

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF THE AERODYNAMIC PERFORMANCE OF Y-SHAPED TALL BUILDINGS WITH CHAMFERED CORNERS MODIFICATION

by Mohd Aftab

Submission date: 11-Jun-2026 03:37AM (UTC+0530)

Submission ID: 2980720250

File name: Mohd_Aftab_Plag.pdf (6.89M)

Word count: 14053

Character count: 74153

ABSTRACT

Modern urbanization has witnessed the emergence of new buildings, particularly tall buildings with unconventional forms like Y-plan buildings. This study tackles the significant problem of wind loading on tall, non-symmetrical structures such as the Y-shaped structure commonly found in urban design of which there are increasing numbers. The main issue investigated is how high wind stresses can create structural instability and structural problems related to strong wind suction (negative forces) affecting the building's facade and structure. To overcome this, the aerodynamic performance of the SY model is considered and the effectiveness of a Chamfered Y-shaped (CY) model as a design modification is evaluated. The central challenge is to define the effects of geometric refinement in case chamfered corners, the control of the wind flow and the reduction of the negative influence of wind loading on high-rise buildings.

Methodology applied is a hybrid method which is a combination of Computational Fluid Dynamic (CFD) simulation and physical wind tunnel test to get comprehensive and validated results. ANSYS CFX software was used to simulate steady-state wind flow around the models at wind incidence angles of 0° , 30° , 60° , 90° , 120° , 150° and 180° . These numerical results were then carefully compared with the experimental results from a boundary layer wind tunnel.

This research clearly shows that the chamfered design outperforms the other designs in terms of aerodynamics. As can be seen in the data, the CY model always has uniform pressure distribution and significantly lower peak suction values than the sharp-edged SY model. The chamfers effectively delay flow separation and smoothen the flow around the structure, as revealed by streamline analysis. The chamfered shape mitigates wind-induced stresses that a building has to endure, and shows how small changes in geometry can make a significant difference in aerodynamic efficiency. The decrease in wind induced stress increase the life span of structure and makes it serviceable. Also, ground level wind reduction in dense urban areas is a common problem, and mitigation of wind vortices will help to create a more pleasant and safe pedestrian environment.

Keywords: Boundary Layer Wind Tunnel, Computational Fluid Dynamic (CFD) Simulation, Chamfer Corner Modification, $k-\epsilon$ Model, Standard Model, Wind Pressure Coefficient, Y-Shape Tall Building

CHAPTER 1

INTRODUCTION

1.1. GENRAL BACKGROUND

Global cities are rapidly growth of population and constant movement towards the cities; modern cities have to be built vertically instead of horizontally. The use of high-rise structures and tall towers has proved to be a necessary solution in making the best use of land space. But the behavior of buildings shifts as they become taller and thinner. The design of a tall building is affected in a way that is very sensitive to lateral environmental forces, as against low-rise buildings, in which the design emphasis is more on dead loads and live gravity loads. Of these lateral loads, wind loading is the most influential, in governing the design and structural integrity of a high-rise building [2].

The forces on a building change when the building grows taller as the height of building increase effect of wind on building also increases. When strong wind flows interacting with a tall building structure create complex fluid-structure interactions. The windward side of the structure is directly affected by positive pressure from the wind. At the same time, the air divides at the sharp outer edges of the structure creating strong negative pressure zones (suction forces) along the leeward face and along the side faces. There are an atmospheric boundary layer effect and less friction with the ground as the height of the atmosphere increases, which is why wind velocity is directly proportional in the atmosphere. The wind's influence on a structure is not a linear function of wind velocity. Therefore, the upper levels of a tall tower experience hugely stronger dynamic air currents than the lower levels [3].

1.2. AERODYNAMIC CHARACTERISTICS OF Y-SHAPE BUILDING

Recently, irregular, asymmetric and tri-axial structures are more frequently seen in iconic high-rise projects by structural engineers and architects. Structural engineers and architects have been using irregular, asymmetric and tri-axial structures. They are more in iconic high-rise projects in recent years. One of these non-standard layouts is especially popular: the Y shaped plan. The Y-shape plan, with its tri-axial symmetry, provides amazing functional benefits including optimal use of natural daylight, excellent views for its residents and a solid structural foundation to resist lateral forces [19].

One the other hand, wind flowing directly into one of the wide entrances into the re-entrant corners creates a huge positive pressure on front face but if the wind strikes the

sharp end of a wing, the air velocity rips at sharp edges, creating deep wake areas and strong local suction on the corner wall surfaces. Such pressure variations cause the standard Y-shaped towers very susceptible to unequal loading conditions.

1.3. AERODYNAMIC MODIFICATIONS

In order to make a tall tower structurally safe when facing extreme wind, engineers have two options: they can either add more steel, concrete, etc. which cost them expensive or they can design the structure to minimize wind forces naturally. Aerodynamic modification (Corner modification) is very effective and economical. These changes are generally considered to be either major global changes (for example, tapering, twisting or cutting out large structural openings into the building), or minor local changes (for example, stepping, rounding or chamfering sharp outer corners). The use of corner modification is a well-recognized elegant and very practical way of changing the flow of fluids around high rise towers [14]. To provide a flat, angled transition zone, the sharp acute corners of the plan of a building are cut at a particular angle, which is called corner chamfering. The physics behind a chamfered corner is that it provides a smooth surface for the wind flow. A strong wind stream hitting a sharp corner causes the flow to separate quickly and violently with a large, turbulent wake following the sharp corner. The sharp turn is thus softened by the cutting into flat chamfered edge. The incoming airflow can now flow around the modified vertex more slowly, postponing the flow separation and enabling the layers of air to adhere to the side walls. The overall width of the turbulent wake region behind the high-rise tower is significantly reduced due to this smooth transition. This smooth transition helps reduce the overall size and intensity of the low-pressure wake zone behind the building, effectively reducing the pressure act on the surface of building [22].

1.4. WIND LOAD

In modern high-rise structures, wind loading is the main lateral force that controls the design and stability of these structures. The force of wind is dynamic and fluctuating and differs in magnitude, direction and intensity based on the building's shape, surface roughness, and direction of the wind relative to the approaching air stream. The higher the building is, the faster the wind will blow because the ground friction will be decreased, and the kinetic energy of the blowing air will be transformed into a powerful surface pressure on impact.

1.4.1. Factors Affecting Wind Loads

Geometric and environmental parameters of structure are main factor for the distribution of wind-induced forces and magnitude:

Building Geometry: The most important is the shape of the structure. The number H/B, Aspect Ratio, reflects the susceptibility of the structure to swaying and resonance. The Y-shaped floor plan used in this study creates complex flow patterns, unlike the simple rectangular planes, which create simple flow paths. In addition, Corner Modifications (such as chamfering or rounding) play a pivotal role by slowing down flow separation and successfully reducing peak suction forces.

Wind Flow Characteristics: The characteristics of the wind are controlled by the surrounding Terrain Category (open, suburban, or urban) and the Atmospheric Boundary Layer (ABL). Turbulence Intensity and Gust Factors are important factors that influence the frequency and size of the air fluctuations coming against the tower.

Angle of Wind Incidence: For non-prismatic structures, it is important to consider the direction of the structure with respect to the wind. Pressure distribution and global force coefficients (C) are highly sensitive to the changes in incidence angle (small changes).

1.5. WIND LOAD EVALUATION

Wind load evaluation is the most important steps ³² in the structural design of tall buildings, and is the major lateral force acting on a building that affects the stability and serviceability of the structure. Increasing height creates increasing wind forces because of the decrease of ground friction and the increase in wind velocity, so it is important to use accurate calculation methods to ensure safety.

Here are different methods of evaluation wind load: -

- a) By Using Standard Code (IS 875 – part 3)
- b) By Using Physical Testing
- c) By Using Numerical Simulation

1.5.1. STANDARD CODE

The evaluation for wind loading begins with a codified empirical procedure. In India using standard wind code IS 875 -part 3 (2015) [6]. Which help to determine the ³³ basic wind speed, risk ³⁵ coefficient, terrain/height factor, and local topography are all factors that influence the design wind speed (V_d), which ¹¹ can then be used with this fundamental relationship to calculate the design wind pressure (P_z) at any height (z).

➤ **Calculation formula of Design Wind Speed (V_z)** (Is 875-part 3 (2015) Cl. 6.3)

$$V_z = V_b k_1 k_2 k_3 k_4 \quad (1.1)$$

Here,

V_z = Design Wind Speed at Height z , in m/s

V_b = Basic Wind Speed

K_1 = Probability Factor (Risk Coefficient)

K_2 = Terrain Roughness and Height Factor

K_3 = Topography Factor

K_4 = Importance Factor for the Cyclonic Region

➤ **Calculation formula of Design Wind Pressure (V_z)** (Is 875-part 3 (2015) Cl. 6.3)

$$p_z = V_z^2 \quad (1.2)$$

Here,

p_z = Wind Pressure at Height z , in N/m²

V_z = Design Wind Speed at Height z , in m/s

These codal provisions work well on regular geometry such as prismatic (rectangular or square) plans, but may not be adequate for irregular geometry (not prismatic), such as a Y-shaped plan. So, we move on Physical Testing and Numerical Simulation.

1.5.2. PHYSICAL TESTING (WIND TUNNEL)

Physical testing of model done through the help of wind tunnel. It is also one of the most reliable means of evaluating wind load on high rise buildings, especially for non-standard building geometries, where design codes may provide limited guidance. This study uses wind tunnel techniques to simulate the atmospheric surface layer on a small-scale model of the Y-shaped structure.

According to Structural layout two type of wind tunnel present such as open and closed. This project work done in open circuit wind tunnel.

The Boundary Layer Wind Tunnel is known as (BLWT) is used because it is able to simulate the exact atmospheric conditions that a high-rise structure would experience. The tunnel is set up to mimic two main flow characteristics of the tunnel: the mean

velocity profile and the turbulence intensity profile, which are important to ensure the results are physically representative [26].

25 1.5.3. NUMERICAL SIMULATION

Computational Fluid Dynamics (CFD) is a high-level numerical simulation technique using the basic governing equations of fluid motion to simulate fluid behaviors, both liquid and gas. CFD done through the help of software which named ANSYS. It has also been a key visualization and analytical tool for wind engineers and offers a comprehensive understanding of the interaction between the wind and the building, the evolution of surface pressures and the creation of more complex flow patterns in the building's wake. At the core of it, CFD numerically solves the continuity, momentum and turbulence transport equations to reproduce the actual wind flow in reality. It has now proven to be essential for analyzing velocity fields, surface pressure distributions, flow separation zones, vortex formation and wake development which are all significant elements that affect how well tall buildings operate aerodynamically [28].

The basic CFD exercise will consist of three phases:

- a) **Pre-processing**
 - The geometry of the buildings is developed
 - The computational domain is defined
 - Generating the mesh
 - Define Boundary conditions (Inlet, Outlet, Ground, Walls etc.)
- b) **Solution**
 - This is done in setup part, select suitable physical and turbulence models.
 - Solution solved by Navier–Stokes equations.
- c) **Post-processing**
 - This is final step to getting visualization of results, Streamlines, Pressure contours and pressure coefficients of building.

1.6. PROBLEM STATEMENT

The lateral forces are most important in the design of tall structure, with wind loading being the most important lateral force act as critical design factor for safety and serviceability. The wind data of pressure coefficient (C_p) for regular cross-sections of buildings, including rectangular, square and circular, as conventional design shape obtained from buildings standards code like IS 875- Part 3 and ASCE 7. These standardized codes have sufficient knowledge of wind data and an accurate prediction of the wind-induced loads for the traditional structures with a high degree of reliability.

In recent years, however, the irregular shape of the building, such as the Y-shaped plan, has tended to become more popular to make it more aesthetically look, to allow more natural daylighting and to provide better view. The issue is that there is no validated pressure coefficient data available for such complex and tri-axial geometry for either Indian Standard (IS 875) or international codes (ASCE 7). To determine this information, this study uses a hybrid investigative approach combining high fidelity numerical simulation and experimental validation. Computational Fluid Dynamics (CFD) using modeling based on the $k-\epsilon$ turbulence model and Reynolds-Averaged Navier-Stokes (RANS) equations which can be used to very precisely calculate the pressure distribution across the different faces of a Standard Y shape (SY) and Chamfered Y shape (CY) configuration. The numerical results of this study are thus compared with experimental data which get from wind tunnels to check the validity of the simulated aerodynamic behavior with respect to the actual physical behavior. This study is an attempt to quantify the exact change in the pressure reeducation caused by corner chamfering, thereby giving a scientific basis for using aerodynamic shape optimization for naturally improving structural stability. The dual effect is that they create a reliable database for the pressure distribution (C_p) on building of the Y-type, which is not addressed by existing codes, and that they also show that how corner modification of the geometry allows to weaken the drag forces and reduce the peaks of suction around the tower, making the tower designs more sustainable, material-efficient, stable and safer.

1.7. RESEARCH OBJECTIVE

- a) To design and test 3D geometric models of same cross-sectional of the Standard Y-shape (SY) high rise building and the modified Chamfered Y-shape (CY) high rise building covering 9 (A-I) and 15 (A-O) structural faces respectively.
- b) To test wind induced flow characteristics and pressure distributions from 0° to 180° at 30° wind incidence interval for both models.
- c) To determine face-wise analysis of the pressure coefficient (C_p) for all the faces of both models.
- d) To determine and compare aerodynamic efficiency of both model with and without chamfering at corners and also quantified by calculating the percentage reduction in the pressure coefficients (C_p), which gives an optimal modification to achieve better stability and safety.
- e) Visualization and mapping of complex velocity streamline patterns around the Y-shaped geometry and pressure contour of the all faces of building. Which identify regions of high turbulence to understand the how chamfering corner modification help to delays flow separation and enhanced building stability.

1.8. ORGANIZATION OF THESIS

There are five chapters in this thesis report:

CHAPTER 1: INTRODUCTION

This section provides a broad overview of wind hazards in high-rise engineering, explains the specific aerodynamic issues and problems associated with Y-shaped floor plan, introduces the mechanisms of the chamfering modification in Y-shape, describe physical and numerical testing approach and describes problem statement of this study.

CHAPTER 2: LITERATURE REVIEW

This chapter provides a present codal provision and comprehensive research review of the historical research and modern studies on building structure aerodynamic, highlights recent research into tri-axial and irregular high-rise shapes, examines previous research regarding aerodynamic corner cuts and reviews the performance of CFD models in structural wind design to identify the clear research gap that will be addressed by this project.

CHAPTER 3: METHODOLOGY

It gives a technical description of the geometric dimensions of the Standard Y-shape (SY) and Chamfered Y-shape (CY) building models, explain the experimental setup of open circuit wind tunnel, description of the governing RANS fluid equations, parameters of the standard $k-\epsilon$ turbulence model and boundary conditions of CFD simulations.

CHAPTER 4: RESULTS AND DISCUSSION

This chapter show the results of wind streamlines which received from numerical simulation across the range of 0° to 180° at 30° wind incidence interval. Also show the effect of wind in the form of pressure contour by the calculation of both physical and numerical testing.

CHAPTER 5: CONCLUSION, FUTURE SCOPE AND SOCIAL IMPACT:

Summarizes the key engineering conclusions from the study; provides practical recommendations for future structural research directions and positive social, economic and safety benefits of the application of optimized chamfered configurations in contemporary city skyscrapers.

23 CHAPTER 2

LITERATURE REVIEW

2.1. GENRAL

The literature review is crucial for understanding the aerodynamic performance of buildings resulting from the interaction between wind forces and the building's geometry. In the past, the structural design of high-rise buildings has been based on the national and international standards IS 875 (Part 3:2015), EN 1991-1-4:2005, ASCE-7 and AS/NZS 1170.2:2011. These standards, however, are mainly set for regular shapes such as rectangular or square plans as briefly explain in chapter 1. As a result, they give limited or no guidance on structures other than prismatic, such as Y-shaped and multi-limb plan structure. To overcome this gap in the code, a number of studies have used numerical and experimental techniques to investigate non-standard plan shapes (Goyal et al., 2022) and (Sanyal & Dalui, 2020) have shown that aerodynamic modifications like chamfering, rounding, tapering and setback have significant effects on the global force coefficients and local pressure peaks. Moreover, validated the efficacy of using the physical wind tunnel testing in conjunction with Computational Fluid Dynamics (CFD) and discovered that simulated and experimental data agreed well [8]. The internal limb angle studies have shown that symmetric plans such as the Y-plan mostly have lower wind forces than skewed plans [19]. This entire collection confirms that although irregular geometries have a tendency to develop distinctive suction zones and torsional effects, they can be optimized with careful aerodynamic treatments. Now we briefly explain each research which help during the project period.

2.2. CODAL INFORMATION

- ¹¹ IS 875- Part 3 (2015)

The Indian Standard (IS 875- Part 3 2015) provides design wind speed for various geographic zones, terrain factors and pressure coefficient (C_p). However, such pressure coefficients are only applicable to standard regular cross-sections, where the cross-section is square, rectangular, triangular or circular. IS 875 (part 3) provides no analytical provisions, equations or pressure distribution maps for: Multi-limb, multi-legged footprints such as Y shaped footprints. Chamfers or corner recessions that have been modified aerodynamically [6].

- **British Standard / Eurocode (EN 1991, 2005)**

The European standard provides for the use of high-level techniques to estimate natural wind actions on simple buildings and bridges within defined geometric limits. It adds advanced formulas for along-wind dynamic responses and structural force factors, but completely fails to provide appropriate analytical guidance or force coefficients for uncharacteristic plan forms, multi-limb structural profiles, and skew wind angle effects [7].

- **American Standard (ASCE 7)**

ASCE 7 gives detailed analytical and wind tunnel testing procedures for the calculation of full-scale wind loads on both low rise and high-rise structures. The information on low rise structure also given in this code. Although it has a good structure for the traditional forms, it does not have wind load parameters for unusual cross-sectional profiles. In addition, it does not handle the non-linear, complex pressure fields and extreme localized suction peaks created by skew wind incidence angles on composite shapes [1].

- **Australian (AS 1170.2:2011)**

This code applies to high-rise structures up to a height of 200m, and structure which roof span less than 100m. It provides specific wind load recommendations, most of which are for rectangular or square plans. It is not suitable for multi-legged, curved or irregular architectural shapes, where it does not provide data or pressure coefficients. It also fails to take into account the intense pressure and changes that can occur when wind is blowing across an irregular frontage [32].

2.3. REVIEW OF PREVIOUS RESEARCH

- **Tse et al. (2012)** Evaluated the structural, performance and financial benefits and drawbacks of enhancing the internal stiffness of a building and adding passive aerodynamic shape changes. The research combined the use of the extensive force-balance data obtained from the wind tunnel tests with the use of structural optimization algorithms to evaluate the amount of material used in high-rise designs. The study demonstrated extremely effective reduction in baseline dynamic wind forces for a building through the use of corner openings and perimeter treatments, resulting in significant savings in structural steel and concrete for the lateral load-resisting system, which was found to be very cost effective.

