

COMPARATIVE ANALYSIS OF CUT-CORNER MODIFICATION ON AERODYNAMIC PERFORMANCE OF Y-SHAPED TALL BUILDING: A COMBINED WIND TUNNEL AND CFD SIMULATION STUDY

**A Thesis Submitted In Partial Fulfillment of the
Requirements for the Degree of**

MASTER OF TECHNOLOGY

in

STRUCTURAL ENGINEERING

by

SHREYANSH SINGH

(2K24/STE/07)

Under the Supervision of

Dr. RITU RAJ

Assistant Professor

Civil Engineering



Department of Civil Engineering

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daultpur, Main Bawana Road, Delhi-110042. India

May,2026



DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daultpur, Main Bawana Road, Delhi-42

CANDIDATE'S DECLARATION

I hereby certify that the work which is being presented in the project report entitled "**COMPARATIVE ANALYSIS OF CUT-CORNER MODIFICATION ON AERODYNAMIC PERFORMANCE OF Y-SHAPED TALL BUILDING: A COMBINED WIND TUNNEL AND CFD SIMULATION STUDY**" in partial fulfillment of the requirements for the award of the degree of **Master of Technology in Structural Engineering** and submitted in the Department of Civil Engineering of the Delhi Technological University, Delhi is an authentic record of my own work carried out during the third semester under the supervision of **Dr. Ritu Raj**, Assistant Professor, Department of Civil Engineering, Delhi Technological University, Delhi, India. The matter presented in this report has not been submitted by me for the award of any other degree or to any other Institution.

A handwritten signature in black ink, reading "Shreyansh Singh". The signature is fluid and cursive, with the first letters of the first and last names being capitalized and prominent.

Candidate's Signature



DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Shahbad Daultpur, Main Bawana Road, Delhi-42

CERTIFICATE BY THE SUPERVISOR

Certified that **Shreyansh Singh** (2k24/STE/07) has carried out their search work presented in this thesis entitled **“Comparative Analysis of Cut-Corner Modification on Aerodynamic Performance of Y-Shaped Tall Building: A Combined Wind Tunnel and CFD Simulation Study”** for award of **Master of Technology** from Department of Civil Engineering, Delhi Technological University, Delhi, under my supervision. The thesis embodies results of original work, and study is carried out by the student himself and the contents of the thesis do not form the basis for the award of any other degree to the candidate or to anybody else from this or any other University.

A handwritten signature in blue ink, appearing to read "Ritu Raj".

Signature

Dr. Ritu Raj
Assistant Professor
Department of Civil Engineering
Delhi Technological University

ABSTRACT

The ongoing vertical growth in urban areas has made wind-induced lateral loads as the determining factors for structural design of tall buildings. In order to overcome negative effects, aerodynamic shape optimization is increasingly being adopted in architecture and structural engineering fields. The tri-axially symmetric Y-shaped building form provides great strength and stability, but the extreme ends and re-entrant corners create abrupt flow separation and wind trapping, respectively. This research focuses on analyzing the performance of Y-shaped buildings that have undergone alteration in terms of having cut-corner on their limbs.

A hybrid approach was used to investigate the surface pressure distribution and flow field characteristics of the conventional sharp-edged Y-shape form versus the cut corner Y-shape form. Experimental studies were performed in an open circuit Boundary Layer Wind Tunnel, employing 1:150 scale rigid Perspex models subjected to an atmospheric boundary layer of Terrain Category II. The experimental results were then used to validate the three-dimensional numerical modeling using ANSYS CFX by solving steady-state RANS equations with k-epsilon turbulence model. In order to explore the entire symmetrical effects, grazing, and asymmetrical vortex shedding, study was performed for varying wind incidence angles ranging between 0^0 and 180^0 at 30^0 intervals.

Comparison of data proves that the introduction of cut-corners on the Y-shaped buildings limbs has drastically altered the behavior of the shear layer and wake flow. In particular, for critical trapping orientations (60^0 and 180^0), the recessed design creates an aerodynamic buffer, facilitating the escape of air in the inner spaces, thus lowering maximum positive pressure by about 10% to 15%, and narrowing the turbulent wake flow. This study also highlights an important aerodynamic compromise when considering highly asymmetric wind incidence (30^0 and 150^0), at these orientations, the modified design induces faster shear layer detachment around the inner limbs, thus creating high localized negative pressures (suction) on the subsequent surfaces. Finally, the results show that while recessed corners effectively decrease base shear stress and provide better stabilization of the primary structure, they require local reinforcement of facade cladding in the re-entrant regions.

Keywords: Shape optimization of buildings, Y-shaped high-rise building, Cut corner modification, Boundary Layer Wind Tunnel, Computational Fluid Dynamics (CFD), Pressure coefficient, Re-entrant corner.

ACKNOWLEDGEMENT

I would like to express my great pleasure and acknowledgement to those people who have supported me and extended their help to complete this project. The completion of this major project was possible with the support of several people.

First of all, I would like to express my deepest sense of gratitude, special thanks, and appreciation to my supervisor **Dr. Ritu Raj**, Assistant Professor, Department of Civil Engineering, DTU, for his kind support. His expert guidance, valuable suggestions, and unwavering cooperation were instrumental in defining the direction of this research. His motivation has been a constant source of strength throughout this study. I would like to thank **Kunwar Navodit Srivastava**, PhD Scholar at DTU for his continuous encouragement, practical guidance, and invaluable assistance throughout every stage of this project.

I truly appreciate the assistance received from the Department Head and faculty members of the **Prof. Awadhesh Kumar**, Department of Civil Engineering, Delhi Technological University. I would also like to express my appreciation to the staff in the Wind Engineering Laboratory for their assistance during the course of the experimental and simulation work.

Finally, I am thankful to my parents and friends for their moral and emotional support throughout this journey.



SHREYANSH SINGH
2K24/STE/07

TABLE OF CONTENTS

CANDIDATE’S DECLARATION	ii
CERTIFICATE BY THE SUPERVISOR	iii
ABSTRACT	iv
ACKNOWLEDGEMENT	v
LIST OF FIGURES	viii
LIST OF TABLES	ix
LIST OF ABBREVIATION	x
CHAPTER 1: INTRODUCTION	1
1.1 General	1
1.2 Wind Loads	2
1.3 Computational Fluid Dynamics (CFD)	2
1.4 Wind Tunnel Testing	3
1.5 The Physics of Wind-Structure Interaction	4
1.6 Aerodynamic Shape Optimization of Y-Plan Configuration	5
1.7 Importance of Corner Modification	6
1.8 Problem Statement	6
1.9 Need of Current Study	6
1.10 Objective of the study	7
1.11 Organization of Thesis	7
CHAPTER 2: LITERATURE REVIEW	8
2.1 General	8
2.2 Codal Provisions	8
2.3 Review of Previous Research	9
2.4 Research Gaps	13
CHAPTER 3: EXPERIMENTAL METHODOLOGY	14
3.1 General	14
3.2 Boundary Layer Wind Tunnel Testing Facility	14
3.3 Atmospheric Boundary Layer Simulation	16
3.4 Model Scaling and Fabrication	16
3.5 Instrumentation and Pressure Measurement Apparatus	19

3.6 Experimental Procedure	21
3.7 Data minimization and characterization	22
CHAPTER 4: NUMERICAL MODELLING	23
4.1 General	23
4.2 Geometric Modelling	23
4.3 Governing Equations	25
4.4 Turbulence Modelling	26
4.5 Computational Domain Design	26
4.6 Discretization and Meshing	27
4.7 Boundary Conditions and Inflow Physics	30
4.8 Solver settings and Convergence criteria	31
4.9 Iterations and Solution	32
4.10 Velocity and Turbulence Intensity Profile	33
CHAPTER 5: RESULTS AND DISCUSSION	34
5.1 General	34
5.2 Validation of Results	34
5.3 Face-wise pressure coefficients (C_p) Analysis by Wind Incidence Angle	35
5.4 Velocity Streamlines	56
5.5 Aerodynamic Mitigation Summary	61
CHAPTER 6: CONCLUSION AND FUTURE SCOPE	62
6.1 Research Summary	62
6.2 Principal Findings	62
6.3 Engineering and Design Implications	63
6.4 Recommendations for Future Research	63
REFERENCES	65
LIST OF RESEARCH PAPERS	69
PLAGIRISM REPORT	71

LIST OF FIGURES

Figure 1.1: Burj Khalifa and Jeddah Tower.....	5
Figure 3.1: Wind Engineering Laboratory at Delhi Technological University.....	14
Figure 3.2: Wind Tunnel at the Wind Engineering Laboratory, DTU.....	15
Figure 3.3: Wind Tunnel Facility Plan View.....	15
Figure 3.4: Flow Roughening Devices placed to simulate Terrain Category-II.....	16
Figure 3.5: Plan view of Standard and Cut-corner Y-shape.....	17
Figure 3.6: Fabricated models of Standard and Cut-corner Y-shape	18
Figure 3.7: Elevation view of a limb of Standard Y-shape model.....	19
Figure 3.8: Elevation view of a limb of Cut-corner Y-shape model.....	20
Figure 3.9: Testo-400 device and PVC tubes below the turn table.....	21
Figure 3.10: Wind angle of attack on Standard and Cut-corner Y-shape models.....	21
Figure 4.1: ANSYS Workbench Home with fluid flow (CFX).....	24
Figure 4.2: Design Modeler for geometric modelling.....	24
Figure 4.3: Standard Y- model and Cut-corner Y-model.....	25
Figure 4.4: Computational domain- Isometric and Top view.....	27
Figure 4.5: Tetrahedra elements used in meshing.....	27
Figure 4.7: Computational Domain Mesh.....	28
Figure 4.8: Fine refined mesh around the building model.....	28
Figure 4.9: Inflation layers around the building models.....	29
Figure 4.10: Meshing with of both the models.....	29
Figure 4.11: Expression window of CFX Solver.....	30
Figure 4.12: Inlet and Outlet Boundary Conditions.....	30
Figure 4.13: Solver control tab in CFX Solver.....	31
Figure 4.14: Iteration tab in solutions section of CFX Solver.....	32
Figure 4.15: Run completion tab.....	32
Figure 4.16: CFX Solver after iterations.....	32
Figure 4.17: Result section of ANSYS CFX	33
Figure 5.1: Plan and fabricated model of square structure.....	34
Figure 5.2: Pressure contours of Standard Y-model at 0° -wind incidence.....	38
Figure 5.3: Pressure contours of Cut-Corner Y-model at 0° -wind incidence.....	40
Figure 5.4 Pressure contours of Standard Y-model at 30° -wind incidence angle.....	41
Figure 5.5: Pressure contours of Cut-Corner Y-model at 30° -wind incidence.....	42
Figure 5.6: Pressure contours of Standard Y-model at 60° -wind incidence.....	43
Figure 5.7: Pressure contours of Cut-Corner Y-model at 60° -wind incidence.....	45
Figure 5.8: Pressure contours of Standard Y-model at 90° -wind incidence.....	46
Figure 5.9: Pressure contours of Cut-Corner Y-model at 90° -wind incidence.....	48
Figure 5.10: Pressure contours of Standard Y-model at 120° -wind incidence.....	49
Figure 5.11: Pressure contours of Cut-Corner Y-model at 120° -wind incidence.....	50
Figure 5.12: Pressure contours of Standard Y-model at 150° -wind incidence.....	51
Figure 5.13: Pressure contours of Cut-Corner Y-model at 150° -wind incidence.....	53
Figure 5.14: Pressure contours of Standard Y-model at 180° -wind incidence.....	54
Figure 5.15: Pressure contours of Cut-Corner Y-model at 180° -wind incidence.....	56

LIST OF TABLES

Table 5.1: Comparison of mean pressure coefficients.....	34
Table 5.2: Pressure Coefficient (C_p) values for Standard Y (WIND TUNNEL)	35
Table 5.3: Pressure Coefficient (C_p) values for Standard Y (CFD Simulation)	36
Table 5.4: Pressure Coefficient (C_p) values for Cut-corner Y (WIND TUNNEL) ...	36
Table 5.5: Pressure Coefficient (C_p) values for Cut-corner Y (CFD Simulation)	37

LIST OF ABBREVIATION

ρ = air density
 μ = dynamic viscosity
 μ_{eff} = effective viscosity
 ε = Epsilon
 α = Power law exponent
 S_M = momentum sources
 F = body forces
 C_p = Pressure Coefficient
 C_f = Force Coefficient
 V_b = Basic wind speed
 V_d = Design Wind Velocity
 C_{pe} = External Pressure Coefficient
 C_{pi} = Internal Pressure Coefficient
 K_1 = Risk factor
 K_2 = Terrain factor,
 K_3 = Topography factor
 K_4 - Importance factor
CFD = Computational Fluid Dynamics
WT = Wind Tunnel

CHAPTER 1

INTRODUCTION

1.1 General

Rapid urbanization and insufficient availability of development land in metro cities have led to transformation in architectural and structural features these areas. To cater the growing populations demand, a sustainable solution is vertical expansion of cities. Advancement in quality of high strength materials like high strength concrete and high-performance steel along with advanced computational analysis software's have enabled structural engineers to design high rise structures with complex geometries more economical and structurally resilient. Earlier, primary concern in design of such structures was identifying best possible load path to safely transfer dead and live loads to the sub-structure [1-3]. However, with increase in height and building slenderness, lateral loads like wind and seismic loads becomes the governing criteria in design of structure. Seismic forces are highly affected by building mass while wind load generally depends on building's plan and elevation, aerodynamic interaction of wind flow with structure. In case of high slender buildings, overturning moments and lateral drift due to wind loads surpasses seismic loading, hence wind loading controls the lateral resistance system of shear walls, core walls, etc. [4-5].

Building shape plays an important role in modifying wind flow interaction with structure, thus affecting wind load governing design criteria for high-rise structure. In this project, Y-shape plan which is highly popular is investigated for mitigation off wind effects by corner modification [6]. Aerodynamic optimization of structures through shape and corner modification, geometric changes is now a well-used practice to mitigate adverse wind conditions. These building modifications reduce pressure on claddings, overturning moments, torsion, base shear and other responses. Total wind force acting on a structure is net value of windward and leeward pressures [7]. To calculate wind loads following parameters need to completely evaluated:

- Building Geometry- Aspect Ratio is used to determine slenderness of the structure thus evaluating if the building will experience greater sway and vortices. Building's plan and shape determine the time of flow separation and wake width. Openings and façade in building affect the pressure and load distribution across the building. Type of roof of structure determines the effect of uplift forces on structure [21].
- Wind Flow Features- Turbulence intensity and gust factor determine the characteristics of wind flow. Boundary layer profile gives the variation of wind effects with increase in height. Terrain category significantly alters wind behavior rough surface (urban areas) act as friction does reducing wind speed while smooth surfaces (open, flat areas) allow wind to flow with faster speed [8].
- Wind Angle of Attack- Pressure distribution, sway, torsion, vortex shedding is affected by the wind incidence angle. A head-on wind impact generates high positive pressure on front face while oblique angles generate high suction forces. At 0° and 90° angles of attack vortex formation is highest while oblique angles tend to generate torsion.

1.2 Wind Loads

Wind load acting on a structure is evaluated for individual elements like a side or wall and for whole building together [35]. Two methods are specified in IS-875 (Part-3) to calculate wind load is (a) Force Coefficient Method and (b) Pressure Coefficient Method. Design wind load is evaluated from following equation:

Wind Force = Pressure x Area

In case of Pressure Coefficient Method,

Pressure = Wind Pressure normal to Area

$$= 0.5 \times C_p \times \rho \times V_d^2$$

ρ = air density

C_p = Coefficient of Pressure

V_d = Design Wind Velocity

For structures with openings, coefficient of pressure (C_p) is,

$$C_p = C_{pe} - C_{pi}$$

Here,

C_{pe} = External Coefficient of Pressure

C_{pi} = Internal Coefficient of Pressure

$$\text{Hence, Pressure} = 0.6 \times (C_{pe} - C_{pi}) \times V_d^2$$

Also, Design Wind Speed (V_d) is

$$V_d = V_b \times K_1 \times K_2 \times K_3 \times K_4$$

Here, V_b is basic wind speed, K_1 is risk factor, K_2 is terrain factor, K_3 is topography factor and K_4 is importance factor for cyclonic region.

In case of Force Coefficient Method, design wind force is calculated as

$$\text{Wind Force} = \text{Area} \times 0.5 \times C_f \times \rho \times V_d^2$$

Here, C_f = force coefficient

1.3 Computational Fluid Dynamics (CFD)

CFD is the science that utilizes computers to simulate flow of liquid and gases through a domain based on governing equations. It can predict fluid flowability around a structure and serves as a tool to reduce dependency on wind tunnel testing. This assists in observing the wind pattern and pressure changes on the building. It provides velocity streamlines to accurately differentiate flow behavior for various wind incidence angles. Steps involved in CFD Analysis:

- PRE-PROCESSING
 - Geometric modelling
 - Domain selection and Meshing
 - Establishing Boundary Conditions
- CFX SOLVER
 - Numerical modelling

- Establishing iterative convergence
- POST-PROCESSING
 - Extracting desired results (Pressure contours, velocity streamlines, etc.)
 - Analyzing the aerodynamic response

Advantages

- Has the capability of analyzing the whole flow field.
- Easier to conduct comparative investigations (for buildings with different shapes).
- High-quality visualization of flow, vortices formed and separation zones.
- Less expensive and less time consuming than wind tunnel tests.

Disadvantages

- Even though CFD techniques are very effective, they cannot completely replace wind tunnel tests
- Numerical solution can involve modelling errors.
- Turbulence modelling doesn't take into account complicated flow separations.
- Boundary layer simulation requires proper calibration.

