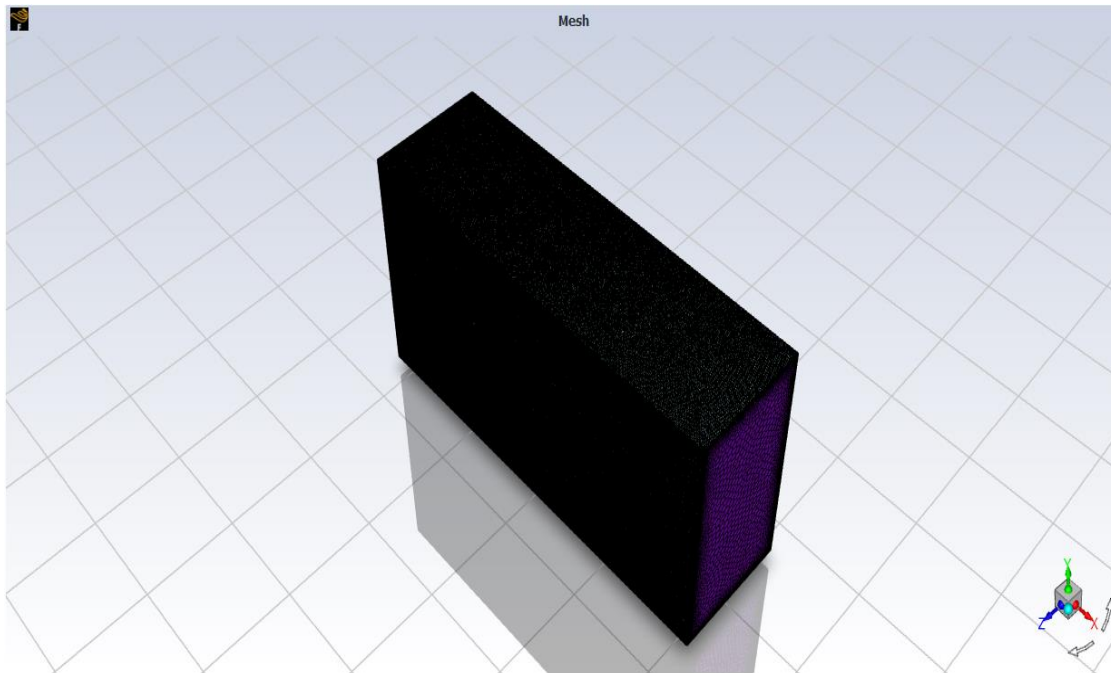


Figure 2. Meshing of the enclosure.

Solution Initialization and Convergence

- *Initialization:* Standard initialization from the inlet
- *Iterations:* Approximately 150–300 iterations
- *Convergence criteria:* Residuals below 10^{-4} to 10^{-5} ; Stabilization of flow parameters

In addition to residual convergence, the stabilization of flow patterns and wake characteristics was monitored to ensure the numerical stability of the solution.

Post-Processing

The results were analyzed using CFD post-processing tools. The following parameters were evaluated.

- *Velocity contours:* to observe flow acceleration and wake
- *Pressure contours:* to identify high and low-pressure regions
- *Pathlines/streamlines:* to study vortex formation and flow separation
- *Wake region characteristics:* to compare aerodynamic behavior

Comparisons were made across all models under identical conditions to ensure consistency.

Comparative Analysis Approach

All building configurations were analyzed using the same domain size, boundary conditions, and solver settings. This provided a fair and direct comparison of the major and minor aerodynamic modifications. The CFD methodology used in the present study was aimed at a comparative aerodynamic assessment and not a precise prediction of the absolute wind loads.

RESULTS AND DISCUSSIONS

Introduction

In this study, CFD simulations were performed to assess the aerodynamic behavior of tall buildings with diverse shapes under identical wind conditions. The analysis focused on comparing the flow pattern, pressure distribution, and wake characteristics of the base model and a few modified configurations.

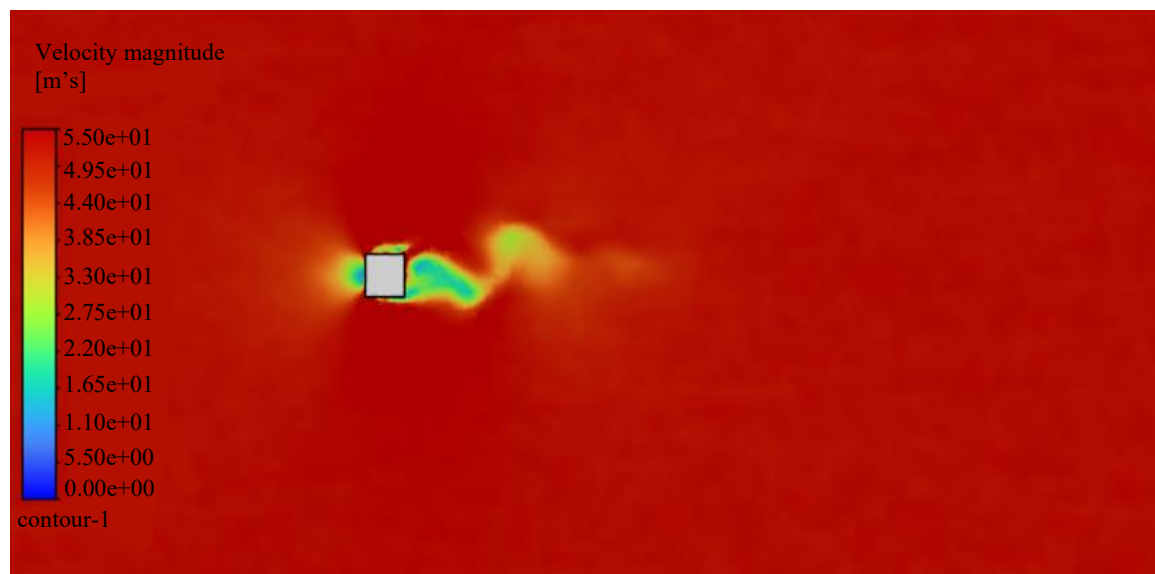


Figure 3. Velocity contour of a rectangular building.

To ensure consistency, each model was tested with the same parameters: velocity contours, pressure contours, pathlines (flow patterns), and wake region behavior. Such a uniform approach allows for a direct comparison of the major and minor aerodynamic modifications.

All figures are shown with the same scale and visualization settings, allowing consistent comparisons between different building configurations. Comparative analysis is useful for understanding the influences of different aerodynamic modifications on flow separation, pressure distribution, and wake formation around tall buildings

Base Model—Rectangular Building

Velocity Distribution

The velocity contours show a well-developed stagnation region on the windward face, where the flow decelerates significantly. The flow detaches sharply at the edges, and a large wake region forms behind the building, as shown in Figure 3.

Pressure Distribution

The front face was subjected to high positive pressure. A large negative pressure (suction) zone is formed on the leeward side. The sharp edges make the pressure change fast Figure 4.

Flow Pattern (Pathlines)

The pathlines revealed well-defined vortices behind the structure. The flow separates at the corners and creates a large recirculation zone, as shown in Figures 5 and 6.

Observation

The rectangular model exhibited significant flow separation, high pressure variation, and a large wake region, indicating relatively low aerodynamic performance. The sharp-edge geometry causes early flow separation and strong vortex formation, resulting in an increase in aerodynamic resistance and unstable wake characteristics.

Circular Building

Velocity Distribution

The velocity contours indicate that the flow around the structure was smoother with less separation than that of the rectangular model (Figure 7).

Pressure Distribution

It has a more uniform pressure distribution with a lower peak pressure on the windward side and lower suction on the leeward side, as shown in Figure 8.

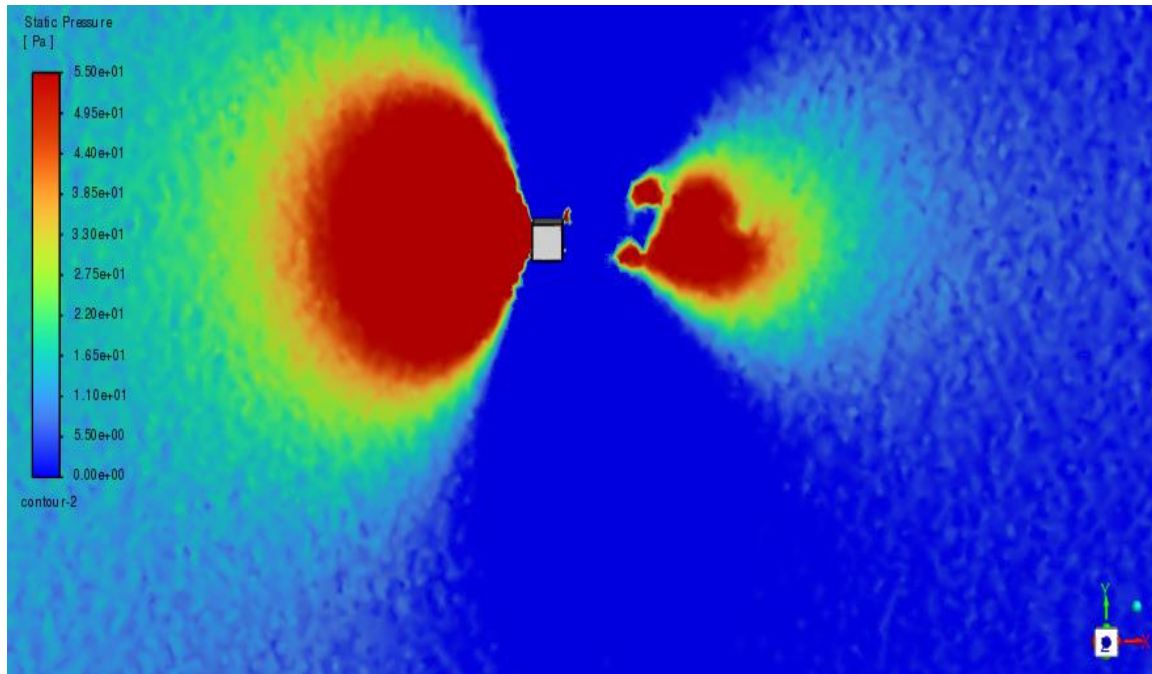


Figure 4. Static pressure contours of rectangular building.

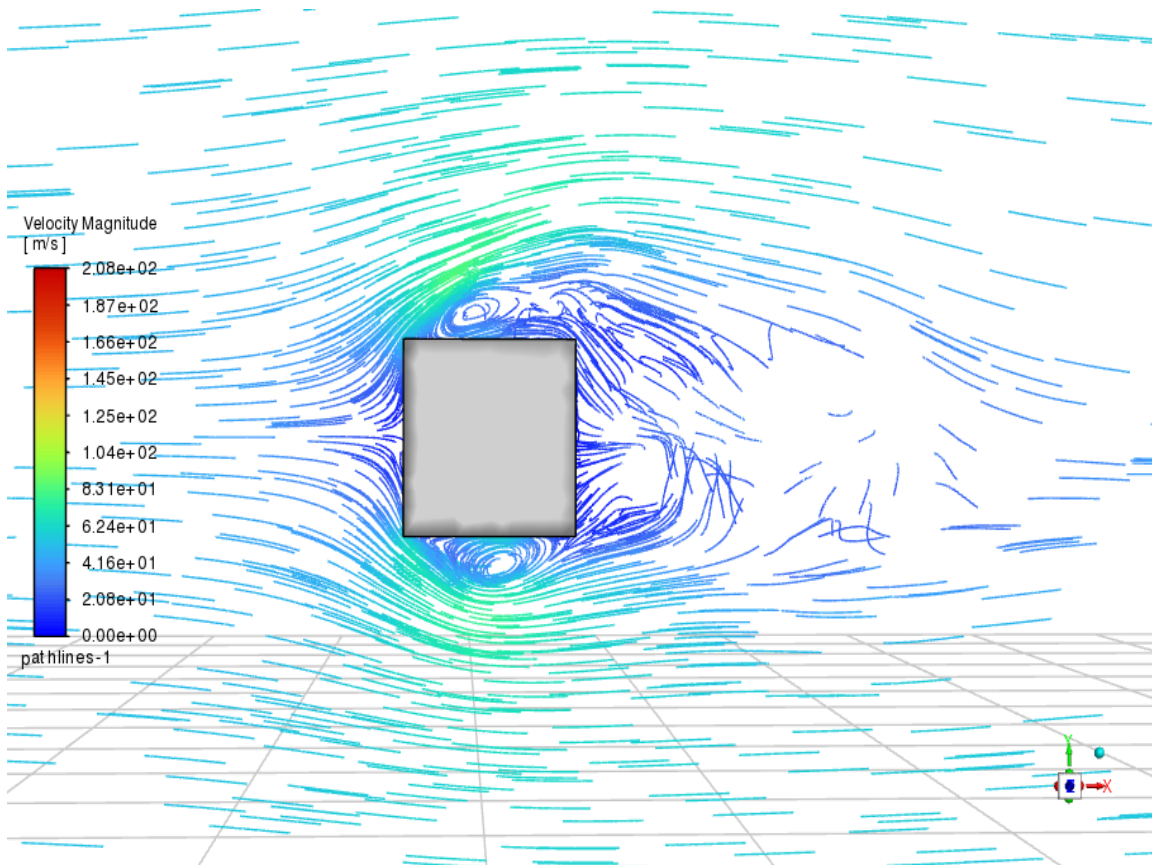


Figure 5. Pathlines around rectangular building (top view).