- **Raj, R. et al (2013)** Carried out a series of basic boundary layer wind tunnel (BLWT) tests at 1:100 scale on a set of rigid and similar models to assess the overall influence of cross-sectional profiles on the basic forces on the structure. Test performed on square, rectangle, circle, octagonal under the simulation on atmospheric boundary layer. The empirical results showed that the traditional sharp-edged geometries create very alternating cross-wind forces as a result of rigid flow separation lines.
- **Mukherjee, S., et al. (2014)** Perform the aerodynamic modification of high-rise building, studied in wind tunnel. Research used boundary layer wind tunnel (BLWT) test with the multi-point pressure scanning to map the reduction of along wind and cross wind dynamic responses on modified architectural profiles. The writers extracted the fluid-structure dynamics for flow separation, and demonstrated that some minor changes of geometry around a building's perimeter are adequate to break the vertical continuity of the wind vectors. These changes result in a dramatic reduction of the diameter of the wake cavity on the lee side, and diminish dynamic lateral forces, all of which require a premature or guided re-attachment of the separated shear layers.
- **Hayashida et al. (2015)** Developed a comprehensive experimental data set rating the ability of several corner treatments to reduce wind forces, with the criterion being the dimensions of the chamfer treatment relative to the overall width of the building, from 5% to 15%. A chamfered perimeter geometry was found to affect the flow separation behavior at leading edges in the boundary layer wind tunnel tests. The chamfers cause the initial separation line to move further downstream, allowing the separated shear layers to cleanly re-attach along the side faces of the building. This reduces the area of the wake depression significantly on the lee side, resulting in a large decrease in drag forces along the wind and cross-wind dynamic swaying.
- **Dalui et al. (2016)** Conducted a study on the use of CFD for studying wind effects on corner modified tall building it is a numerical simulation technique for simulating the response of corner modified high-rise buildings under wind loads. The wind flow characteristics surrounding the models of square forms with altered outside borders are simulated using the Reynolds-Averaged Navier-Stokes (RANS) computational fluid dynamics (CFD) modeling framework. The external pressure coefficients (C_p) and their mean and Root-Mean-Square (RMS) along the side faces were reduced significantly by the use of corner modifications, demonstrating the effectiveness of corner modifications to eliminate sharp pressure gradients and giving a validated computational procedure for virtual wind tunnel testing.

- ²² • **Meng et al. (2018)** In the analysis, wind pressure coefficients of typical tall buildings under CFD simulation were examined using sensitivity analysis carried out a detailed sensitivity investigation of the effects of the numerical model parameters on the precision of wind pressure coefficient results around high-rise structures. The study compared some of the computational grid topologies, near-wall mesh refinement methods, localized boundary layer grid inflation factors, and specific criteria to the standard international experimental data contained in the CAARC datasets for tall buildings. The results provided the essential best-practice computational guidelines with a clear demonstration that near-wall grid resolution plays a crucial role in capturing flow separation, wake recirculation phenomena and peak surface suction in RANS virtual wind tunnel simulations.
- **Simiu et al. (2019)** Delivers the most theoretical and mathematical basis for the mechanics of the structural wind research of the atmospheric boundary layer (ABL). Use model terrain roughness in civil engineering application and derivate the power-law and logarithmic velocity scaling profiles. It covers, in detail, the physics of the instabilities of aeroelasticity, wake-induced and vibration induced by the vortex (VIV) which can flexible and slender structures. In addition, the basic equations which transform the dynamic fluid kinetic energy into the static surface pressure field are described which are fundamental to the current international design codes and regulations for controlling structural serviceability, cladding design and structural safety limits.
- **Sheng et al. (2019)** Tested the micro-climate flow characteristics and wind-structure interactions around complex cross-sectional high-rise configurations in a 1:300 scale wind tunnel testing program. Vertical wind shear profiles, turbulence intensity variations and power-law exponents were monitored around multi-limb structural shapes to see how they change velocity and pressure gradients. The study showed that complex cross-sections result in very non-linear flow patterns, with velocity vectors being funneled into recesses, leading to local wind speed formation of non-uniform pressure fields, highlighting the importance of modelling realistic.
- **Ahuja et al. (2020)** Perform research on analysis of Y-shaped tall building systematically evaluated the external wind loading properties and climate flow behaviors of a tri-branch Y-shaped tall building model by using advanced computational fluid dynamics. Using fine structure mesh for the simulation of structure tailing wake vortex formation and resolve windward stagnation zones. The study finds out the mean and peak pressure contours, and found that there are significant localized suction changes between the three radiating limbs depending on the angle of the oncoming wind.

- **Sanyal et al. (2020)** The research analysis of various changes to Y-plan high-rise towers to achieve the aesthetic of space and aerodynamic shape. The study employed a numerical modelling within the ANSYS CFX code to simulate six different geometric configurations (spiral, tapered, vertical setback, recessed, chamfered and rounded corner). The structural analysis showed that the geometric alteration of the structure on the macro-scale (setbacks and continuous vertical tapering) is very effective to reduce the global base shear forces and overturning moment coefficients. Micro treatments such as corner chamfers, on the other hand, are primarily used to control the actions of the boundary layer in local areas, thereby limiting high pressure peaks in the local boundary layer and reducing the vulnerability of the structures' cladding.
- **Dalui et al. (2020)** Research finds internal angle effect between limb of Y shapes tall structure, the effect characteristics of tri-branch high-rise structures from the effects of geometry. The researchers then conducted numerical simulations in an ANSYS CFX environment to assess how the drag, lift and torsional coefficient change for different amounts of wind incidence angle for various internal angles between the three building legs. The study has shown that the flow distribution is most stable for the perfectly symmetrical branching configuration of 120° - 120° - 120° and there with the minimal overall wind forces and base moments. The building's capacity to balance wind paths is diminished by the limited deviation from symmetry or skewing of the limbs, causing a significant amount of flow channeling, local suction at the back corners and high torsion on the structure.
- **Sanyal et al. (2020)** This research finds out the ³⁴effect of corner modification to Y plan shape tall building it compared the aerodynamic performance of three passive corner treatments (recessed, chamfered and rounded) on the edges of a Y-shaped tall building model. The study was carried out using computational fluid dynamics and the force coefficients as well as ²⁸detailed pressure distributions were obtained along the perimeter and height of the building for various wind angles of attack. The analysis showed that there are different design compromises; rounded corners reduced the overall wind forces most but created high localized negative pressure peaks (suction forces) along the curved lines but chamfered corner modification gives optimized structural balance, reduce wind forces and provide low local peak suctions for safe design.
- **Bhattacharyya et al. (2020)** Presented a Dual Experimental and Numerical Wind Engineering Study on an asymmetric E-shaped high-rise building. To test the flow, a series of pressure tap arrays was installed on physical scale models in a boundary layer wind tunnel, followed by CFD simulations on a parallel platform, using the

two-equation standard $k-\varepsilon$ turbulence model, and the empirical data collected through these tests was validated. The study found very good consistency between the physical and computational approaches. The study revealed that the non-uniform flow fields with unexpected high intensity suction zones and localized pressure gradients away from the main windward face call for dual-method validation for complex geometries, highlighting the importance of such a task.

- **Assainar et al. (2021)** This study helps to find out the effect of wind on pentagon shape tall building when applied to multi-face pentagonal high-rise shapes, extended passive aerodynamic shape optimization principles. These modifications to the building structure and the changes in the corners were analyzed to see how they affect the global force coefficients, overturning moments, and the micro climate wake profile around the building perimeter in a systematic manner, using ANSYS CFX. The numerical results showed that the multi-faceted and optimized geometric edges were effective in breaking wind paths for a uniform wind and in disrupting the vertical coherence of the cross-wind vortex structures. This study shows that the plan-shape and corner optimization methods performed on Y-shaped towers are also very effective for reducing wind loads in a variety of non-rectangular high-rise shapes.
- **Kumar et al. (2021)** Explored the local wind flow mechanisms and surface pressure distributions of deep re-entrant corners and concave facade areas typical of multi-limb building geometries. The study used high fidelity CFD simulations to follow the flow patterns through different angles of attack. The figures were clearly showing that when the wind blows at a skew angle to a multi-limb structure, the air that comes in is compressed and increases in velocity quickly inside the internal facade recesses. This phenomenon forms serious local pressure gradients and high velocity channeling effects between the inner core and outer tips of the limb, establishing critical parameters for local cladding design.
- **Gaur et al. (2022)** Compared computed and physical boundary layer wind tunnel data for a 1:100 scale rigid model structure in detail. The focus of the project was to test the wind load reductions of a baseline square model with sharp corner cut-backs at varying wind attack angles within a continuous arc. The numerical model used was based on the two-equation standard $k-\varepsilon$ turbulence closure model, which is used to solve the Reynolds-Averaged Navier-Stokes equations. The research showed that the numerical simulation is accurate in predicting external surface pressure coefficients (C_p), base moments and wake turbulence field; thus, strategic corner cuts reduce the width of the wake on the leeward side and offer a reliable prediction of aerodynamic load mitigation.

- **Meena et al. (2022)** Systematic measurement of external surface pressure coefficients (C_p), drag and lift fluctuations and wake vortex shedding lengths for non-traditional tall building shapes. The study evaluated a range of models with sharp edges with optimized models using corner cuts, slots, and recessions using a validated numerical framework. The research showed the effectiveness of windward corner treatments in reducing aeroelastic instabilities via reduced strength of the detached shear layers. This modification tunes the periodic vortex formation frequency which can effectively manage the cross-wind dynamic responses and improve the safety of the structure.
- **P. k Goyal et al. (2022)** Carried out detailed computational fluid dynamics analysis with the software ANSYS CFX to determine the interactions between wind and the structure of a multi-limb Y-shaped high-rise building. The numerical analysis determined the performance of a sharp-edged baseline Y-plan and modified Y-plan with optimized rounded corners. The velocity profiles and surface pressure patterns were generated by the simulated wind field on the building faces. The study found that rounding the corners of the tri-branch geometry changed the separation characteristics of wind streams moving towards it, stopping the air from detaching out of the flow suddenly and resulting in a dramatic decrease in the global wind forces and flow dynamic pressures applied to the Y-shaped object.
- **Rajabi et al. (2022)** Perform aerodynamic force coefficient measurements during 1:100 scale Y-shaped high-rise building physical wind tunnel testing and CFD numerical simulation of the structural deformations. This scale model was systematically rotated to capture wind pressure distributions and structural responses at various wind directions throughout a complete range of wind directions. The study followed drag, lift and lateral displacement behaviors and results were compared with the numerical result data obtained, which were found to be very close.
- **Verma et al. (2022)** Compared the aerodynamic efficiency of traditional rectangular skyscraper with the triaxial and multi-limb symmetrical structural system under the same boundary conditions. The structural effects of plan geometry were isolated from the other variables by keeping the total floor area and the model height constant for all of the configurations. The numerical analysis results showed that the aerodynamics of a triaxial symmetrical shape, like a Y shape, are inherently better. Their radiating limbs are designed to break up uniform vortex formation by naturally diffusing oncoming wind streams, and to result in lower baseline drag and lift coefficients than sharp-edged rectangular alternatives.

- **Al-Masoodi et al. (2023)** Explored passive corner optimizations on multi-face, octagonal high-rise shapes through an advanced computational process. The study examined the effects of the geometry of corner modifications on existing lateral loads that usually control the design of the structural systems of tall structures. The influence of the changes in the force coefficient was followed by the numerical models and it was found that the changes that are made along the outer edges of the perimeter compromise the intensity of the detached shear layers considerably. This optimization will help reduce the width of the wake behind the boat, minimize the pressure difference between the side walls, and minimize any suction forces inside the leeward zone.
- **Wijesooriya et al. (2023)** Provided an extensive state-of-the-art review showing the development, testing issues and future directions of CFD for tall building wind design. The paper presents a critical review of common turbulence closures, which are shown to be very effective for mean windward pressure predictions, but are found to have difficulty in capturing highly unsteady cross-wind wake oscillations. The authors describe key requirements for sizing the computational domain, the constraints on blockage ratio and boundary layer modeling, providing a template for standardized procedures in industrial wind design.
- **Kim et al. (2024)** Designed an innovative hybrid system of traditional virtual wind tunnel CFD testing coupled with deep unsupervised neural network analysis on localized facade forces. This research collected a large amount of high-frequency pressure database metrics for rigid multi-limb and corner-cut building profiles at a wide range of wind orientation angles. The predictive algorithms predicted with accuracy the high-resolution stagnation windward fields and extreme leeward C_p wake suctions without the necessity to constantly generate a mesh. The results showed that adding micro scale changes in the boundary, such as chamfering at the front surfaces, reduces turbulence intensity near the leading separation boundaries, which decreases dynamic force oscillations through complex structural axes.
- **Nagar et al. (2025)** Study about the non-linear surface pressure pattern prediction for aerodynamically modified high-rise give a structural investigation that addressed the effect of wind on a 1:150 scale, tri-branch Y-shaped tower with systematic variations of the corner chamfer sizes. They performed a complete set of wind angles of attack, from 0 to 180 degrees, at increasingly intervals of 10° and tracked the base shear, overturning moment and multi-face pressure contours. Results of the structural analytics showed that the width of the corner chamfer can be increased up to an optimized geometric ratio to soften the separation lines at the boundary.

2.4. LIMITATION

The existing literature on the aerodynamic performance of tall buildings indicates that there are some limitations in the current knowledge, which necessitates the need for this study. Firstly, little research has been carried out regarding the specific advantages of corner chamfer for y-shaped floor plan; most studies have been limited to simpler shapes, such as rectangular or square buildings. Additionally, many existing studies have been realized using either wind tunnel testing or numerical simulations but there is a lack of studies that incorporate high fidelity physical testing as well as comprehensive numerical simulation of these modified Y-shaped geometries. Numerous studies are also limited in their wind load analysis to a specific set of incidence angles, typically less than 180-degree, at rather broad intervals.

Many studies do not also cover the entire range of incidence angles, usually less than 180-degree, at meaningful and fine spatial resolution, which is important to capture the complex vortex shedding patterns that are characteristic of Y-shaped plans. Thus, the lack of such gaps in these designs requires designers to depend on incomplete data to achieve an optimum structural safety and material efficiency of Y-shaped skyscrapers, which emphasizes the need for the systematic, data-driven approach presented in this study.

CHAPTER 3

RESEARCH METHODOLOGY

3.1. EXPERIMENTAL SETUP

The experimental examination was carried out in an open circuit boundary layer wind tunnel of Wind Engineering Laboratory, Delhi Technological University, Delhi, India (Fig.3.3). This facility is specially designed to simulate the Atmospheric Boundary Layer (ABL) conditions which are very important for realistic estimation of wind loads on high-rise structures. The tunnel is powered by a 60 horse-power blower fan at the exit which creates a steady and continuous suction. The test section where the model place has a cross-sectional area of approx. $1.6 \text{ m} \times 1.6 \text{ m}$ and a total length of 14 m (Fig.3.1), which is suitable for the study of wind building interaction in a controlled environment.

A honeycomb flow straightener ($3.6 \text{ m} \times 3.6 \text{ m}$) is used at the inlet to the tunnel the openings of which are in the form of squares, eliminating the turbulence and stabilizing the flow. Vortex generators are placed in front of the test section to give the boundary layer the desired features. The set up is designed for Terrain Category II as per the Bureau of Indian Standards (Is 875 Part 3,2015). To get the simulation, square cubes of differing size (70mm, 50mm and 35m) are placed along the floor of the upstream section to provide the surface roughness that simulates urban terrain (Fig.3.2).

It is mounted in the middle of a manually operated turntable with a distance of 12 m from the effuse section so that it can be rotated across the complete range of wind incidence angles with a high accuracy. The wind speed inside the tunnel is kept constant at 52 m/s during tests, however the system is highly flexible and can be adjusted to a range from 10 m/s to 80 m/s by changing the wind speed of the fan or with a high precision. The data is taken observe, with a Pitot tube downstream 7.8 m from the effuse section to measure the velocity and record the reference static pressure. The pressure data is collected by a pressure transducer installed on the tapping points of the model and all outputs are passed through the data and collected to the device named as 'Testo 400' (Fig.3.5). This device allows the raw data to be precisely transformed into graphic and numerical data sets which enables a comprehensive analysis of the surface pressure distribution in the building.

Fig.3.3: - Wind Engineering Laboratory at Delhi Technological University



Fig.3.4.1: - Open Circuit Wind Tunnel



Fig.3.4.2: - Floor Roughness Block

Fig.3.4.: - Boundary Layer Wind Tunnel

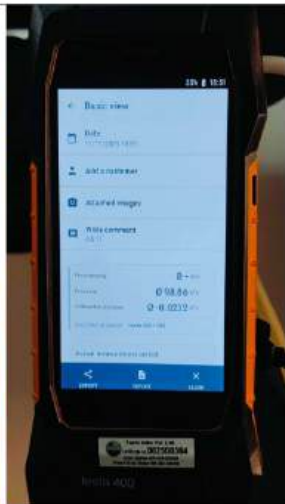


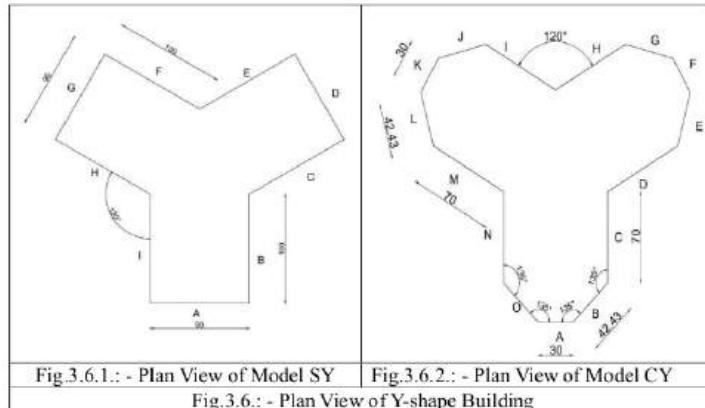
Fig.3.5.: - TESTO 400 (Measurement Device)

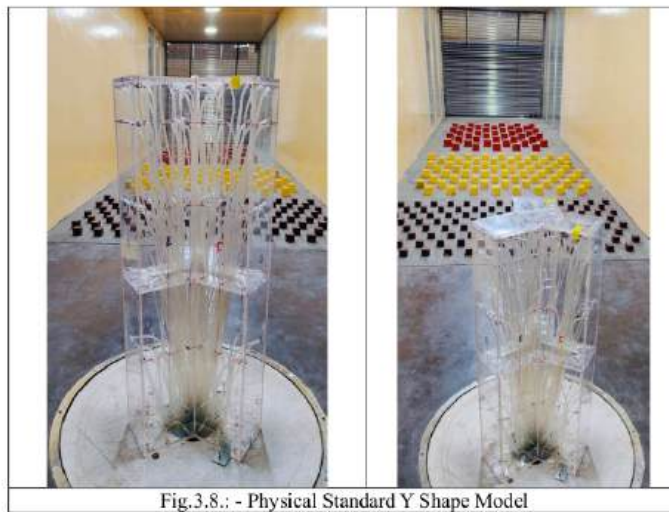
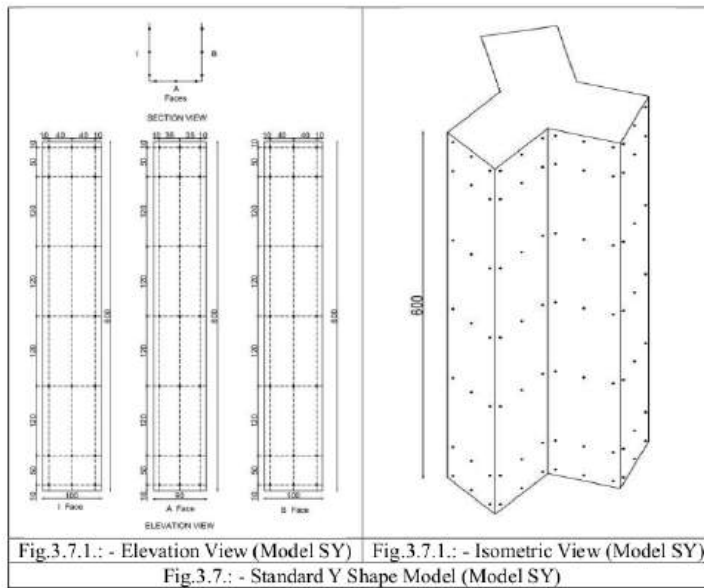
3.2. MODEL OVERVIEW