1.4 Wind Tunnel Testing

Wind tunnel testing is among the most accurate ways of obtaining wind loadings on tall buildings in case of complex geometries and where standard code suggestions are inadequate. It provides experimental results regarding wind pressures, forces, and flow behavior [8].

For instance, a wind tunnel test is done to produce the atmospheric boundary layer flow over the model with varied wind speeds and direction. The results of the wind tunnel tests are recorded from varied angles of incidence of winds. The wind tunnel is employed in measuring surface pressures, global forces, and flow behavior around the Y shaped tall building model, both with and without the cut corner design modification.

Objective of Wind Tunnel Testing:

- Establishing pressure coefficients (C_p) on all exterior surfaces.
- Measuring global force and moment coefficients (drag, lift, torsion, overturning moments).
- Identifying critical wind directions and aerodynamic effects such as vortex shedding, flow separation, and wake development.
- Obtaining experimental results that can be used to validate Computational Fluid Dynamics (CFD) modeling.

Wind tunnel tests are essential in composite or multi-legged structures, including Y plan structures, where standard code do not provide adequate information.

Boundary Layer Wind Tunnel Setup

Boundary Layer Wind Tunnel (BLWT) is employed since it is able to replicate the atmosphere boundary layer (ABL) over scaled model terrains [25]. It is essential in simulating:

- Mean velocity profile
- Turbulence intensity

This will be done via spire elements at inlet of tunnel and flow roughness elements before the test section. Profile will be adjusted depending on the type of terrain to be represented-urban/suburban conditions.

Findings from wind tunnel tests

- Identification of critical areas where suction forces occur.
- The observation that cut corner helps reduce flow separation and reduces peak suction force.
- Determination of maximum wind sensitive orientation by considering incidence angles.
- Generation of experimental C_p values for validation of the CFD simulation adopted in this study.

Advantages

- Physically measured pressure data.
- Validation of CFD simulations and modeling.
- Capacity to assess wind effects in non-regular building geometry where standard building codes cannot be applied.
- Data obtained suitable for structural design and cladding purposes.

1.5 The Physics of Wind-Structure Interaction

Wind Engineering is a specialized complex branch of civil engineering that brings together fluid dynamics, meteorology, and structural mechanics. Wind load is different from other loadings such as gravity loads because it is dynamic in nature and depends on the topology of surroundings. As wind flows above the surface of the earth, it produces friction due to roughness (trees, buildings, and terrain). This leads to creation of an Atmospheric Boundary Layer (ABL) [9]. In this boundary layer, the speed of wind increases logarithmically while its turbulence intensity reduces. Once the turbulent wind hits a skyscraper, the structure behaves like a bluff body causing a great disturbance in the wind flow. These disturbances result in the following aerodynamic effects:

- **Along-Wind Behavior (Drag):** The first effect is the stagnation point where the wind directly impacts the windward face, thus causing very high positive pressures. After separation at the sharp edges of the structure, the wind flow causes very strong negative pressures (suction) on the leeward face. Pressure difference between the two faces causes drag [4].

- Across-Wind Response (Lift & Vortex Shedding): Vortex generation occurs behind buildings due to separation of air flow near the edge of the building. The vortices formed in this manner are called Von Kármán vortex street, and if the frequency of this vortex shedding phenomenon is close to natural frequency of the building, resonance occurs [4].
- Torsional Response: Oblique or asymmetrical wind incidence causes asymmetric pressure distribution and hence a twisting moment around the vertical axis. Torsion damages the structural core and perimeter columns [13].

1.6 Aerodynamic Shape Optimization of Y-Plan Configuration

For improvement of aerodynamic behavior of tall structures, two major techniques are used: structural damping (increase of mass or mechanical dampers) and aerodynamic shape optimization. Because the implementation of mechanical dampers involves considerable costs and space, aerodynamic optimization, which means altering the design of the structure to either confuse or streamline the airflow, is considered the most cost-effective option. Lately, architects and engineers are moving away from traditional square and rectangular shapes and opting for irregular ones, which are aesthetically more pleasing as well as aerodynamically efficient. Tri-axially symmetrical “Y” shape has become a hallmark of modern skyscraper planning [29]. The most prominent examples of the Y-shape configuration include the Burj Khalifa (Dubai, 828m) and the Jeddah Tower (Saudi Arabia, planned for around 1000m).



Fig.1.1: Burj Khalifa and Jeddah Tower

The Y-shape configuration provides great advantages from the structural and aerodynamic point of view. Three limbs of the structure provide stability by supporting the central core against overturning moments and torsion. When air flows through the

structure, it hits different flow angles as well as changes in geometry as a result the shedding takes place at various frequencies [21].

1.7 Importance of Corner Modification

The tip of each limb in the Y-shape has sharp edge corners but from fluid dynamics perspective, the presence of sharp 90° or even acute corners will result in instantaneous flow separation. This instant flow separation causes high suction (which affects the choice of cladding material for the facade) and increases the width of turbulent wake, hence increasing the drag coefficient and to mitigate this, local aerodynamic modifications to the building corners are employed [21]. Such modifications include-chamfering: angling the sharp corner, rounding: filleting the corners to allow the wind to hug the building for longer duration and cut-corners (or recessing): taking out a stepped rectangular or square portion of the corner. Instead of merely streamlining the flow like in a rounded corner, cut corners create a local stagnation region where small, high-frequency eddies get trapped. These eddies act like an aerodynamic cushion that forces the shear layer to separate much farther from the surface, hence reducing the width of the wake [26].

1.8 Problem Statement

Many studies have been conducted regarding the aerodynamic behavior of regular square and rectangular tall buildings, but there are few studies that investigate the tri-axially symmetrical Y-shaped buildings. In all existing studies conducted on the Y-plan buildings, most of the research has been conducted using either the regular sharp-edge shape of the Y-building or the rounded and chamfered variations of the building. The critical problem area that needs to be researched is the quantitative aerodynamic behavior of Y-shaped tall buildings with cut-corner modifications. Standard Y-shaped building shows great aerodynamic weakness for asymmetric wind incidences especially when the angle of attack is at 30° and 60° . Wind flow bypasses the leading limbs of the building and flows through the inner re-entrant corner thereby generating significant differential pressures. Oblique wind flow causes high torsional moments. Cut-corner modification postpones flow separation and minimizes peak suction force and base shear force acting on the building; however, the effects of modified Y-shaped model on the re-entrant corner pressures and building torsional forces is unknown. A standalone numerical investigation is inadequate owing to the sophisticated turbulence modelling; hence, a combined method that uses both CFD and physical BLWT investigations is necessary in order to thoroughly assess this particular fluid-structure interaction.

1.9 Need of Current Study

Shortcomings in Standard Design Codes: Although the Y shape provides great structural advantages, it suffers from aerodynamic weaknesses that cannot be taken

care of by the existing standards (IS: 875 Part-3) due to insufficient data available on determination of pressure coefficients for irregular shapes.

Aim of Present Study: The proposed thesis aims to fill those academic gaps through an extensive hybrid approach which will quantify the aerodynamic pros and cons of cut corner Y shape structure. The study involves direct correlation between wind tunnel results and ANSYS CFX flow field visualization for wind incidence angles ranging between 0° to 180° .

1.10 Objectives of the study

1. To perform experimental evaluation and determining mean pressure coefficient (C_p) values for all faces of the Standard Y-plan and Cut-corner Y-plan models by physical tests in Open-Circuit Boundary Layer Wind Tunnel.
2. To corroborate the three-dimensional computational framework in ANSYS CFX based on k- ϵ turbulence model against the experimental findings to ensure accurate predictions of building aerodynamics.
3. To study the vulnerability of both building models against wind loads for different wind incidence angles from 0° to 180° with a 30° interval and determine critical orientations.
4. To analyze the flow field characteristics using numerical visualizations (contours and streamlines), in particular the locations of shear layer separation, wind-trapping in re-entrant cavity and downstream wake widths.
5. To quantify the extent of aerodynamic mitigation achieved by the cut-corner modification and the percentage decrease of stagnation pressure, while identifying severe suction regions.

1.11 Organization of Thesis

The thesis is structured as follows: **Chapter 1** introduces wind loading concepts and the explains need of investigating Y-shaped buildings using wind tunnel and CFD methods. **Chapter 2** reviews existing literature on tall building aerodynamics, shape optimization, and testing methods. **Chapter 3** details the experimental setup, including the wind tunnel facility, 1:150 scaled Perspex model fabrication, and data collection procedures. **Chapter 4** outlines the ANSYS CFX computational approach, domain setup, meshing, and the k-epsilon turbulence model. **Chapter 5** validates numerical results against experimental data, analyzing pressure coefficients and flow characteristics across a 0° to 180° wind incidence range. Finally, **Chapter 6** summarizes the quantitative findings, discusses the engineering impact of recessed corners, and suggests future research ideas.

CHAPTER 2

LITERATURE REVIEW

2.1 General

In the last decades of the 20th century, several scholars have studied the effect of wind on high-rise structures, starting from a basic assumption that wind pressure exhibits a static behavior. Wind engineering research was conducted using a variety of wind tunnel tests for estimating mean pressure distribution around buildings. The development of measurement technologies and computer modeling techniques resulted in the understanding of wind load as a highly dynamic, oriented, and geometrically complex phenomenon.

It should be noted that the evolution of tall buildings is a history that goes back centuries. Numerous advancements in architectural and structural fields allowed the erection of ever-taller and more sophisticated tall buildings. In conjunction with urbanization and lack of available space, a great demand for tall buildings of unusual forms emerged worldwide.

As a result, modern tall buildings are getting higher and slimmer, and the study of their cross-section geometries is crucial nowadays. Longitudinal and transverse responses, vortex shedding, interferences, and pressure concentrated on the corners are among the aspects affecting comfort level and safety of the structure. Seismic loads may still be of significance in some cases, but typically, wind loads play dominant role because of fundamental time periods of structures.

Y-shaped structure is one of the evolving structures that are increasingly being adopted in modern architectural design due to their functional and aesthetic advantages. The three-limbed plan, sharp edges, and re-entrant corners of this kind of structure contributed to a pronounced flow separation and localized suction, hence necessitating comprehensive wind tunnel tests and CFD analysis. Various aerodynamic treatments, including chamfering, rounding, tapering, and cut-back, have received considerable interest lately.

2.2 Codal Provisions

- **IS-875 Part 3-2015:** Primary wind design code for structures in India. It divides India into various wind zones having distinct basic wind speed for each zone. Design wind speed is evaluated using four factors- Risk factor, Terrain factor, Topography factor and Cyclonic Importance factor. External and Internal pressure coefficients are calculated to get net pressure on the structure. Special provisions are added for chimneys, open structures, etc. [31].
- **ASCE 7:** This code works on the phenomena of determining the minimum load requirement for structures due to wind. Probabilistic hazard analysis is preferred to determine wind speed using different wind maps categorizing ordinary buildings, essential facilities and hurricane-prone area depending on the risk level.

Separate methods are used for low and high-rise buildings; wind tunnel testing is recognized as important method in wind analysis of irregular shape buildings [32].

- **EN 1991-1-4:** This code is used for wind load calculation throughout Europe with local parameters for each country. Wind action is based on basic wind speed, terrain category, turbulence. It emphasizes on wind response, turbulence intensity, atmospheric boundary layer. Pressure coefficients are evaluated for walls, roofs and other structures [33].
- **AS/NZS 1170.2:** This code is used for evaluating wind loads in Australia and New Zealand regions which frequently experience cyclones and storms. Wind speed is categorized in cyclonic, non-cyclonic and thunderstorm regions. There is a separate section for serviceability conditions and ultimate limit state design. This code has specific provisions for houses, industries, tall structures and open framed structures [34].

2.3 Review of Previous Research

Singh, N., & Verma, A. (2025) - This article presents a detailed review of different methods employed to reduce wind loads in tall buildings. The paper explains the development of aerodynamic design modifications ranging from corner cutting to tapering and discusses the use of CFD modeling in predicting flow separation at early design phases. In conclusion, the study finds that CFD together with wind tunnel tests represents the most reliable methodology [1].

Sharma, N., Yadav, H., & Roy, A. K. (2024) - In this paper, dynamic analysis of tall buildings designed in a Y-shape is conducted with respect to their interference effects on each other. As a result, the authors find out that the presence of other structures near the tall building influences its vibration frequency and increases torsional oscillation. Hence, analysis of isolated tall buildings fails to show real peak dynamic responses [2].

A.H.H. Al-Masoodi, Y.M. Abbas, F. Alkhatib, Mohamed ElGawady, et al. (2023) - In this paper, the focus is made on evaluating the effect that corner modifications have on an octagonal-plan-shaped model. Specifically, the study analyzes the effect produced by changing sharp corners into smoother contours through a computational fluid dynamics analysis, which helps delay shear-layer separation. The optimized geometries can be recommended for use in reducing drag and across-wind dynamic forces [6].

Al-Masoodi, A. H. H., Alkhatib, F. H., Shafiq, N., & Wahab, et al. (2022) - This paper reviews the literature on the use of different aerodynamic modifications in buildings. Specifically, the review includes corner modifications, tapering, twisting, and setbacks as means to improve the aerodynamic performance of tall buildings. This paper could serve as a basic guideline for structural engineers to implement aerodynamic optimization during the conceptualization phase of buildings [7].

Gaur, N., & Raj, R. (2022) - The paper examines the aerodynamic mitigation resulting from the application of corner cut technique to square tall buildings' models. Using the combined technique of 1:100 scale boundary layer wind tunnel tests and the conventional k-epsilon CFD analysis, the work analyses the force coefficient, base moment and the power spectra at varied wind incidence angles. The author conclusively demonstrates that corner modification delays the separation of the shear layer and minimizes aerodynamic drag. In addition, the study demonstrates the high validity of the computational technique in relation to physical empirical results [8].

Goyal, P. K., Kumari, R. K., Meena, R. K., & Raj, R. et al. (2022) - Carrying out numerical analysis on CFD software, the above researchers investigated the wind loads on tall Y plan-shaped buildings. The velocity stream lines, pressure coefficients and turbulence kinetic energies at different incidence angles from 0° to 180° are studied. The results identify the asymmetry angles that produce highest torsional moments and show high aerodynamic vulnerability of the inner re-entrant corners [9].

Meena, R. K., Raj, R., & Anbu Kumar, S. (2022) - In this paper, the effect of wind-excited actions on tall buildings characterized by different types of corners (i.e. chamfered corners, rounded corners and recessed corners) has been examined. It has been determined that the rounded corners lead to the most effective reduction in along-wind drag while the recessed corners have the best performance in breaking down the vortex shedding [11].

Rajabi, E., Sadeghi, H., & Hashemi, M. R. (2022) - The wind effects on a Y-shaped plan building have been studied in this paper through the use of a hybrid technique that involves conducting wind tunnel experiments on the 1:500 scaled prototype alongside the application of CFD modeling. It is shown in the findings that the maximum structural displacement and the pressure coefficient peak values will be experienced when the wind hits at 30°. Their success is proved by the high correlation exhibited between the experimental and numerical results [13].

Srivastava, K. N., & Raj, R. (2022) - In this study, the authors examine the aerodynamic characteristics of tall buildings having composite configurations, for example, square-octagonal or circular-hexagonal configurations. According to the authors, even though design codes from around the world specify pressure coefficients for regular shapes, there is little information available regarding composite shapes. Thus, the authors emphasize the importance of carrying out specific wind load studies using computation or experiment when using these unique shapes [15].

Meena, R. K., Raj, R., & Anbu Kumar, S. (2022) - In this computational study, the authors used ANSYS CFX software to carry out an aerodynamic analysis of different high-rise building shapes at 1:200 scale models. With the help of flow visualization and pressure coefficient calculation, the authors were able to verify their computational results with international standards. It was observed that even though pressure

distributions over the windward faces are similar in all cases, geometric parameters play a very important role in wake behavior [16].

Sanyal, P., & Dalui, S. K. (2022) - In this paper, the forecast of the aerodynamic force and moment coefficients of the tri-axially symmetrical Y-plan shaped building was conducted with respect to the Artificial Neural Networks. With the aid of an abundance of data collected by applying CFD simulation across different corner cut percentages and wind incidence angles, ANN predicts fluid-structure interaction [17].

Sanyal, P., & Dalui, S. K. (2021a) - In order to investigate the effect of changing the internal angle of Y-shaped tall building, this study attempts to identify different critical wind incidence angles for asymmetrical geometric configurations. The paper makes it quite clear that by modifying the angle of limbs away from the usual 120 degrees symmetric geometry, the process of capturing the wind flow becomes quite prominent [18].

Sanyal, P., & Dalui, S. K. (2021b) - This paper examines the aerodynamic behavior in terms of different side ratios of Y-shaped tall building at constant plan areas. Using ANSYS CFX, force and moment coefficients were studied. The paper also suggests some mathematical equations to estimate these coefficients from the side ratios and validates them by comparing them with wind tunnel results [19].

Sanyal, P., & Dalui, S. K. (2020) – For the comparative analysis of aerodynamic coefficients, various types of shapes and designs have been analyzed in Y-shaped high-rise buildings possessing sharp corners along with corner geometry modifications. Variation of drag, lift and torsional coefficients within 180° wind incidence is mapped in the study and the paper highlights the findings of a conclusion drawn which suggests that even though modification of corners significantly reduces base shear, the moment may increase depending upon wind angles [21].

Raj, R., Rana, T., Anchalia, T., & Khola, U. (2020) - The aerodynamic deficiency of the high-rise building, particularly the slender and H-shaped type, is analyzed in this analysis through surface pressure distribution using ANSYS CFX. A considerable emphasis is put on studying the interference effect that arises due to placing another building near the first one, which changes the natural pattern of wind flow. This study proves that the complex cavity inside an "H" structure experiences tremendous pressure difference due to interference and needs attention [20].