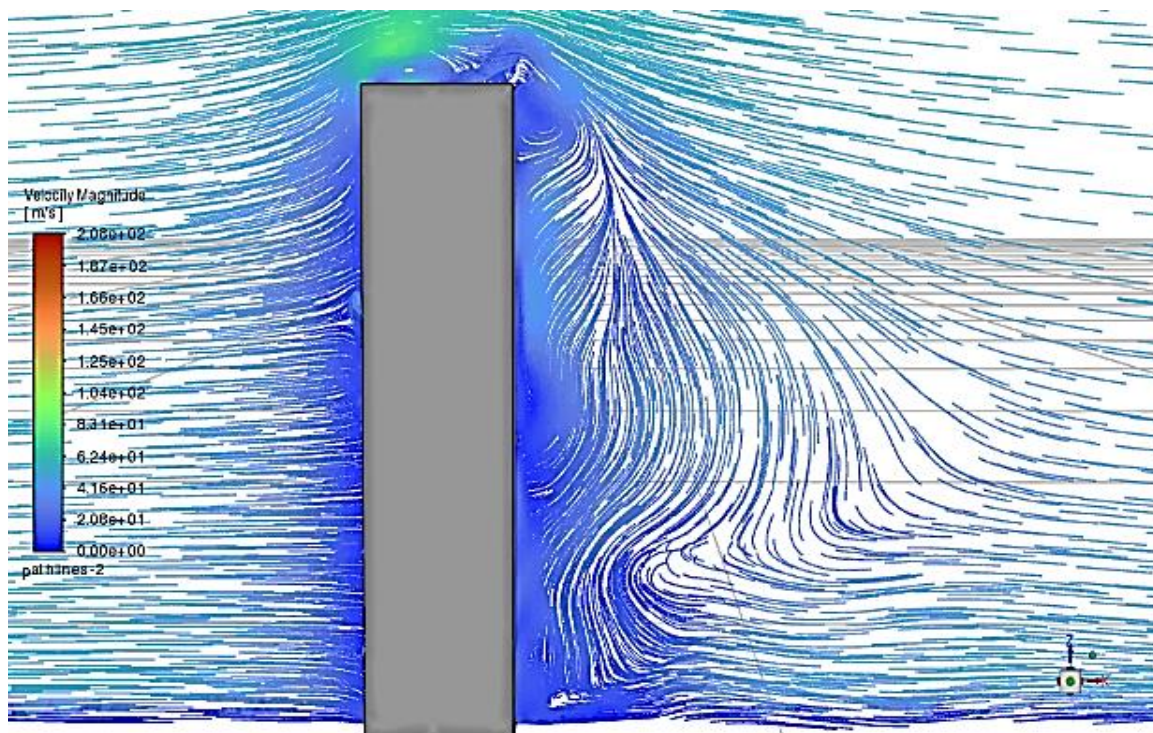


Figure 6. Pathlines around rectangular building (Side view).

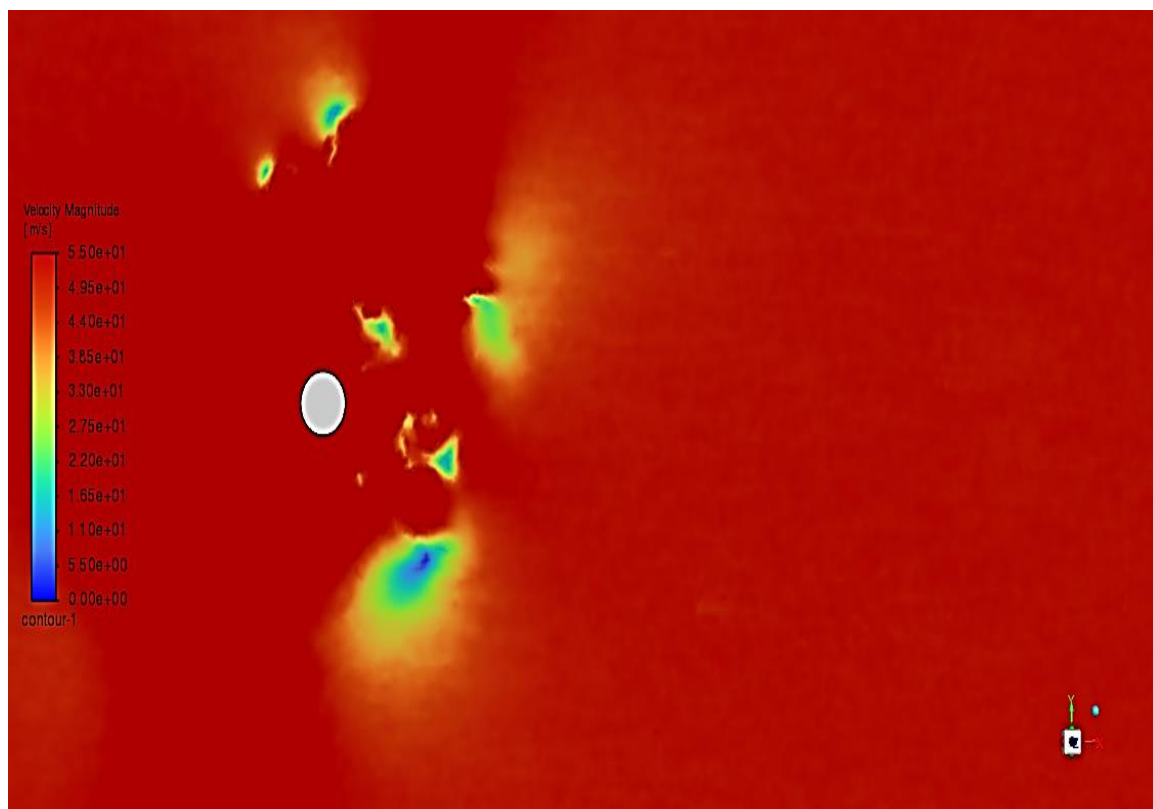


Figure 7. Velocity contours of circular building.

Flow Pattern (Pathlines)

The pathlines indicate delayed flow separation and a narrower wake region, as shown in Figures 9 and 10.

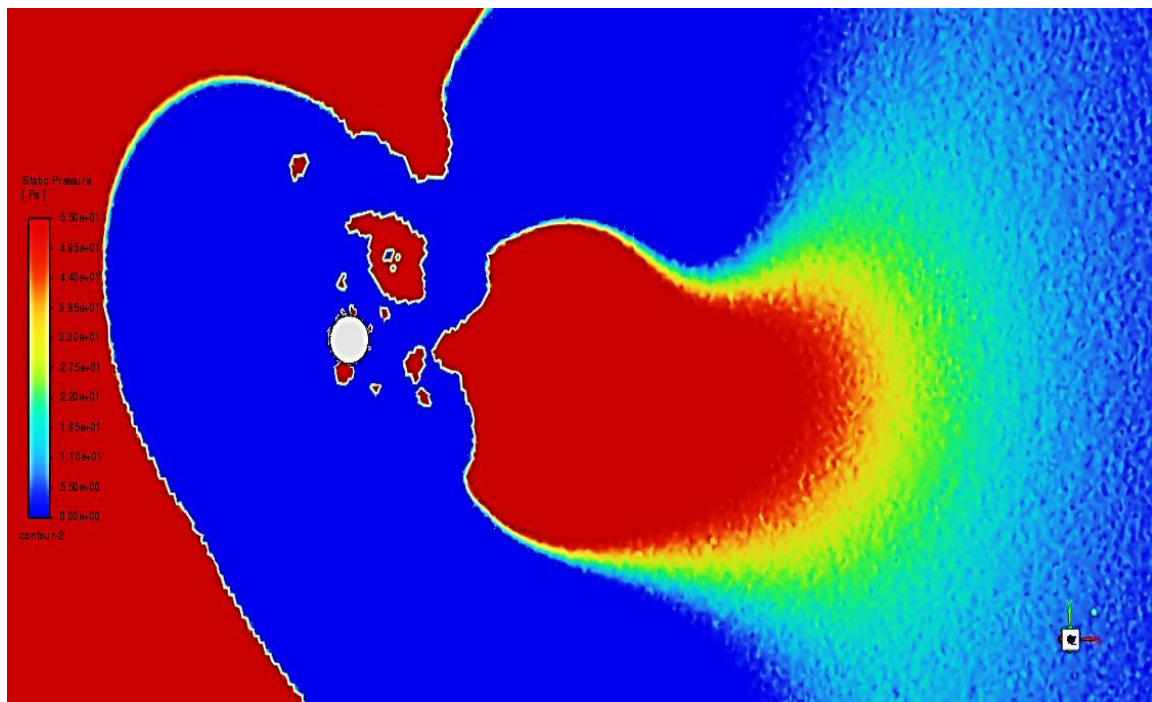


Figure 8. Static pressure contours of circular building.

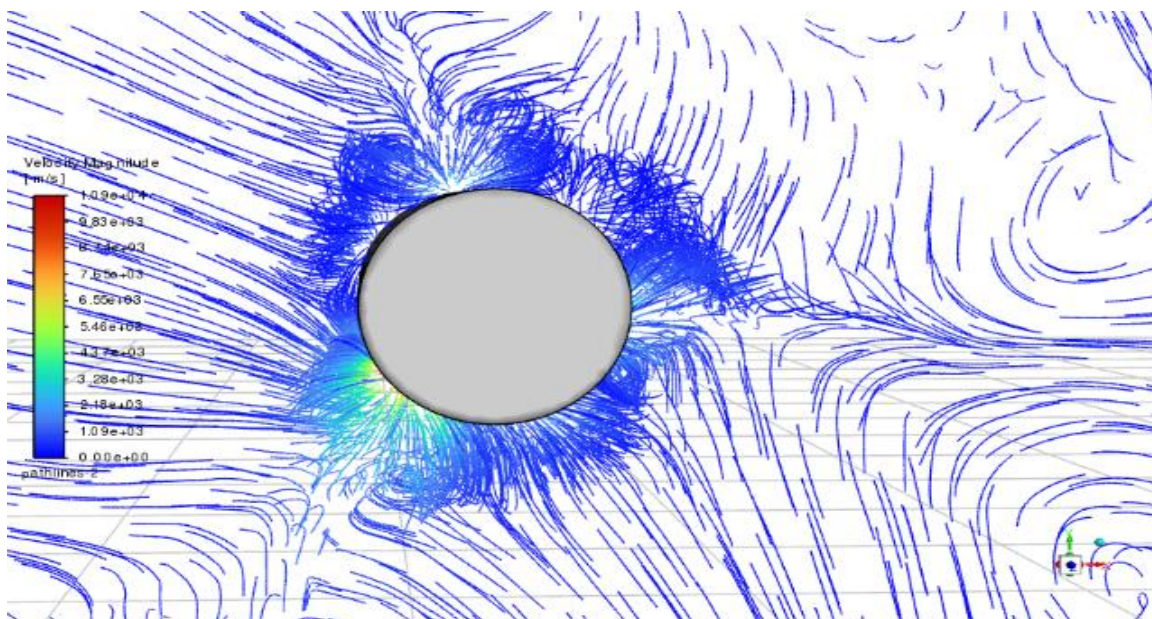


Figure 9. Pathlines around circular building (top view).

Observation

The circular shape improved the flow behavior by reducing the separation and wake sizes. This led to better aerodynamic performance compared to the base model. The smooth curved surface of the structure allows the airflow to smoothly transition around it. This helped delay the flow separation and reduce the wake intensity.

Elliptical Building *Velocity Distribution*

The flow accelerated smoothly along the curved surface, showing reduced turbulence compared with both the rectangular and circular shapes (Figure 11).

Pressure Distribution

The pressure distribution is more gradual, with less intense suction zones Figure 12.

Flow Pattern (Pathlines)

The pathlines show a streamlined flow with minimal vortex formation and a smaller wake region, as shown in Figures 13 and 14.