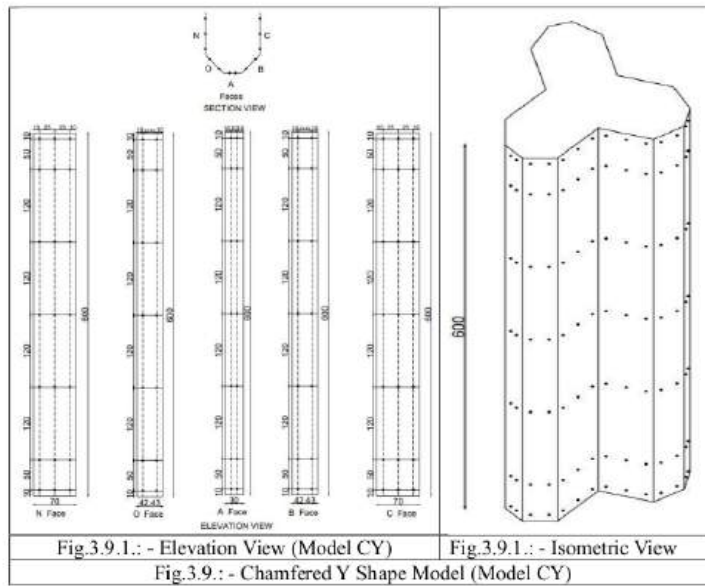
The study is conducted using two main configurations: Standard Y shaped model (Model SY) and Chamfered Y shaped model (Model CY). The Model SY has nine faces (A - I) and a total of 189 pressure points (Fig.3.7). The other model, the Model CY, chamfered corners at 45° degree (with a 135° interior angle) to make it more aerodynamic, and contains 252 pressure points, whereas the other has 15 faces (A -O) (Fig.3.9). These points are strategically placed on seven different elevations above the base level of model, which are 10 mm, 60 mm, 180 mm, 300 mm, 420 mm, 540 mm and 590 mm, so as to obtain a comprehensive surface pressure profile (Fig.3.7 & 3.9).

For physical testing a scaled physical model was carefully fabricated from 4 mm thick Perspex sheet to obtain high precision in the experimental pressure measurement. Perspex was chosen for this study because it has good structural strength, is light and easy to fabricate and is very transparent, thus enabling accurate placement of pressure-tapping points (Fig.3.8 & 3.10). The scale used was a geometric scale of 1:150, or ratio of the size of the physical model to the building. The actual height of the building was 90 m, so the scaled model height was determined to be 600 mm.

The pressure is measured using through a pressure measurement tube with the inner diameter of 1 mm for data collection. Each tube is securely inserted at the particular pressure point on the building's surface with the other end attached to the 'Testo' device (fig.3.3). The system can be used to record data of wind pressures all over the building surface simultaneously and then send it to computer for detailed numerical and graphical analysis. This is an instrumentation setup that allows for a high-fidelity surface pressure distribution to be measured, offering a good data set for research into the aerodynamic advantages of the chamfered Y-plan geometry.







3.3. COMPUTATIONAL FLUID DYNAMIC (CFD)

In this research Computational Fluid Dynamics (CFD) is used to simulate wind flow around the building models, based on the standard $k-\epsilon$ (k-epsilon) turbulence model of the ANSYS CFX platform. The numerical approach is very effective for wind engineering applications, and the equations that describe the behavior of wind turbulence particularly the kinetic energy of turbulence (k) and the rate at which wind turbulence dissipates (epsilon) are simplified. These governing equations can be solved to accurately simulate flow patterns and pressure distribution around the surfaces of the building with different wind incidence angles [28].

For this study the numerical simulations will only consider the "steady flow" condition which considers the aerodynamic response of the wind when it blows at a steady constant speed, not suddenly varying with time. The approach is a steady-state approach, which gives a stable and reliable method to evaluate mean wind pressures and global force coefficients. The $k-\epsilon$ model is well grounded on its reliability in the literature as a number of researchers have successfully validated it against empirical information obtained from physical wind tunnel tests, including [22]. The results they have obtained show this numerical method to be accurate for studying the aerodynamic performance of complex building geometries, and to yield results that are in good agreement with experimental observations in the real world. Using this proven method allows for the computational simulations to be based on well-established methodologies which has been validated and provides a solid foundation for the comparison of the aerodynamic efficiency between the Standard Y (SY) and Chamfered Y (CY) configurations.

3.4. NUMERICAL PROCEDURE: STEP BY STEP CFD SIMULATION

The aerodynamic analysis of the Y-shaped high-rise structure is performed by the numerical simulation following the systematic and well-defined process to ensure accuracy and consistency of the analysis. This procedure aims to transition from physical representation to a numerically validated solution for the pressure coefficient (C_p) that will be reliable in every step of the way.

Step by step procedure of CFD simulation are following: -

- Geometry (Selection of Building Shape)
- Selection of Dimension of Domain
- Type and size of meshing elements
- Define Boundary Conditions and Flow Characteristics
- Run the simulation
- Process the output in CFX post processing and gets results.

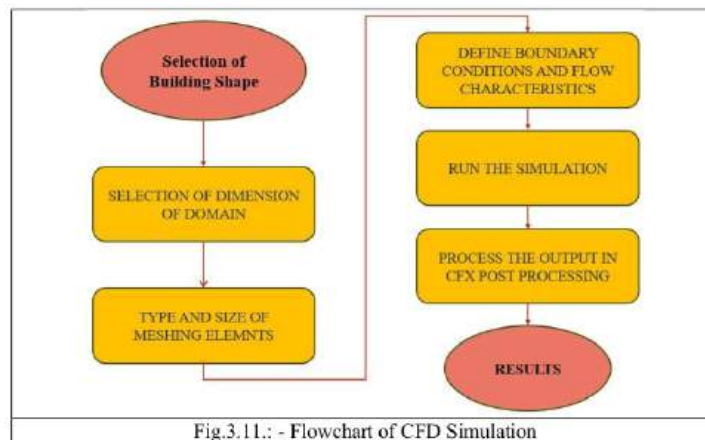


Fig.3.11.: - Flowchart of CFD Simulation

3.4.1. GEOMETRY AND MODEL DOMAIN OF CFD SIMULATION

The first step is to import the 3D geometry of the building into the ANSYS workbench in geometry part. The model is scaled down to 1:150 and is 600 mm high, after that the making of domain start. For an accurate estimation of the aerodynamic effects of the wind on the Y shaped standard and chamfered high-rise structure, a strong computational domain was established by using the ANSYS CFX. The dimensions of the domain were carefully considered, allowing the model to represent the entire domain of airflow interactions without coming into contact with artificial boundary constraints that are often referred to as "blockage effects. Taking into account the international standards for tall building simulations, the computational domain was established according to (Frank et al, 2004) with an inlet located 5H (3,000 mm) upstream of the building and an outlet which was extended 15H (9,000 mm) downstream and lateral distance and top clearance should be 5H (3,000 mm). This extended layout allows sufficient physical space for the incoming wind to interact with the structure, and provides enough space for the complex wake region created as air passes around the structure to settle before it reaches the edge of the domain, avoiding edge errors [19].

A "free-slip" condition was used for the upper and lateral domain boundaries, which allows wind to flow through this boundary without frictional drag of the tunnel walls, allowing the building's aerodynamics to be separated from the tunnel walls. On the other hand, "no slip" wall conditions were given to the exterior faces of the building and the ground surface. It induces a condition in which wind velocity becomes zero at

Normal V - Workbench

File View Tools Units Extensions Jobs Help

Project

Import... Reconnect Refresh Project Update Project ACT Start Page

Tools

Analysis Systems

- Coupled Field Harmonic
- Coupled Field Model
- Coupled Field Static
- Coupled Field Transient
- Eigenvalue Buckling
- Electric
- Explicit Dynamics
- Fluid Flow (CFX)
- Fluid Flow (Fluent with Fluent Meshing)
- Fluid Flow (Fluent)
- Fluid Flow (Polyflow Classic)

Project Schematic

1. Fluid Flow (CFX)

2. Geometry

3. Mesh

4. Setup

5. Solution

6. Results

Fluid Flow (CFX)

Fig.3.12.- Workbench Interface of CFD Simulation

The screenshot displays the ANSYS Design Modeler (Geometry) interface. The top menu bar includes File, Edit, Graph, Tools, Edit, Help, and Plug. The left sidebar shows the 'Geometry' tree with a list of objects: Plane, Edge, Face, Volume, and a selected 'Duct'. The main workspace displays the 3D model of the duct, which is a rectangular prism. The bottom status bar shows dimensions: 0.000000, 0.000000, and 0.000000 mm.

Fig.3.13.: - Design Modeler (Geometry) Interface of CFD Simulation

As explained size of computational domain defined as: -

- Inlet Distance: - $5H$
- Outlet Distance: - $15H$
- Lateral Distance: - $5H$
- Top Clearance: - $5H$

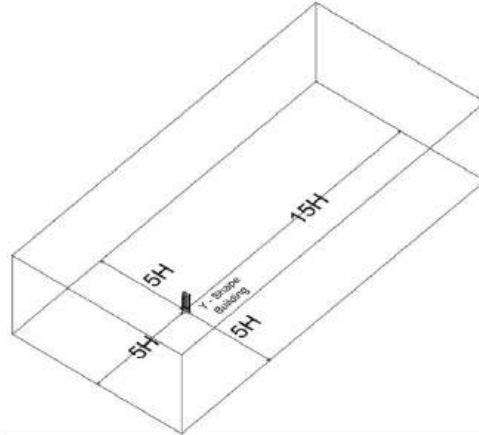


Fig.3.14.: - Size of Domain Used in CFD Simulation

3.4.2. MESHING AND DISCRETIZATION

The boundary was meshed with a high-quality tetrahedral mesh to represent the complex aerodynamics of the building geometry in the shape of a Y. The complicated chamfered edges were discretized by employing a much finer mesh size near the building surfaces and near the near-wake region, which has high pressure and velocity gradients. However, more coarsely meshed regions were used in the far-field parts of the domain to allow for computational efficiency and to minimize the number of cells required while maintaining the accuracy of the primary flow features. In order to further improve the resolution of the mesh in the critical transition areas between buildings, like building corners and chamfer lines, additional local mesh refinement was used to reduce numerical diffusion and improve the accuracy of the surface pressure coefficient (C_p) calculations [22].

Prism layers were made along the ground floor and the building wall surfaces to ensure correct resolution of the boundary layer. This mesh has been thoroughly evaluated based on industry standards, including element skewness and smoothness in cell size transitions. A comprehensive mesh independence study was then conducted using four

density levels; the Coarse (165,790 elements), medium (645,578 elements), fine (1,389,916 elements) and very fine (6,715,291 elements). During this validation process, the “fine” mesh configuration was chosen to be used in all subsequent simulations, because it achieved the best compromise between the cost of computations and the convergence of the results, while being consistent with both experiments and standards in wind engineering. This particular size of the mesh is important because the research notes that a bad mesh will give a wrong result but a too fine of mesh may result in a lack of stability in the calculations. The dimensions of the grid were carefully designed according to the recommended practices outlined in the previous literature in chapter 2, which were 0.008 m for faces of buildings, 0.08 m for the surrounding domain and 0.09 m at the bottom. This systematic meshing method allows the study to capture reliable and reproducible motion parameters and aerodynamic properties around the building, which helps to provide a solid basis for comparing the performance of the standard Y-plan configuration and chamfered Y-plan configuration.

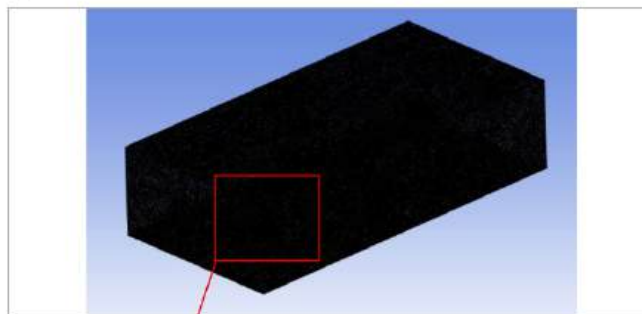


Fig.3.15.1.: - Domain Meshing

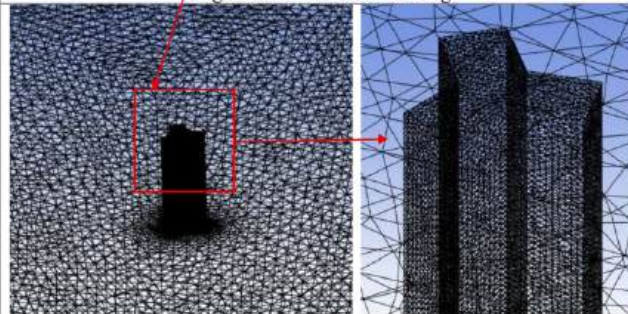


Fig.3.15.2.: - Detail Meshing of Standard Y Shaped Model (SY)

Fig.3.15.: - Meshing of Model SY

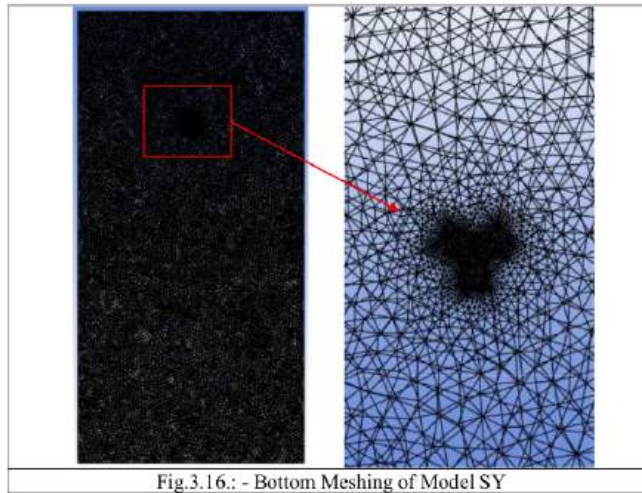


Fig.3.16.: - Bottom Meshing of Model SY

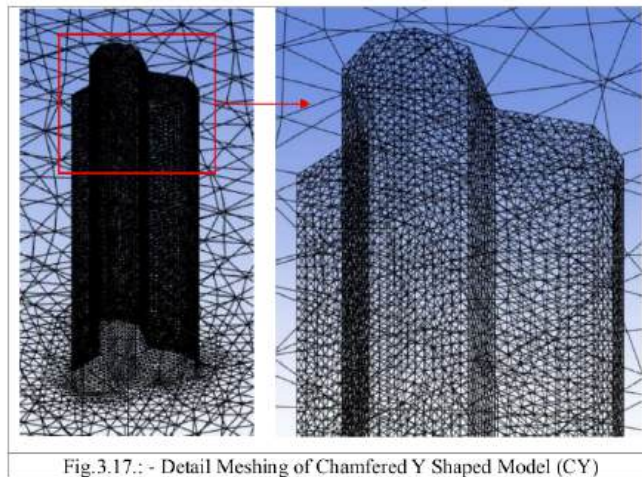


Fig.3.17.: - Detail Meshing of Chamfered Y Shaped Model (CY)

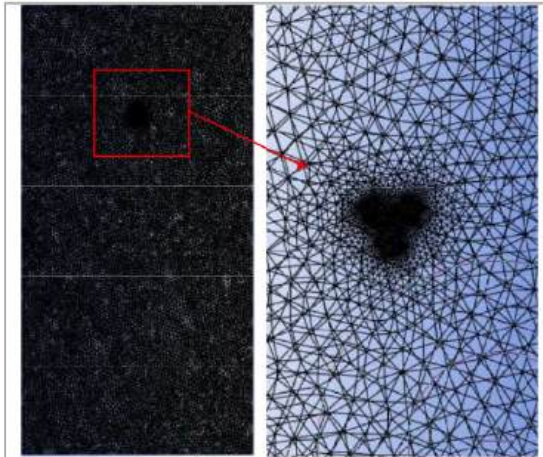


Fig.3.18.: - Bottom Meshing of Model CY

3.5. CHARACTERISTICS OF VELOCITY AND TURBULENCE INTENSITY PROFILES:

The inflow wind characteristics were carefully modelled to match the requirement of Terrain Category II given in IS 875 (Part 3): 2015 to ensure that the simulation represents the atmospheric boundary layer condition of an urban area. These profiles are important because they affect the pressure distribution and aerodynamic loads on the building surfaces directly, which in turn critically affects the development of these profiles.

The equations using in this model function are: -

a) Navier stokes equation:

$$\frac{\partial(\rho u_i)}{\partial t} = -\frac{\partial(\rho u_i u_j)}{\partial x_j} - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + F \quad (3.1)$$

b) Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (3.2)$$

c) Moment Equation:

$$\frac{\partial(\rho u_i)}{\partial t} = -\frac{\partial(\rho u_i u_j)}{\partial x_j} - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu_{\text{eff}} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + S_M \quad (3.3)$$

Here

ρ = Air Density,

u_i = x, y and z directions velocity components,

u_j = j^{th} direction velocity component,

x_j = j^{th} direction spatial coordinate,

P = Static Pressure

F = Body Forces

S_m = Source term momentum

3.5.1. VELOCITY PROFILE

A power law was used to determine the vertical profile (Fig.3.19) of mean ⁶wind speed, where the wind velocity (V_z) at any height (z) depends on the surface roughness characteristics. The experimental setup successfully created a velocity gradient along the tunnel floor by placing roughness cubes of three different sizes, namely 70mm, 50mm and 35mm, on the tunnel floor to mimic the wind shear effect that tall buildings would experience in urban and built-up areas.