Mallick, M., Kumar, A., & Patra, K. C. (2019) - In this paper, wind pressure on "C"-shaped buildings was experimentally explored using boundary layer wind tunnel testing. The scientists carried out an extensive study on the highly complex process of separation and suction areas that exist in the "C" cavity zone. This paper provides important data that can be applied when designing cladding systems for façades of "C"-shaped buildings [22].

Ahlawat, R., & Ahuja, A. K. (2015) - The objective of this research is to conduct wind tunnel tests on wind forces acting on “T” shaped tall buildings by studying their base shear force, overturning moment, and the influence of torsion effect. The building is analyzed in both isolated states as well as when it is subject to interference effects from surrounding buildings. In this research, it is seen that based on wind direction and positioning of interfering building structure, nearby buildings can be either shields or enhancers of load on the “T” shaped building [24].

Ahlawat, R., & Ahuja, A. K. (2015) - This research is carried out to investigate the wind loads on Y-shaped tall buildings through an open circuit wind tunnel test under isolated and interference cases. The study carefully measures the base shears, overturning, and torsional moments under various incidence angle of winds and spacing. It is shown that the wind direction plays a very crucial role in determining the forces on the structure, as the interference condition may act both favorably as well as unfavorably for torsion [25].

Elshaer, A., Bitsuamlak, G., & Damatty, A. El. (2015) - In this study, authors develop an innovative methodology based on a combination of ASO and CFD along with neural network technologies that aims at finding out optimal geometry of tall buildings' corners in order to reduce structural drag forces. It represents a highly sophisticated approach to aerodynamic structural form-finding that uses advanced computing methods [26].

Dalui, S. K. Ashok Kumar Ahuja, Sourav Mukherjee. (2014) - In studying the pressure distribution and interference effects on complicated geometrical shapes of buildings, this paper employs physical scale model techniques to determine the peak localized pressures. The study focuses particularly on high localized negative pressures that are experienced by buildings, and the results underscore the need to account for architectural characteristics because they affect pressure coefficients in the building code [27].

Raj, R., & Ahuja, A. K. (2013) - Wind Loads on Cross Shape Tall Buildings. This experimental study conducts a boundary layer wind tunnel test on rigid model buildings with cross-shaped plan. The rigid model buildings have been compared to other models having different sections but having equal floor area. From this experimentation, it is evident that base shear force, overturning force, and particularly twisting force are highly dependent on wind incidence angle and section shapes. [28].

Irwin, P.A. (2005) - Wind Engineering of the Burj Dubai Tower. This article describes the wind engineering and aerodynamic design optimization techniques used in the design of the Burj Khalifa Dubai. Using sophisticated aeroelastic wind tunnel testing methods, the author explains how the stepped and triaxial symmetry design of the building creates a condition where the wind is fooled into not generating organized

vortices around the building. It clearly highlights how architectural design is an effective means of reducing across-wind behavior in tall buildings [29].

2.4 Research Gaps

- There is an abundance of literature focused on optimizing shapes for traditional square and rectangular tall buildings. There is a significant deficiency when it comes to the research involving sophisticated shapes such as the Y-shape.
- Previous corner modification research mostly involves rounded or chamfered designs on rectangular shapes or general parameter changes in traditional Y-shapes. No quantitative information about the fluid dynamics and localized zones of stagnation in recessed modification on Y-shapes exists yet.
- Available geometric modifications concentrated on minimizing the total base shear force, it ignored that oblique winds tend to create pressure imbalance in the modified limbs thus, significantly increasing the torsional effect.
- The current literature focuses mostly on large-scale structural behaviors such as total drag, lift, and moment loads in order to achieve overall stability. It does not offer face-wise pressure coefficient maps required for design of cladding system.
- Existing literature depends on either CFD or Wind Tunnel Tests individually. CFD results without any validation test especially for wake flows whereas wind tunnel tests cannot visualize the underlying flow fields making an integrated approach essential.

CHAPTER 3

EXPERIMENTAL METHODOLOGY

3.1 General

Despite the highly accurate visualization offered by CFD, there is no doubt that CFD analysis is based on mathematical models, which are approximate representations of turbulent flows. Therefore, BLWT studies are still considered the best means of estimating wind pressures acting on tall buildings. Experimental modeling is necessary to obtain reliable information about fluid-structure interaction, especially flow separation, wake formation, and vortex shedding [5].

In this chapter, the experimental procedure used to test the aerodynamic performance of standard and cut-corner Y-shaped tall buildings is presented. The description includes the specifications of the wind tunnel facility, boundary layer atmosphere simulation, fabrication of the rigid scaled model, measurement instruments, and method of acquiring data on pressure distribution.

3.2 Boundary Layer Wind Tunnel Testing Facility

Experimentation for this study were performed in an advanced open circuit boundary layer wind tunnel, which is housed at Wind Engineering Laboratory of Delhi Technological University (DTU), Delhi, India. The facility is a suction type wind tunnel and is especially designed to simulate the atmospheric boundary layer along a considerable length of fetch.



Fig.3.1: Wind Engineering Laboratory at Delhi Technological University

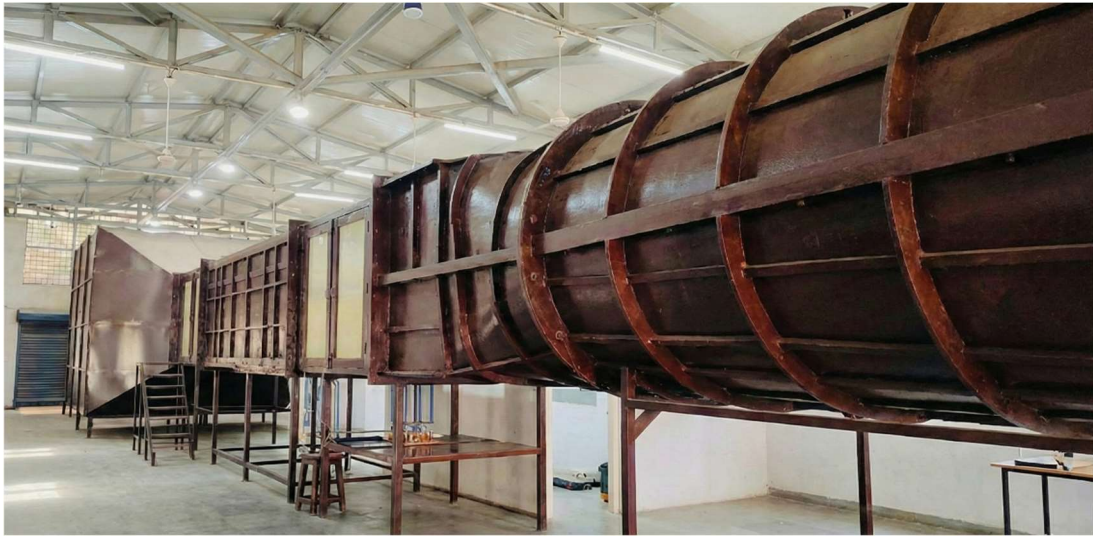


Fig.3.2: Wind Tunnel at the Wind Engineering Laboratory, DTU

The airflow is created by a powerful 60-horse power axial exhaust fan that is placed at the downstream end of the tunnel. Suction mode of operation prevents any contamination of airflow to the test section due to turbulence created by the rotating blades of the fan. The dimensions of the wind tunnel are carefully chosen for structural aerodynamic testing purposes. The test section of the wind tunnel has a cross-section of 1.6m x 1.6m, length of test section is 1.97m and a total length of wind tunnel facility is 18.2m. In order to establish a uniform and low turbulence free stream flow prior to the creation of the boundary layer, a huge honeycomb flow straightener of dimension 3.6m x 3.6m has been fitted at the inlet of wind tunnel. Further downstream is a vortex generator connected to honeycomb wind entrance through a contracting section with contracting ratio of 9:5:1 and length of this elliptical effuse is 2m. It is provided to smoothen the flow of air before it reaches the main testing section area.

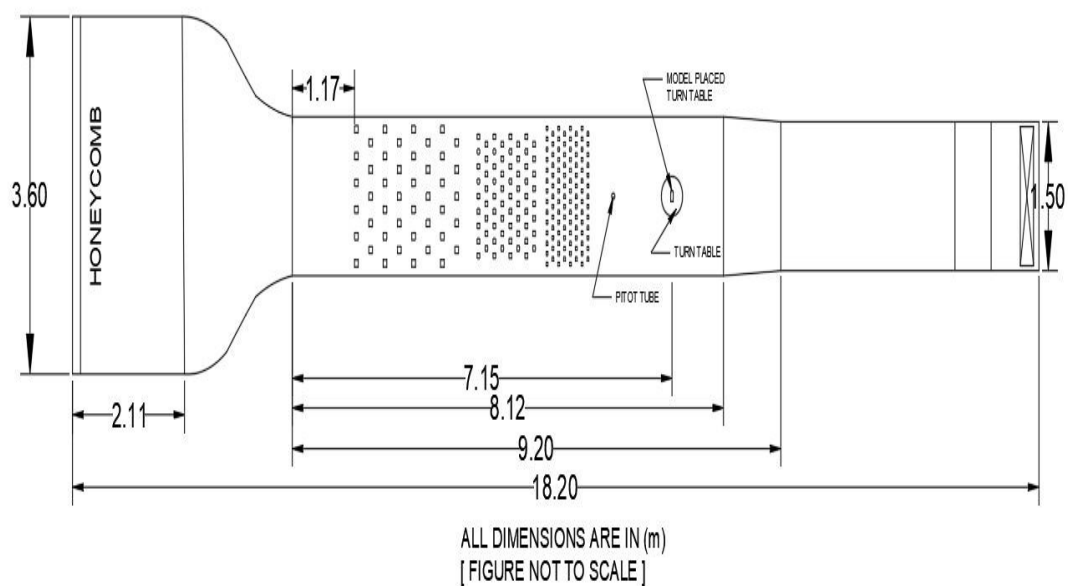


Fig.3.3: Wind Tunnel Facility Plan View

3.3 Atmospheric Boundary Layer Simulation

In natural situations, when wind flows over the surface of the earth, due to the friction experienced by the wind, its velocity reduces to nearly zero near the earth's surface and increases non-linearly with respect to height up to the gradient height and this particular zone is termed as Atmospheric Boundary Layer (ABL). In order to achieve similitude between the laboratory situation and natural phenomena, it becomes essential to generate an artificial ABL inside the wind tunnel [24].

In this investigation, the type of terrain considered was Terrain Category II, implying that it is an open terrain where the obstacles distributed evenly with a height range from 1.5m to 10m, as per Indian Standard Code IS:875 (Part-3). In order to physically create such a terrain configuration, flow-roughening devices were strategically placed in a geometrically staggered fashion over the floor of the development section of the wind tunnel. Wooden square cubes of different sizes were used for this purpose [25]. The uniformly flowing air from the honeycomb straightener after passing through the vortex generating devices passes over these roughness elements, the air stream gets tripped and develops a velocity profile and turbulence intensity profile as would exist in a Category II terrain type.

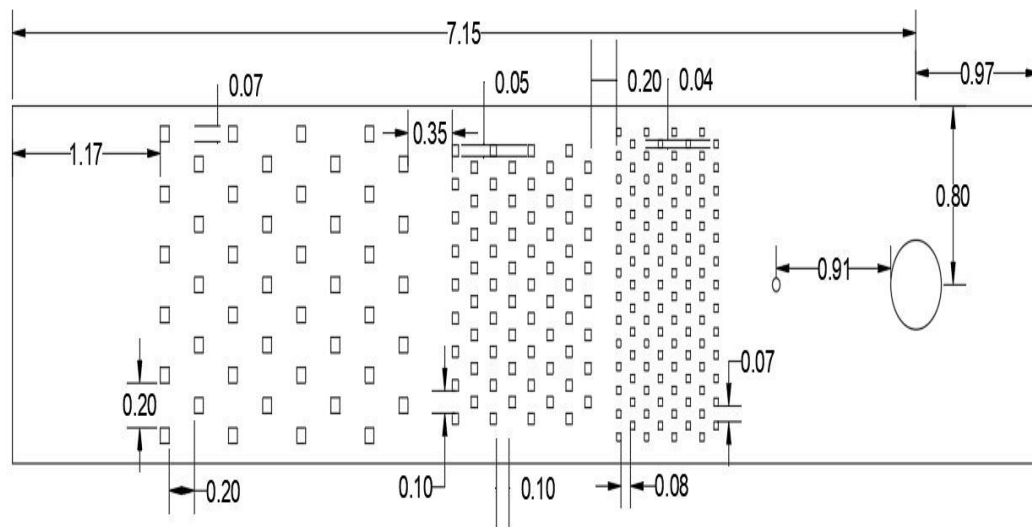


Fig.3.4: Flow Roughening Devices placed to simulate Terrain Category-II

3.4 Model Scaling and Fabrication

In order to obtain quantitative information through physical experiments, it is imperative to maintain exact geometric and kinematic similarity between the actual tall building being studied and the model inside the wind tunnel.

Geometric Scaling

The height of the actual prototype tall building considered in this study is 90 meters. Given the cross-sectional dimension of 1.6mx1.6m for the test section in the wind tunnel, a scale of 1:150 was employed. The choice of the specific scale is in order to maintain the correct ratio of the frontage area of the model and the cross-sectional area of the wind tunnel does not exceed the critical value of 5%. A low blockage ratio ensures that there is no artificial acceleration of the flow past the model, hence avoiding mathematical correction factors.

Scale: 1:150

Prototype Height: 90 meters

Model Height (H): 600 mm

Model Construction

Two different model configurations were made for the experiment:

Model S (Standard Y-shape): A traditional tri-axially symmetric Y shape with sharp 90° and 120° re-entrant angles.

Model C (Cut-Corner Y-shape): A standard Y-shaped base design with recessed (cut corner) modification at the ends of each of the three limbs.

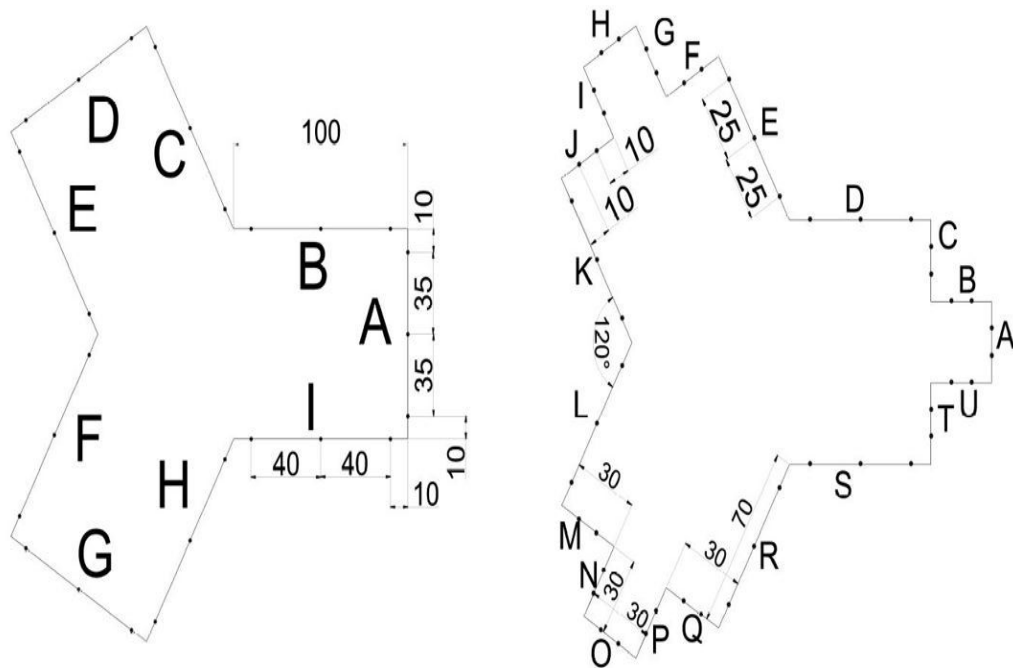


Fig.3.5: Plan view of Standard and Cut-corner Y-shape



Fig.3.6: Fabricated models of Standard and Cut-corner Y-shape

Both models were carefully built out of rigid Perspex (acrylic) sheets. There are two reasons why the Perspex sheets have been chosen. Firstly, because of their rigidity, they will guarantee that the structure does not flex when the wind load is maximal, thus eliminating the structural component of the aeroelastic effect. Secondly, Perspex sheets are clear, and so they will help to ensure that all pressure tubes inside the models are intact.

3.5 Instrumentation and Pressure Measurement Apparatus

The main parameter for determining the aerodynamic wind pressures acting on cladding and facade members is the local surface pressure. This was achieved by dividing the exterior surface of the two model buildings into certain testing faces. In the case of Model S, Faces A to I were used, whereas in Model C, due to its modified geometry, Faces A to U were required.

Pressure Tapping's

In order to determine the surface pressure, tiny circular holes called pressure tapping's were drilled on the faces of the Perspex models at seven levels along the height of model. Firstly, small steel tubes were inserted in these holes then flexible PVC tubes were inserted into the back side of the pressure tapping locations within the hollow model. The function of these tubes was to serve as the pneumatic pipe that conveys the instantaneous surface pressure at the location of the tapping hole to the bottom of the turntable and then further to the measuring device (Testo-400). To capture the variation in flow separation accurately, pressure tapping's were placed as close as possible to the ends of limbs.