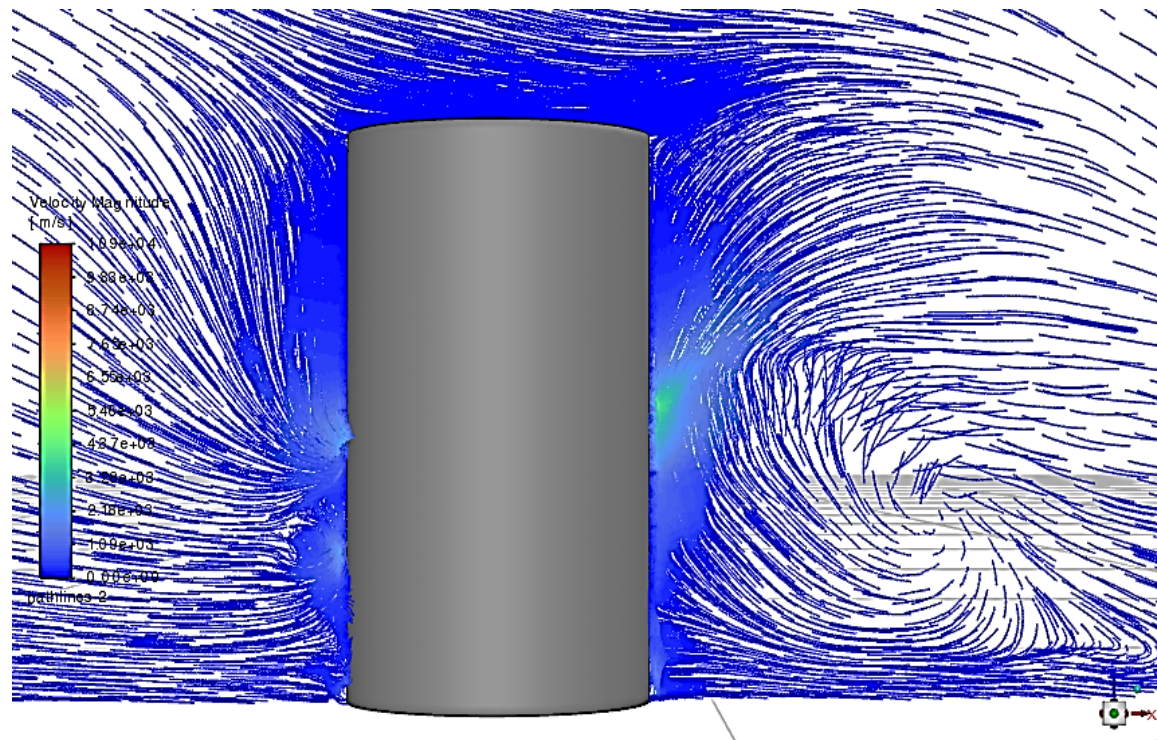


Figure 10. Pathlines around a circular building (side view).

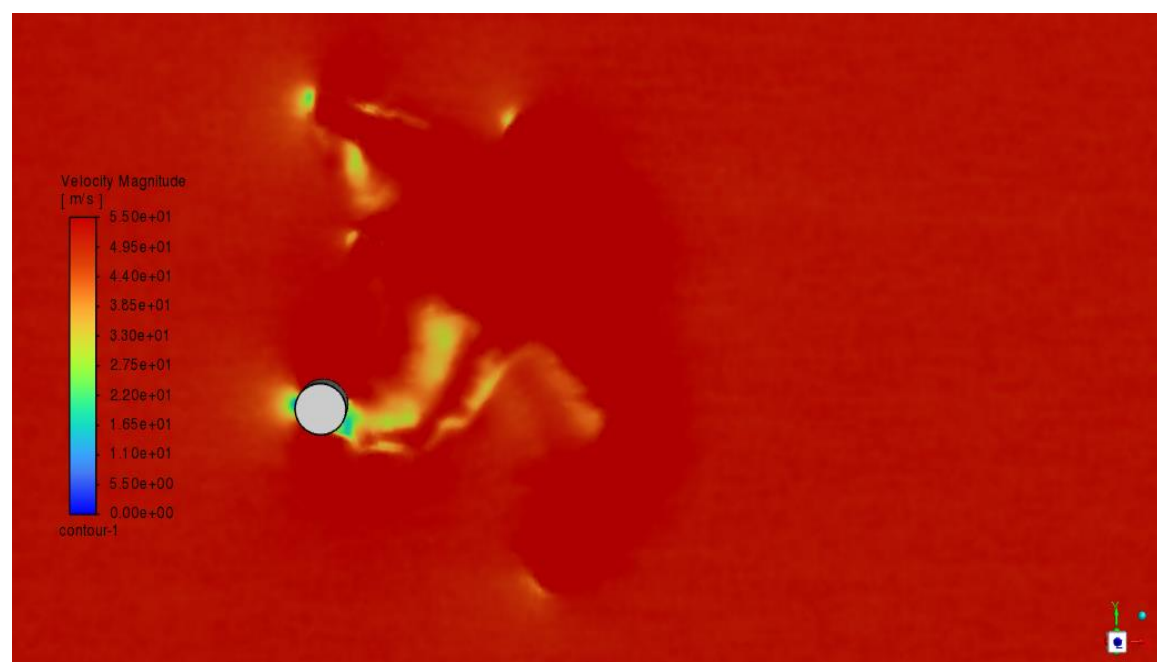


Figure 11. Velocity contours of elliptical building.

Observation

The elliptical model offers better aerodynamic efficiency, less turbulence, and a smooth flow transition. Simplified geometry led to smaller pressure gradients and less turbulence, resulting in relatively better aerodynamic behavior.

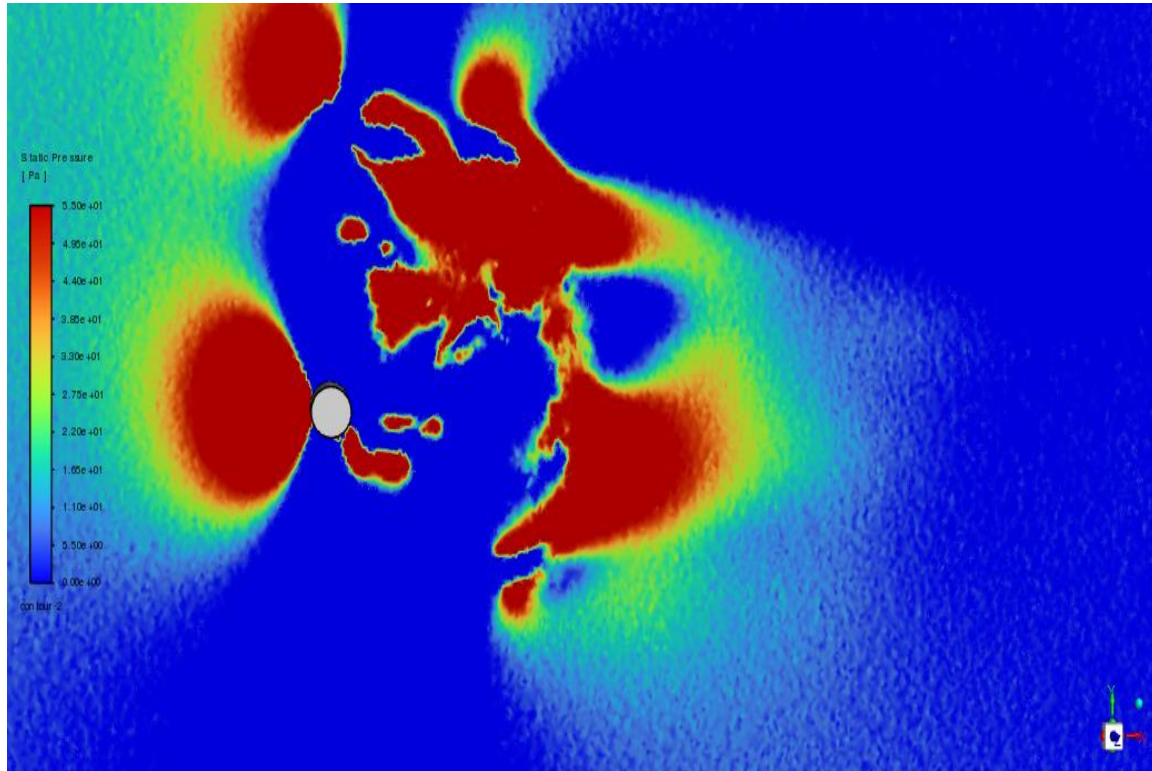


Figure 12. Static pressure contours of elliptical building.

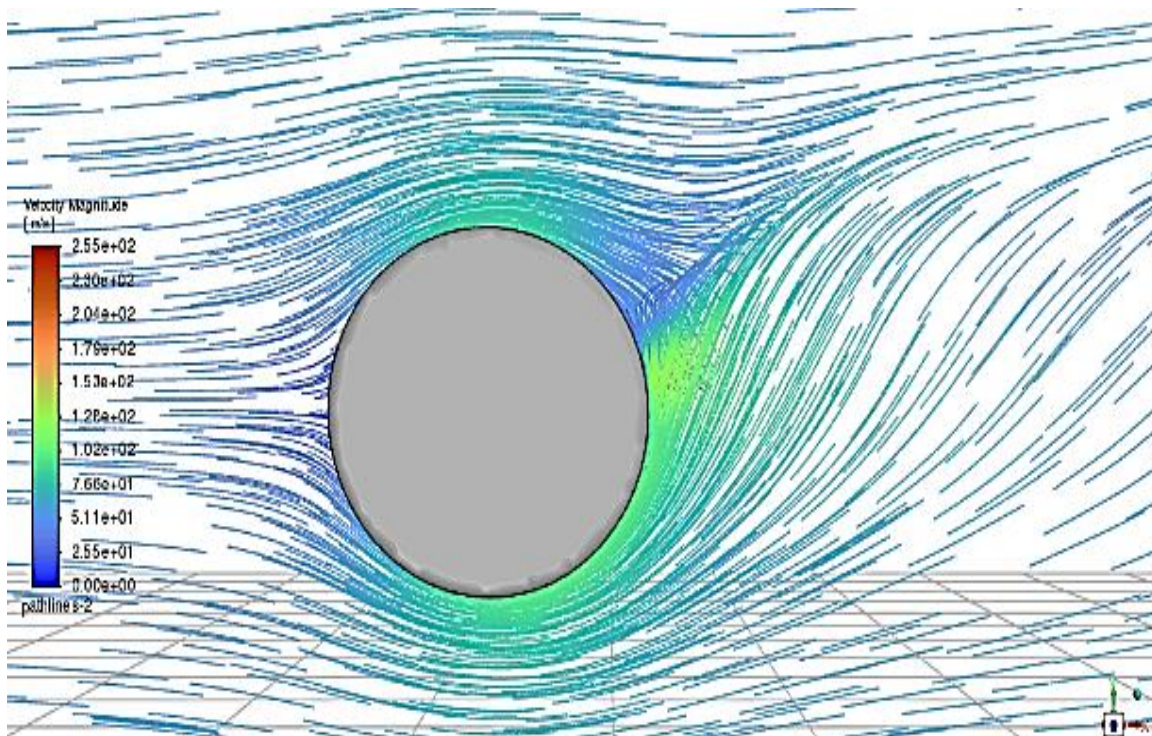


Figure 13. Pathlines around elliptical building (top view).

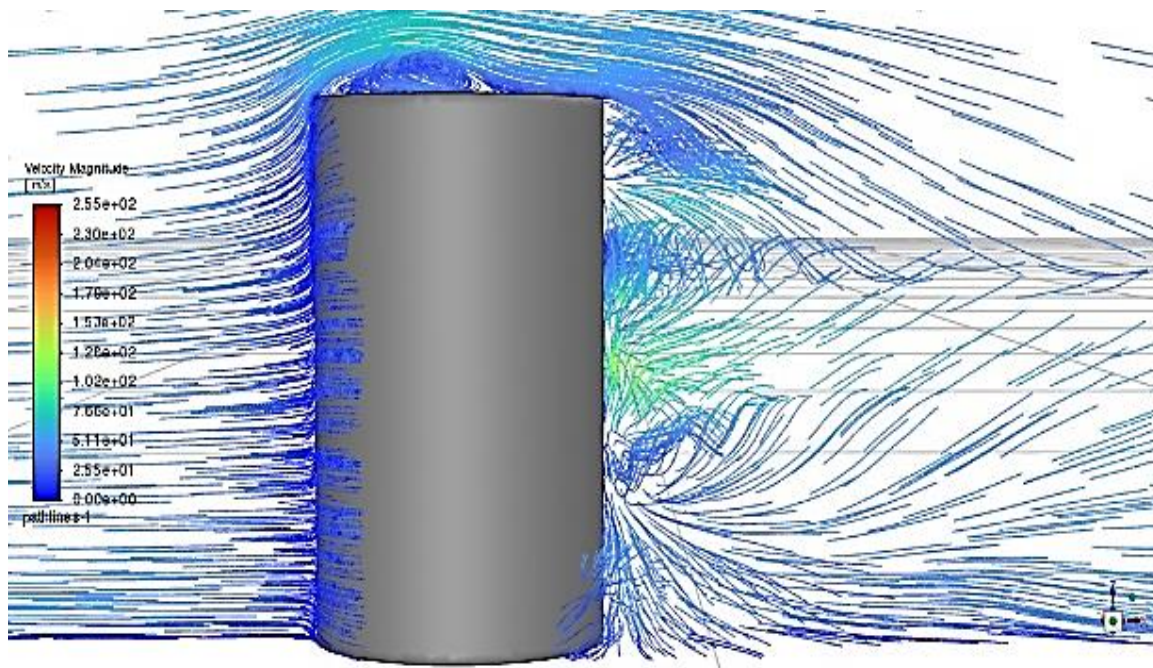


Figure 14. Pathlines around elliptical building (side view).