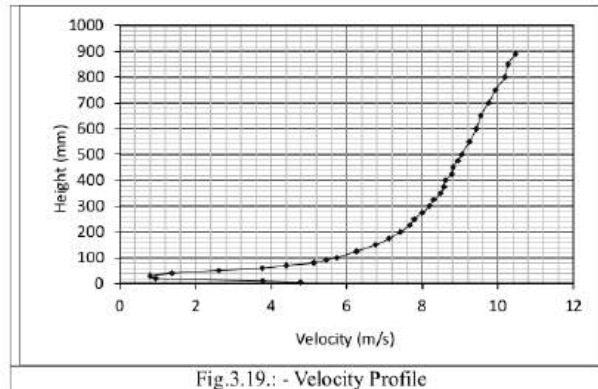


Fig.3.19.: - Velocity Profile

In velocity profile wind varies with height express in form of: -

$$U_z = \frac{U_o}{K} \ln \left(\frac{Z}{Z_o} \right) \quad (3.4)$$

Where K is denoted as Karman constant, U_o is coefficient of friction acceleration, U_z is the wind velocity at height Z , and Z_o is the detectable altitude of the outermost hardness.

Power law formula helps to control wind patterns with match the Terrien category: -

$$U_z = U_o \left(\frac{Z}{Z_o} \right)^n \quad (3.5)$$

U_o is the average air velocity at beginning height Z_o , and U_z is the wind speed at height Z . "n" is alpha which decide the terrain category.

3.5.2. TURBULENCE INTENSITY PROFILE

The ratio of the root-mean-square of the wind speed fluctuations to the mean wind speed is an important parameter for the understanding of the unsteady wind forces acting on the structure i.e. turbulence intensity (Fig.3.20.). The turbulence intensity profile was created for the purpose of this study to simulate the high turbulence conditions of an urban area. This alignment allows the numerical simulation to be able to simulate the realistic fluctuation of the wind flow that contributes to the variations of the pressure around the chamfered Y-shaped building faces. This validation of velocity and turbulence intensity profiles ensures the accuracy of the CFD environment in reproducing complex wind characteristics essential for a high-fidelity aerodynamic analysis.

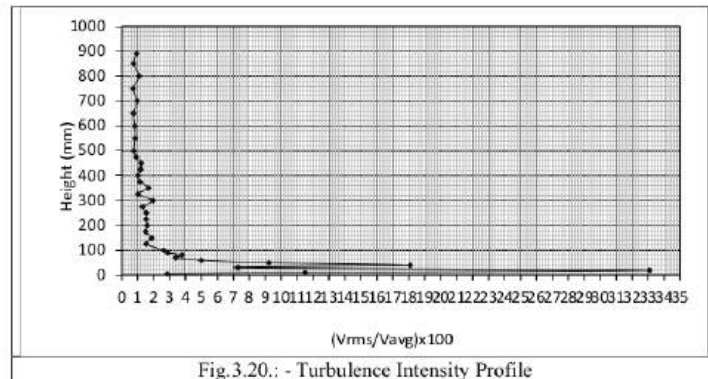


Fig.3.20.: - Turbulence Intensity Profile

3.6. VALIDATION

Validation is a very important role in any research in this study, the validation procedure is done in a rigorous manner by comparing the pressure coefficient (C_p) of the ANSYS CFX simulation with the pressure coefficient values mentioned in the Indian Standard IS 875 (Part 3): 2015 (Bureau of Indian Standards, 2012). Before testing the accuracy of our computational model, a preliminary validation test was first carried out on a standard rectangular building model which is a classical model widely used in wind engineering. The model was used in a rectangular shape and was modeled and analyzed using the ANSYS CFX as well as the experimental data of the boundary layer wind tunnel. These results showed a high correlation between the numerical simulation, the measurements in the wind tunnel and the values given in the standard. The agreement between the three data sets validates our numerical configuration to be able to simulate complex interactions between wind and structure. For a structured comparison, the building surfaces were divided into three different aerodynamic zones: windward side (toward the wind), sidewalls and leeward side (rear side). The average C_p for each of these zones was calculated at a range of height levels using ANSYS CFX and wind tunnel. The computed values were then systematically compared to the average values given in different wind design codes in various countries. There is an excellent degree of correlation between the results of our simulation and the established code and experimental data, which confirms the validity of our CFD and physical testing. This successful cross-verification proves that the numerical methodology is able to capture the complex aerodynamic behavior of the standard and chamfered Y-shaped geometry.

Table 3.1: Comparison of CFD and Experimental Results with Different Codes

International standards codes	Wind angle	Windward side	Sidewall	Leeward side
CFD Results by (present work)	0°	0.84	-0.57	-0.58
	90°	0.78	-0.63	-0.47
Experimental test Results	0°	0.78	-0.64	-0.30
	90°	0.70	-0.61	-0.45
IS 875 (Part III)	0°	0.70	-0.70	-0.40
	90°	0.80	-0.50	-0.10
ASCE/SEI 7-16	0°	0.80	-0.70	-0.50
	90°	0.80	-0.70	-0.50
AS/NZS 1170.2:2011	0°	0.80	-0.65	-0.50
	90°	0.80	-0.65	-0.50
EN ISO 4354: 2012	0°	0.80	-0.65	-0.70
	90°	0.80	-0.65	-0.70

shaped at 0-degree wind incidence with data obtained from CFD, experimental data and existing research. The correlation is very high and values obtained from our ANSYS CFX simulations, obtained from the physical wind tunnel experiments, and obtained by Mukherjee et al., (2014) are in close agreement. The good overall correlation obtained from three independent data sources is indicative of the high accuracy of our numerical setup, both in terms of the meshing strategy, boundary conditions, and turbulence modeling used. The pressure distribution plots on the windward, sidewall, and leeward faces of the building are similar, which is a strong validation of our approach.

Table 3.2: Comparison of CFD and Experimental Results with Research Paper

Data	A	B	C	D	E	F	G	H	I
0° (Wind Tunnel Present Study)	0.54	0.03	0.25	-0.39	-0.36	-0.36	-0.39	0.01	-0.18
0° (CFD Present Study)	0.51	0.01	0.1	-0.3	-0.26	-0.27	-0.33	0.09	-0.01
0° (Mukherjee et al. 2014)	0.666	0.189	0.33	-0.347	-0.279	-0.279	-0.347	0.33	0.189

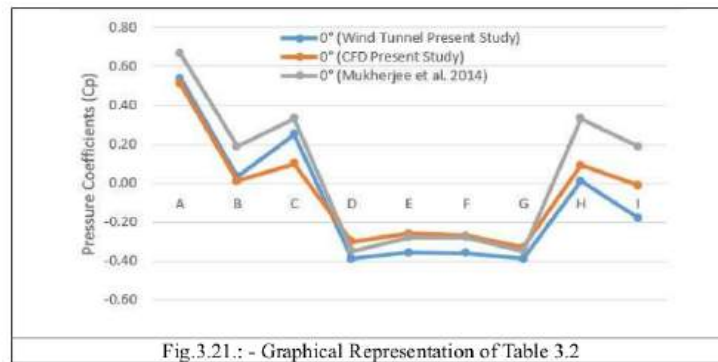


Fig.3.21.: - Graphical Representation of Table 3.2

3.7. PRESSURE COEFFICIENT (C_p)

Pressure coefficient is denoted by " C_p ". It is a basic dimensionless parameter in wind engineering to describe the wind pressure acting on the body surface compared to the wind flow around the body. It gives the normalized understanding of the interaction between wind speed and the geometry of the building to understand the positive (pressure) force and the negative (suction) force on different parts of the building. The difference between the local surface pressure and the reference static pressure and the dynamic pressure of the oncoming wind are used to express C_p mathematically. This study translates the pressure distribution into a pressure contour map of the exterior face of the building, by calculating the C_p value for each area.

High positive C_p values on the windward side reflect the effect of direct flow of wind, where the air velocity is slowed down on impact with the building surface, increasing the pressure. The sidewalls of the building as well as the leeward side exhibit negative C_p values (suction) due to the separation of the flow from the edges of the building and formation of low-pressure areas.

These coefficients can be recorded and used to determine the most wind-stressed parts of the building. The analysis of C_p is important to understand the effect of corner changes on airflow separation and pressure recovery in our chamfered Y-shaped models (Model SY and Model CY). After comparing these experimental and numerical data of C_p we can verify that our models are able to capture the aerodynamic behavior very well. This data is used as the fundamental data to analyze the structural stability and overall aerodynamic performance of the proposed high-rise configurations.

$$C_p = \frac{\text{Pressure at any point (pa)}}{0.6 V_z^2} \quad (3.6)$$

Where V_z is the design wind speed in m/s

CHAPTER 4

RESULT AND DISCUSSION

Chapter 4 show result and discuss about the analyze aerodynamic performance of two building shapes, Standard Y-shaped building (Model SY) and Chamfered Y-shaped building (Model CY). To get the result from hybrid approach was used, in which numerical simulation was performed using ANSYS CFX and experimental testing was done in a BLWT. Analysis was systematically carried out for a large range of wind incidence angles 0-degree to 180-degree at 30-degree interval. This broad spectrum of angles enables us to see how wind loads vary when the wind direction changes, relative to the main faces of the building, giving a full picture of the building's aerodynamic profile.

The wind-structure interaction was analyzed using two key diagnostic tools, flow streamlines and pressure contours. Numerical simulation was used to generate streamlines to visualize airflow path, which helped understand how the wind hits the building, where the wind separates at corners etc. At the same time, experimental testing was carried out in the wind tunnel to map out the pressure contours, which gave a visual indication of the pressure distribution over the building surface. These contours can then be used to distinguish high pressure (windward) from high suction (sidewalls and leeward).

One of the key points in this discussion is the determination and comparison of the pressure coefficient (C_p). The dimensionless value can be used to quantify the wind actions on both models in a common sense and it was easily seen the effect of the overall wind load of the geometric changes of the chamfered model (Model CY). For getting high accuracy, the numerical results obtained from ANSYS are compared with experimental results throughout this section. The detailed comparison shows how geometric modifications, like chamfering corners, change the airflow pattern and can help reduce the peak suction pressure that can adversely affect the stability of a high-rise structure. The analysis of the data provided from both the numerical and experimental aspects of the research establish a clear relationship between the building shape, wind orientation and the structural wind loading, which is valuable for the design of the modern architecture of tall buildings.

4.1. STREAMLINE AND FLOW CHARACTERISTIC ANALYSIS

A key component of the numerical analysis is the flow visualization, which is performed using streamlines, which gives a good qualitative picture about the interaction between the wind and the Y-shaped building geometry. These lines can be used to trace the path of air particles as they come closer to the building, pass through it and go away, which can be useful in the CFD post processing environment. The streamlines analysis from angle of attack (AOA) 0° to 180° at 30° wind incidence interval (Fig.4.1.) showing the stagnation of flow at the windward side where the wind has to bend around the structure. This separation leads to specific flow separation points at the corners and along the edges and has a direct effect on the distribution of surface pressures.

The analysis of the streamlines also shows the development of recirculation areas, especially on the leeward side and sidewalls. Low velocity vortices are present in these zones and the air is trapped behind the building creating the negative pressure (suction) regions measured in a quantitative manner. The streamlines of the Standard Y (SY) model reveal separation at sharp edges indicating high turbulence and high suction intensity. The chamfered Y (CY) model, on the other hand, has a more controlled flow transition (the streamlines at 45° chamfered corners with 135° interior angle are smoother), because the building geometry is able to reduce the attachment of turbulence flow from the surface of building and minimize excess suction. This visual demonstration of airflow is important to understand and communicate the design clues to aerodynamic improvements.

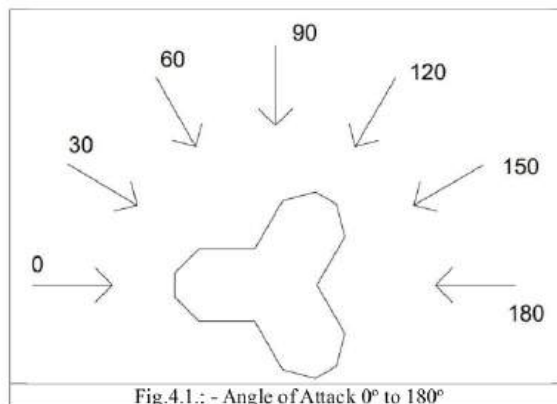


Fig.4.1.: - Angle of Attack 0° to 180°

4.1.1. STREAMLINE AT DIFFERENT DEGREE OF WIND INCIDENCE

The Standard Y (SY) and Chamfered Y (CY) models were tested at 30-degree intervals between 0-degree and 180-degree. The following notes briefly summarize the behavior of each one of the models at these particular orientations:

➤ AT 0-DEGREE WIND INCIDENCE

When wind act at 0-degree wind incidence the main leading face is struck directly by the wind. The flow separation is significant at the leading corners of Model SY (Fig.4.2) because of the sharp edges, whereas Model CY has a chamfer, leading to a smoother flow transition. The chamfer in Model CY helps to minimize peak suction on sidewalls next to the chamfer (Fig.4.3).

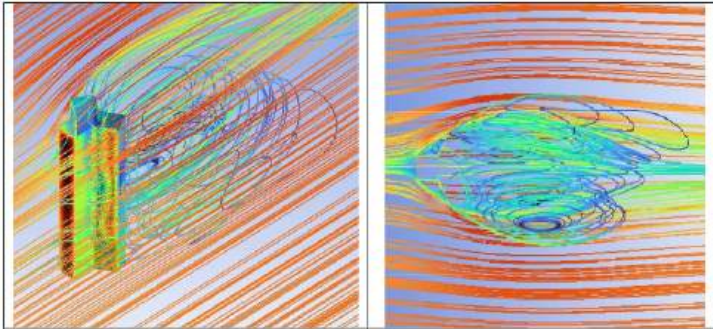


Fig.4.2.1.: - Horizontal Streamline (SY)

Fig.4.2.2.: - Bottom Streamline (SY)

Fig.4.2.: - Streamline of Model SY at 0-Degree Wind Incidence

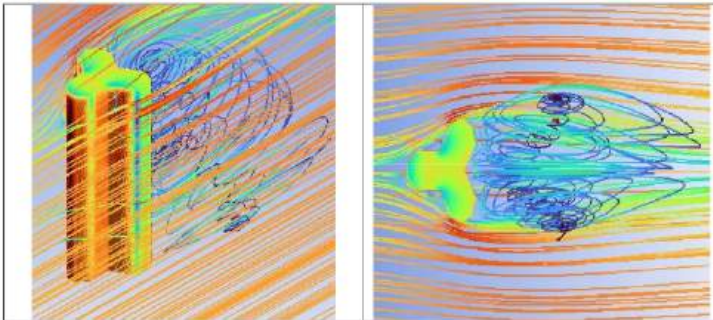


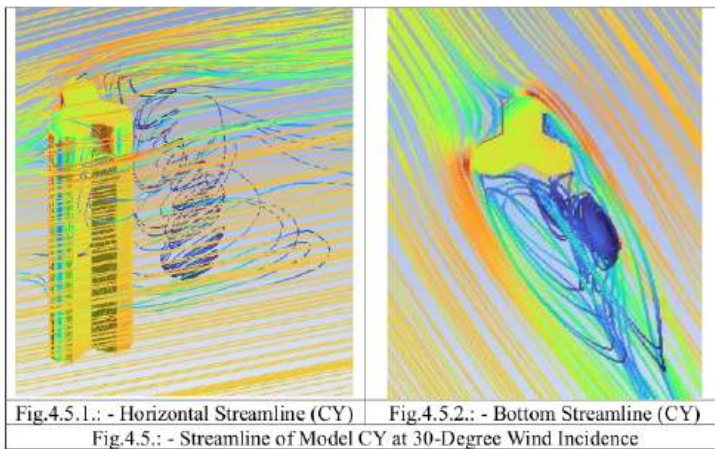
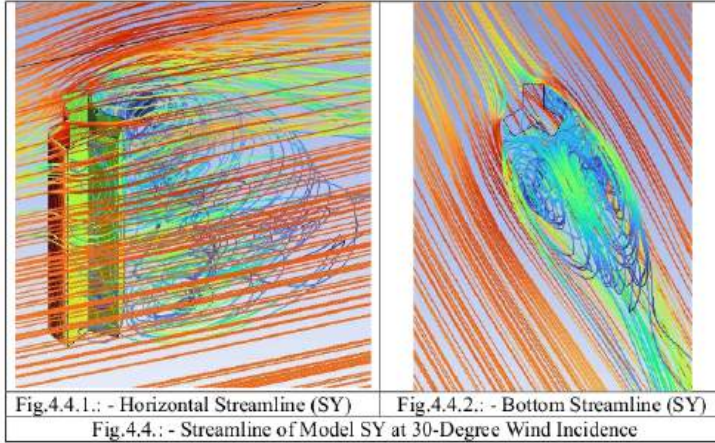
Fig.4.3.1.: - Horizontal Streamline (CY)

Fig.4.3.2.: - Bottom Streamline (CY)

Fig.4.3.: - Streamline of Model CY at 0-Degree Wind Incidence

➤ AT 30-DEGREE WIND INCIDENCE

When wind act at 30-degree wind incidence wind flow starts to develop an angle to the main faces. Model SY demonstrates high turbulence at the leading edge (Fig.4.4) because wind strike direct to corner while Model CY has a more stable flow along the faces (Fig.4.5). Model CY exhibits more even pressure distribution as compared to SY with higher suction concentration.



➤ AT 60-DEGREE WIND INCIDENCE

When wind act at 60-degree wind incidence the wind strikes the building at an angle, which leading to a turbulent separation that is very energetic on one side, whereas it attaches to the other side. The chamfered corners of Model CY provide a delayed separation (Fig.4.7), and there is appreciable vortex formation in Model SY (Fig.4.6). The total drag in Model CY is slightly reduced because the flow is reattached better.

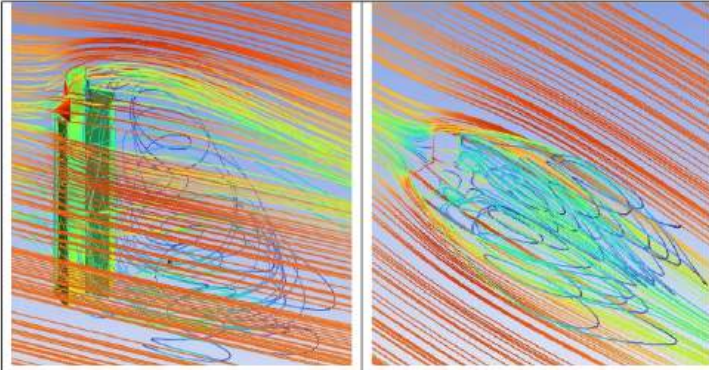


Fig.4.6.1.: - Horizontal Streamline (SY)	Fig.4.6.2.: - Bottom Streamline (SY)
Fig.4.6.: - Streamline of Model SY at 60-Degree Wind Incidence	

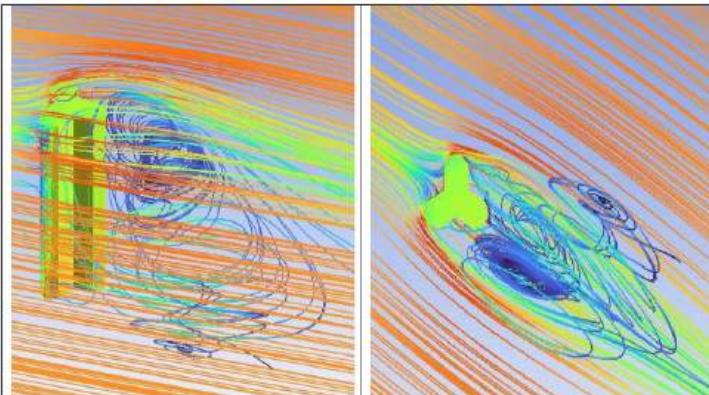


Fig.4.7.1.: - Horizontal Streamline (CY)	Fig.4.7.2.: - Bottom Streamline (CY)
Fig.4.7.: - Streamline of Model CY at 60-Degree Wind Incidence	

➤ AT 90-DEGREE WIND INCIDENCE

When wind act at 90-degree wind incidence its means wind acts perpendicular to the building. In the leeward direction of Model SY, there is a large wake zone behind the building, resulting in strong leeward suction (Fig.4.8). A similar wake but less pronounced occurs at the corners of model CY (Fig.4.9). The aerodynamic stability of the model CY is improved through minimization of the loss of pressure at the corners.