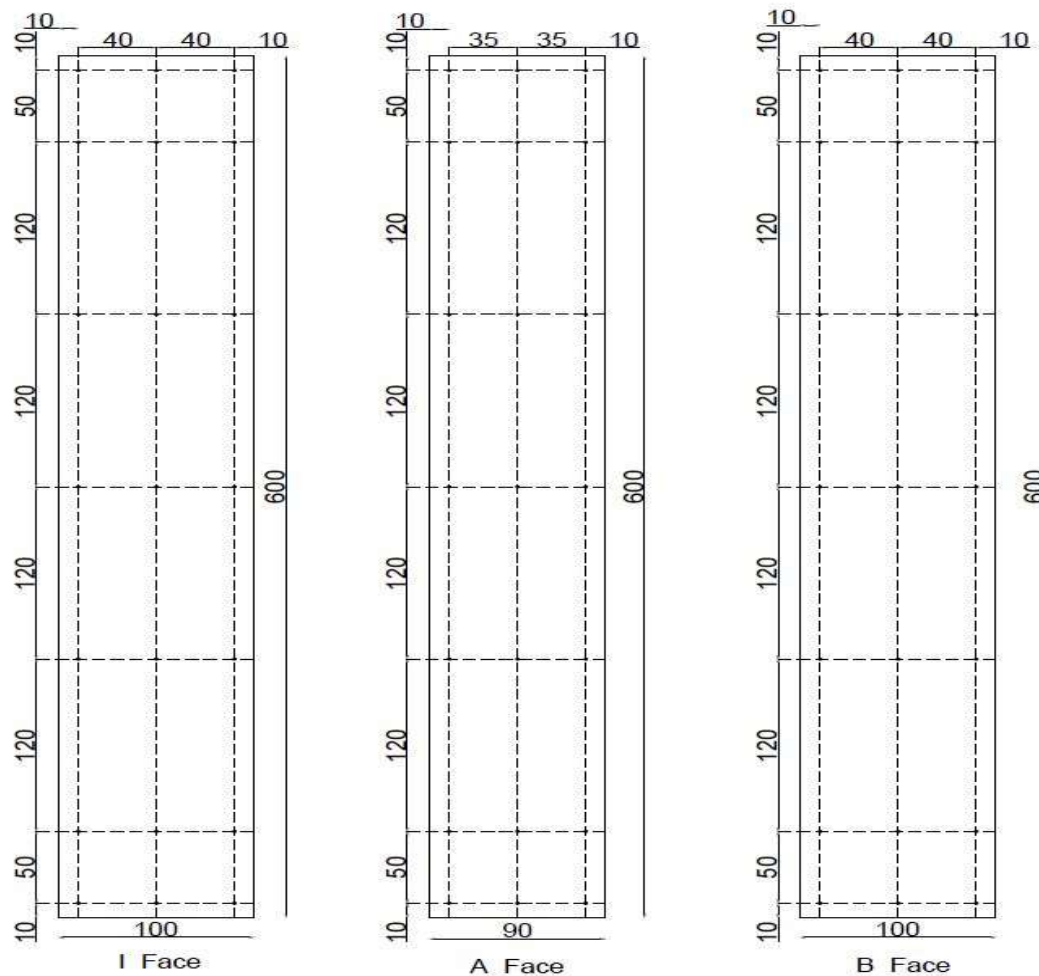


Fig.3.7: Elevation view of a limb of Standard Y-shape model

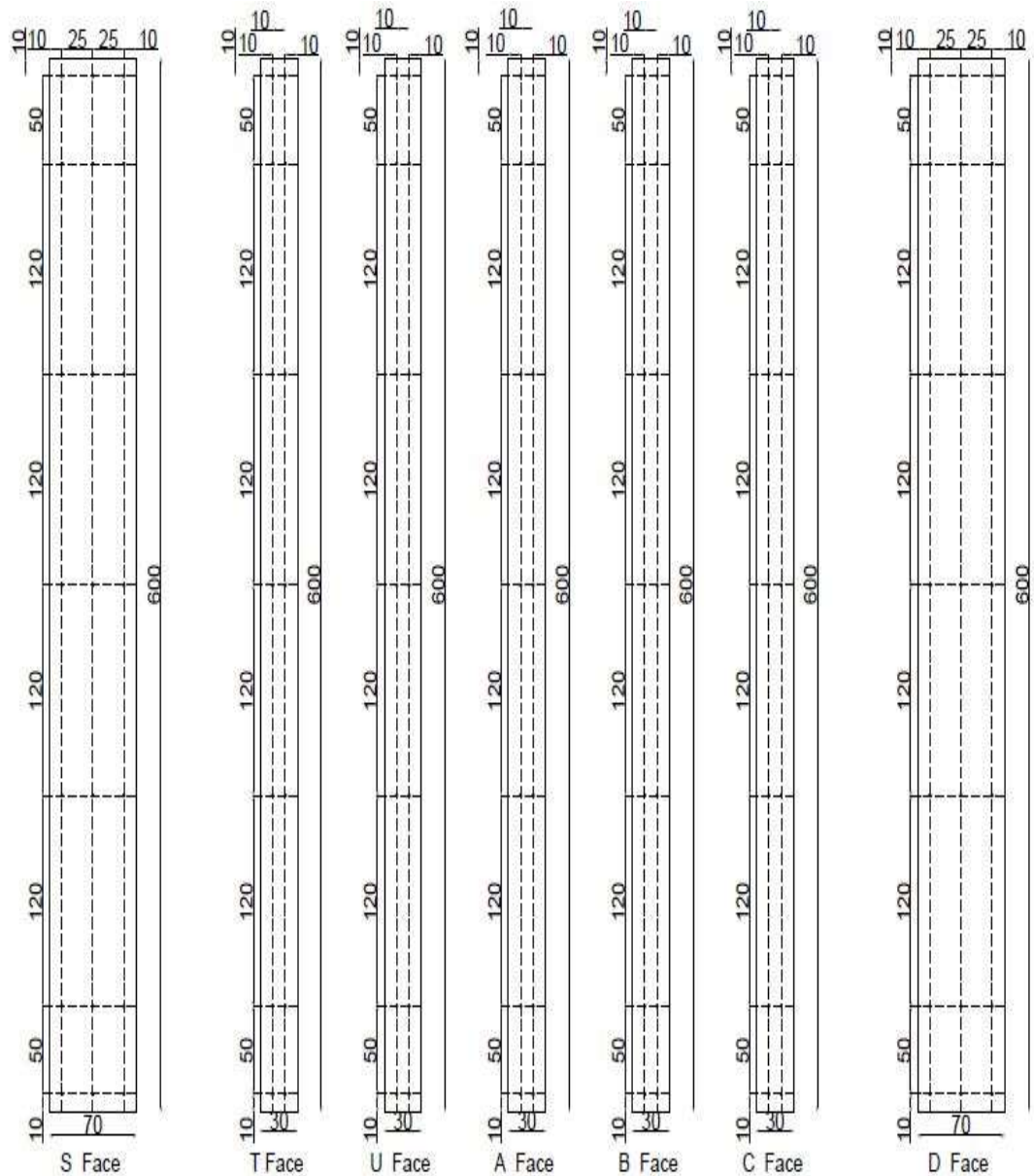


Fig.3.8: Elevation view of a limb of Cut-corner Y-shape model

In the above two figures portraying elevation view of the models, each intersection represents a pressure tapping point. All dimensions are in mm.

Data Acquisition Instrumentation

Pressure measurements were taken using the Testo 400 digital measurement and analysis instrument. The Testo 400 works as an extremely sensitive differential pressure transducer. It compares the total pressure measured at the building surface through the tapping tubes with the static pressure obtained from the other tube connected to the free stream in the wind tunnel. Pitot tube was used in coordination with Testo device to get the velocity profile and turbulence intensity profile of the wind tunnel facility for varying frequencies.

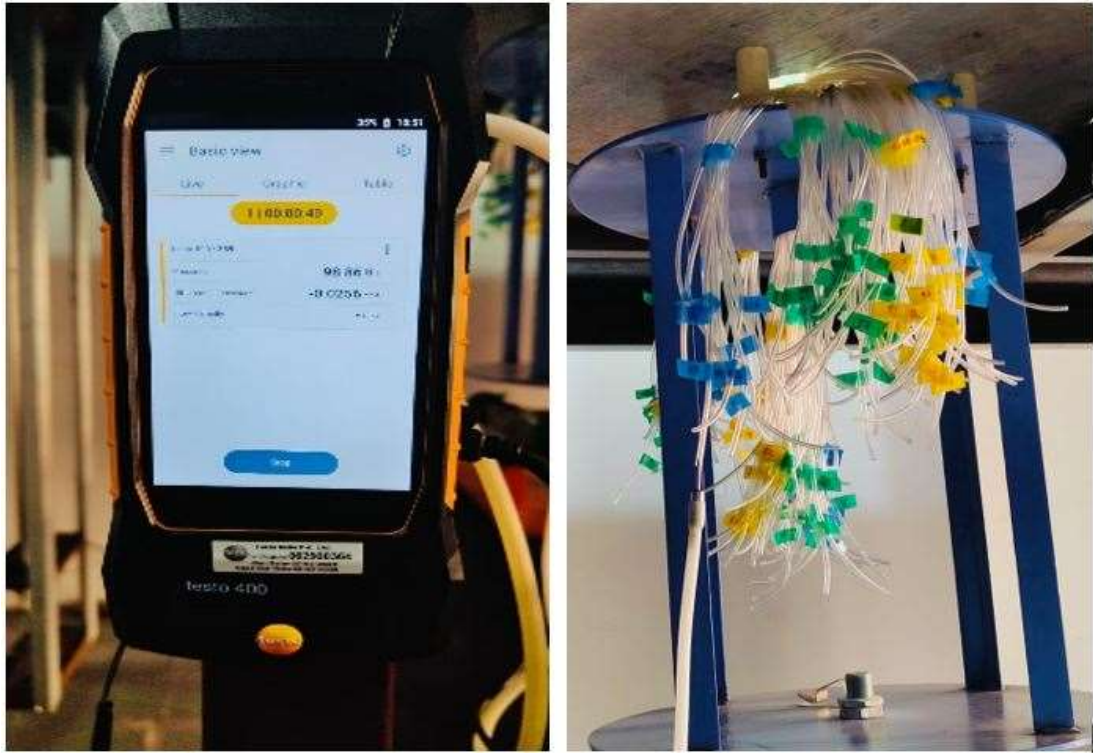


Fig.3.9: Testo-400 device and PVC tubes below the turn table

3.6 Experimental Procedure

The test procedure was designed to obtain the aerodynamic behavior of the building at a full range of Wind Incidence Angles (WIA). As the Y-shaped design is axially symmetric (each limb is at 120° difference to one another), testing angles ranging from 0° to 180° will be sufficient in capturing all the aerodynamic phenomena including impact loads, grazing, and asymmetric vortex shedding.

The fabricated models were mounted securely on the center of the rotating turntable in the test section. All the pneumatic tubing inside was connected to the Testo 400 underneath the floor of the wind tunnel. The turntable was manually rotated such that the models would be tested at angles of 0° , 30° , 60° , 90° , 120° , 150° and 180° .

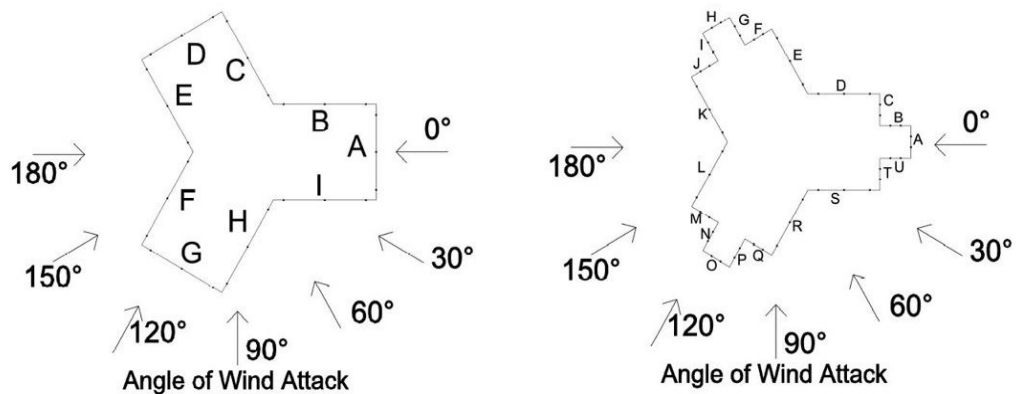


Fig.3.10: Wind angle of attack on Standard and Cut-corner Y-shape models

The wind tunnel fan was turned on and air flow was let stabilized. Once the boundary layer had stabilized, pressure values were measured. Sufficient sampling time was taken in order to filter out high frequency turbulent fluctuations in order to obtain mean pressure values.

3.7 Data minimization and characterization

The differential pressure readings evaluated from the wind tunnel test environment are quite unique to the wind speed and air density that existed in the wind tunnel test conditions. For the purpose of universal applicability of this data and to be comparable with the CFD analysis data, the readings need to be converted into dimensionless average pressure coefficients (C_p). The coefficient of pressure for each face was determined using the basic aerodynamic equation:

$$C_p(\text{mean}) = \frac{P - P_o}{\frac{1}{2}\rho U_z^2} \dots\dots\dots (3.1)$$

where:

P = Mean surface pressure at the tapping point (Pa)

P_o = Reference static pressure of free air stream flow (Pa)

ρ = Air density in the wind tunnel testing facility (kg/m^3)

U_z = Reference mean wind velocity at the gradient height of the building model (m/s).

This dimensionless parameter, C_p , is the main parameter used in this analysis, which will serve as the main basis for validating the subsequent ANSYS CFX numerical simulations and quantifying pressure reduction.

CHAPTER 4

NUMERICAL MODELLING

4.1 General

Computational Fluid Dynamics (CFD) has proven to be an extremely important predictive analysis technique in structural wind engineering due to its ability to perform a very accurate three-dimensional analysis of flow fields around bluff bodies. While experimental wind tunnel studies focuses on determining pressures, on the other hand, the numerical method facilitates mapping of fluid properties such as separation streamlines, shear layers formation, velocity distribution, and wake turbulence kinematics over the entire computational domain [9-12]. This chapter provides information on the numerical procedure conducted via ANSYS CFX, which is a commercial finite volume solver software used to conduct aerodynamic study of standard and cut-corner Y-shaped high-rise buildings. This chapter will focus on the mathematical basis of the governing equations, turbulence model, discretization of the computational domain, boundary conditions, and convergence criteria of the solver.

While pressure evaluation in the wind tunnel tests gives accurate results, CFD can act as an equally valuable tool as it is capable of offering a full three-dimensional view of the flow field. CFD is also capable of modeling continuously the velocity streamlines, wakes, and shear layer separation, which cannot be measured in their entirety by pressure tapping's in a conventional wind tunnel [15]. The current chapter describes the numerical technique used in the study of aerodynamic interactions between the atmospheric boundary layer and Y-shaped tall buildings. Simulations were carried out using ANSYS CFX finite volume solver. This chapter also discusses the process of geometry creation, definition of the computational domain, meshing methods to be used in detail for the analysis of both the structures.

4.2 Geometric Modelling

In order to have an accurate comparative analysis, the geometry of the building was modeled to be precisely in the same proportions as the actual Perspex models created in 1:150 scale for experimental purposes.

Model S (Standard Y-Shaped): This is the standard tri-axially symmetrical model with sharp edges having a height of 600mm.

Model C (Cut-Corner Y-Shaped): This is the modified model with the same height of 600mm, but with cut-corners at the end of each of the three limbs of the Y-shape structure.

Both models were modelled in AUTOCAD and then imported to ANSYS CFX considering them as rigid solid structures.