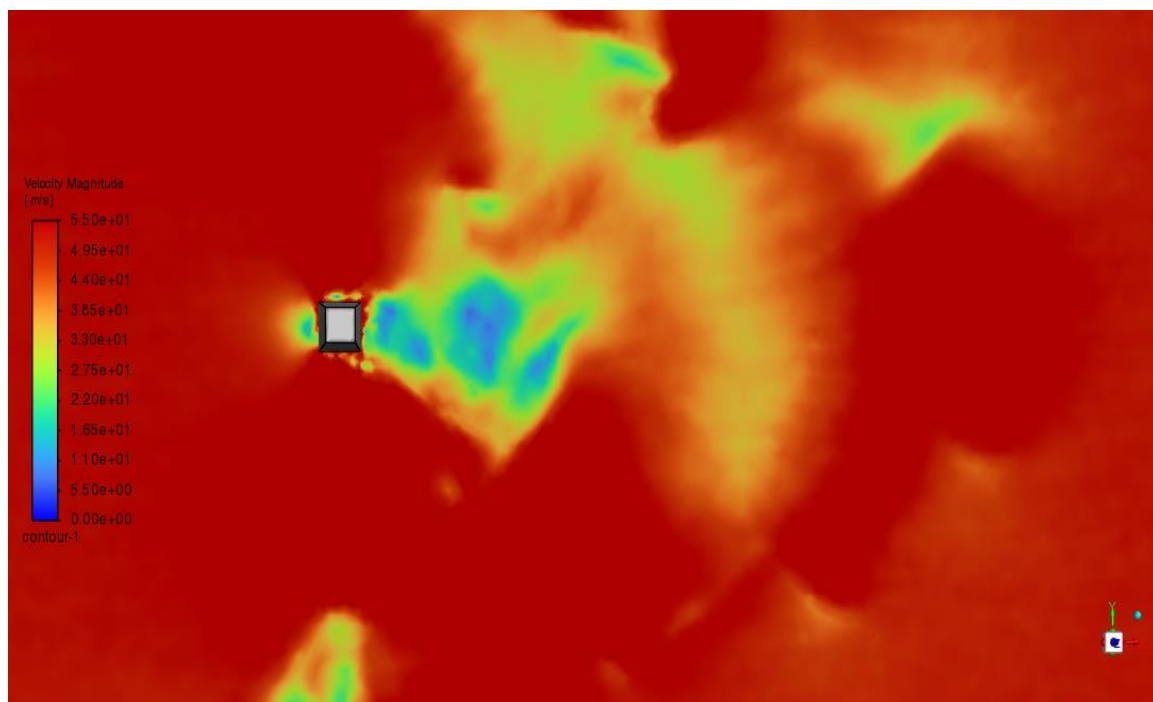


Figure 15. Velocity contours of tapered building.

Tapered Building ***Velocity Distribution***

The velocity contours show that the higher the height, the lower the flow obstruction and the smoother the flow, as shown in Figure 15.

Pressure Distribution

The pressure decreased progressively with height, thereby reducing the concentration in any one region, as shown in Figure 16.

Flow Pattern (Pathlines)

The pathlines show disrupted vortex formation and a reduced wake region, as shown in Figures 17 and 18.

Observation

The tapered shape reduces the aerodynamic effects by changing the flow interaction along the height, thereby improving the performance. Changes in the cross-sectional area along the height modified the vortex development and reduced the concentrated wind interaction.

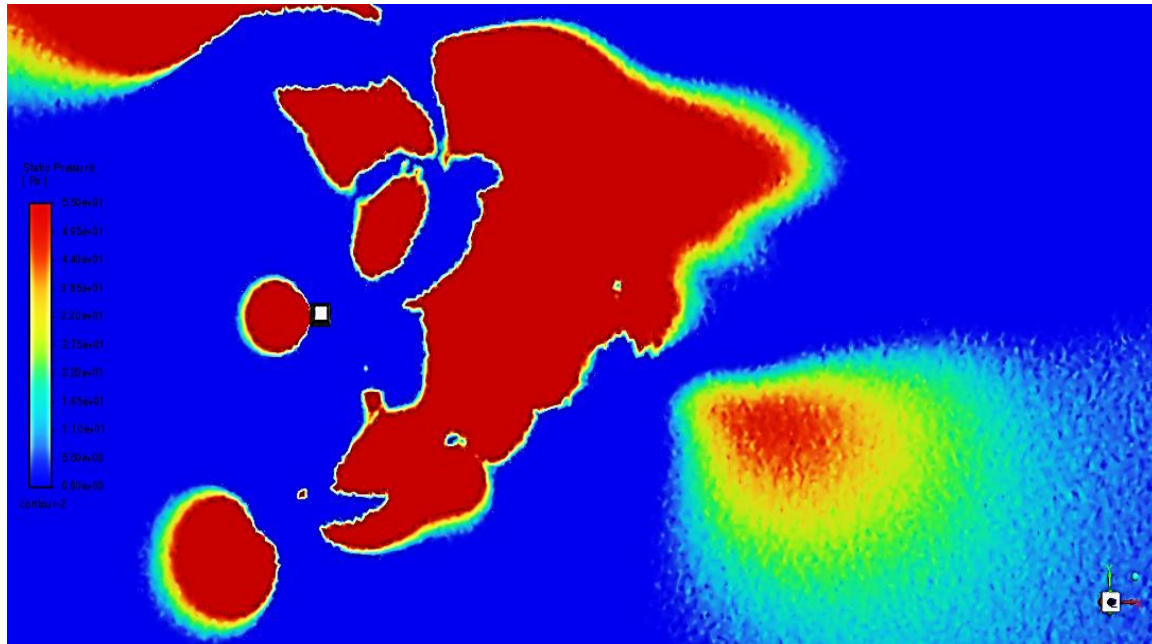


Figure 16. Static pressure contours of tapered building.

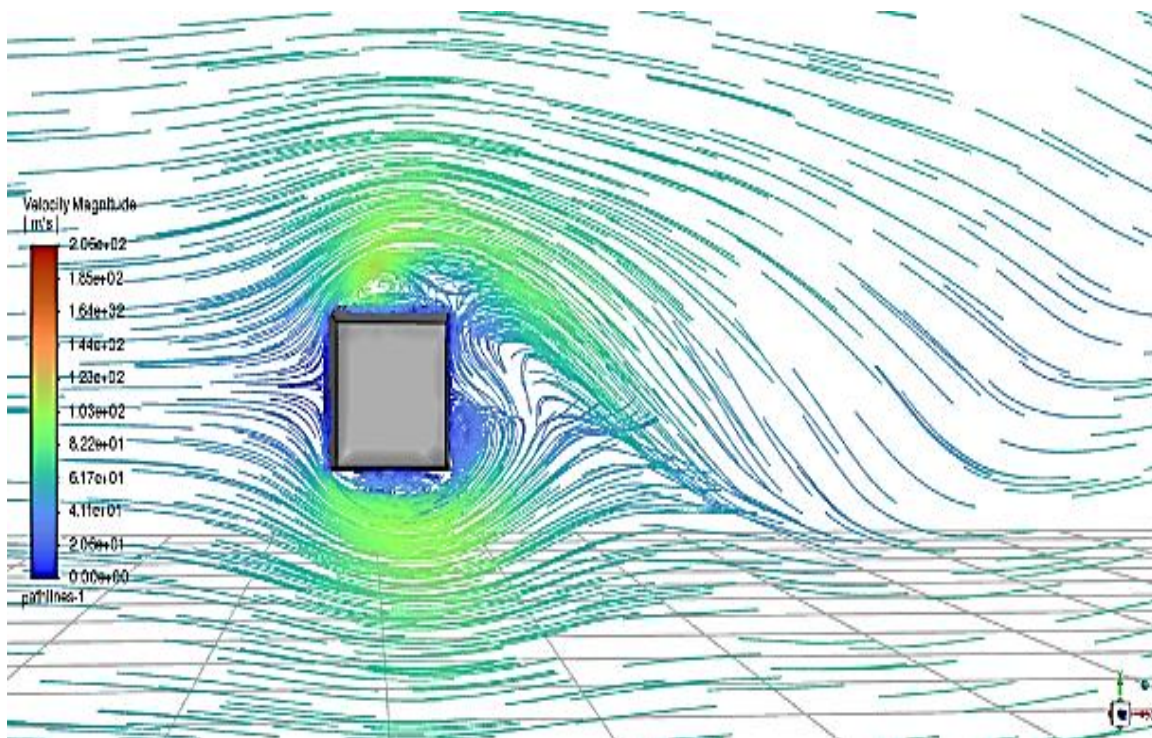


Figure 17. Pathlines around tapered building (top view).

Twisted Building

Velocity Distribution

The velocity distribution shows variations along the height owing to the twisting geometry, which continuously alters the wind interaction, as shown in Figure 19.

Pressure Distribution

The pressure zones were distributed unevenly but with a reduced peak concentration compared with the rectangular model, as shown in Figure 20.

Flow Pattern (Pathlines)

The pathlines indicate disrupted vortex shedding and irregular wake formation (Figures 21 and 22).

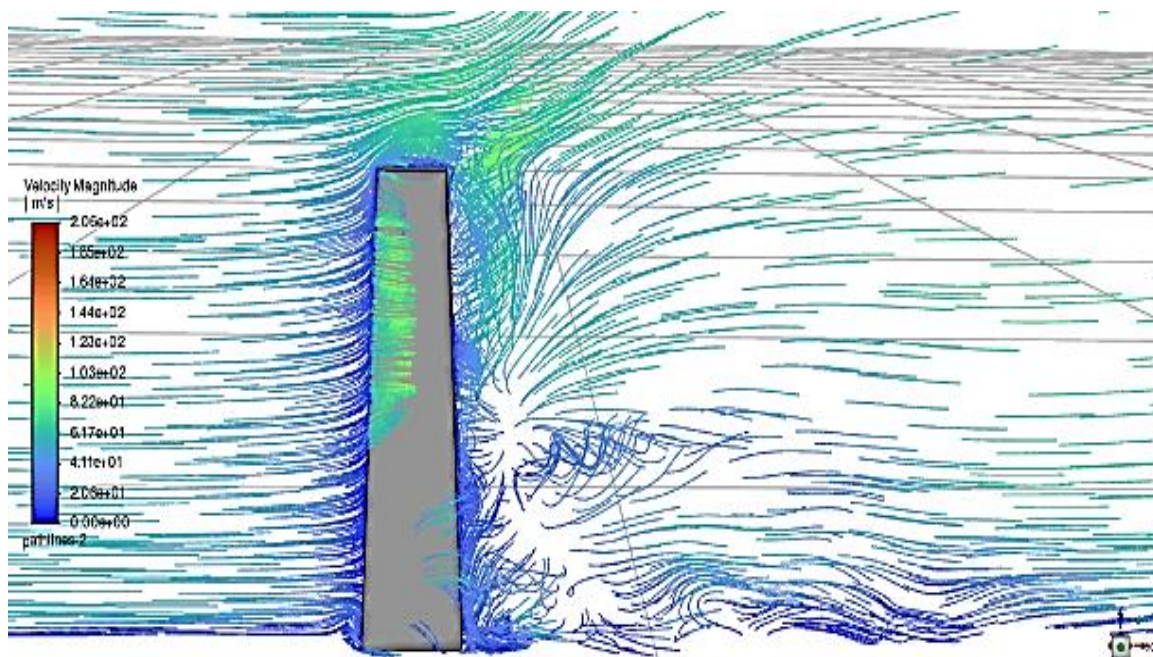


Figure 18. Pathlines around tapered building (side view).