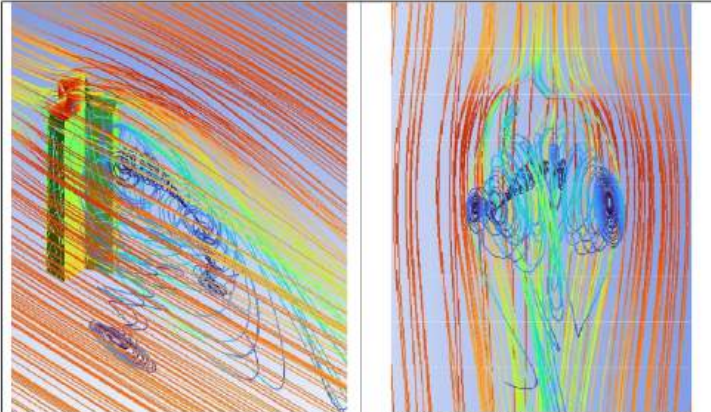


Fig.4.8.1.: - Horizontal Streamline (SY) Fig.4.8.2.: - Bottom Streamline (SY)
Fig.4.8.: - Streamline of Model SY at 90-Degree Wind Incidence

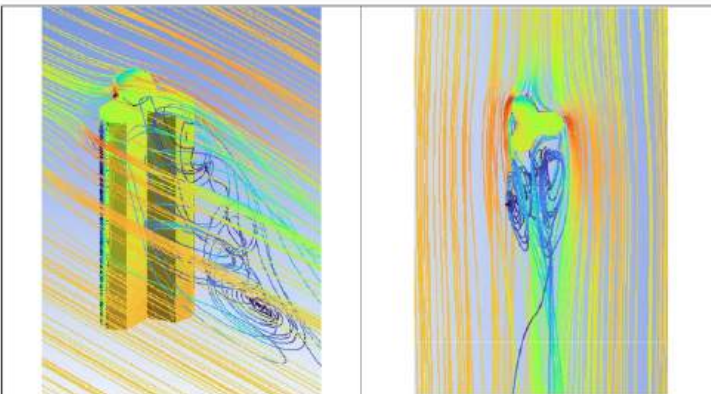
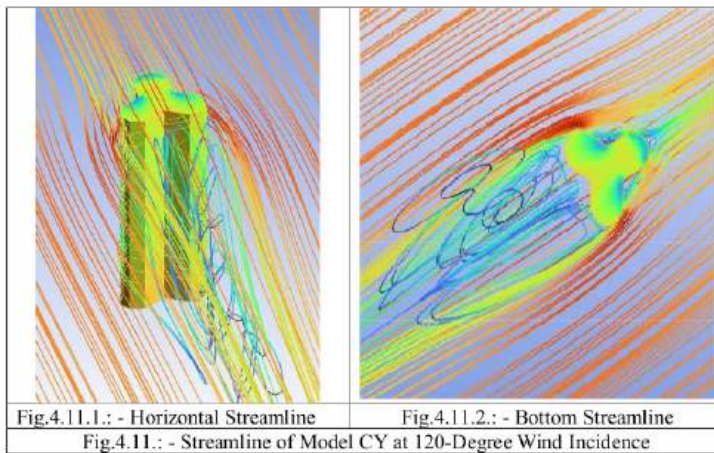
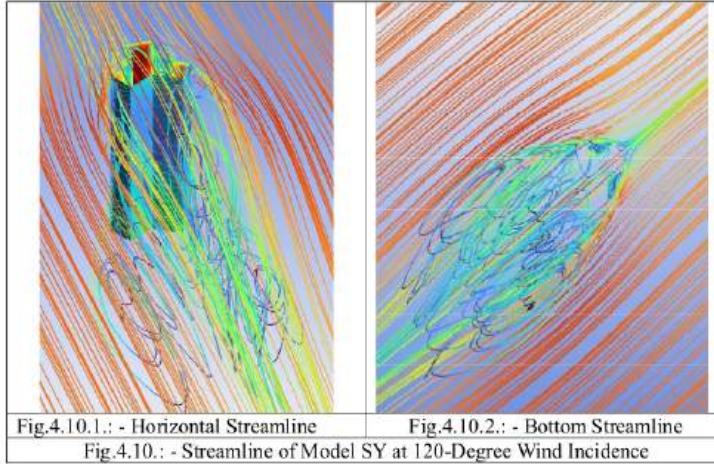


Fig.4.9.1.: - Horizontal Streamline (CY) Fig.4.9.2.: - Bottom Streamline (CY)
Fig.4.9.: - Streamline of Model CY at 90-Degree Wind Incidence

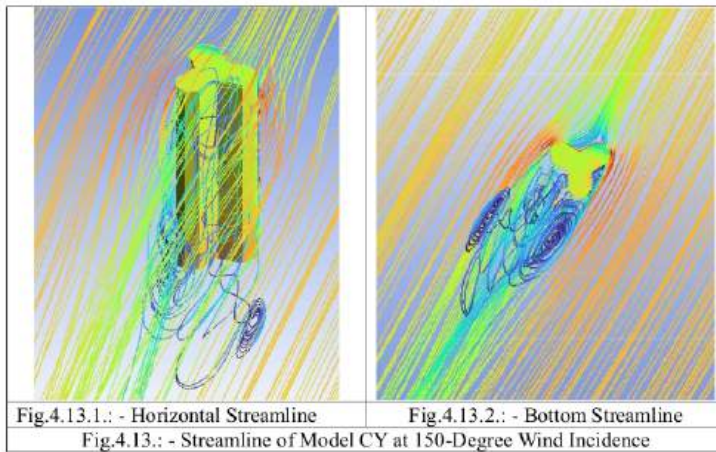
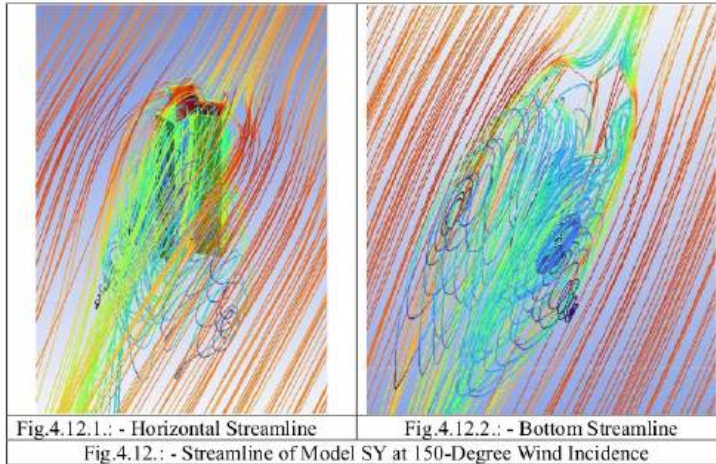
➤ AT 120-DEGREE WIND INCIDENCE

When wind act at 120-degree wind incidence it means the flow meets the building from the backside angle. Complex vortex shedding behind the trailing faces is observed in model SY (Fig.4.10). Model CY helps to direct the flow and reduces flow fluctuations (Fig.4.11). With chamfering, turbulent wake is smaller than standard model.



➤ AT 150-DEGREE WIND INCIDENCE

When wind act at 150-degree wind incidence wind blows at an angle to the back of the building. There is a greater difference in pressure between the faces of Model SY (Fig.4.12). The modified geometry helps to keep Model CY's pressure profile stable. Model CY is more evenly distributed pressure back (Fig.4.13).



➤ AT 180-DEGREE WIND INCIDENCE

When wind act at 180-degree wind incidence it means the wind strikes the back of the building. The wide wake in front of Model SY quickly wraps around the building (Fig.4.14), whereas the chamfered edges in Model CY allow the air to flow more efficiently (Fig.4.15). The net wind loads are smaller with Model CY because the flow separation occurs less abruptly than with Model SY.

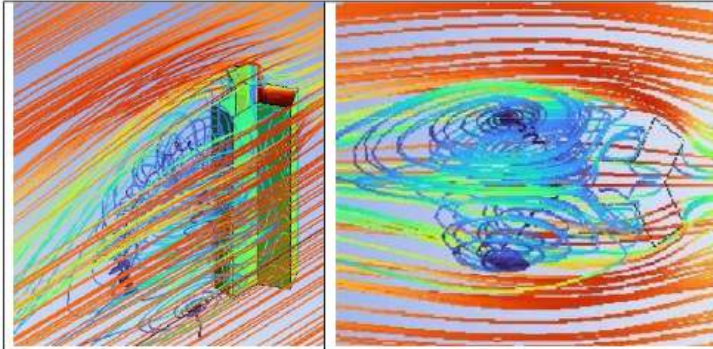


Fig.4.14.1.: - Horizontal Streamline Fig.4.14.2.: - Bottom Streamline (SY)
Fig.4.14.: - Streamline of Model SY at 180-Degree Wind Incidence

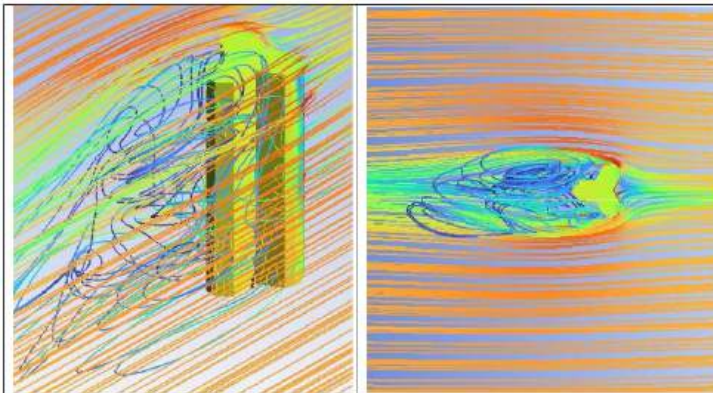


Fig.4.15.1.: - Horizontal Streamline Fig.4.15.2.: - Bottom Streamline
Fig.4.15.: - Streamline of Model CY at 180-Degree Wind Incidence

4.2. ANALYSIS OF PRESSURE CONTOURS

Pressure contours give the building a visual representation of the distribution of wind pressure over its surfaces, and are one of the key indicators of the aerodynamic loading. The maps are color-coded, with red zones representing windward faces that the wind impacts directly, resulting in high positive pressure by the wind's kinetic energy being transferred to impact. In contrast, blue zones are low pressure zones, usually on the sidewalls and the back (leeward) surfaces of buildings, where the air stream detaches from the edges of the buildings, forming low pressure areas.

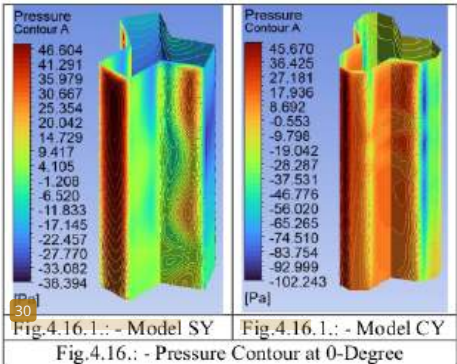
The SY model and the CY model can be easily distinguished by looking at these contours. The SY model shows sharp edges forming intense patches of color, indicating sudden patches of flow separation and higher stress areas. The chamfered corners in the CY model ease the transition of these colors, suggesting a smoother flow of the wind around the building.

4.2.1. PRESSURE CONTOUR AT 0° TO 180° OF WIND INCIDENCE

Following describe the pressure contour behaviors at different degree of wind incidence. These contour results get through the ANSYS CFD and from Experimental data.

➤ AT 0-DEGREE WIND INCIDENCE

The leading face of both models indicates a strong red zone at 0-degree. The blue patches at the leading edges of the models are sharp and concentrated for Model SY but are softer for Model CY because of the abrupt flow separation. Model CY chamfered corners are effective in opening up the suction zones.



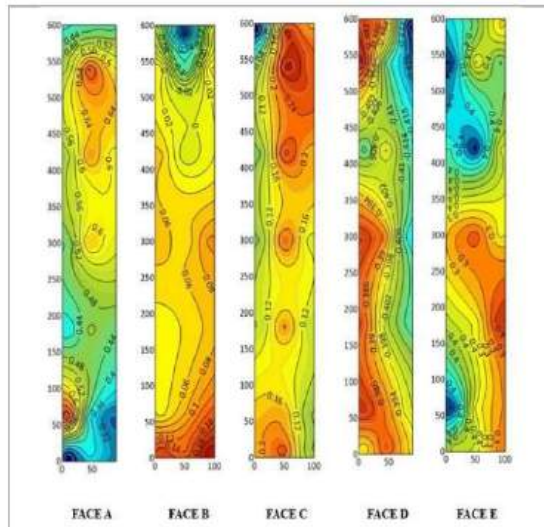


Fig.4.17.1.: - Pressure Contour of Model SY (Face A -E)

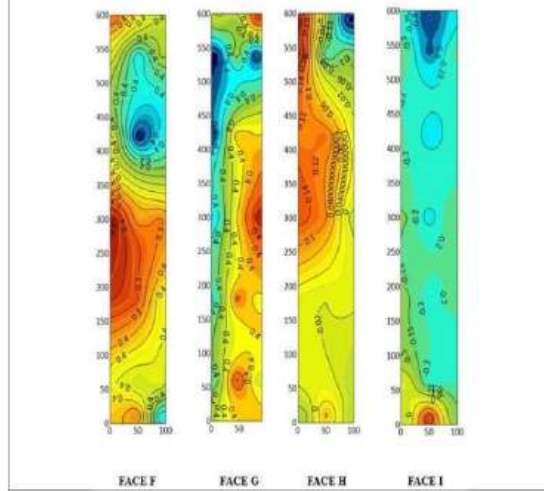


Fig.4.17.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.17.: - Pressure Contour of Model SY At 0-Degree

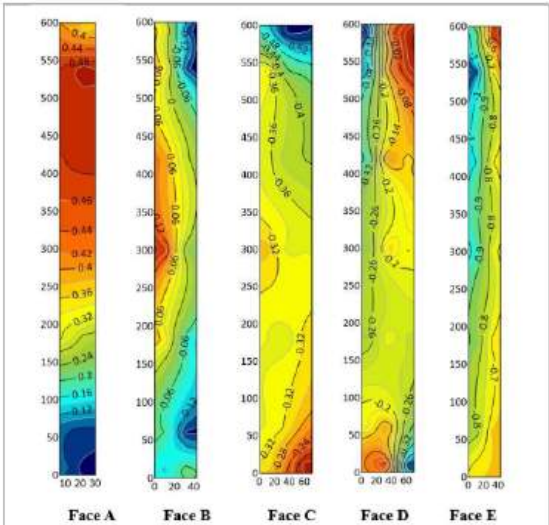


Fig.4.18.1.: - Pressure Contour of Model CY (Face A -E)

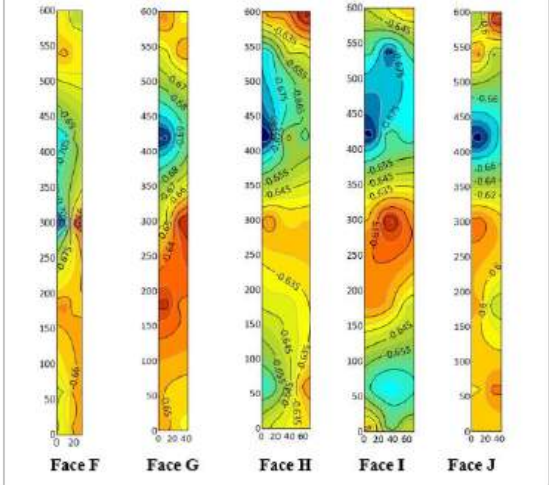


Fig.4.18.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4.18.: - Pressure Contour of Model CY At 30-Degree

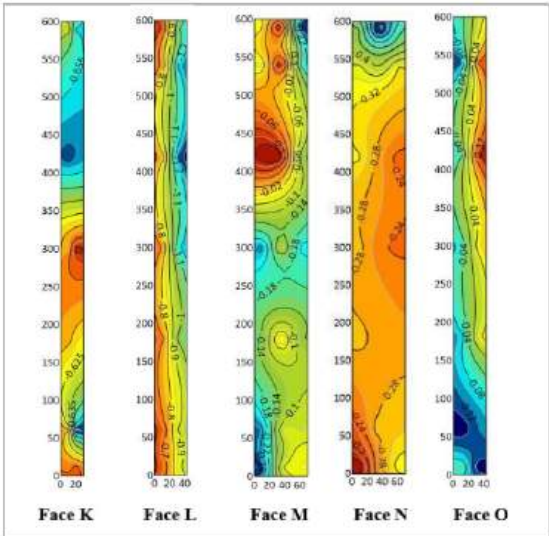


Fig.4.18.3.: - Pressure Contour of Model CY (Face K -O)

Fig.4.18.: - Pressure Contour of Model CY At 0-Degree

➤ AT 30-DEGREE WIND INCIDENCE

The wind is at 30-degree angle act at one part of building. Model SY has an extreme red mark on the corner to the windward side, and Model CY spreads this pressure out. Consideration: The CY model avoids an excessive suction build up at the corner as compared to the SY model.