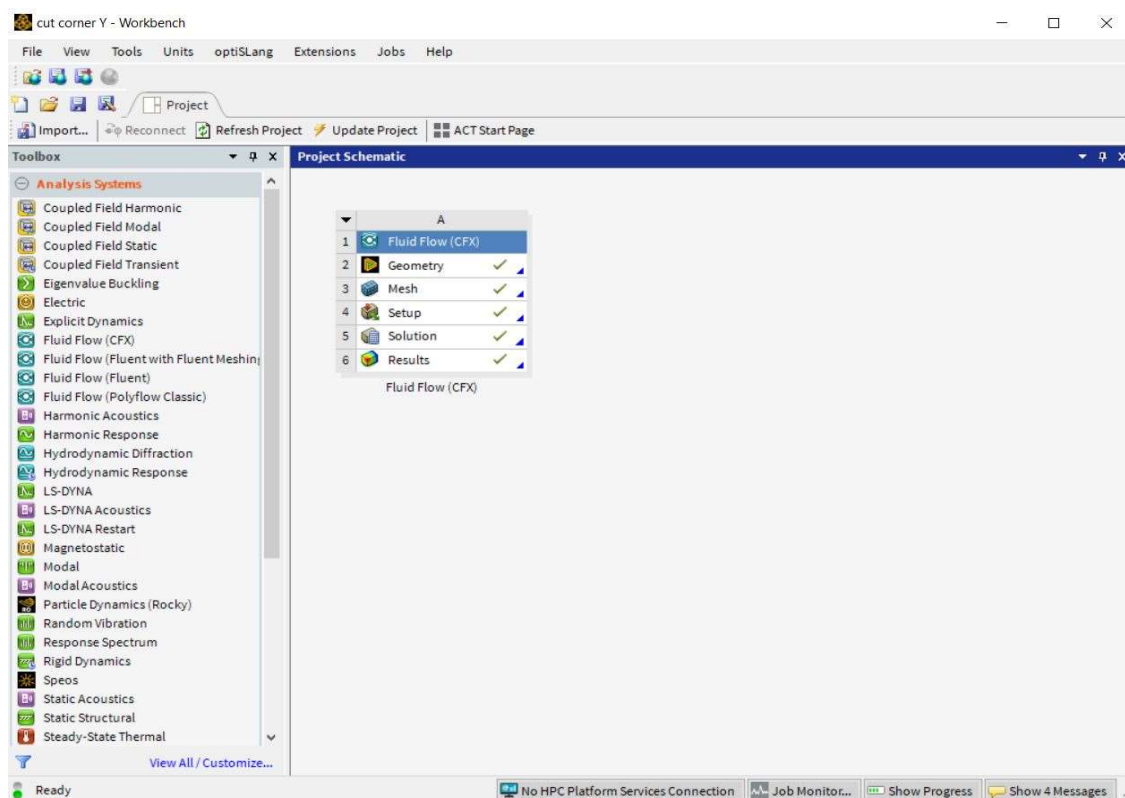


Fig.4.1: ANSYS Workbench Home with fluid flow (CFX)

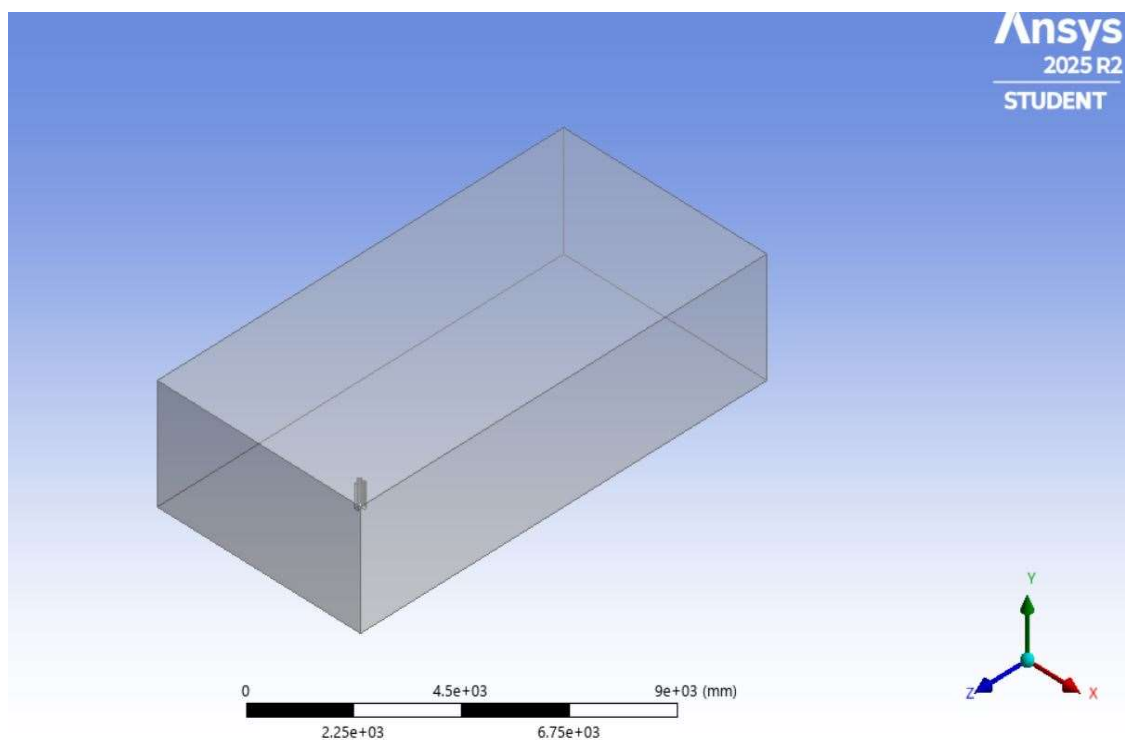


Fig.4.2: Design Modeler for geometric modelling

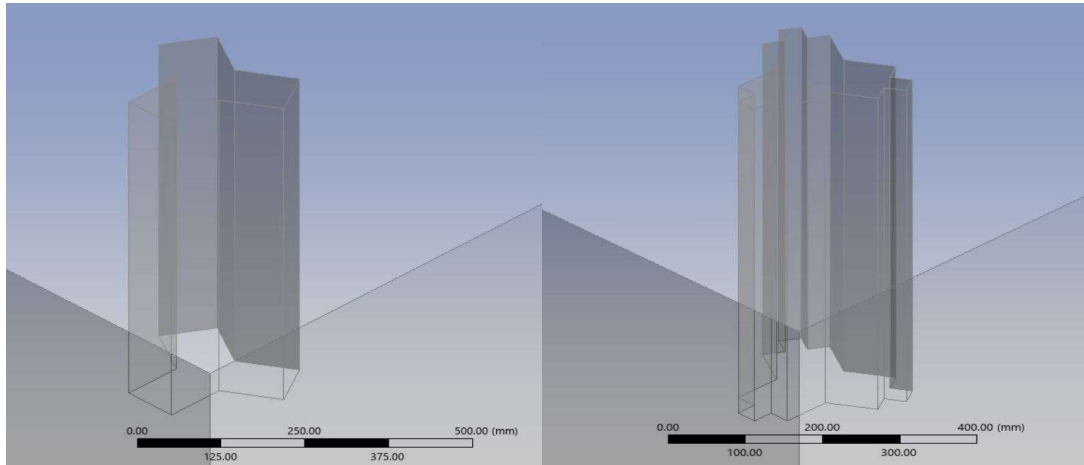


Fig.4.3: Standard Y- model and Cut-corner Y-model

4.3 Governing Equations

The basic physics which governs the fluid flow behavior of an incompressible, isothermal fluid flow around a rigid bluff body can be described by two conservation laws of physics, the conservation of mass and conservation of momentum. In this research work, the wind flow has been simulated as a steady-state, three-dimensional incompressible fluid flow, governed by the Reynolds-Averaged Navier-Stokes (RANS) equations [8]. The mathematical equations which describe these two physical phenomena are as follows:

Continuity Equation (Conservation of Mass): The continuity equation indicates that the mass entering into a controlled volume is equal to the mass leaving the controlled volume. In case of a steady incompressible three-dimensional flow, the continuity equation will take the form:

$$\frac{\partial \rho}{\partial \tau} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad \dots\dots\dots (4.1)$$

The density of subsonic wind engineering flow is assumed to be constant, and the steady state condition has been considered in the analysis. Here, u_i denotes the mean velocity component in the direction of Cartesian coordinates $x_i = 1, 2, 3$.

Navier-Stokes Equation and Moment Equation: Conservation of momentum, which comes from the Newton's Second Law, relates to the forces applied to the fluid element. Steady-state RANS momentum equation can be expressed mathematically as follows.

Navier-Stokes equation:

$$\frac{\partial(\rho u_i)}{\partial t} = -\frac{\partial(\rho u_i u_j)}{\partial x_j} - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + F \quad \dots\dots\dots (4.2)$$

Moment Equation:

$$\frac{\partial(\rho u_i)}{\partial t} = -\frac{\partial(\rho u_i u_j)}{\partial x_j} - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left[\mu_{\text{eff}} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + S_M \dots\dots\dots (4.3)$$

where ρ = density of air (kg/m^3), u_i and u_j = velocity vector in the i^{th} and j^{th} direction respectively (m/s), P = Static fluid pressure (Pa), μ = dynamic viscosity, μ_{eff} = effective viscosity, S_M = momentum sources and F = body forces acting on the fluid.

4.4 Turbulence Modelling

In order to close the RANS equations, turbulence modeling is required for the calculation of the Reynolds stress tensor. In this comparative study, standard two-equation k-epsilon semi-empirical turbulence modelling is selected. The k-epsilon standard model is often preferred in aerodynamic design due to its ideal tradeoff between computational cost and robust, stable estimation of the average pressure distribution over complex and unconventional geometries. In this model, turbulent eddy viscosity (μ_t) depends on turbulent kinetic energy (k) and its dissipation rate (ϵ):

$$\mu_t = \rho C_u \frac{k^2}{\epsilon} \dots\dots\dots (4.4)$$

where C_u is constant. The transport equations of k and epsilon (ϵ) are then solved along with continuity and momentum equations for modeling the production, transport, and dissipation of turbulence over the sharp and recessed edges of Y-shaped structures.

4.5 Computational Domain Design

It is important that the physical boundaries of the virtual wind tunnel are set accurately in order to avoid any computational mistakes. When the computational domain is too small, the artificial boundary conditions squeeze the flow lines around the model, resulting in unrealistic acceleration of velocities and backflow issues. As per the international validation criteria recommended by Franke et al. (2004) [30], the size of the computational domain was scaled relative to the building height ($H = 600\text{mm}$), in order to allow an unrestricted growth of the flow field.

Upstream distance (inlet to model): $5H$ (3000mm), in order to allow the establishment of the atmospheric boundary layer profile before its collision with the model.

Downstream distance (model to outlet): $15H$ (9000mm), in order to provide enough space for unrestricted growth of the downstream wake region and alternating vortices.

Side clearance (model to walls): $5H$ (3000mm) on both sides, in order to minimize the blockage effects.

Top clearance (model to ceiling): $5H$ (3000mm), in order to prevent artificial downward flow acceleration over the roof of the building.

Thus, the domain size limits the total blockage ratio to approx. 3% (<5%).

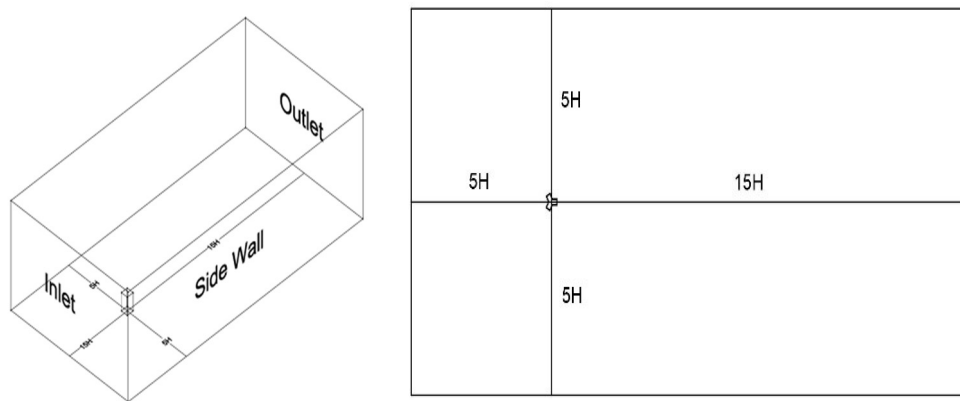


Fig.4.4: Computational domain- Isometric and Top view

4.6 Discretization and Meshing

Due to the complex geometrical characteristics of the standard and cut corner Y-shaped structures which have multiple re-entrant corners and localized cuts, an unstructured meshing technique was employed using mesh option in the CFX toggle bar.

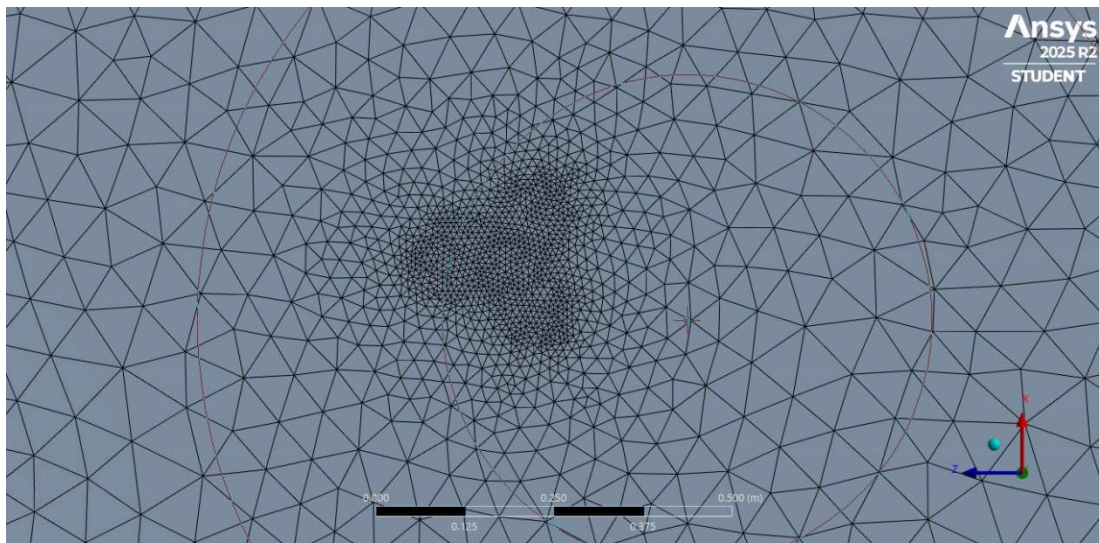


Fig.4.5: Tetrahedral elements used in meshing

Tetrahedral elements were used as the elements for the global domain because of their ability to adapt easily to complex boundaries [8]. For efficient computation, a multi-zoning meshing strategy was used as follows:

Global Domain Mesh: A relatively coarse mesh of tetrahedrons was used to mesh the far field region of the domain region where pressure gradient is minimal to save computational memory [9].

Domain element size – 0.08mm.

Ground layer element size – 0.09mm.

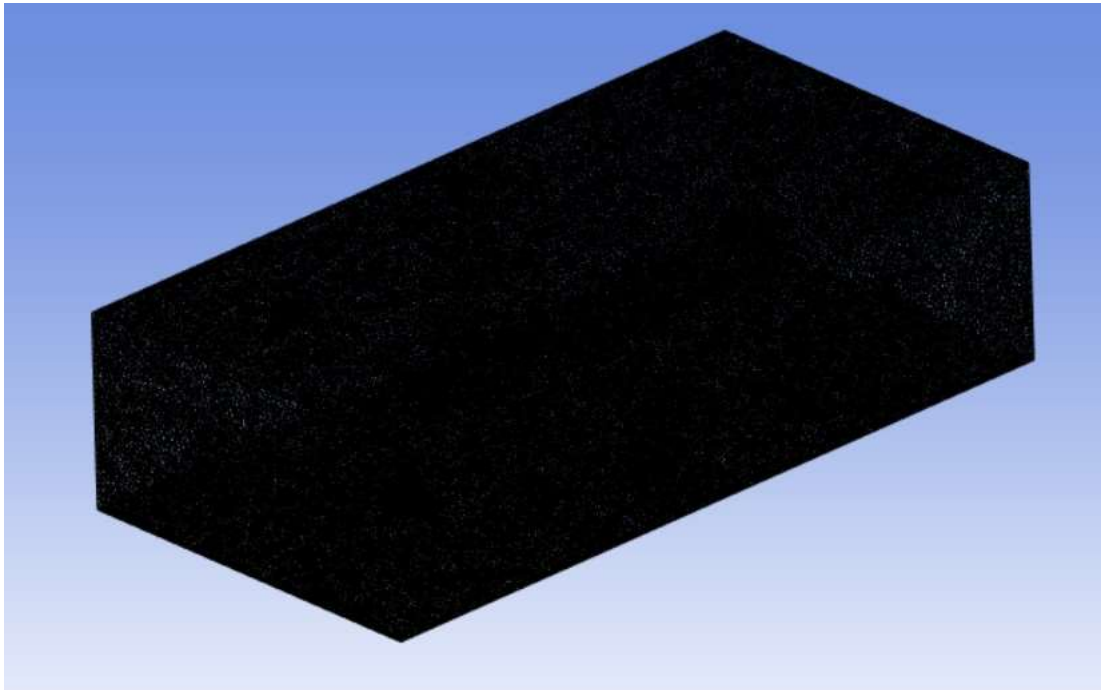


Fig.4.7: Computational Domain Mesh

Model Refinement: High density refinement was done around the building models by applying more fine sizing on the sharp edges of Model S and the recessed geometry of Model C. High density refinement is important in capturing the highly steep pressure gradient and flow separation zones.

Model element size – 0.008mm.

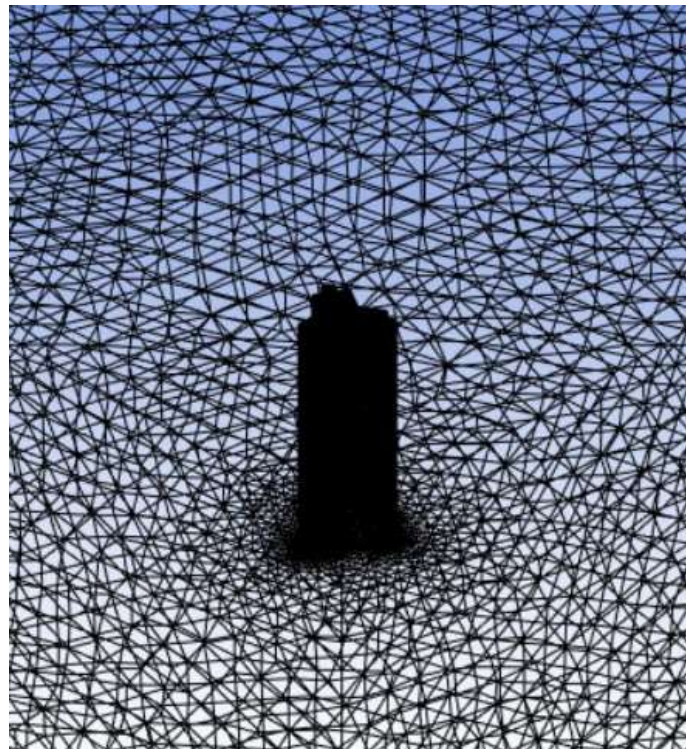


Fig.4.8: Fine refined mesh around the building model

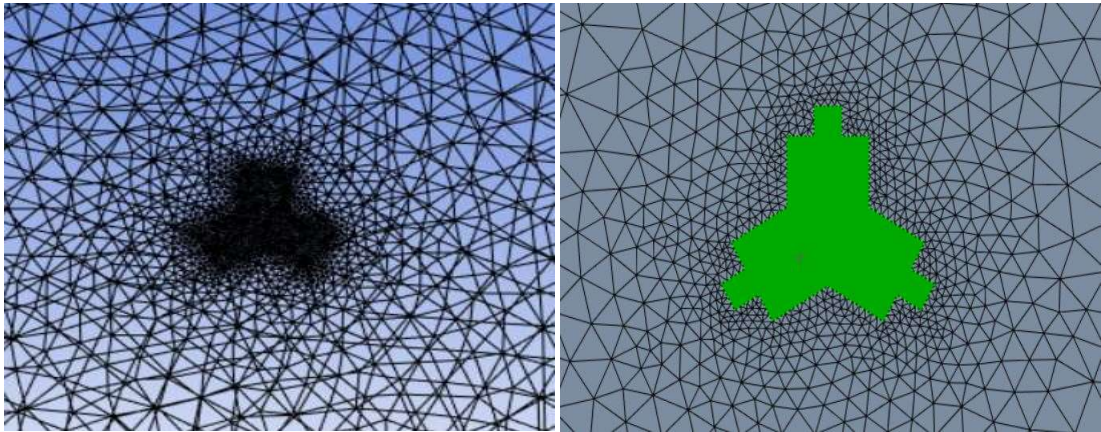


Fig.4.9: Inflation layers around the building models

Boundary Layer Inflation Layers: Explicit prismatic inflation layers were added on all faces of the models to capture the high shear laminar and turbulent transition in the boundary layers of the models [8].