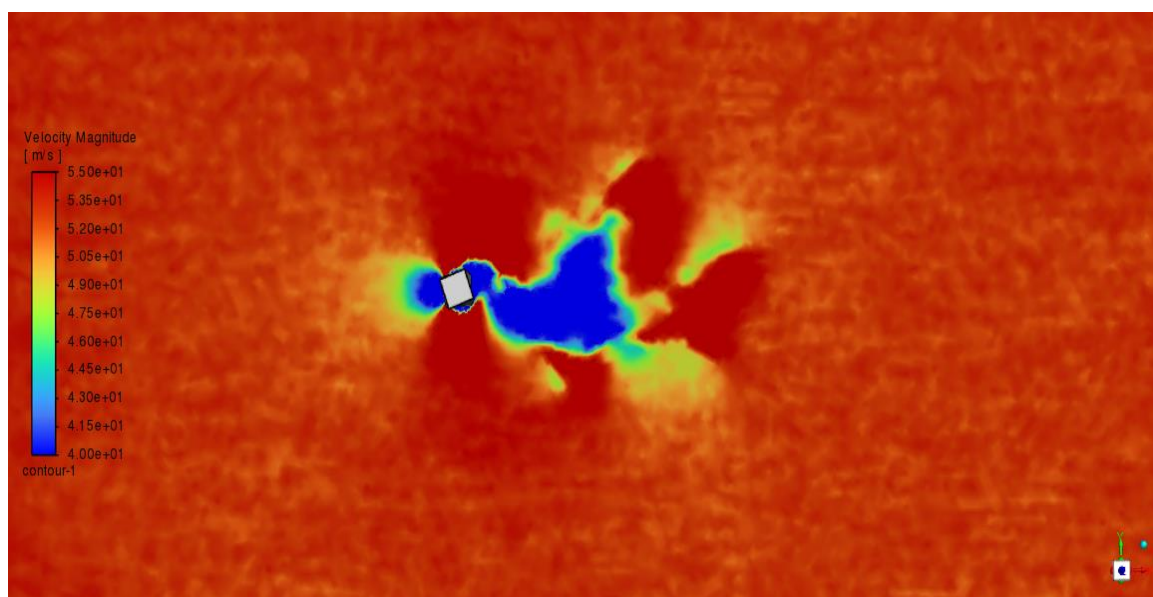


Figure 19. Velocity contours of twisted building.

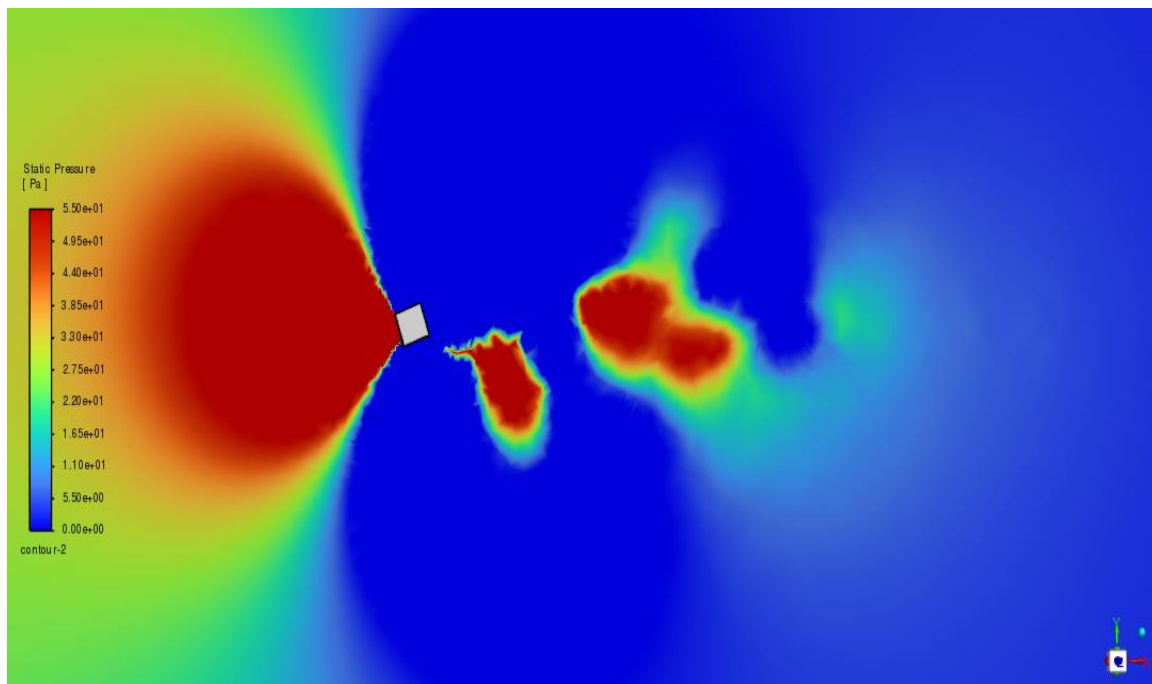


Figure 20. Static pressure contours of twisted building.

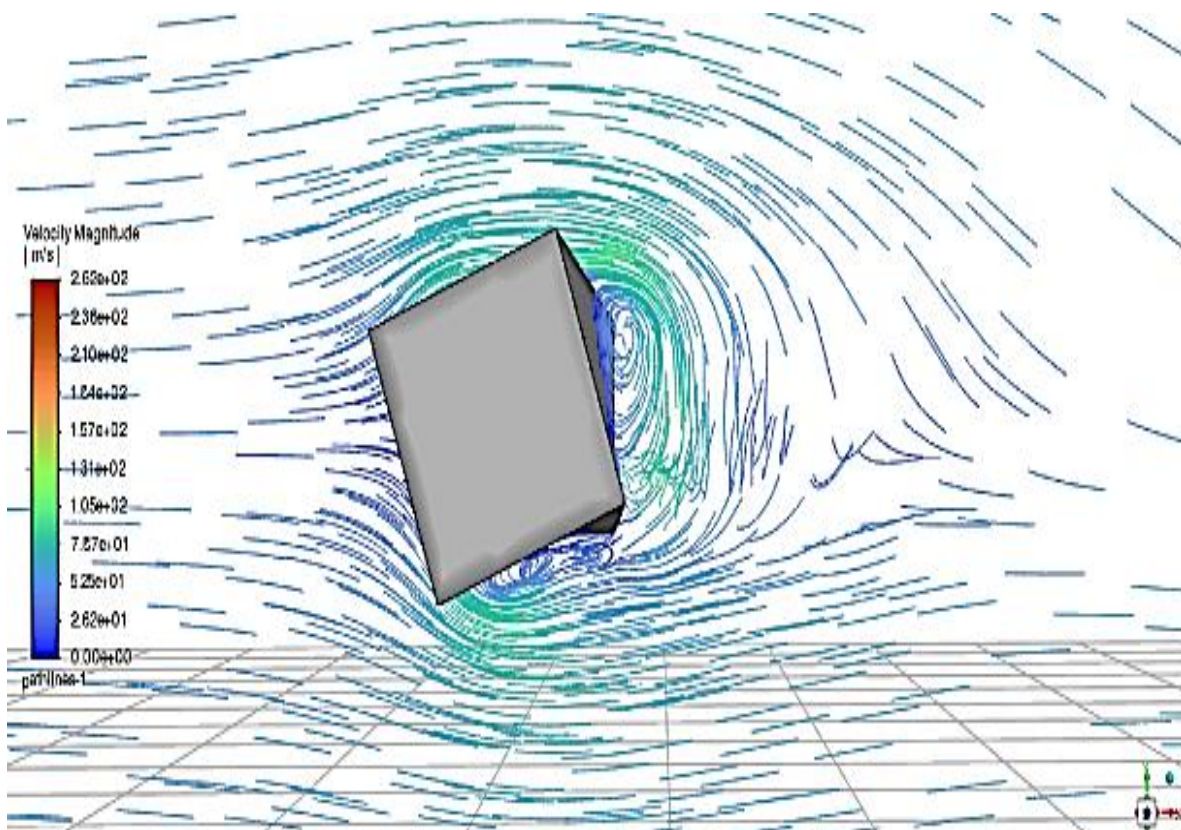


Figure 21. Pathlines around twisted building (top view).

Observation

The twisted geometry helps break up the vortex formation, thus achieving relatively stable flow characteristics. The continuously changing orientation disrupts the organized vortex shedding, which contributes to the comparatively stable flow behavior.

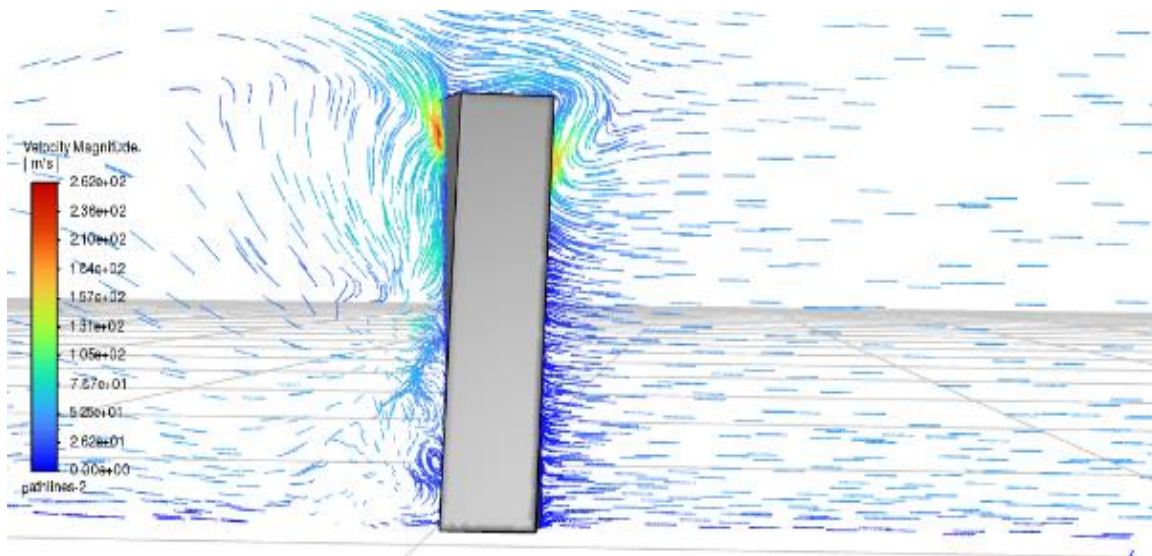


Figure 22. Pathlines around twisted building (side view).

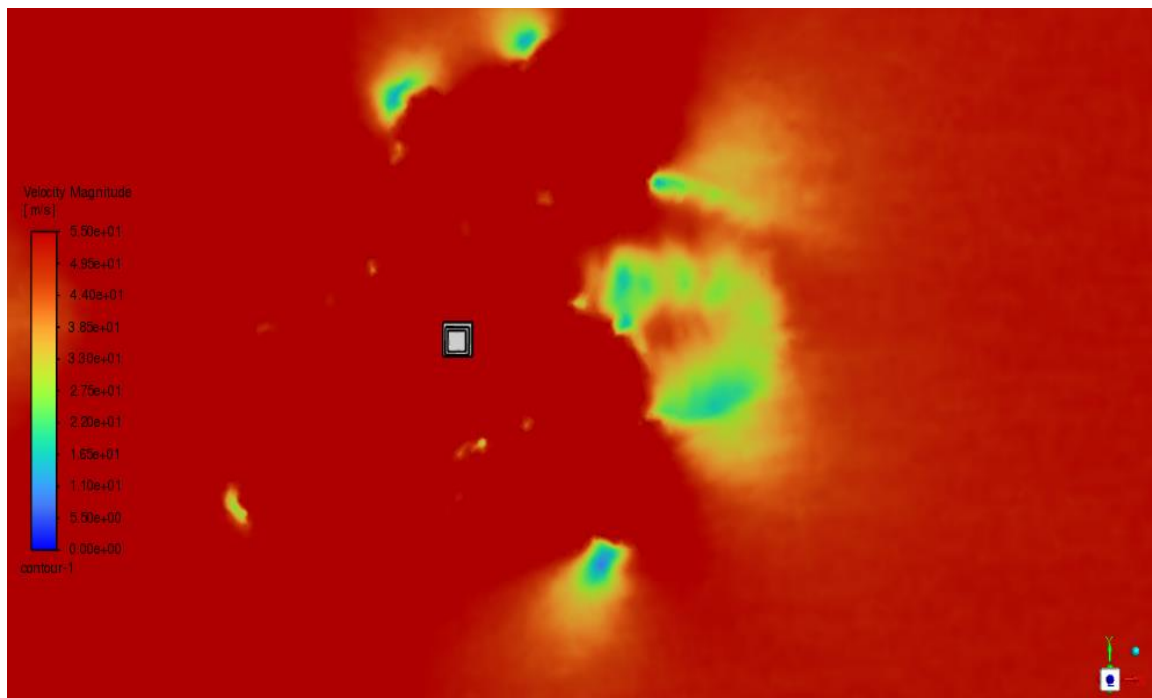


Figure 23. Velocity contours of setback configuration building.

Setback Building

Velocity Distribution

The velocity contours for the setback building showed a non-uniform distribution of flow along the height owing to geometric discontinuities. The stepped profile causes local acceleration and deceleration of the airflow at each setback level. The flow was partially redistributed compared with the rectangular model, but localized high-velocity zones were observed close to the edges of each step, as shown in Figure 23.

Pressure Distribution

The pressure contours show high-suction regions in the setback regions, particularly at the upper levels. Unlike the smoother geometries of elliptical or rounded models, geometric discontinuity causes the pressure to concentrate at the step edges, leading to a non-uniform pressure distribution.

Flow Pattern (Pathlines)

Pathline visualization clearly illustrated the different heights of the multiple vortex formations. The rectangular building created a large wake, whereas the setback design created intermittent recirculation zones at each step. This led to a disintegrated wake structure instead of a single continuous wake (Figures 24 and 25).