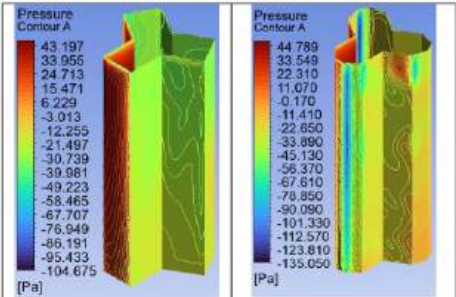


Fig.4.19.1.: - Model SY

Fig.4.19.2.: - Model CY

Fig.4.19.: - Pressure Contour at 30-Degree

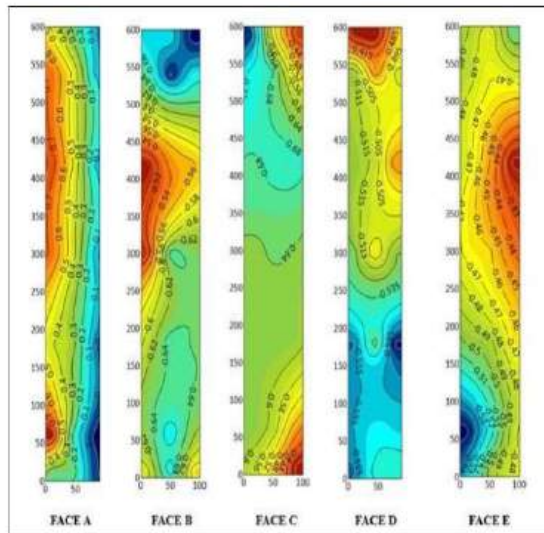


Fig.4.20.1.: - Pressure Contour of Model SY (Face A -E)

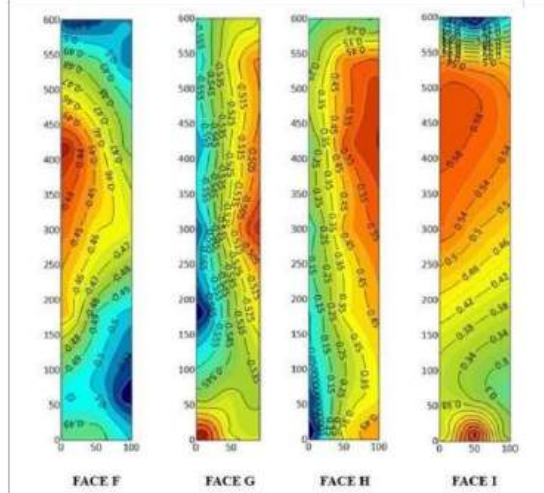


Fig.4.20.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.20.: - Pressure Contour of Model SY At 30-Degree

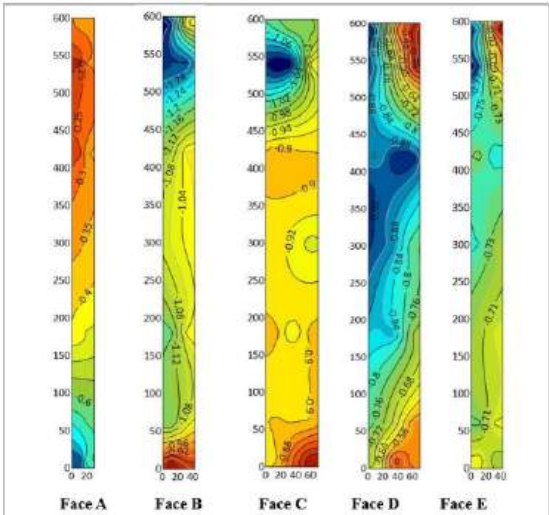


Fig.4.21.1.: - Pressure Contour of Model CY (Face A -E)

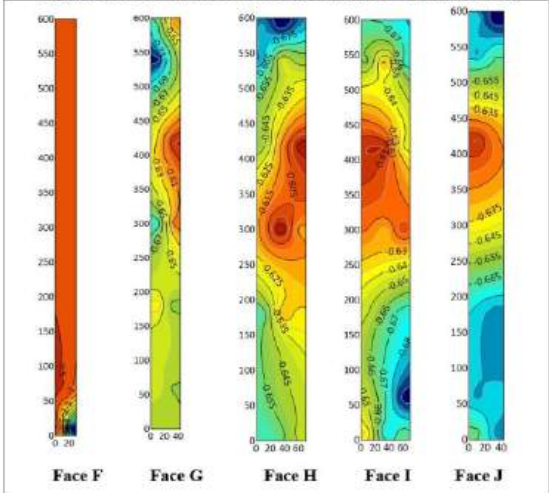


Fig.4.21.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4.21.: - Pressure Contour of Model CY At 30-Degree

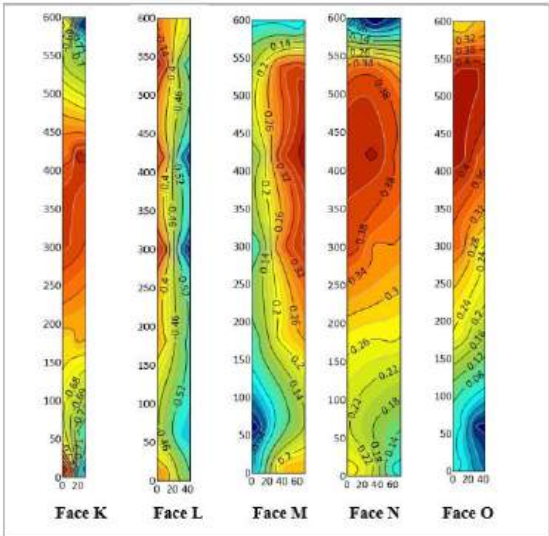


Fig.4.21.3.: - Pressure Contour of Model CY (Face K -O)

Fig.4.21.: - Pressure Contour of Model CY At 30-Degree

➤ AT 60-DEGREE WIND INCIDENCE

The flow starts to circulate around the building. The blue suction area along the side is larger and deeper in Model SY than in Model CY, as can be seen in the Fig.. Chamfered edges help to keep the air from separating from the building to a greater extent, so the total suction area is lowered.

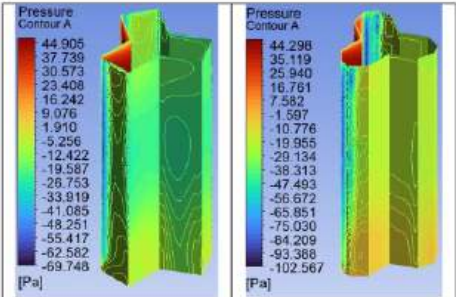


Fig.4.22.1.: - Model SY

Fig.4.22.2.: - Model CY

Fig.4.22.: - Pressure Contour at 60-Degree

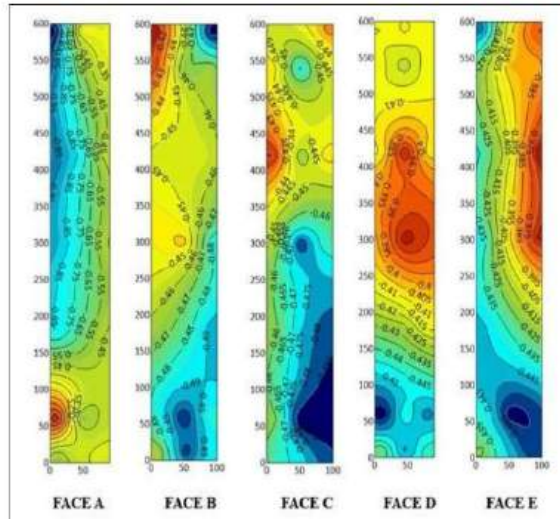


Fig.4.23.1.: - Pressure Contour of Model SY (Face A -E)

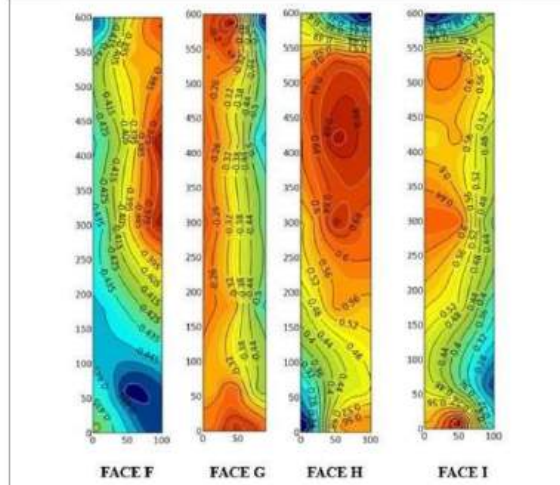


Fig.4.23.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.23.: - Pressure Contour of Model SY At 60-Degree

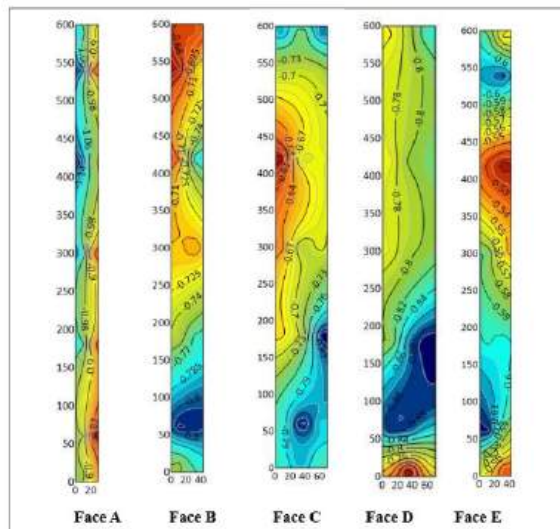


Fig.4.24.1.: - Pressure Contour of Model CY (Face A -E)

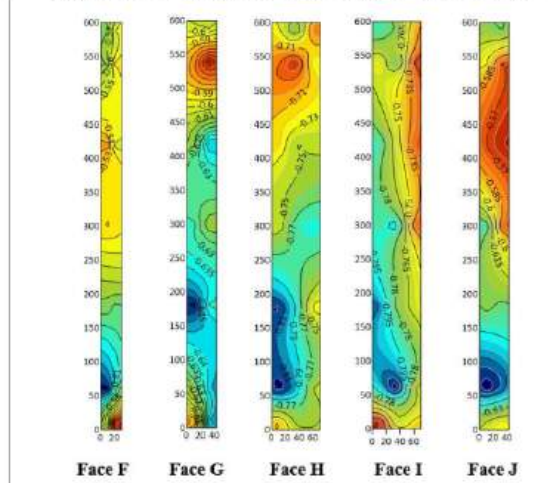
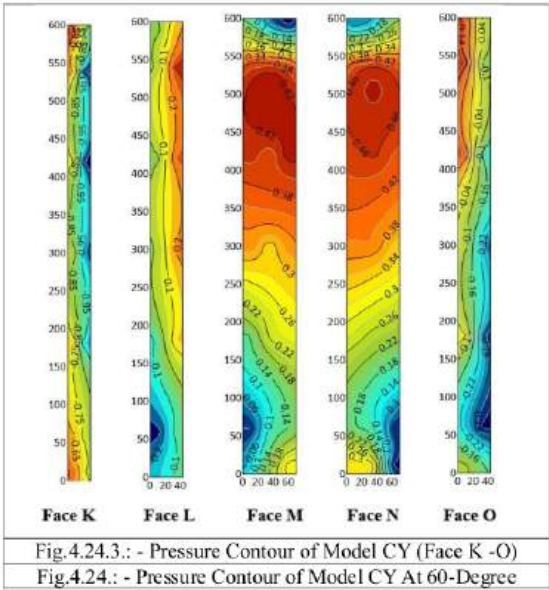


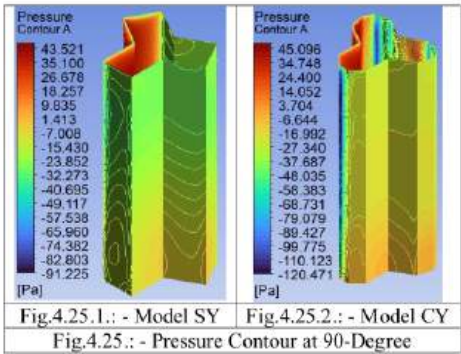
Fig.4.24.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4.24.: - Pressure Contour of Model CY At 60-Degree



➤ AT 90-DEGREE WIND INCIDENCE

The wind is perpendicular to a building. On top of the front of Model SY is a distinct red nose. A similar red nose is evident on the Model CY, although the blue suction areas at the rear edges are greatly reduced. Model CY makes the separation point less abrupt, thus reducing the overall pressure drag.



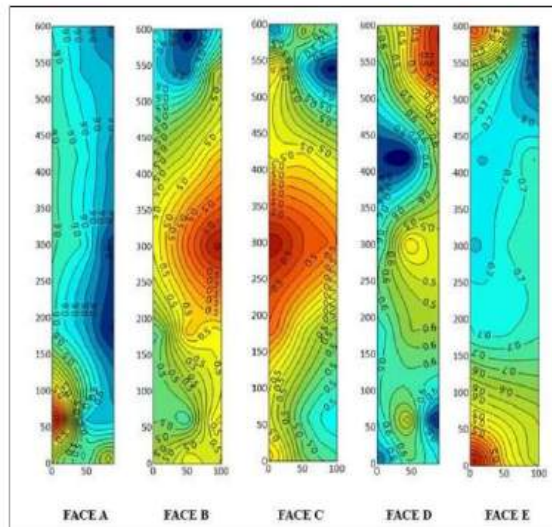


Fig.4.26.1.: - Pressure Contour of Model SY (Face A -E)

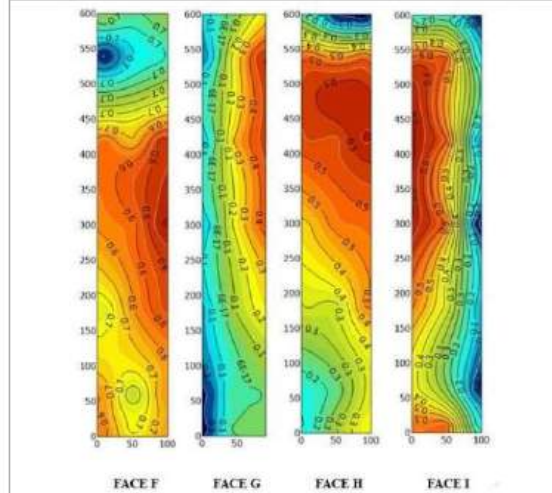


Fig.4.26.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.26.: - Pressure Contour of Model SY At 90-Degree

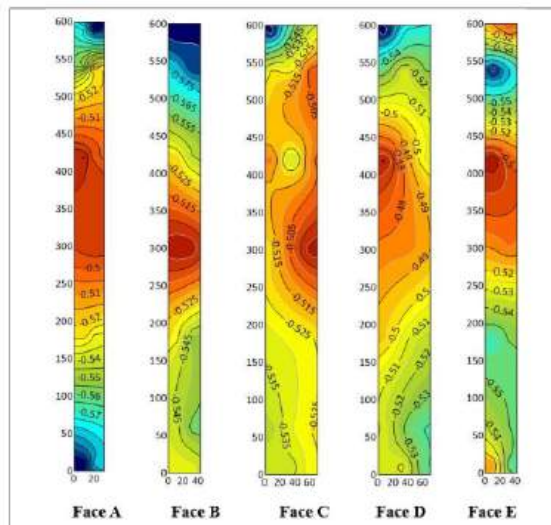


Fig.4.27.1.: - Pressure Contour of Model CY (Face A -E)

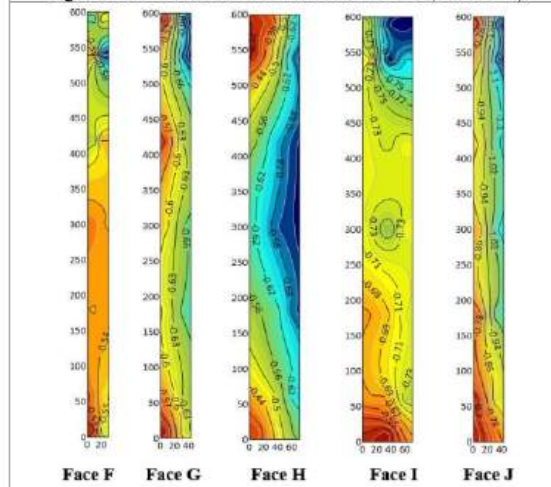


Fig.4.27.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4.27.: - Pressure Contour of Model CY At 90-Degree

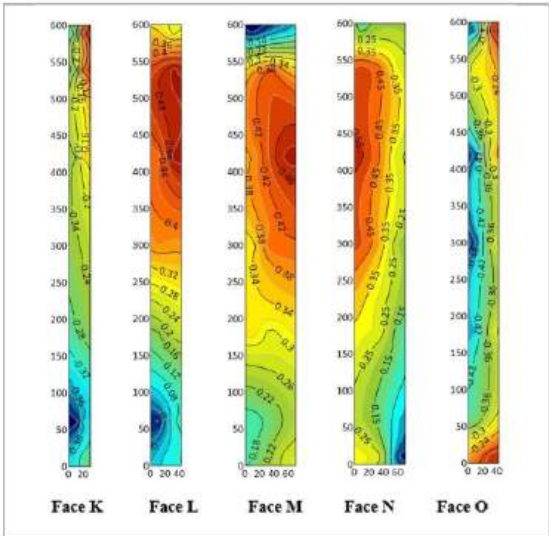


Fig.4.27.3.: - Pressure Contour of Model CY (Face K -O)

Fig.4.27.: - Pressure Contour of Model CY At 90-Degree

➤ AT 120-DEGREE WIND INCIDENCE

The flow is coming closer to the rear-side angle. Model SY shows a broken blue area in front side. Model CY has a more regular, consistent pressure contour on these faces. Model CY produces a more consistent flow pattern, which has less suction distribution, than either of the other models.