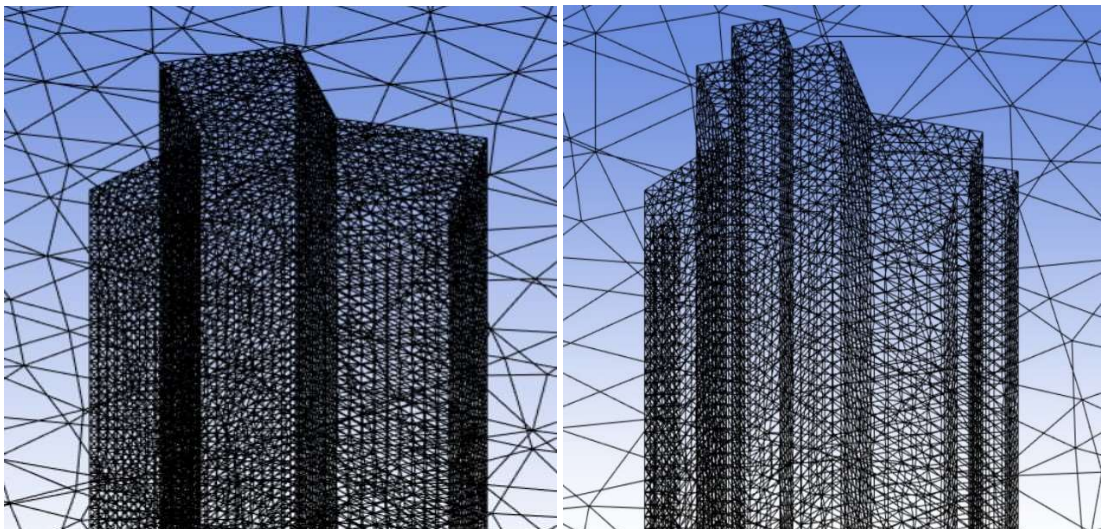


Fig.4.10: Meshing with of both the models

Mesh Independence Study for a Rectangular model: Coarse (167898 elements), Medium (654478 elements), fine (1409816 elements) and very fine (4820391 elements) mesh size were considered for meshing configurations. Fine and very fine mesh size provided nearly same C_p values, due to computing efficiency fine mesh was adopted to get accurate result.

Values	IS 875 Part-3	Coarse Mesh	Medium Mesh	Fine Mesh	Very fine Mesh
Windward Side	+0.58	+0.75	+0.68	+0.61	+0.57
Leeward Side	-0.5	-0.62	-0.53	-0.49	-0.48

4.7 Boundary Conditions and Inflow Physics

In order to achieve a simulation of the physical wind tunnel experiment under Terrain Category II calibration, relevant boundary conditions mathematically were applied to the faces of the computational domain:

Inlet Boundary Condition

An atmospheric boundary layer (ABL) profile was formulated at the inlet through the use of the power law formula:

$$\frac{U_z}{U_{ref}} = \left(\frac{Z}{Z_{ref}} \right)^\alpha \dots\dots\dots (4.5)$$

where: α (α) = Power-law exponent, equal to 0.147 for an exact simulation of roughness characteristics of Terrain Category II, U_{ref} = Free-stream velocity calibrated to 10 (m/s) [36], Z_{ref} = Reference height equal to 1m, U_z = Mean wind speed at reference height (Z).



Fig.4.11: Expression window of CFX Solver

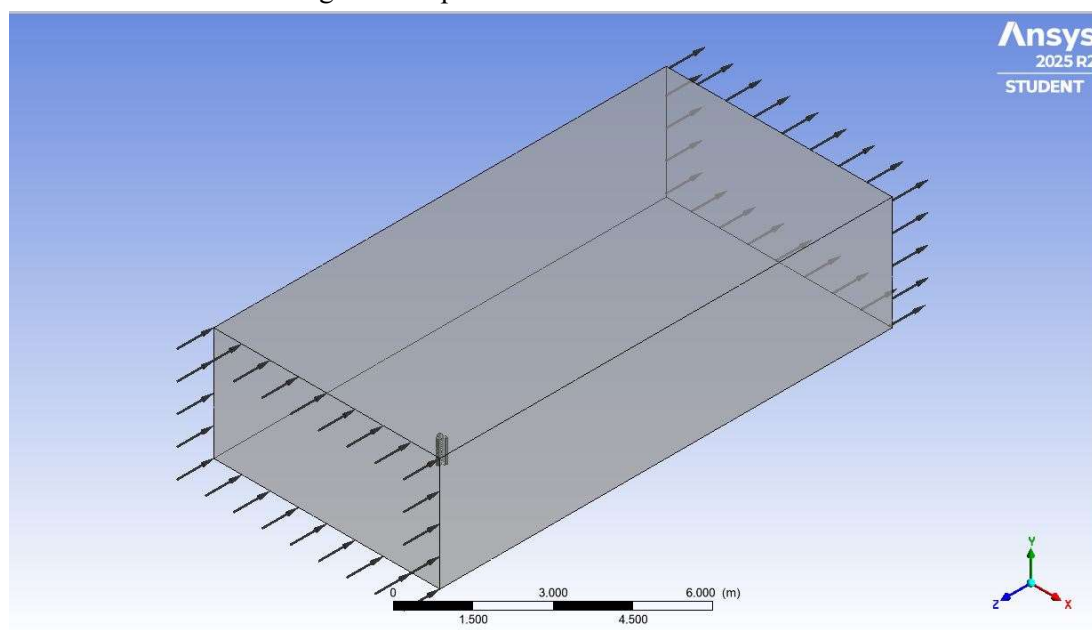


Fig.4.12: Inlet and Outlet Boundary Conditions

Outlet and Surface Boundary Conditions

Outlet: A static pressure condition was used with relative pressure equal to zero (Pa), which allowed the unrestricted fluid mass flow from the outlet.

Boundary conditions for the Top and Side walls: Symmetric boundary (free-slip) condition was chosen for both walls to prevent any friction or physical constraints from occurring in the ceilings and side walls of the domain.

Surface boundaries for Building and Ground floor: Rigid no-slip wall boundary conditions were chosen with appropriate roughness for the domain floor surface [15].

4.8 Solver settings and Convergence criteria

The governing equations were solved using the steady-state, pressure-based coupled solver from ANSYS CFX. Both the advection and turbulence equations were discretized using high-resolution second-order upwind schemes in order to reduce numerical diffusion errors. For absolute mathematical stability and accuracy prior to data extraction:

- The Root-Mean-Square (RMS) residual target for mass, momentum, and turbulence parameters had to fall below a convergence criterion of 1×10^{-5} and conservation target value of 1×10^{-4} .
- As well as monitoring residuals, net imbalances in mass and momentum throughout the entire flow field had to be less than 0.1%.
- Monitor points were strategically selected on the stagnation regions facing the windward direction and suction regions on the leeward direction in order to capture pressure coefficient (C_p) values in real time.
- The simulation process was halted when the desired engineering parameters reached a perfectly steady plateau after 50 iterations.

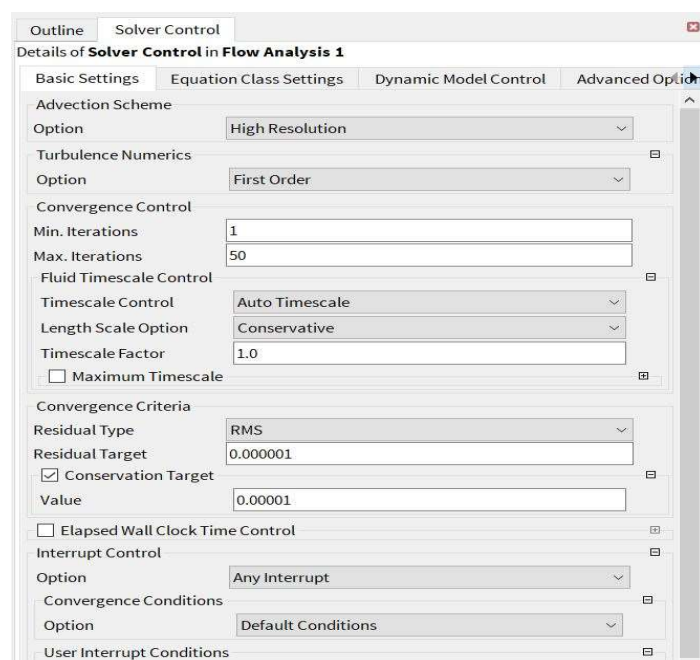


Fig.4.13: Solver control tab in CFX Solver

4.9 Iterations and Solution

- In the solution tab of CFX toggle bar, double precision and large problem are selected in type of run and then run is started. Iterations go on for 50 times and after its completion results section is ready to display pressure distribution, etc.

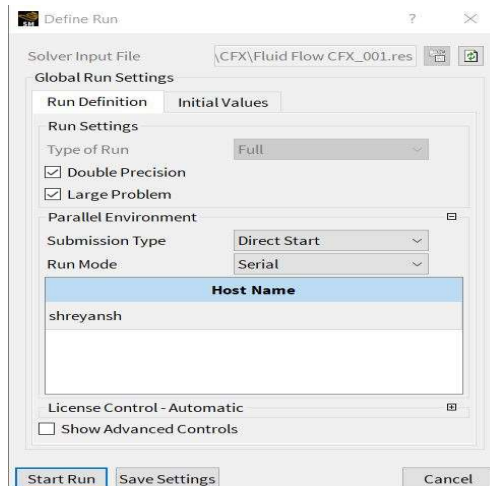


Fig.4.14: Iteration tab in solutions section of CFX Solver

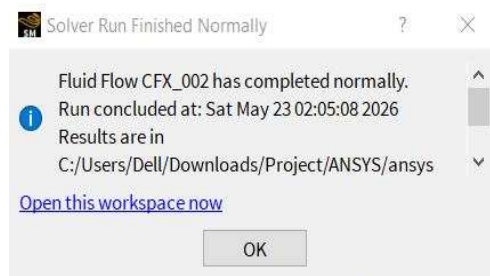


Fig.4.15: Run completion tab

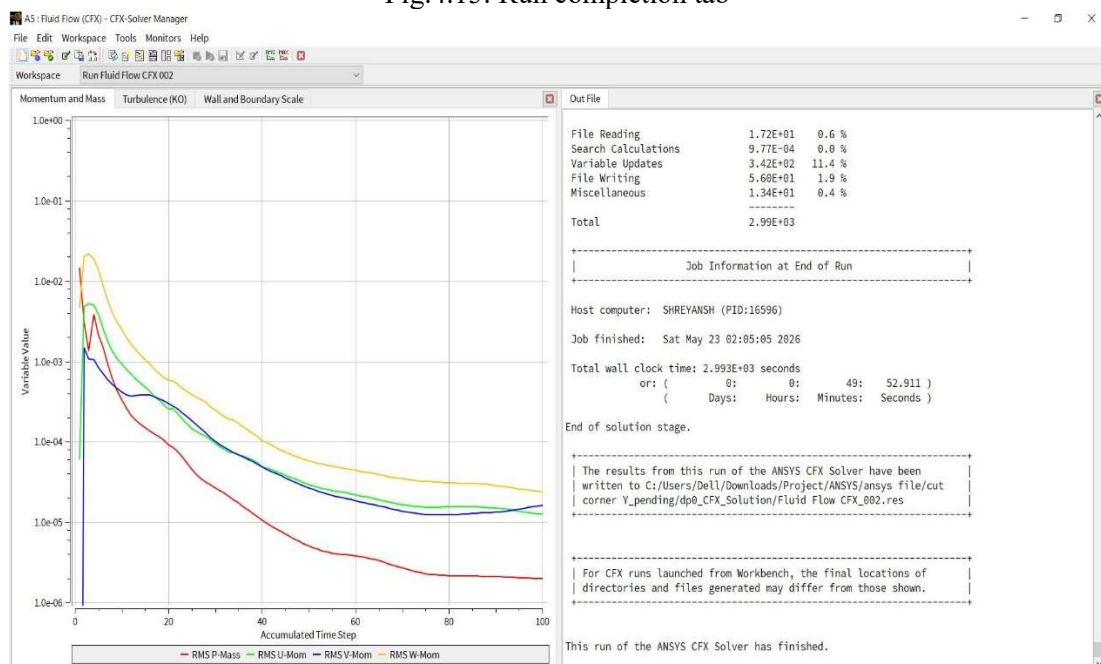


Fig.4.16: CFX Solver after iterations

- In results section of CFX, velocity streamlines are used to display the wind flow using Z-X Streamlines, Y-Z Streamlines. Also pressure contours is generated for each face to describe the pressure distribution along the surface of models. Coefficient of pressure (C_p) is calculated using the excel sheets of each face of the models. C_p is the average value of differential pressure divided by $\frac{1}{2}\rho U_H^2$. Utilising these results further analysis of the standard and cut-corner Y-shape models is done in next chapter.

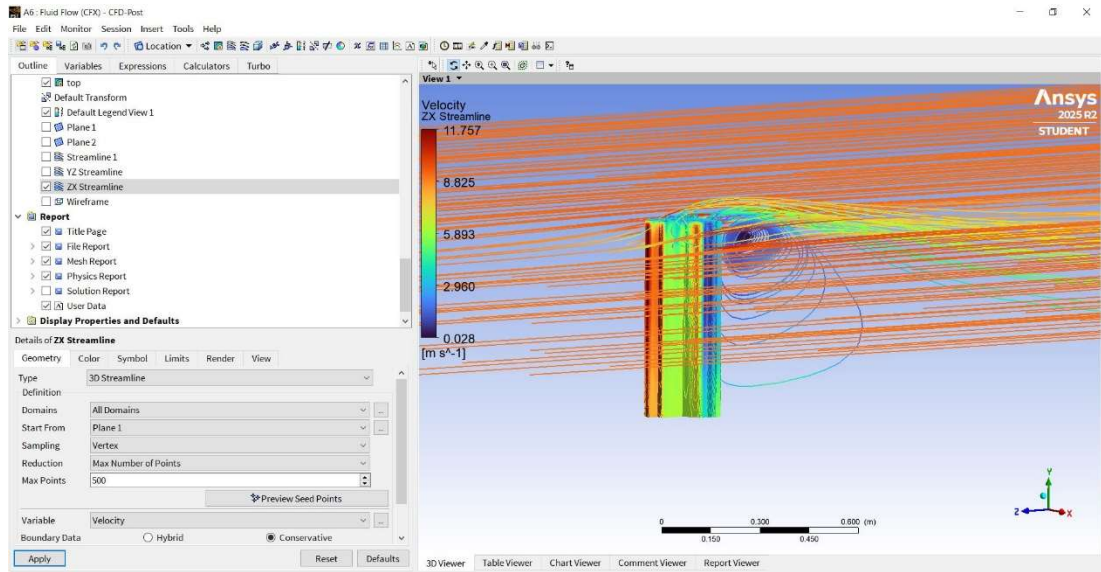
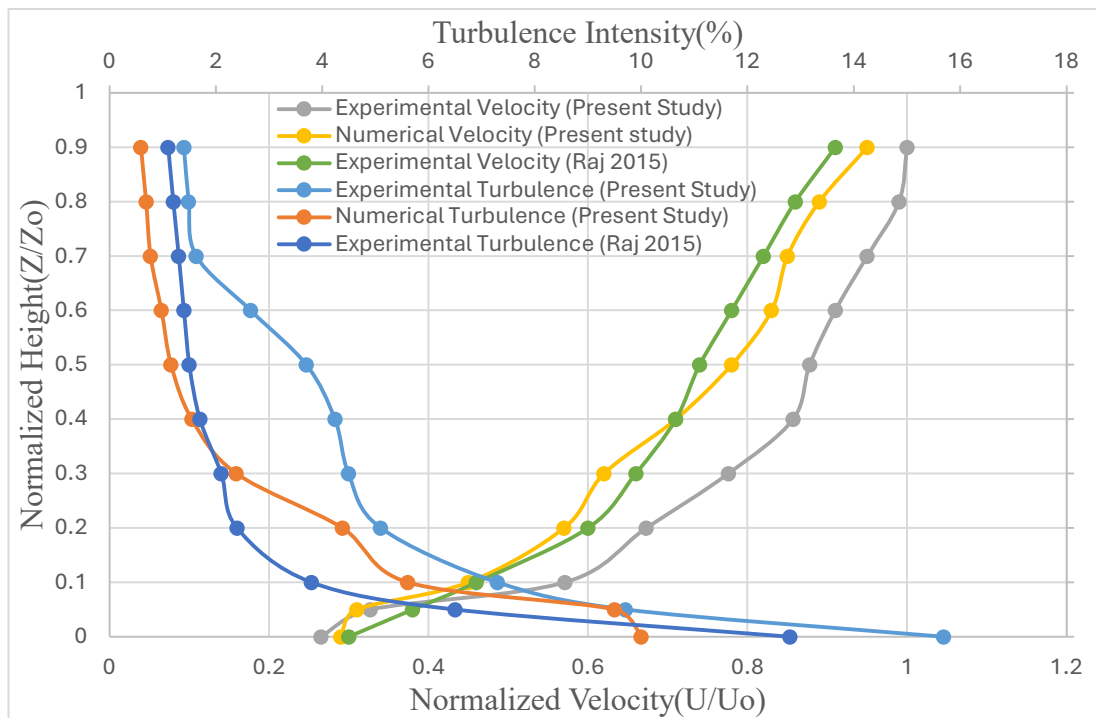


Fig.4.17: Result section of ANSYS CFX

4.10 Velocity and Turbulence Intensity Profile



CHAPTER 5

RESULTS AND DISCUSSION

5.1 General

The chapter contains a comparative discussion on the average values of C_p obtained for both the Standard Y-shape and the Cut-Corner Y-shape models for different angles of attack (AOA) starting from 0° up to 180° with 30° intervals. The purpose of the analysis will be to assess the aerodynamic efficiency under different AOAs, and comparing the experimental results obtained from the Boundary Layer Wind Tunnel (BLWT) with that of ANSYS CFX (k-epsilon turbulence model). The cut-off corners were assessed for efficiency of mitigating aerodynamic effects against sharp corners.