Observation

This behavior shows that setback configurations tend to shift the aerodynamic effects rather than consistently reducing them, as is the case for streamlined geometries. Simplified geometries have a uniform effect on aerodynamics, whereas backstep configurations redistribute the aerodynamics into multiple separation zones.

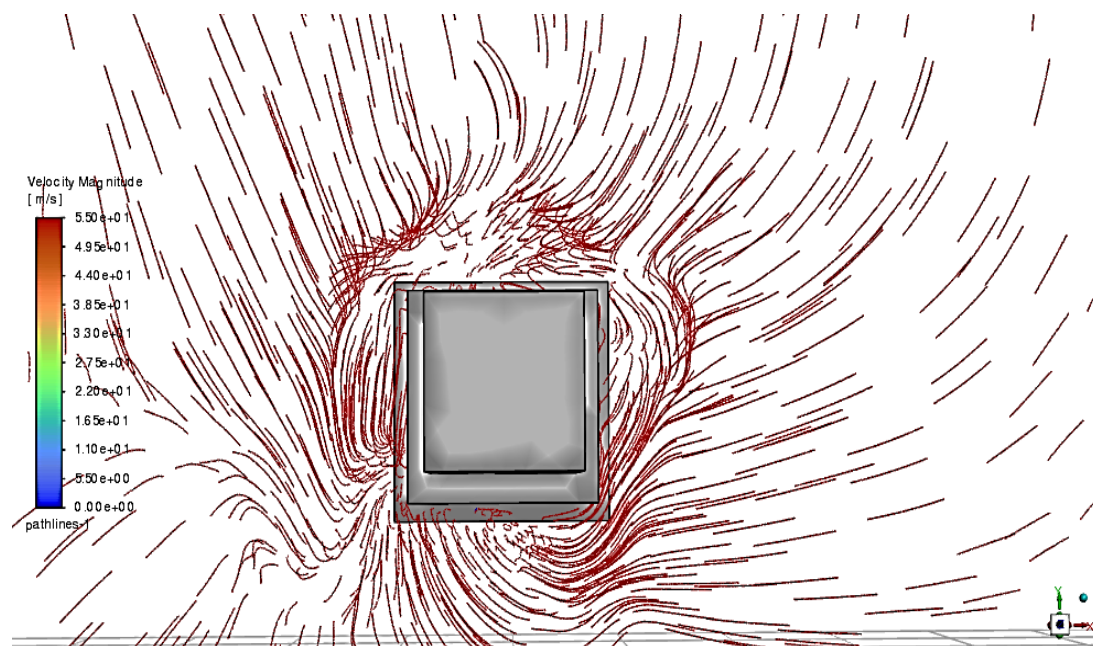


Figure 24. Pathlines around setback building (top view).

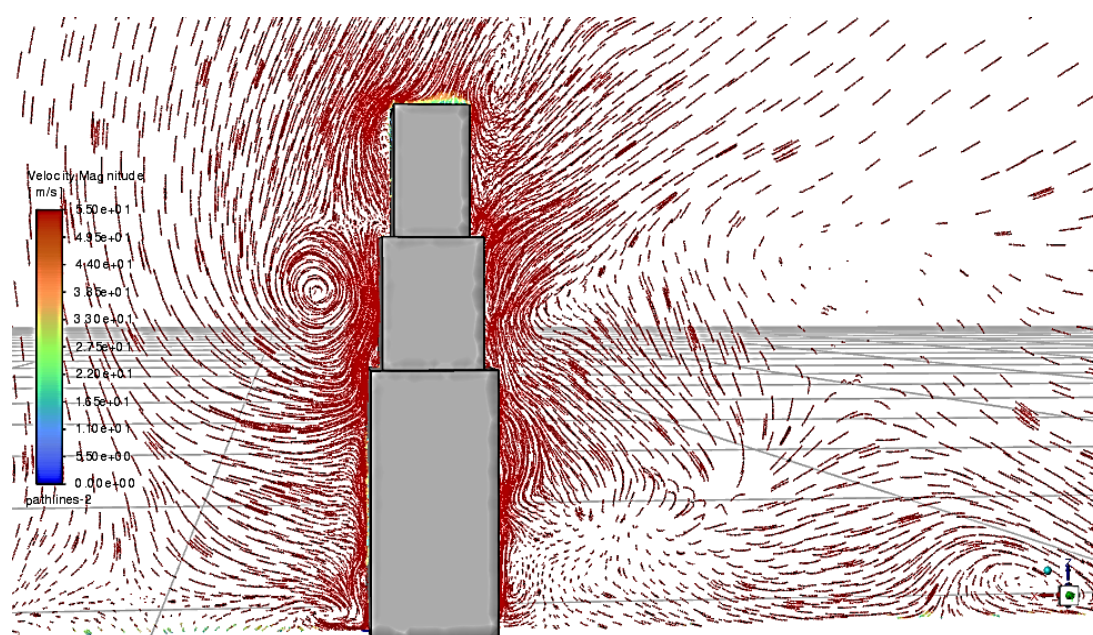


Figure 25. Pathlines around setback building (side view).

Rounded-Corner Building (Minor Modification)

Velocity Distribution

The velocity contours show a smoother transition of airflow at the corners, reducing abrupt changes, as shown in Figure 26.

Pressure Distribution

The pressure concentration at the sharp edges was significantly reduced, leading to a more uniform distribution, as shown in Figure 27.

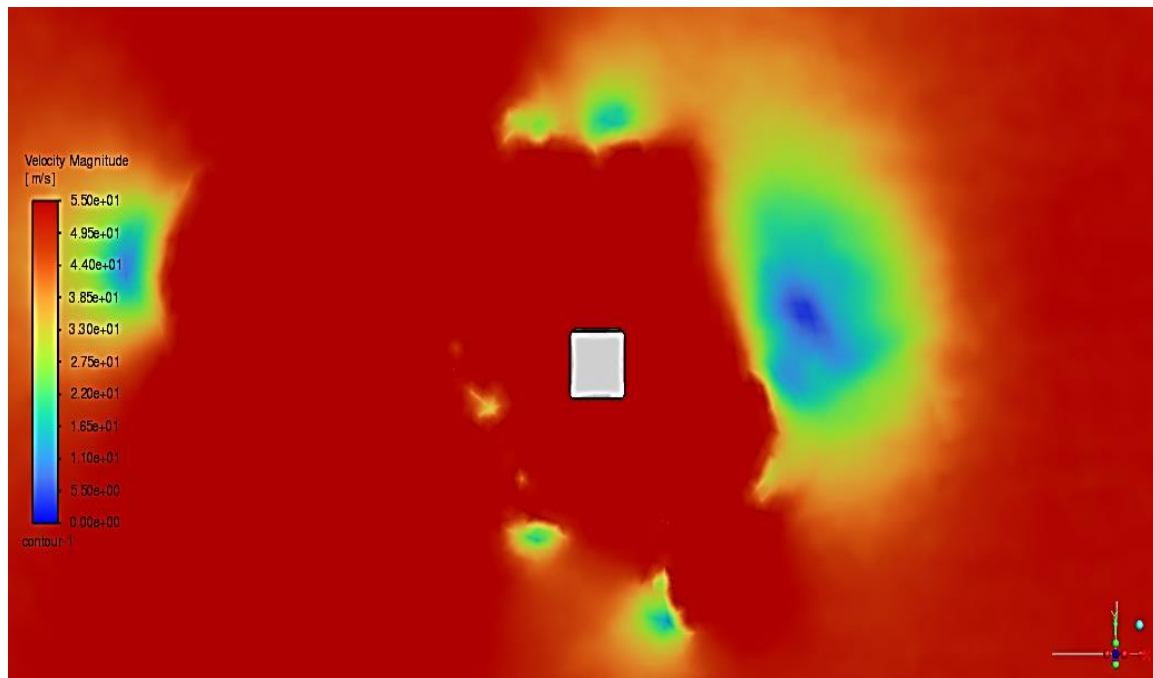


Figure 26. Velocity contours of rounded-corner building.

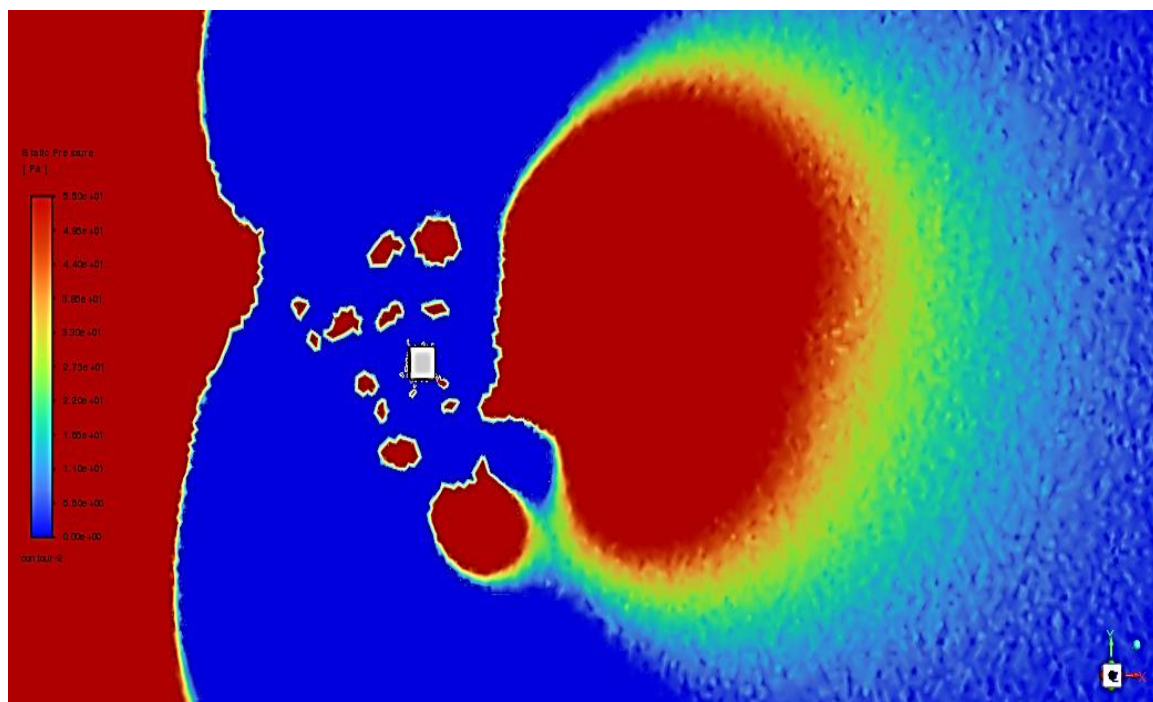


Figure 27. Static pressure contours of rounded-corner building.

Flow Pattern (Pathlines)

The pathlines indicate delayed separation and a reduced wake region compared to the base model, as shown in Figures 28 and 29.

Observation

Rounded corners are very effective for reducing flow separation and turbulence and improving aerodynamic behavior without major changes to the structure. This change results in an aerodynamic improvement and is relatively small in terms of the overall geometry of the building.

Chamfered-Corner Building (Minor Modification)

Velocity Distribution

The velocity contours show a moderate improvement in the flow behavior compared to the rectangular model (Figure 30).

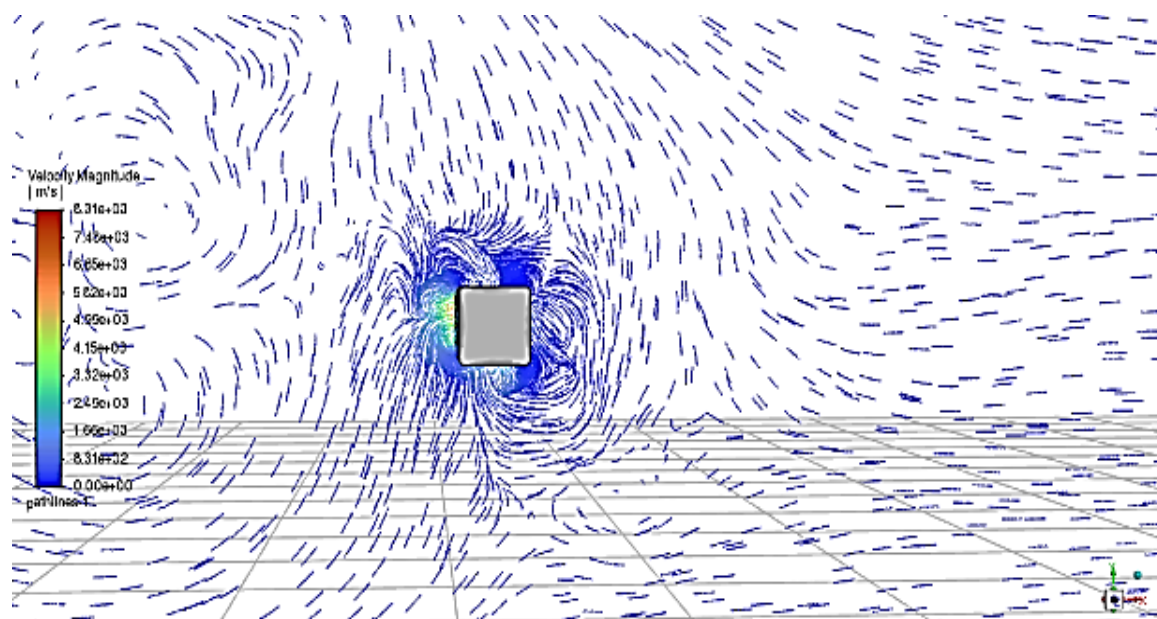


Figure 28. Pathlines around rounded-corner building (top view).