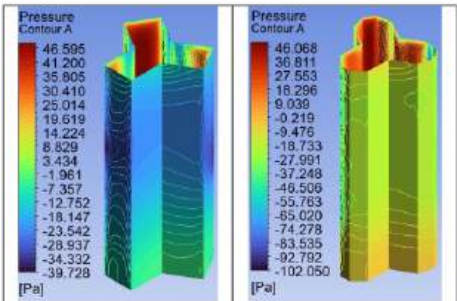


Fig.4.28.1.: - Model SY

Fig.4.28.1.: - Model CY

Fig.4.28.: - Pressure Contour at 120-Degree

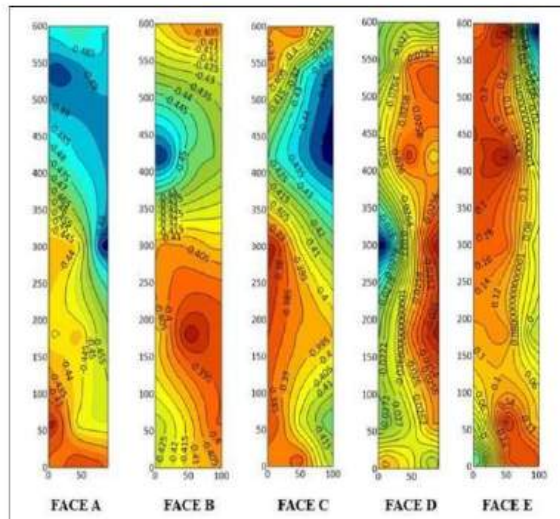


Fig.4.29.1.: - Pressure Contour of Model SY (Face A -E)

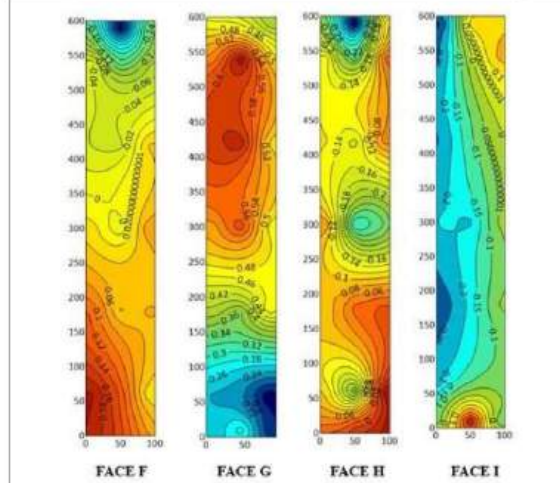


Fig.4.29.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.29.: - Pressure Contour of Model SY At 120-Degree

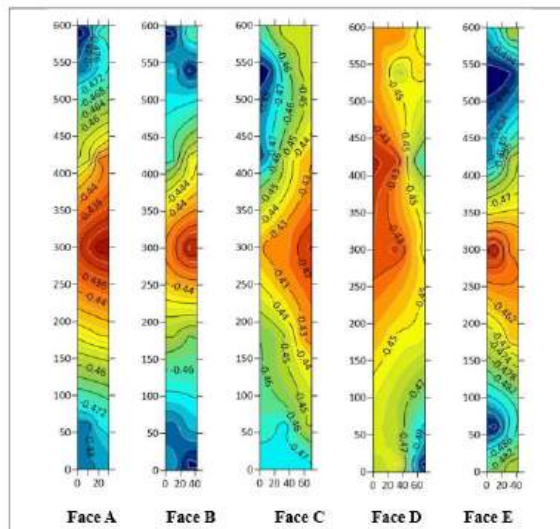


Fig.4.30.1.: - Pressure Contour of Model CY (Face A -E)

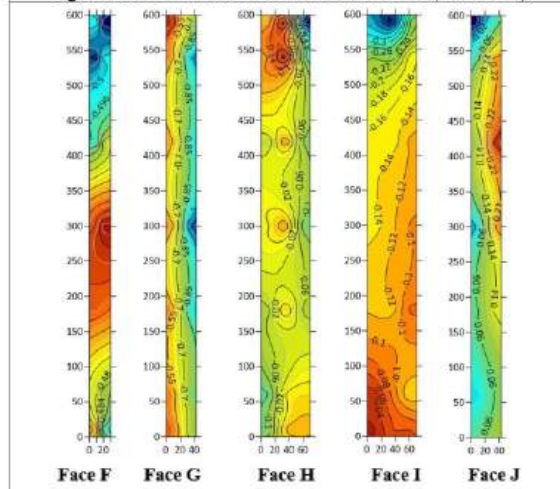
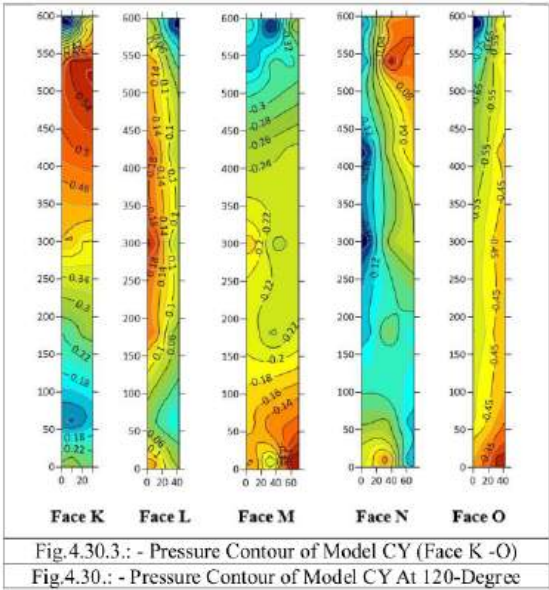


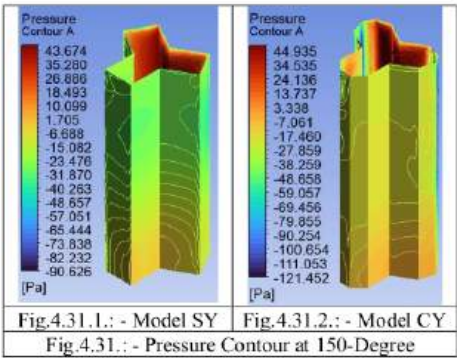
Fig.4.30.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4.30.: - Pressure Contour of Model CY At 120-Degree



➤ AT 150-DEGREE WIND INCIDENCE

The wind is also shallow on the backs. The pressure colors are not the same on Model SY, which can be used to determine if the flowing is not steady. Model CY has less sensitivity to the wind direction and more uniform color gradients. In comparison, the chamfering forms a more aerodynamically "calm" profile for the structure.



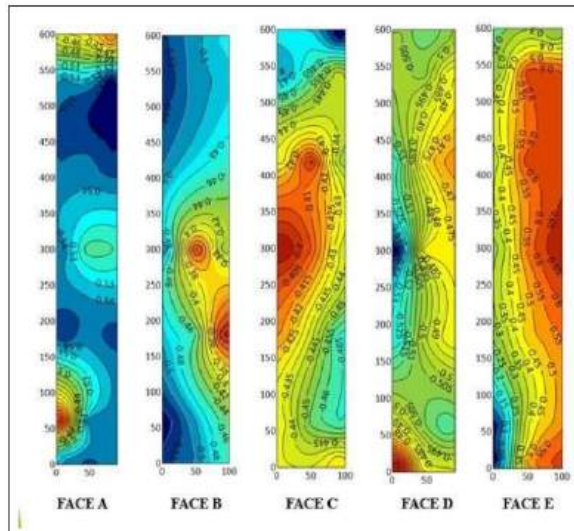


Fig.4.32.1.: - Pressure Contour of Model SY (Face A -E)

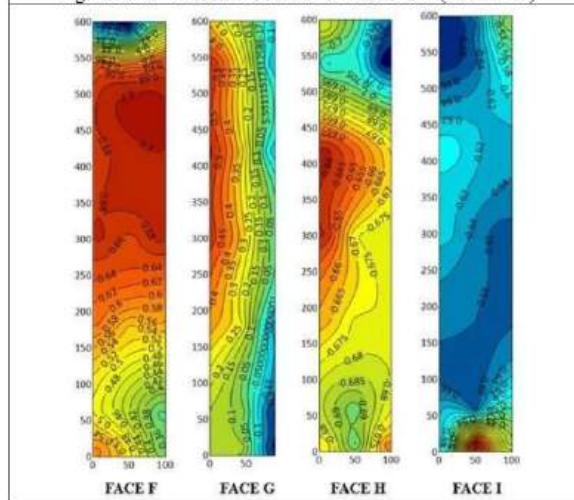


Fig.4.32.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.32.: - Pressure Contour of Model SY At 150-Degree

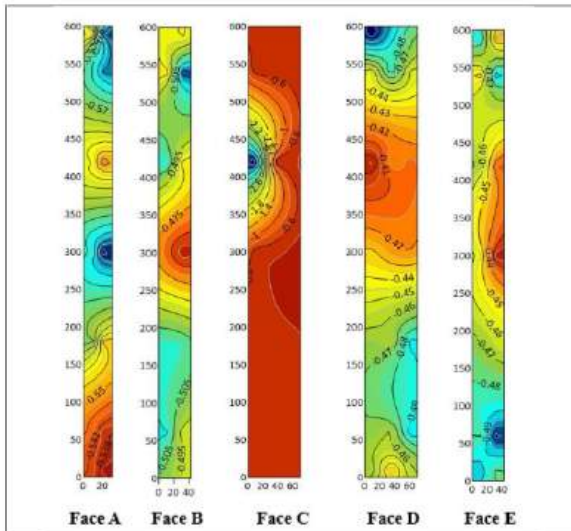


Fig.4.33.1.: - Pressure Contour of Model CY (Face A -E)

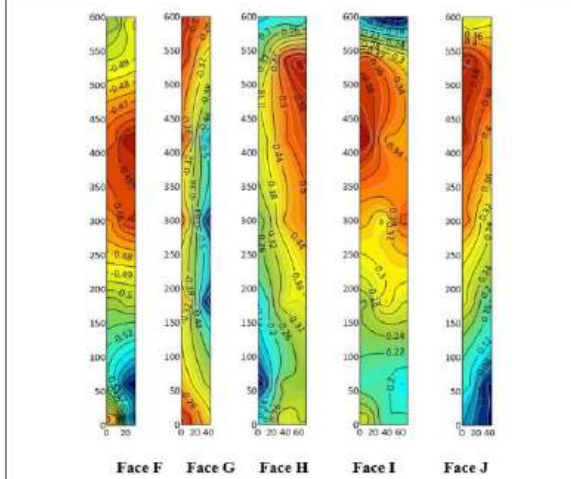
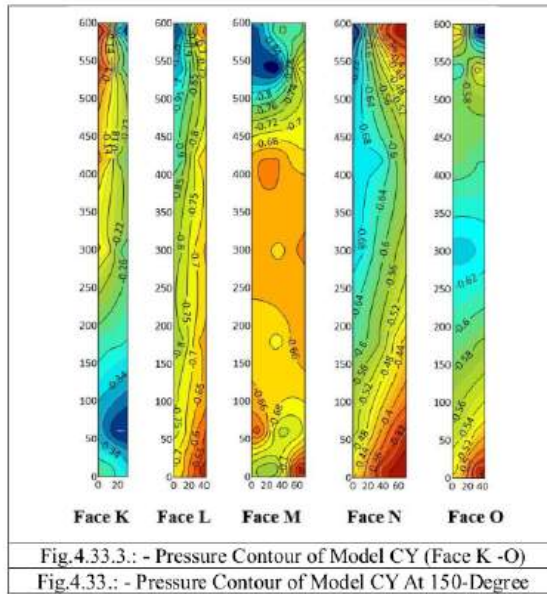


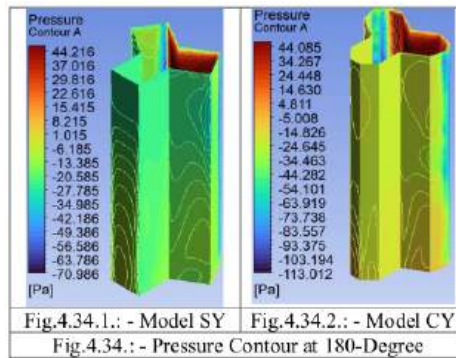
Fig.4.33.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4.33.: - Pressure Contour of Model CY At 150-Degree



➤ AT 180-DEGREE WIND INCIDENCE

The wind blows into the back of the building. Model SY shows a wide, diffuse blue zone representing a large wake. Model CY has a less pronounced wake and blue suction areas. The model CY offers better aerodynamic efficiency as the wind can leave the building profile cleaner.



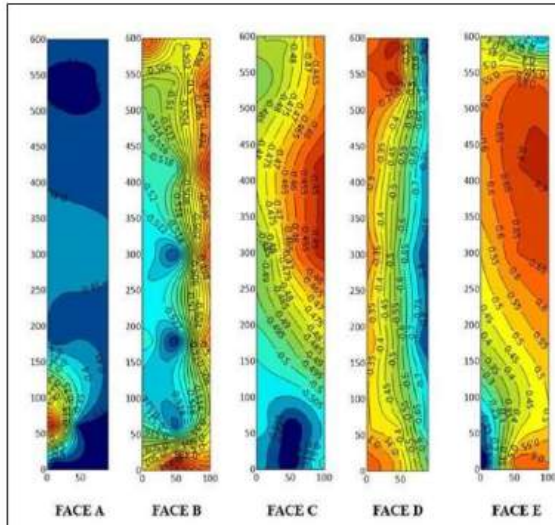


Fig.4.35.1.: - Pressure Contour of Model SY (Face A -E)

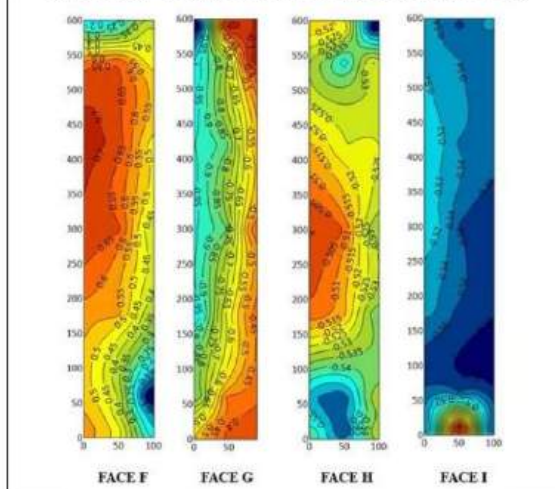


Fig.4.35.2.: - Pressure Contour of Model SY (Face F -I)

Fig.4.35.3.: - Pressure Contour of Model SY At 180-Degree

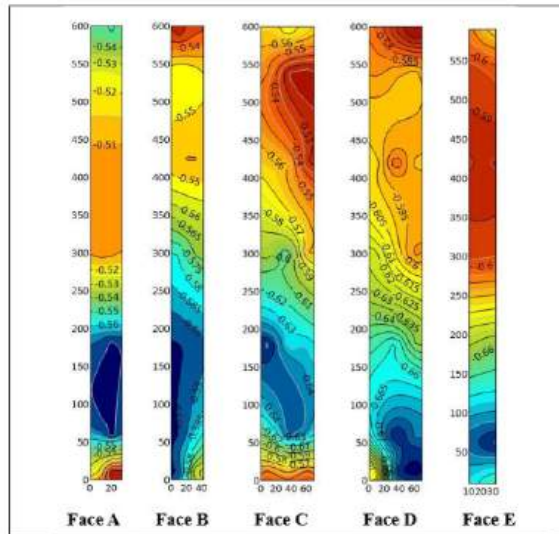


Fig.4.36.1.: - Pressure Contour of Model CY (Face A -E)

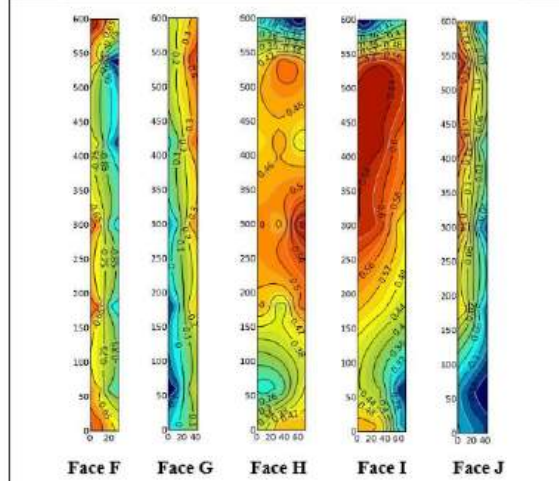
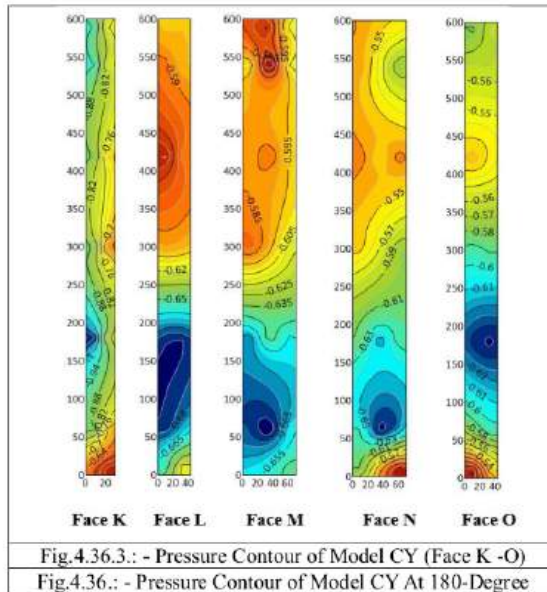


Fig.4.36.2.: - Pressure Contour of Model CY (Face F -J)

Fig.4. 36.: - Pressure Contour of Model CY At 180-Degree



4.3. COMPARATIVE ANALYSIS OF PRESSURE COEFFICIENT

The pressure coefficient (C_p), is a dimensionless, fundamental parameter in wind engineering that is used to quantify and compare the wind pressure that affects the surface of a building as briefly explain in Chapter 3 (Section 3.7). C_p scales down the data, so that researchers can make comparisons and study the performance of the different building shapes and models, like the Standard Y (SY) and Chamfered Y (CY) models, under different wind speeds and environmental conditions.

For the purposes of this study, C_p is the most important to determine aerodynamic regions that are critical, positive indicates windward side where the air act directly, negative indicates the suction regions, typically on the sidewalls and the leeward side. These coefficients can be plotted out on the exterior of the building to visualize the intensity of the wind forces.

In this section the pressure coefficient (C_p) distribution is shown for both the Standard (SY) and Chamfered (CY) models in Table 4.1 and Table 4.2 respectively. The data has been arranged in a logical sequence that helps to clearly illustrate how each model performed aerodynamically and the effect of the chamfered design.