5.2 Validation of Results

- The CFX model exhibits excellent correlation with the experimental data, showing the symmetrical splitting of flow and wake formation with negligible deviations.
- To validate the results, a square cross-section structure with dimensions 150mm x 150mm and height 600mm was investigated in the wind tunnel facility and simultaneously analyzed in ANSYS CFX.

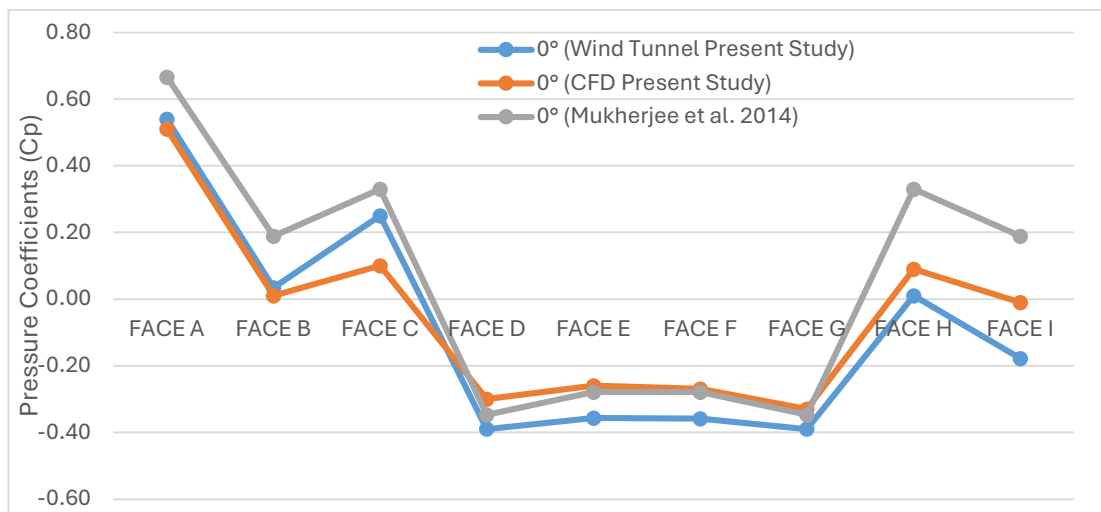


Fig.5.1: Plan and fabricated model of square structure

Table 5.1: Comparison of mean pressure coefficients

Analysis Method	Wind angle	Windward side	Sidewall	Leeward side
ANSYS CFX	0°	0.81	-0.57	-0.58
Wind Tunnel Tests	0°	0.78	-0.64	-0.47
IS 875 (Part-3)	0°	0.70	-0.61	-0.40

- The steady state CFD results indicate are consistent with the experimental wind tunnel results at all wind angles of attack.
- Employing the standard k- ϵ turbulence model and utilizing an inflation layer meshing method led to accurate determination of complex wind flow separation locations, wake thickness, and pressure concentration zones.
- These results confirm that the current CFD approach is a credible predictor of localized aerodynamic changes to complex structures.
- Pressure coefficients (C_p) of experimental wind tunnel study and CFD study of Standard Y-shape model were evaluated for all faces from A to I at 0° wind incidence and compared with available empirical data of Mukherjee et al. (2014). Data obtained is graphically represented below:



5.3 Face-wise pressure coefficients (C_p) Analysis by Wind Incidence Angle

Pressure Coefficient (C_p) values for Standard Y-model

Table 5.2: Pressure Coefficient (C_p) values for Standard Y (WIND TUNNEL)

AOA	0°	30°	60°	90°	120°	150°	180°
FACE A	0.49	0.32	-0.56	-0.56	-0.46	-0.52	-0.44
FACE B	0.03	-0.62	-0.47	-0.52	-0.42	-0.47	-0.51
FACE C	0.15	-0.6	-0.46	-0.51	-0.41	-0.44	-0.49
FACE D	-0.4	-0.52	-0.42	-0.55	-0.44	-0.5	-0.47
FACE E	-0.36	-0.48	-0.45	-0.65	0.09	0.42	0.49
FACE F	-0.36	-0.48	-0.42	-0.66	0.006	0.53	0.47
FACE G	-0.4	-0.53	-0.33	0.076	0.43	0.15	-0.64
FACE H	0.003	0.32	0.49	0.36	-0.13	-0.68	-0.53
FACE I	-0.18	0.42	0.49	0.34	-0.07	-0.59	-0.52

Table 5.3: Pressure Coefficient (C_p) values for Standard Y (CFD Simulation)

AOA	0°	30°	60°	90°	120°	150°	180°
FACE A	0.51	0.28	-0.53	-0.49	-0.38	-0.5	-0.38
FACE B	0.01	-0.58	-0.39	-0.47	-0.35	-0.42	-0.47
FACE C	0.1	-0.54	-0.37	-0.4	-0.33	-0.38	-0.44
FACE D	-0.3	-0.49	-0.34	-0.5	-0.4	-0.46	-0.42
FACE E	-0.26	-0.5	-0.37	-0.61	0.09	0.43	0.53
FACE F	-0.27	-0.45	-0.32	-0.59	0.01	0.58	0.53
FACE G	-0.33	-0.52	-0.34	0.12	0.52	0.21	-0.59
FACE H	0.09	0.38	0.54	0.42	-0.004	-0.63	-0.48
FACE I	-0.01	0.45	0.55	0.44	-0.095	-0.52	-0.54

Pressure coefficient (C_p) values for Cut-corner Y-model

Table 5.4: Pressure Coefficient (C_p) values for Cut-corner Y (WIND TUNNEL)

AOA	0°	30°	60°	90°	120°	150°	180°
FACE A	0.49	-0.17	-0.78	-0.58	-0.56	-0.64	-0.56
FACE B	0.06	-0.76	-0.67	-0.57	-0.53	-0.57	-0.59
FACE C	0.15	-0.76	-0.68	-0.58	-0.54	-0.57	-0.63
FACE D	-0.21	-0.35	-0.70	-0.58	-0.53	-0.57	-0.64
FACE E	-0.03	-0.51	-0.69	-0.56	-0.51	-0.52	-0.60
FACE F	-0.71	-0.55	-0.67	-0.60	-0.55	-0.53	-0.67
FACE G	-0.64	-0.51	-0.68	-0.61	-0.53	-0.54	-0.63
FACE H	-0.47	-0.51	-0.64	-0.68	-0.55	-0.60	-0.79
FACE I	-0.46	-0.47	-0.71	-0.68	-0.81	-0.35	0.24
FACE J	-0.45	-0.47	-0.67	-0.71	-0.85	-0.46	0.07
FACE K	-0.43	-0.52	-0.69	-0.65	-0.05	0.37	0.31
FACE L	-0.44	-0.51	-0.61	-0.71	-0.24	0.37	0.37
FACE M	-0.45	-0.55	-0.71	-0.65	0.12	0.41	0.07
FACE N	-0.46	-0.54	-0.71	-0.67	0.03	0.40	0.18
FACE O	-0.41	-0.58	-0.88	-0.24	0.30	-0.31	-0.77
FACE P	-0.72	-0.37	0.15	0.30	-0.12	-0.81	-0.63
FACE Q	-0.73	-0.43	0.05	0.29	-0.01	-0.78	-0.63
FACE R	0.01	0.24	0.31	0.23	-0.34	-0.71	-0.63
FACE S	-0.11	0.69	0.25	0.23	-0.16	-0.68	-0.64
FACE T	0.30	0.68	-0.09	-0.66	-0.78	-0.71	-0.60
FACE U	0.09	0.59	0.11	-0.57	-0.74	-0.68	-0.60

Table 5.5: Pressure Coefficient (C_p) values for Cut-corner Y (CFD Simulation)

AOA	0°	30°	60°	90°	120°	150°	180°
FACE A	0.57	-0.14	-0.58	-0.45	-0.41	-0.47	-0.4
FACE B	0.19	-0.62	-0.48	-0.44	-0.48	-0.44	-0.45
FACE C	0.27	-0.6	-0.5	-0.45	-0.46	-0.48	-0.5
FACE D	-0.12	-0.37	-0.55	-0.47	-0.47	-0.43	-0.5
FACE E	0.02	-0.41	-0.53	-0.41	-0.44	-0.4	-0.45
FACE F	-0.55	-0.43	-0.5	-0.49	-0.46	-0.41	-0.48
FACE G	-0.47	-0.36	-0.49	-0.48	-0.4	-0.42	-0.47
FACE H	-0.32	-0.35	-0.43	-0.51	-0.41	-0.46	-0.56
FACE I	-0.27	-0.28	-0.5	-0.49	-0.56	-0.19	0.28
FACE J	-0.28	-0.27	-0.43	-0.5	-0.65	-0.28	0.25
FACE K	-0.27	-0.35	-0.45	-0.47	0.03	0.45	0.5
FACE L	-0.27	-0.32	-0.42	-0.52	-0.15	0.57	0.59
FACE M	-0.3	-0.36	-0.57	-0.49	0.22	0.62	0.2
FACE N	-0.32	-0.34	-0.58	-0.57	0.13	0.58	0.27
FACE O	-0.33	-0.4	-0.63	-0.15	0.42	-0.2	-0.56
FACE P	-0.59	-0.22	0.23	0.43	0.19	-0.67	-0.53
FACE Q	-0.57	-0.31	0.17	0.42	0.07	-0.65	-0.53
FACE R	0.01	0.34	0.4	0.4	-0.21	-0.61	-0.51
FACE S	-0.06	0.57	0.36	0.38	0.02	-0.54	-0.54
FACE T	0.36	0.63	0.15	-0.51	-0.63	-0.59	-0.51
FACE U	0.2	0.58	0.28	-0.42	-0.59	-0.55	-0.52

0° Wind Incidence Angle (Head-on Impact)

In the case of 0° incidence angle, the incoming wind impacts the leading limb of the Y-shape models.

Standard Y-Model:

- Face A serves as the main pressure stagnation area, showing the maximum positive pressure ($C_p = 0.49$ in WT, 0.51 in CFX).
- The flow separation takes place from the sharp leading edges, resulting in moderate suction region on the sides and leeward faces.
- The highest value of negative pressure appears on Face D (-0.40 WT, -0.30 CFX) and Face G (-0.40 WT, -0.33 CFX).
- Pressure contours clearly showcase the positive pressure on Face A and negative pressure zones on Face D, G, I.

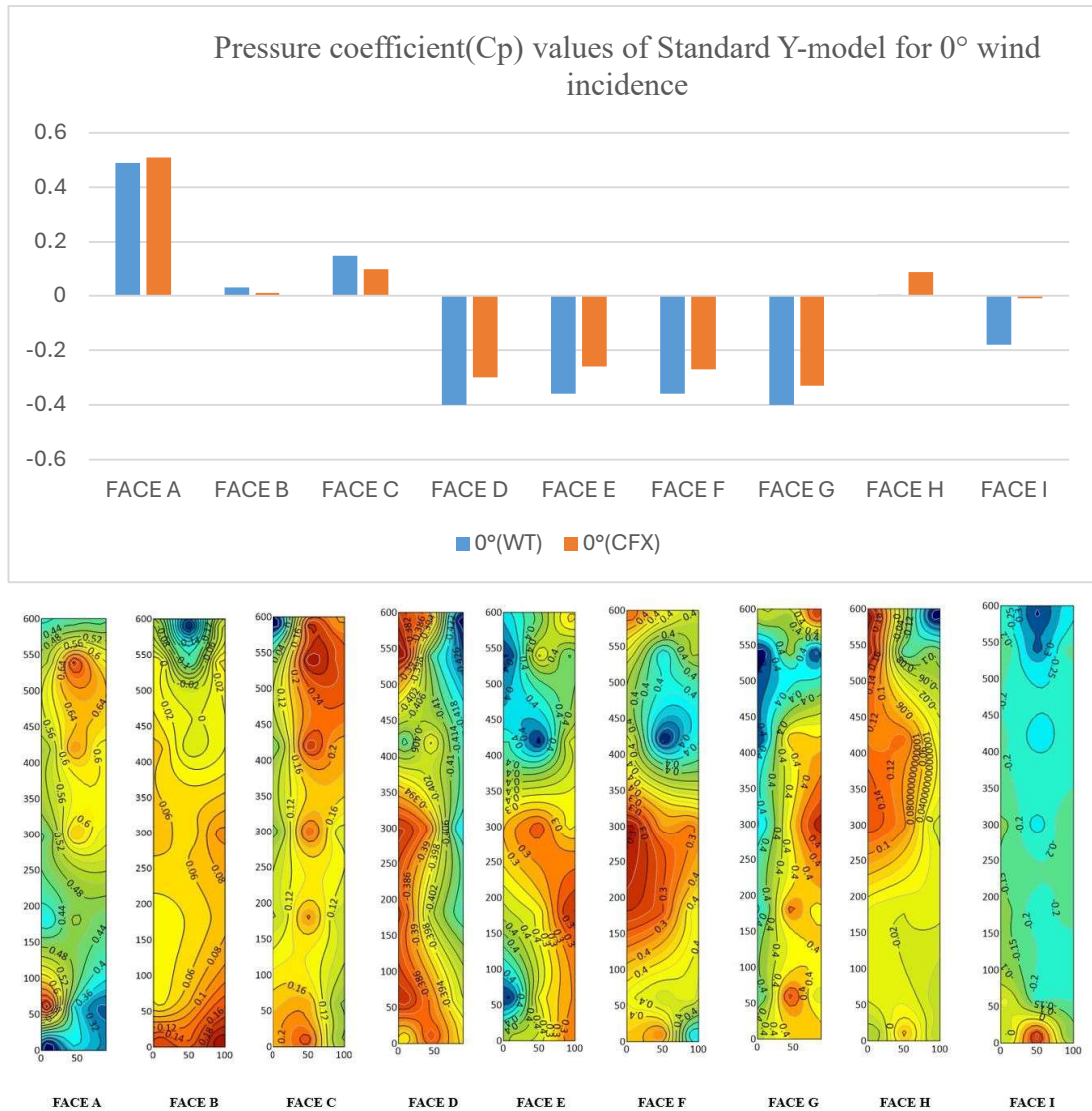
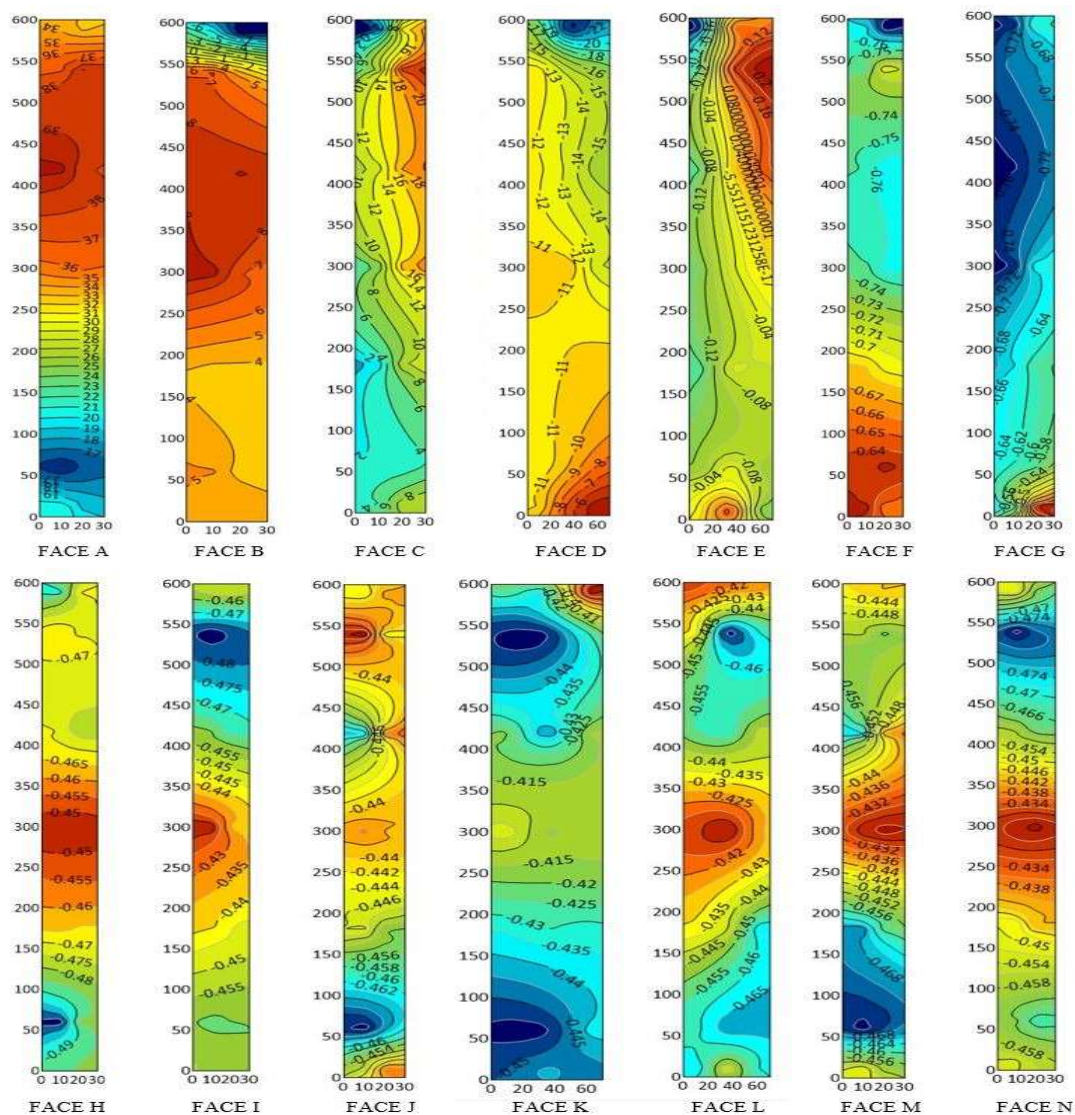
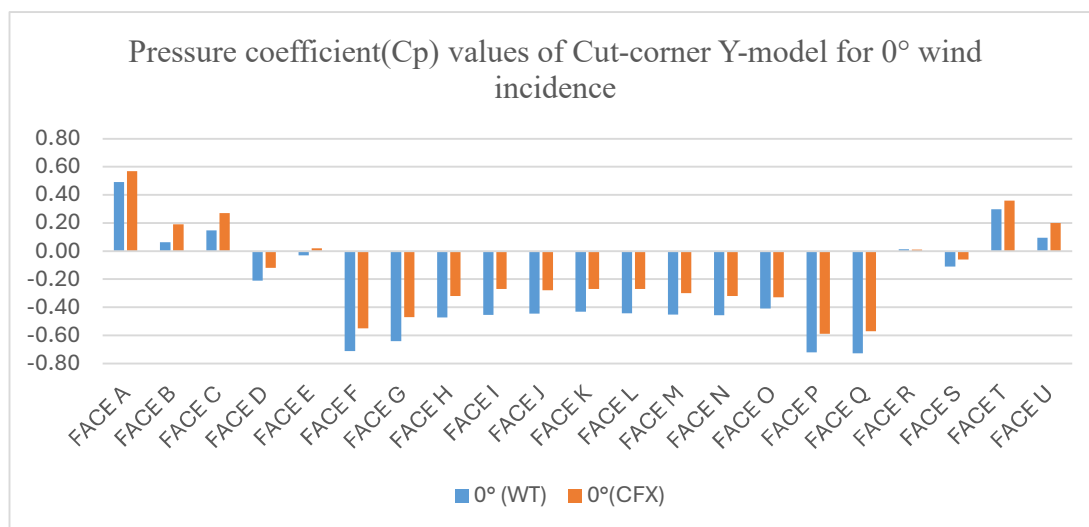