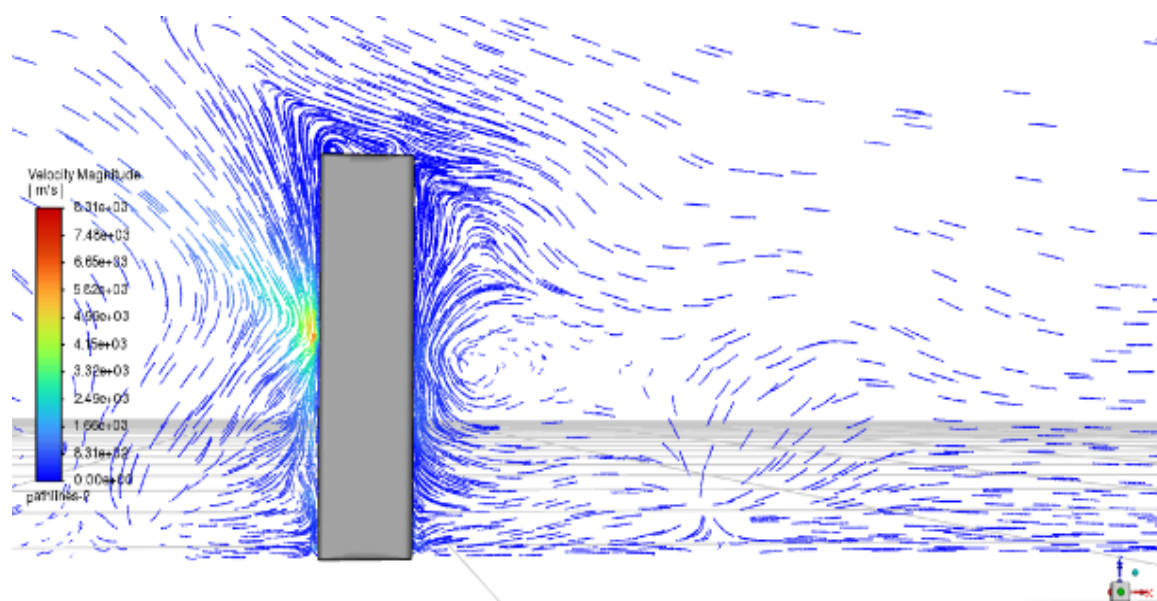


Figure 29. Pathlines around rounded-corner building (side view).

Pressure Distribution

The pressure distribution was improved, although it was slightly less uniform than that of the rounded configuration, as shown in Figure 31.

Flow Pattern (Pathlines)

The wake region was reduced, but some vortex formation was still visible, as shown in Figures 32 and 33.

Observation

The chamfered corners provided a moderate increase in aerodynamics, but not as much as the rounded corners. Corners were chamfered to eliminate sharp edges from separation; however, the flow was not as completely altered as with smoother corner modifications.

Key Findings

The results of the comparison analysis show that significant changes in the aerodynamic properties of the cross-section mainly result in changes in the global flow field, whereas small changes mainly result in improvement of the local flow field near the edge and corners. These observed aerodynamic characteristics were found to be consistent with previous wind engineering studies that showed that streamlined and tapered shapes reduce wake formation and increase aerodynamic efficiency.

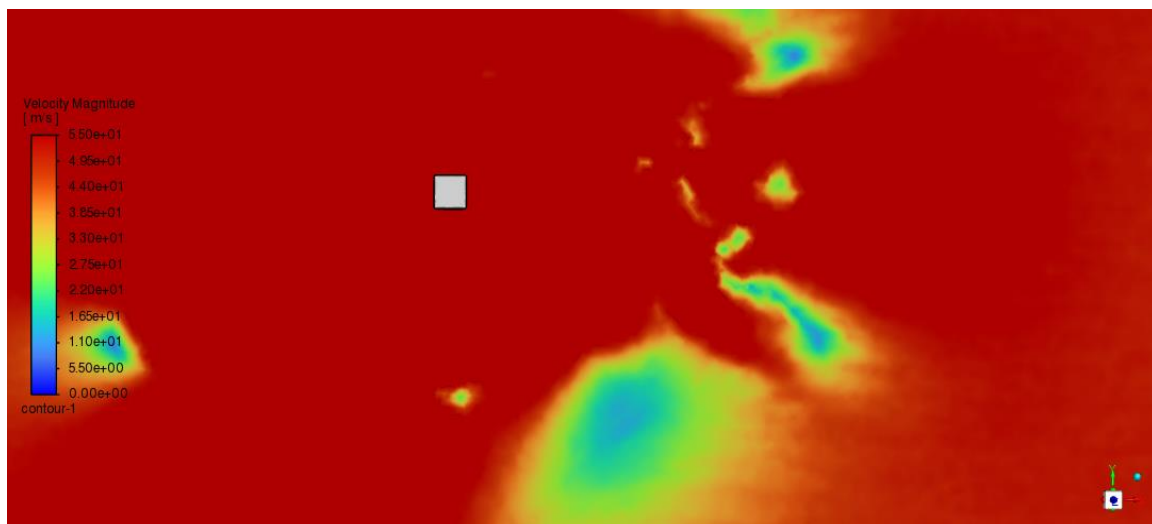


Figure 30. Velocity contours of chamfered-corner building.

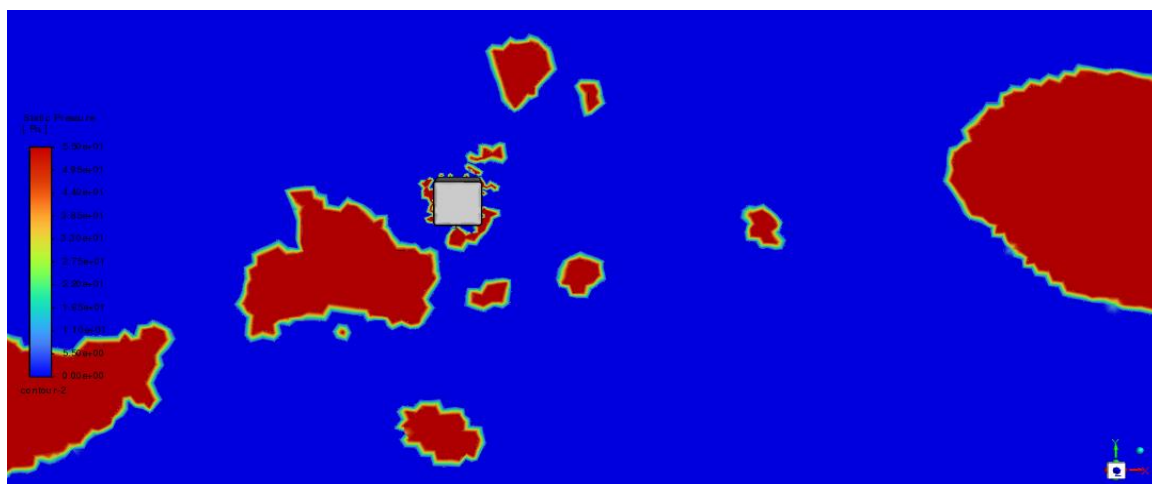


Figure 31. Static pressure contours of a chamfered-corner building.

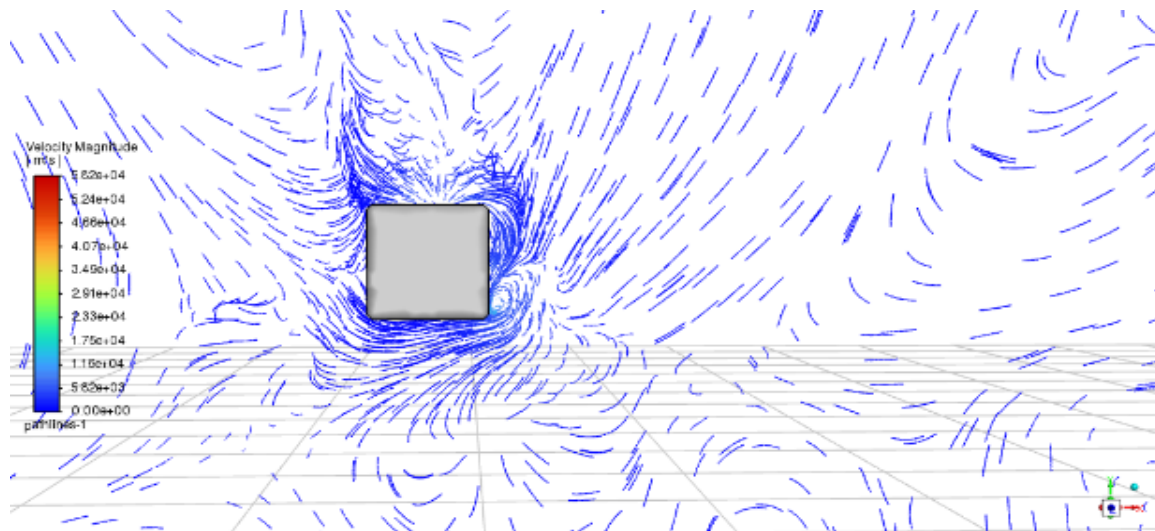


Figure 32. Pathlines around chamfered-corner building (top view).

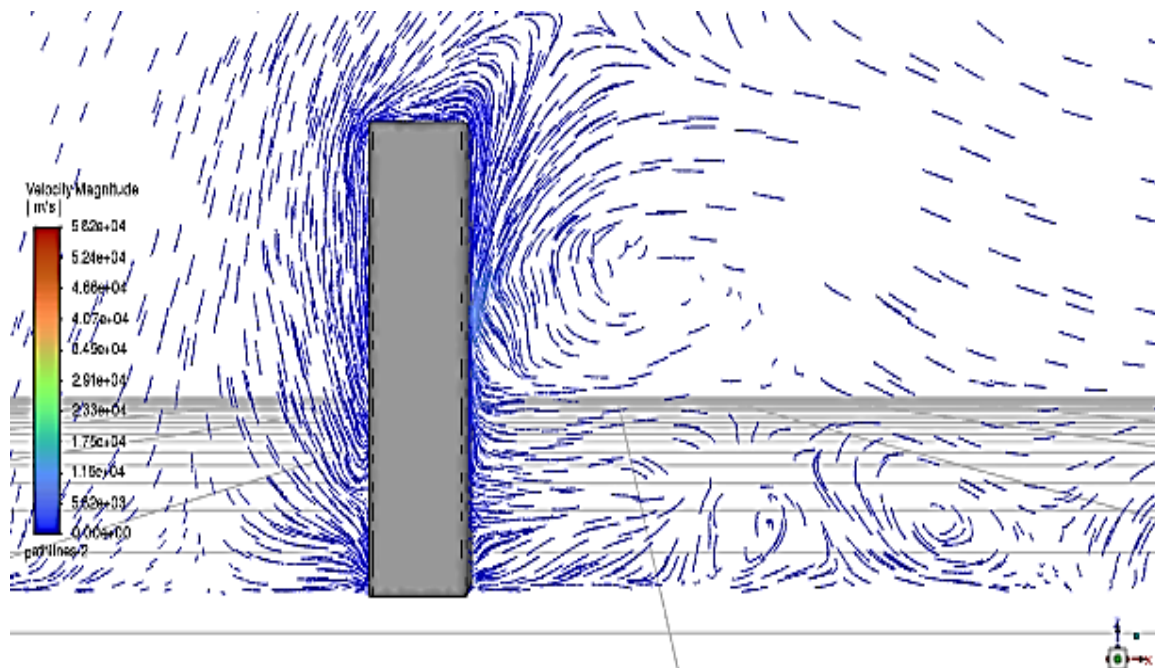


Figure 33. Pathlines around chamfered-corner building (side view).

The absolute drag forces may not be comparable when used for direct comparison in CFD simulations because of different force scaling and reference parameters. Thus, an approach using the normalized drag coefficient (C_d/C_{d0}) was employed, with the rectangular building used as the reference building. This technique allows for a consistent and meaningful comparison of the aerodynamic performance of various configurations. Qualitative flow properties, such as wake size, pressure distribution, and flow separation behavior, observed from CFD visualization further support the observed trends (Table 3).