Table 4.1 Pressure Coefficient (C_p) of Model SY

FACE	STANDARD Y SHAPE MODEL (MODEL SY)															
	0 DEGREE		30 DEGREE		60 DEGREE		90 DEGREE		120 DEGREE		150 DEGREE		180 DEGREE			
	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED
A	0.51	0.49	0.28	0.32	-0.53	-0.56	-0.49	-0.56	-0.38	-0.46	-0.50	-0.52	-0.38	-0.44		
B	0.01	0.01	-0.58	-0.62	-0.39	-0.47	-0.47	-0.52	-0.35	-0.42	-0.42	-0.47	-0.47	-0.51		
C	0.10	0.15	-0.54	-0.60	-0.37	-0.46	-0.40	-0.51	-0.33	-0.41	-0.38	-0.44	-0.44	-0.49		
D	-0.30	-0.40	-0.49	-0.52	-0.34	-0.42	-0.50	-0.53	-0.40	-0.44	-0.46	-0.50	-0.42	-0.47		
E	-0.26	-0.36	-0.50	-0.48	-0.37	-0.45	-0.61	-0.65	0.09	0.09	0.43	0.42	0.53	0.49		
F	-0.27	-0.36	-0.45	-0.48	-0.32	-0.42	-0.59	-0.66	0.01	0.01	0.58	0.53	0.53	0.47		
G	-0.33	-0.40	-0.52	-0.53	-0.34	-0.33	0.12	0.08	0.52	0.43	0.21	0.15	-0.59	-0.64		
H	0.09	0.06	0.38	0.32	0.54	0.49	0.42	0.36	0.00	-0.13	-0.63	-0.68	-0.48	-0.53		
I	-0.01	-0.03	0.45	0.42	0.55	0.49	0.44	0.34	-0.10	-0.07	-0.52	-0.59	-0.54	-0.52		
CFD - CFD DATA (From ANSYS)																
ED - Experimental data (From Wind Tunnel)																

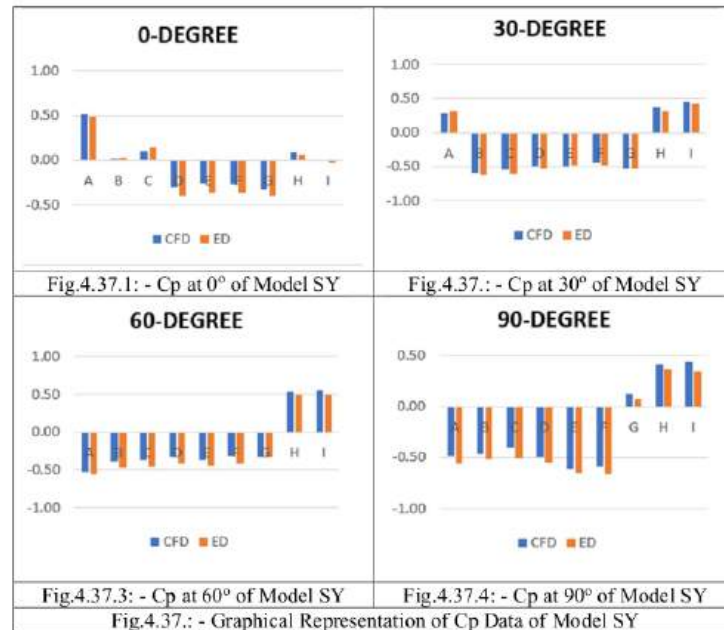
Table 4.2 Pressure Coefficient (C_p) of Model CY

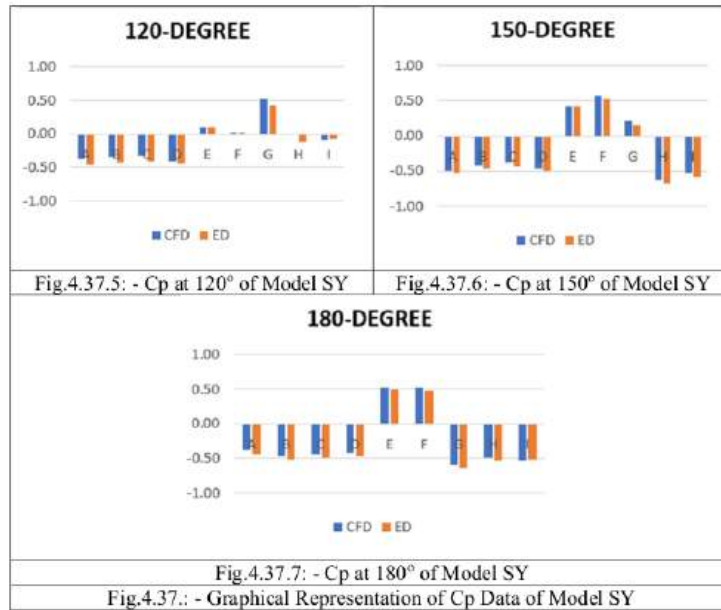
FACE	CHAMFERED Y SHAPE MODEL (MODEL CY)															
	0 DEGREE		30 DEGREE		60 DEGREE		90 DEGREE		120 DEGREE		150 DEGREE		180 DEGREE			
	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED	CFD	ED
A	0.38	0.32	-0.42	-0.44	-0.79	-0.88	-0.94	-0.84	-0.41	-0.46	-0.34	-0.46	-0.31	-0.42		
B	-0.20	-0.10	-0.96	-1.10	-0.39	-0.44	-0.03	-0.02	-0.29	-0.36	-0.31	-0.40	-0.36	-0.43		
C	0.09	0.06	-0.49	-0.55	-0.28	-0.34	-0.29	-0.39	-0.27	-0.35	-0.29	-0.39	-0.35	-0.48		
D	0.08	0.40	-0.58	-0.65	-0.41	-0.46	-0.30	-0.41	-0.27	-0.35	-0.29	-0.36	-0.32	-0.45		
E	-0.69	-0.78	-0.39	-0.46	-0.31	-0.44	-0.29	-0.43	-0.28	-0.38	-0.31	-0.44	-0.48	-0.53		
F	-0.34	-0.48	-0.32	-0.43	-0.30	-0.41	-0.32	-0.45	-0.40	-0.48	-0.54	-0.69	-0.93	-0.83		
G	-0.29	-0.34	-0.29	-0.38	-0.36	-0.48	-0.40	-0.55	-0.73	-0.69	-0.68	-0.63	0.02	0.04		
H	-0.26	-0.36	-0.29	-0.33	-0.34	-0.45	-0.36	-0.45	0.06	0.02	0.40	0.32	0.60	0.53		
I	-0.26	-0.38	-0.28	-0.36	-0.32	-0.47	-0.45	-0.53	0.09	0.04	0.56	0.47	0.60	0.48		
J	-0.29	-0.35	-0.30	-0.34	-0.45	-0.51	-0.50	-0.51	0.19	0.11	0.52	0.47	0.01	0.01		
K	-0.38	-0.42	-0.49	-0.60	-0.88	-0.80	-0.37	-0.25	0.58	0.43	-0.42	-0.34	-0.95	-0.80		
L	-0.71	-0.73	-0.61	-0.56	0.06	0.04	0.53	0.46	0.19	0.12	-0.97	-0.84	-0.49	-0.58		
M	0.08	0.69	0.40	0.80	0.60	0.55	0.56	0.49	0.10	0.04	-0.49	-0.57	-0.33	-0.48		
N	0.10	0.20	0.56	0.40	0.60	0.57	0.39	0.27	0.07	0.03	-0.38	-0.45	-0.38	-0.43		
O	0.19	0.20	0.52	0.45	0.09	0.08	-0.69	-0.53	-0.74	-0.60	-0.40	-0.51	-0.34	-0.47		
CFD - CFD DATA (From ANSYS)																
ED - Experimental data (From Wind Tunnel)																

When comparing the pressure coefficient data, the improvements in aerodynamics can be seen clearly for the Standard Y (SY) model versus the Chamfered Y (CY) model. The pressure distribution is studied and observed that the chamfered corners of the Model CY are able to reduce the intensity of suction as compared to the sharp edged model SY. Examples of this include the concentrated high magnitude suction zones on the sidewalls, sometimes greater than C_p values of -0.60, that occur at the critical wind incidence angles, such as 60-degree and 90-degree, in the SY models as shown in table. On the contrary, the CY model has a medium and progressive pressure distribution. The chamfers help control the flow around the structure so that the flow does not separate off the surface as much, and the wake region is reduced, the latter being the main cause of excessive suction.

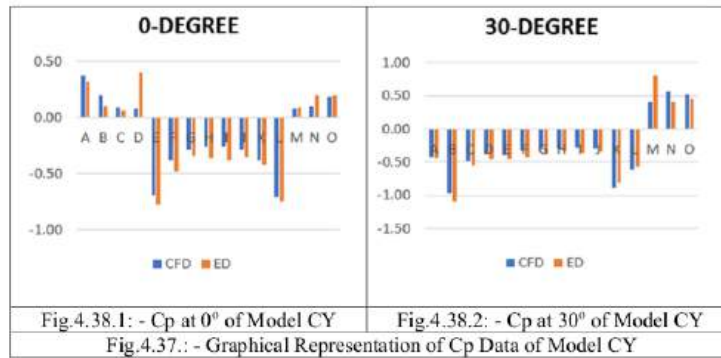
These results are consistent and reliable, as the model results from the numerical simulation using ANSYS CFX software well compare to the experimental data taken on the wind tunnel. The CFD results are very close to the experimental data (ED) results for all incidence angles and building faces with differences generally within a small and acceptable margin. For example, for model SY at 0-degree, the CFD value for face A was 0.51 compared to 0.49 (experimental value) and for model CY at 180-degree, the CFD value for face H was 0.60, and the experimental value was 0.53. There is good agreement between the numerical and experimental suction pressures. The layered validation of both reduction in suction from SY to CY and consistency between simulation and experiment make the conclusion from the research robust and based on data. It illustrates how significant architectural modifications, such as chamfering the corners, can be to stabilize the distribution of pressures and to reduce the wind forces, and therefore help achieve a more aerodynamically efficient design for a tall building. Also for the better understanding here the graphical representation of Cp data for both Model SY and maodel CY from the of Table 4.1 and Table 4.2 repectively.

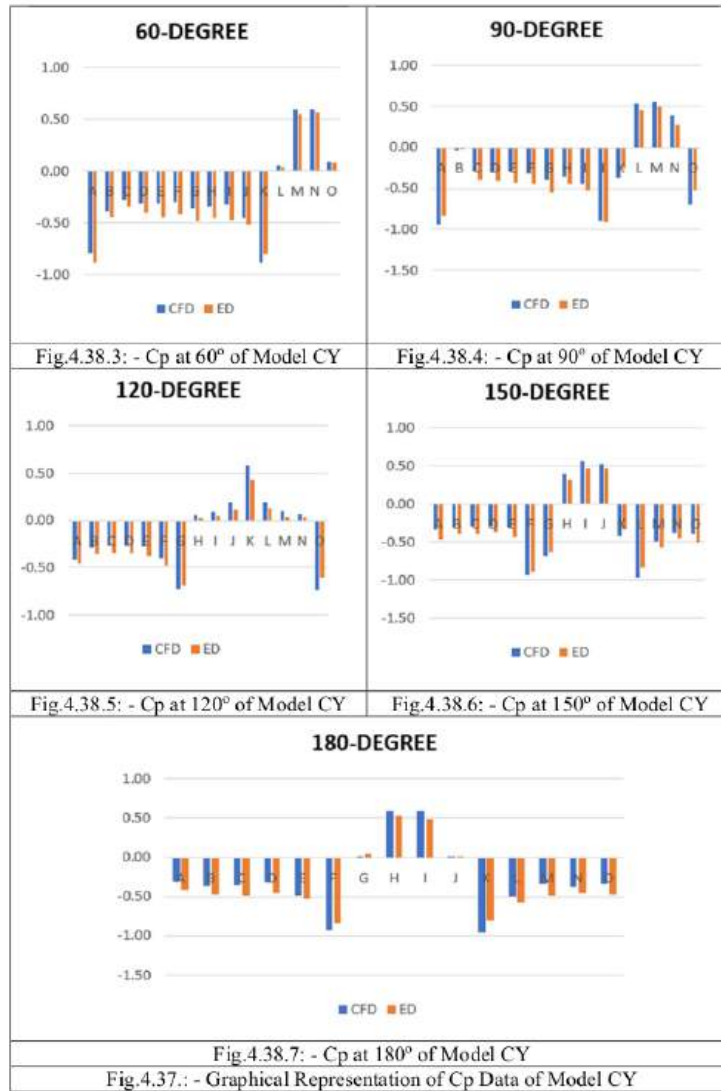
➤ GRAPHICAL REPRESENTATION OF Cp DATA FROM TABLE 4.1





➤ **GRAPHICAL REPRESENTATION OF Cp DATA FROM TABLE 4.2**





21 CHAPTER 5 CONCLUSION

5.1. RESEARCH SUMMARY

This study aims to assess the aerodynamic characteristics of Y-shaped tall structures, and its effects on aerodynamic resistance by changing the geometric parameters. A Standard Y-shaped (SY) model is compared with a Chamfered Y-shaped (CY) model over the entire range of wind incidence angles 0-degree to 180-degree at 30-degree wind incidence interval. The study adopts a hybrid approach combining computational fluid dynamics (CFD) with ANSYS CFX and physical testing in a boundary layer wind tunnel.

Key findings show that chamfered comers are an improved design approach. The comparative study reveals that chamfering can help prevent flow separation, facilitate a smooth wind flow around the building, and markedly reduce the magnitude of peak suction (negative pressure) which is generally the cause of structural instability. The research reveals that the CY model has a more even pressure distribution than the SY model, which produces excessive pressures at the sharp edges at the front.

The validation process shows that there is a good agreement between a numerical simulation result and experimental data. The high correlation at all angles tested. Finally, this study has a robust engineering justification for the use of chamfers in modern architecture. Engineers can make substantial reductions in lateral wind loads through optimization of the shape of the building, which results in safer structural designs, reduced material usage and enhanced pedestrian comfort in the street. The findings from this study will be a building block to future development of "green", "resilient" and "economical" design of a skyscraper.

5.2. KEY FINDING IMPLICATION

- **Aerodynamic optimization:** The addition of chamfers at the comers of the Y-shaped building, greatly enhances its aerodynamic performance. The numerical and experimental results have been seen to be the same; that is, the peak suction of the model CY is lower than the model SY with sharp edges.
- **Validation of Methodology:** The ANSYS CFX results compared well with wind tunnel experimental data (ED), thus ensuring the accuracy of our method. The numerical model successfully predicts the C_p value at different incidence angle

values 0-degree to 180-degree, which gives a good basis for future structural design.

- **Streamline analysis:** Streamline analysis indicates chamfering further delays flow separation and promotes a smooth flow of air around the building, termed Flow Mitigation. This decrease in the intensity of the vortexes directly results in a decrease of wind-induced stress on the building's facade.
- **Structural Efficiency:** Chamfered corners promote a more even pressure distribution. This balance helps to avoid excessive pressure peaks, enabling better planning of structures and possibly decreasing the amount of material needed for the facade and lateral load resistance systems.

5.3. FUTURE RESEARCH RECOMMENDATION

The potential for further research in the field outlined below: -

- **Dynamic Loading:** Future research should include dynamic wind effects including vortex-induced vibration across the wind to gain insights on the effect of chamfering on the long-term fatigue life of the building.
- **Further experimental testing:** The effects of different chamfer angles or architectural features, such as tapering or twisting, could be investigated for additional optimization of wind loading in high-density urban settings.
- **Multi-Building Interaction:** Further research is needed to take into account the "interference effect" at 0-degree to 180-degree wind incidence, which is the effect of neighboring high-rise buildings on wind flows, and obtain a more realistic estimation of wind loads in an urban environment.

5.4. SOCIAL, ECONOMIC, AND SAFETY IMPACT

Future work should incorporate these aerodynamic findings with structural material optimization and determine how these effects can minimize the carbon footprint of tall buildings by designing structures that also require less structural reinforcement.

- **Improved structural safety:** Chamfered configurations lower the suction pressure, thus minimizing damage to the facade and structural fatigue, and boosting the safety and service life of the skyscraper.
- **Economic Viability:** The wind loading optimized buildings, geometrically designed, need less structural reinforcement which can be heavy. This results in

substantial material benefits for the construction process, and hence economic benefits and cost reduction for tall building developments.

- **Better urban comfort:** The improvement of wind flow reduces the generation of uncomfortable wind flow situations leading to a safer and more pleasant urban environment for urban residents.
- **Aerodynamically efficient shapes:** This is a development in architecture towards sustainable development. This method reduces the need for cumbersome structural elements, thereby enabling the creation of more environmentally friendly, resource-efficient high-rise buildings that help to shape the modern urban environment.

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF THE AERODYNAMIC PERFORMANCE OF Y-SHAPED TALL BUILDINGS WITH CHAMFERED CORNERS MODIFICATION

ORIGINALITY REPORT

5%

SIMILARITY INDEX

2%

INTERNET SOURCES

5%

PUBLICATIONS

1%

STUDENT PAPERS

PRIMARY SOURCES

- | | | |
|---|--|-----|
| 1 | A. Larsen, G. L. Larose, F. M. Livesey. "Wind Engineering into the 21st Century", CRC Press, 2026
Publication | 1% |
| 2 | A. Larsen, G. L. Larose, F. M. Livesey. "Wind Engineering into the 21st Century - Proceedings of the Tenth International Conference on Wind Engineering, Copenhagen, Denmark, 21-24 June 1999", CRC Press, 2026
Publication | 1% |
| 3 | iaeme.com
Internet Source | <1% |
| 4 | Ching-Jen Chen, Chiang Shih, Jeffrey Lienau, Robin J. Kung. "Flow Modeling and Turbulence Measurements VI", CRC Press, 2026
Publication | <1% |
| 5 | Submitted to Universiti Sains Malaysia
Student Paper | <1% |
| 6 | www.ijamtes.org
Internet Source | <1% |
| 7 | ebin.pub
Internet Source | <1% |
| 8 | link.springer.com
Internet Source | |

<1 %

9 irrigate.whu.edu.cn
Internet Source

<1 %

10 Mohammad Mohsin Khan, Amrit Kumar Roy.
"Numerical investigation of interference
effects on tall power station chimney
subjecting varying wind incident angles",
Journal of Building Pathology and
Rehabilitation, 2024
Publication

<1 %

11 Submitted to Jawaharlal Nehru Technological
University
Student Paper

<1 %

12 core.ac.uk
Internet Source

<1 %

13 Mohammad Mohsin Khan, Amrit Kumar Roy.
"Interference effect of buildings on high rise
power station chimney subjected to wind: a
numerical modelling approach", Innovative
Infrastructure Solutions, 2024
Publication

<1 %

14 "Proceedings of the Indian Structural Steel
Conference 2020 (Vol. 1)", Springer Science
and Business Media LLC, 2024
Publication

<1 %

15 Rahul Kumar Meena, Ritu Raj, S. Anbukumar,
Mohammad Iqbal Khan, Jamal M. Khatib.
"Fluid Dynamic Assessment of Tall Buildings
with a Variety of Complicated Geometries",
Buildings, 2024
Publication

<1 %

Centrifugal Pumps, 2014.

16	Publication	<1 %
17	www.mdpi.com Internet Source	<1 %
18	Submitted to University of Pune Student Paper	<1 %
19	Submitted to University of Queensland Student Paper	<1 %
20	KRUSHNARTH CHONDIKAR. "Design of Overhead Tank with Carbon Fibre/Fiber Rebar as Reinforcement and Time History Analysis using STAAD-Pro software.", Research Square Platform LLC, 2022 Publication	<1 %
21	honors.libraries.psu.edu Internet Source	<1 %
22	Submitted to Institute of Technology, Sligo Student Paper	<1 %
23	Keyser, J. P.B.. "Evaluation of a Personal Growth Model for Leadership Development.", University of Johannesburg (South Africa), 2021 Publication	<1 %
24	Submitted to Universitas Brawijaya Student Paper	<1 %
25	pure-oai.bham.ac.uk Internet Source	<1 %
26	eprints.hud.ac.uk Internet Source	<1 %
27	www.msc-sahc.org Internet Source	<1 %

28 "Advances in Construction Management", Springer Science and Business Media LLC, 2025
Publication

29 Abdul Haseeb Wani, Rajendra Kumar Varma, Ashok Kumar Ahuja. "ON EVALUATION OF THE INDIAN STANDARD TOPOGRAPHY FACTOR FOR DESIGN WIND SPEED ON ESCARPMENTS", ASEAN Journal on Science and Technology for Development, 2023
Publication

30 Submitted to City University
Student Paper

31 Da-Wen Sun. "Computational Fluid Dynamics in Food Processing", Routledge, 2018
Publication

32 Ebrahim Rajabi, Hossein Sadeghi, Mohammad Reza Hashemi. "Wind effect on building with Y-shaped plan", Asian Journal of Civil Engineering, 2022
Publication

33 Padmakar Maddala, Leela Priyanka Mudda, Barhmaiah Borigarla, Rakesh Pydi. "Computer aided seismic analysis and design of a G+10 multistoried residential structure", Materials Today: Proceedings, 2021
Publication

34 Prasenjit Sanyal, Sujit Kumar Dalui. "Comparison of aerodynamic coefficients of various types of Y-plan-shaped tall buildings", Asian Journal of Civil Engineering, 2020
Publication

35

Sushil Kumar Dhawan, Abhinav Bindal, Suresh Bhalla, Bishwajit Bhattacharjee. "Expected residual service life of reinforced concrete structures from current strength considerations", Advances in Structural Engineering, 2019

Publication

<1 %

36

Masanori Izumi, Tsutomu Nakamura, Ronald L. Sack. "Snow Engineering: Recent Advances", CRC Press, 2026

Publication

<1 %

Exclude quotes Off

Exclude matches Off

Exclude bibliography Off