The maximum aerodynamic resistance was achieved for the rectangular building because the flow separated from the surface of the building early owing to the presence of sharp edges, and a large wake region was formed. On the other hand, streamlined shapes such as elliptical and tapered shapes are preferred because they have comparatively less drag owing to the ease of flow and delay in separation. Moreover, geometric changes, particularly rounding of corners, will greatly reduce the local level of turbulence and cause pressure concentration at the edges, which will be beneficial. The chamfered corners help control the flow, but only moderately, as they do not completely smoothen the flow.

Table 3. Comparative aerodynamic performance of different building shapes.

Shape	Normalized drag coefficient (Cd/Cd ₀)	Relative wake size	Pressure intensity	Aerodynamic performance
Rectangular	1.00	High	High	Poor
Circular	0.85–0.90	Moderate	Moderate	Moderate
Elliptical	0.65–0.75	Low	Moderate	Good
Tapered	0.70–0.80	Low	Low–Moderate	Good
Twisted	0.80–0.90	Moderate	Moderate	Moderate
Setback	0.90–1.05	Fragmented	High (localized)	Moderate
Rounded corners	0.75–0.85	Low	Low	Good
Chamfered corners	0.85–0.95	Moderate	Moderate	Moderate

The setback shape has an aerodynamic behavior that is different from the other shapes. The novel approach introduces multiple separation points along the height instead of a uniform reduction in drag owing to geometric discontinuities. This leads to a wake being split and local pressure differences, leading to modest global aerodynamic performance. This geometry also exhibited a twisted disorganization of vortex shedding, which caused moderate flow stability.

Finally, the results indicate that substantial changes have a significant impact on the global flow characteristics, whereas small changes result in better local flow characteristics. Under the conditions of the present simulation, the elliptical and tapered configurations were relatively better aerodynamically among all the configurations considered.

The trends for the aerodynamic responses revealed in this study are qualitatively consistent with those observed in other wind engineering investigations, in which streamlined and tapered buildings were observed to be less prone to wake development and exhibited improved aerodynamic characteristics compared with traditional rectangular buildings.

Approximate Drag Force Estimation

The standard aerodynamic drag equation was employed to approximate the drag forces for a comparable understanding of the aerodynamic resistance of different building configurations. The present study is mainly for comparative CFD flow behavior under the same computational conditions; therefore, the drag force estimation is for relative aerodynamic evaluation rather than a highly precise force prediction.

The drag force was estimated by:

$$FD = \frac{1}{2} \rho V^2 CD A$$

Where:

- FD = Drag force (N)
- ρ = Air density (1.225 kg/m³)
- V = Wind velocity (55 m/s)
- CD = Drag coefficient
- A = Projected frontal area of building

The frontal area of the building: $A = 75 \times 300 = 22500 \text{ m}^2$

$$FD \approx 4.17 \times 10^7 \text{ (CD)}$$

The approximate values of the drag coefficients were used based on the comparative aerodynamic behavior observed in the CFD simulations and trends reported in the literature for similar building geometries, as shown in Table 4.

Table 4. Approximate drag force estimation for different building shapes.

Shape	Approx. drag coefficient (CD)	Estimated drag force (MN)	Comparative aerodynamic behavior
Rectangular	1.20	50.0	High drag and large wake formation
Circular	0.80	33.4	Reduced flow separation
Elliptical	0.60	25.0	Streamlined flow with reduced turbulence
Tapered	0.70	29.2	Reduced vortex formation
Twisted	0.85	35.4	Disrupted vortex shedding
Setback	1.00	41.7	Fragmented wake structure
Rounded corners	0.75	31.3	Reduced edge separation
Chamfered corners	0.90	37.5	Moderate aerodynamic improvement

The calculated aerodynamic drag force data indicate that the aerodynamic drag of streamlined geometries, such as elliptical and tapered configurations, is comparatively lower than that of the normal rectangular model. Small changes, such as rounding the corners, also show significant drag reduction without a major change in the overall building geometry. These values are approximate in nature but provide additional comparative insight into the aerodynamic effectiveness of different geometric modifications under the same wind conditions.

The trends of the estimated drag were qualitatively consistent with the wake characteristics and pressure distributions observed in the CFD simulations.

Limitations of Study

The present study is based on steady-state CFD simulations with a uniform inlet velocity profile. The transient vortex shedding behavior, wind direction, terrain conditions, and effects of the atmospheric boundary layer were not considered. Furthermore, the evaluation of structural response parameters, such as displacement and acceleration, was not conducted. The conclusions were based only on numerical simulations without experimental validation, which could affect the accuracy of the predicted aerodynamic behavior. The steady-state approach may not fully capture the transient vortex shedding phenomena associated with across-wind excitation in tall buildings.

CONCLUSIONS

In this study, a comparative CFD analysis was performed to evaluate the aerodynamic behavior of tall building configurations with major (global) and minor (local) geometrical modifications under wind loads. Therefore, all configurations were analyzed under the same computational conditions to ensure consistency and provide a fair basis for comparison.

The base rectangular building exhibited significant flow separation at the sharp edges, causing a large wake region and large pressure gradients between the windward and leeward faces. This indicates relatively larger aerodynamic effects than those of the modified configurations.

Several major modifications, such as circular, elliptical, tapered, twisted, and setback geometries, exhibited altered flow behavior. Elliptical and tapered configurations with streamlined shapes showed better flow transition with fewer wake regions, indicating better aerodynamic performance under the present conditions. The twisted geometry disturbed the regular vortex shedding, causing moderate flow stability. In contrast, the setback configuration generated several separation points along the height, leading to fragmented wake structures and localized pressure variations instead of a uniform reduction in the aerodynamic effects.

Further minor modifications, such as rounding and chamfering of corners, improved the aerodynamic behavior and local flow characteristics. Moderate improvement was observed in the chamfered corners, whereas the rounded corners were more effective in delaying flow separation and reducing pressure concentration at the edges.

The study showed that the major changes mainly influenced the global flow field, whereas the minor changes improved the local flow behavior, particularly at the edges and corners. Major modifications may provide a higher level of aerodynamic improvement, but minor modifications are a more straightforward and practical way to improve performance without major changes to the overall building form.

The results show that aerodynamic shape optimization at the preliminary design stage can significantly influence the wind performance of tall buildings. Overall, the results show that the flow pattern around tall buildings is strongly affected by geometry. The results were obtained from steady-state CFD simulations and provided comparative insights into the aerodynamic performance of the different configurations. The results are useful for preliminary design assessments, where shape selection is a key parameter for the aerodynamic response.

Small geometric changes, such as rounded corners, could be a pragmatic trade-off between aerodynamic improvement, architectural feasibility, and structural simplicity.

Future Scope

The purpose of the present study was to gain a comparative understanding of aerodynamic behavior, but there are many extensions that could further improve the applicability and depth of the analysis.

Further work is required to simulate time-dependent vortex shedding and crosswind effects. The use of experimental data or benchmark studies would also add confidence to numerical results.

Various wind directions, terrain types, and atmospheric boundary layer profiles can be used for a better representation of real-world conditions. Further research could involve the use of combined aerodynamic modifications, where major and minor aerodynamic modifications are used together to study the combined effect.

Moreover, optimization techniques or machine learning-based approaches can be investigated to identify effective building shapes with less computational effort. Using a combination of CFD and structural analysis tools, the parameters of the structural response, such as displacement, acceleration, and comfort criteria, can be evaluated.

They can contribute to the creation of better aerodynamic design strategies for tall buildings subjected to wind loads that are more reliable and performance-oriented.

REFERENCES

1. Yoga M, Vijaya S, Jyothi MB. Aerodynamic shape optimization for high-rise buildings using ETABS. *Int Res J Eng Technol*. 2025;12(8):416–419.
2. Maletha KP, Verma A. Analysis of wind load on various shape of building: a review. *Int J Struct Des Eng*. 2025;6(1):21–29. doi:10.22271/27078280.2025.v6.i1a.39.
3. Dhami RS, Verma A. Aerodynamic performance of high-rise buildings with orifices: a CFD study. *Int J Res Civ Eng Technol*. 2025;6(1):46–52. doi:10.22271/27078264.2025.v6.i1a.80.
4. Shanthi S, Vidjeapriya R, Jaya KP. Influence of across-wind on rectangular tall buildings. *Buildings*. 2024;14(12):3902. doi:10.3390/buildings14123902.
5. Anusuya Devi VS, Anjaneyulu M, Nagpal A, Chandra PK, Jabbar ASAA, Tyagi LK, et al. A comparative study of analyzing the impact of various RCC building shapes on wind performance. *E3S Web Conf*. 2023;[volume unknown]:1–8.
6. Parihar AS, Parihar S, Dubey V. Aerodynamic shape optimization of a tall building. *Res Militaris*. 2023;13(2):7370–7380.
7. Paul R, Dalui SK. Aerodynamic shape optimization of a high-rise rectangular building with wings. *Wind Struct*. 2022;34(3):259–274.

8. Zhu H, Yang B, Zhang Q, Pan L, Sun S. Wind engineering for high-rise buildings: a review. *E3S Web Conf.* 2022;355:1–9.
9. Li J, Du X, Martins JRRA. Machine learning in aerodynamic shape optimization. *Prog Aerosp Sci.* 2022;134:100849. doi:10.1016/j.paerosci.2022.100849.
10. Wang Z, Zheng C, Mulyanto JA, Wu Y. Aerodynamic shape optimization of a square cylinder with multiparameter corner recession modifications. *Atmosphere.* 2022;13(11):1782. doi:10.3390/atmos13111782.
11. Verma A, Meena RK, Dubey H, Raj R, Anbukumar S. Wind effects on rectangular and triaxial symmetrical tall building having equal area and height. *Complexity.* 2022;2022:4815623. doi:10.1155/2022/4815623.
12. He Z, Liang T, Lai X, Gao M, Tu X, Lu Y. Vibration acceleration-integrated parameterized aerodynamic shape optimization of super high-rise buildings with spiral configurations. *Struct Multidiscip Optim.* 2022;65:291. doi:10.1007/s00158-022-03387-7.
13. Jafari M, Alipour A. Aerodynamic shape optimization of rectangular and elliptical double-skin façades to mitigate wind-induced effects on tall buildings. *J Wind Eng Ind Aerodyn.* 2021;213:104586. doi:10.1016/j.jweia.2021.104586.
14. Bairagi AK, Dalui SK. Distribution of wind pressure around different shape tall building. In: Vinyas M, Loja A, Reddy KR, editors. *Advances in Structures, Systems and Materials.* Singapore: Springer; 2020. p.31–38. doi:10.1007/978-981-15-3254-2_4.
15. Nagar SK, Raj R, Dev N. Experimental study of wind-induced pressures on tall buildings of different shapes. *Wind Struct.* 2020;31(5):441–453.
16. Yeretizian A, Partamian H, Dabaghi M, Jabr R. Integrating building shape optimization into the architectural design process. *Archit Sci Rev.* 2020;63:63–73. doi:10.1080/00038628.2019.1689912.
17. Hosseini SM, Mohammadi M, Guerra-Santin O. Interactive kinetic façade: improving visual comfort based on dynamic daylight and occupant's positions by 2D and 3D shape changes. *Build Environ.* 2019;165:106396. doi:10.1016/j.buildenv.2019.106396.
18. Mallick M, Mohanta A, Kumar A, Raj V. Modelling of wind pressure coefficients on C-shaped building models. *Model Simul Eng.* 2018;2018:6524945. doi:10.1155/2018/6524945.
19. Bairagi AK, Dalui SK. Comparison of aerodynamic coefficients of setback tall buildings due to wind load. *Asian J Civ Eng.* 2018;19(8):205–221. doi:10.1007/s42107-018-0018-3.
20. Asghari Mooneghi MA, Kargarmoakhar R. Aerodynamic mitigation and shape optimization of buildings: review. *J Build Eng.* 2016;6:225–235. doi:10.1016/j.jobbe.2016.01.009.
21. Elshaer A, Bitsuamlak G, El Damatty A. Aerodynamic shape optimization for tall buildings using CFD and genetic algorithms. In: *Proceedings of the International Conference on Wind Engineering.* 2015. p.1–10.
22. Bairagi AK, Dalui SK. Evaluation of interference effects on parallel high-rise buildings for different orientation using CFD. In: *Proceedings of the World Conference on Applied Science, Engineering and Technology.* 2014. p.1–6.
23. Tse KT, Hitchcock PA, Kwok KCS, Thepmongkorn S, Chan CM. Economic perspectives of aerodynamic treatments of square tall buildings. *J Wind Eng Ind Aerodyn.* 2009;97:455–467. doi:10.1016/j.jweia.2009.07.005.
24. Kim YM, You KP. Dynamic responses of a tapered tall building to wind loads. *J Wind Eng Ind Aerodyn.* 2002;90:1771–1782. doi:10.1016/S0167-6105(02)00286-6.
25. Dulikravich GS, Martin MM, Dennis J. Aerodynamic shape design and optimization. *Aerosp Sci Technol.* 1992;1(1):1–12.