Fig.5.2: Pressure contours of Standard Y-model at 0° -wind incidence

Cut-Corner Y-Model:

- Pressure concentration for Face A is similar in both the wind tunnel ($C_p = 0.49$) and CFX analysis (0.57).
- Introduction of the cut corners (Faces T and U) causes creation of secondary stagnation regions ($C_p = 0.29$ and 0.09 in WT, respectively). In this regard, shear layer separation is forced to occur at a much earlier stage, though there is increased suction effect on the adjacent faces such as Faces P and Q (-0.72 WT).
- Positive pressure regions are shown through orange to brownish colour in the contours below while the negative pressure regions are shown in shades of blue colour.



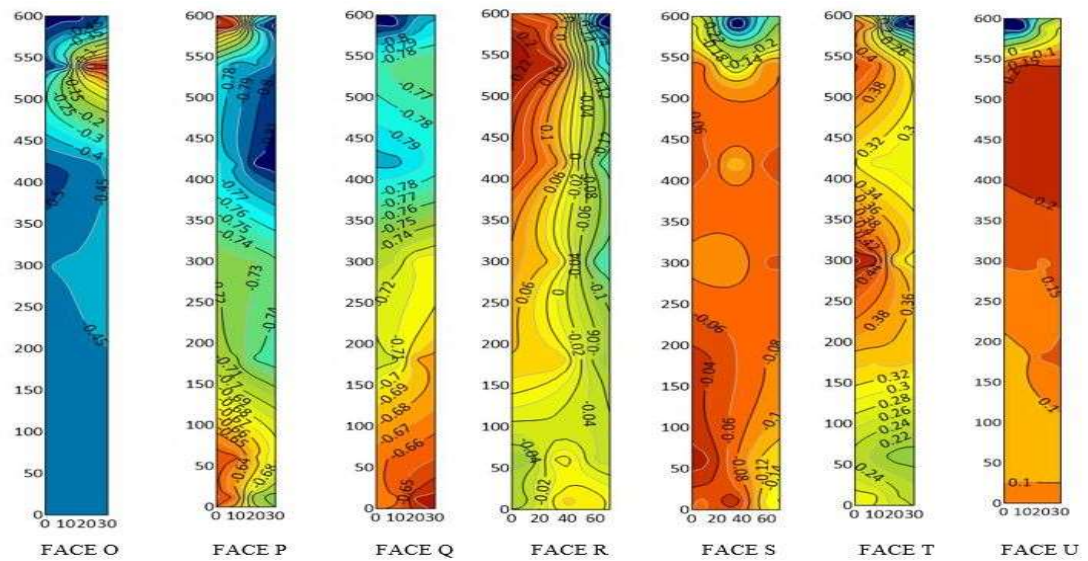


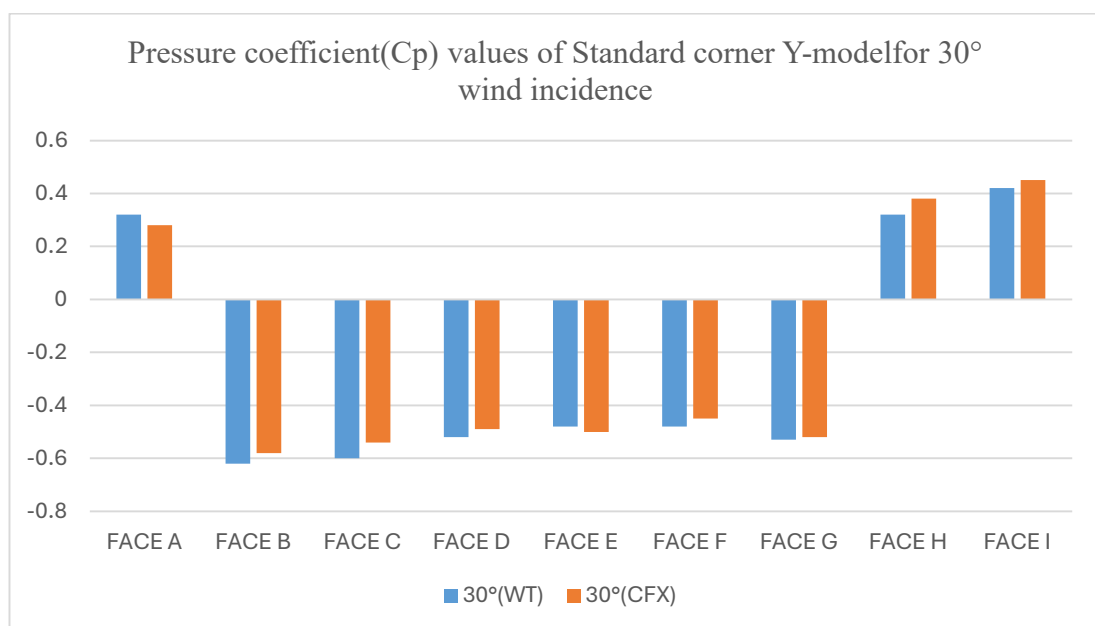
Fig.5.3: Pressure contours of Cut-Corner Y-model at 0° -wind incidence

Wind Incidence Angle: 30° (Critical Asymmetrical Flow)

This angle describes the critical asymmetrical loading state, whereby flow bypasses one limb and is entrapped in the re-entrant corner.

Standard Y-Model:

- Pressure stagnation region is displaced to Face H and Face I which witness the highest positive pressures (0.32 and 0.42 in WT; 0.38 and 0.45 in CFX).
- There is considerable flow separation caused by the sharp edges, hence massive localized suction on Faces B (-0.62 WT) and C (-0.60 WT).
- This pressure distribution is clearly visible through the contour plots.



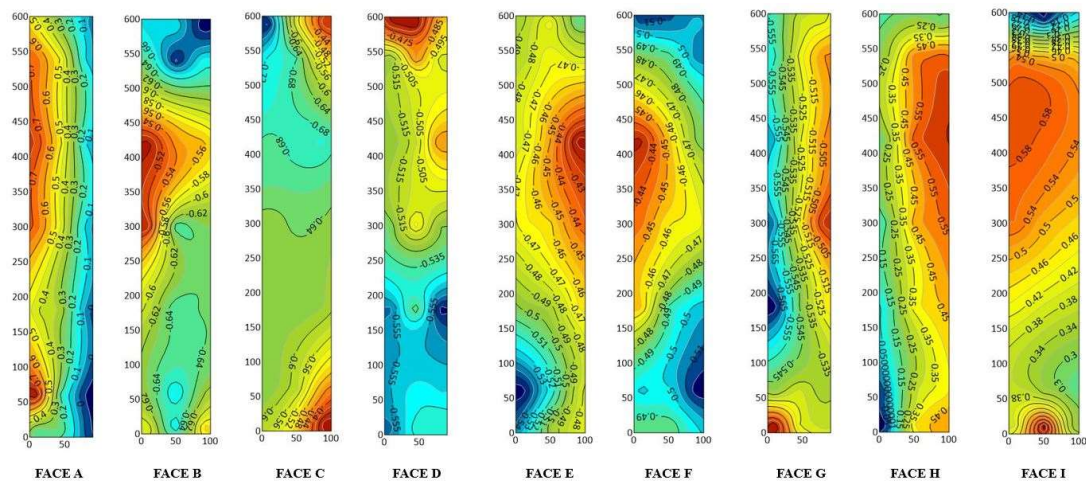
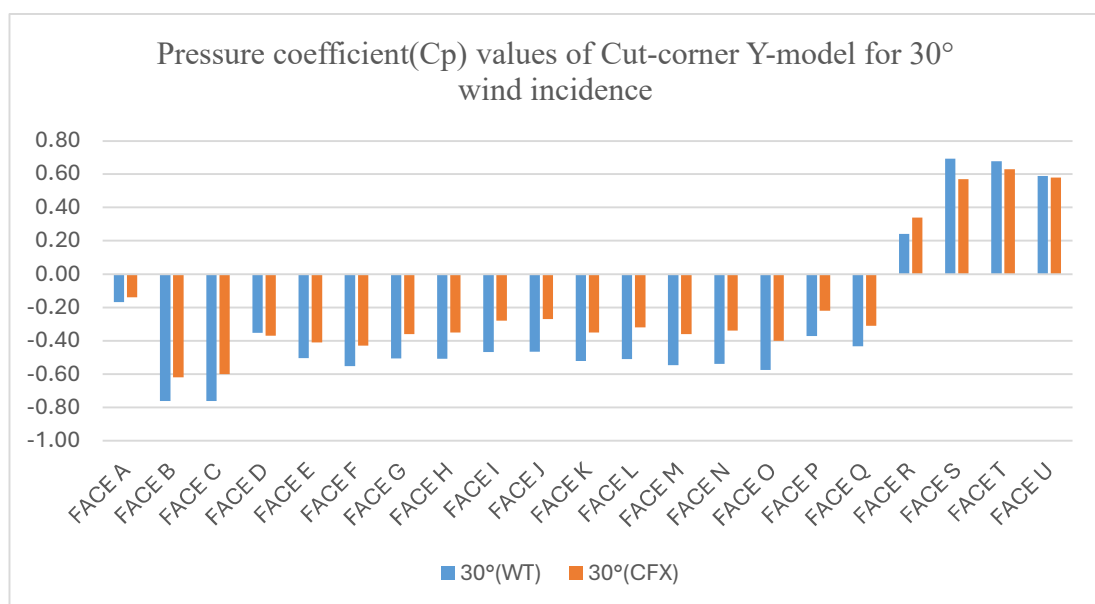


Fig.5.4 Pressure contours of Standard Y-model at 30°-wind incidence angle

Cut-Corner Y-Model:

- The cut corner design significantly changes the pressure distribution. The positive stagnation points move to the recessed surfaces S, T and U (0.69, 0.68, 0.59 in WT, respectively).
- As can be seen, although the overall base shear is reduced, the localized negative pressure on Faces B and C increases (-0.76, -0.76 WT respectively). It implies that the cut corner design accelerates the detached shear flow along the inner limb, and therefore proper localized cladding design must be considered.
- The overall suction reduces on other faces which is also visible through the pressure contours.



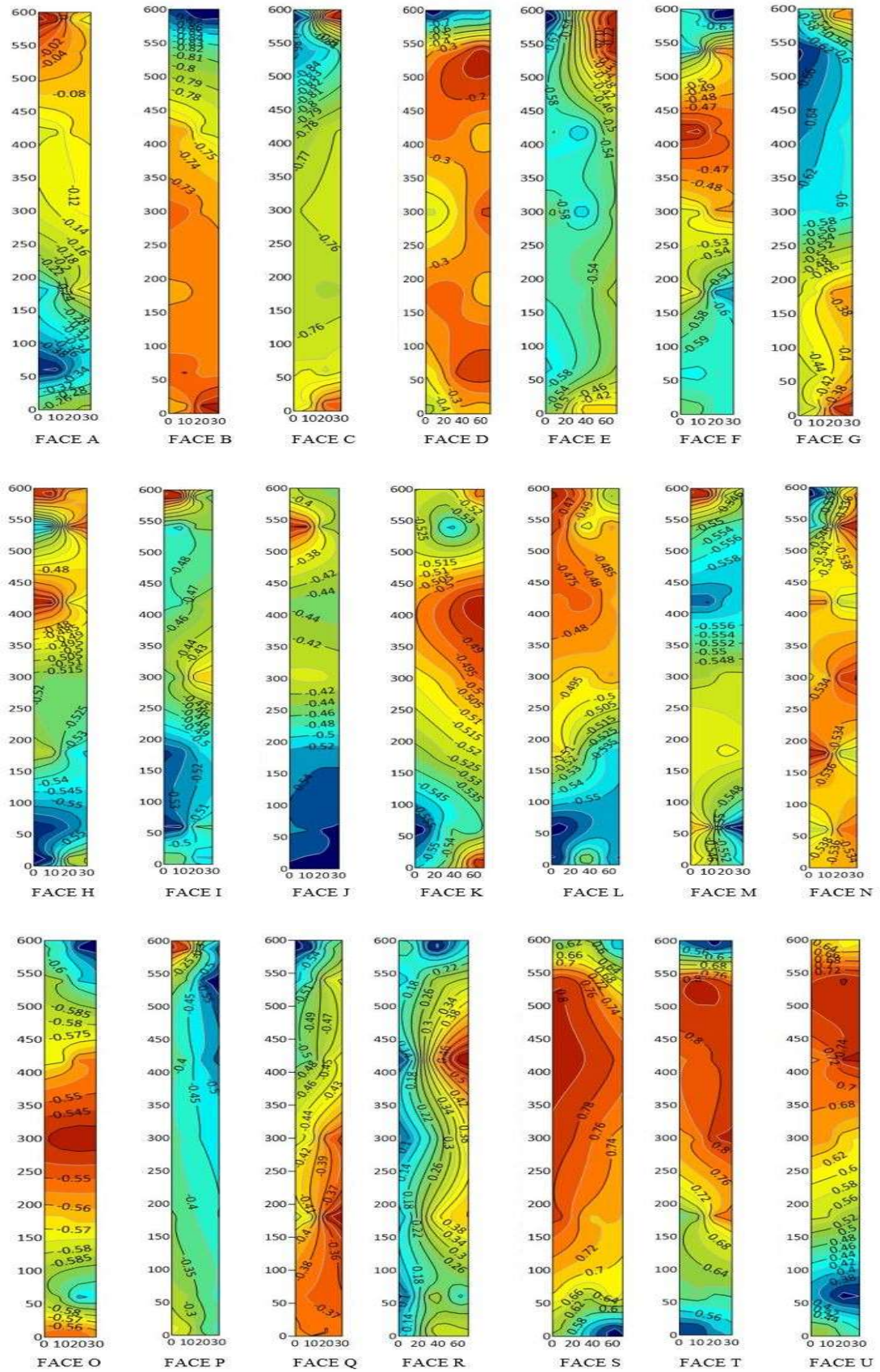


Fig.5.5: Pressure contours of Cut-Corner Y-model at 30°-wind incidence

Wind Incident Angle: 60° (Grazing and Re-Entrant Trapping)

At an incidence angle of 60°, the wind grazes the leading limb of the model and impinges on the inner re-entrant faces strongly.

Standard Y-Model:

- The maximum positive pressure points occur on the inner limb on Faces H and I (0.49 for both in WT; 0.54 and 0.55 in CFX).
- Faces A and B of the outer limb lie in the maximum suction region (-0.56 and -0.47 WT).

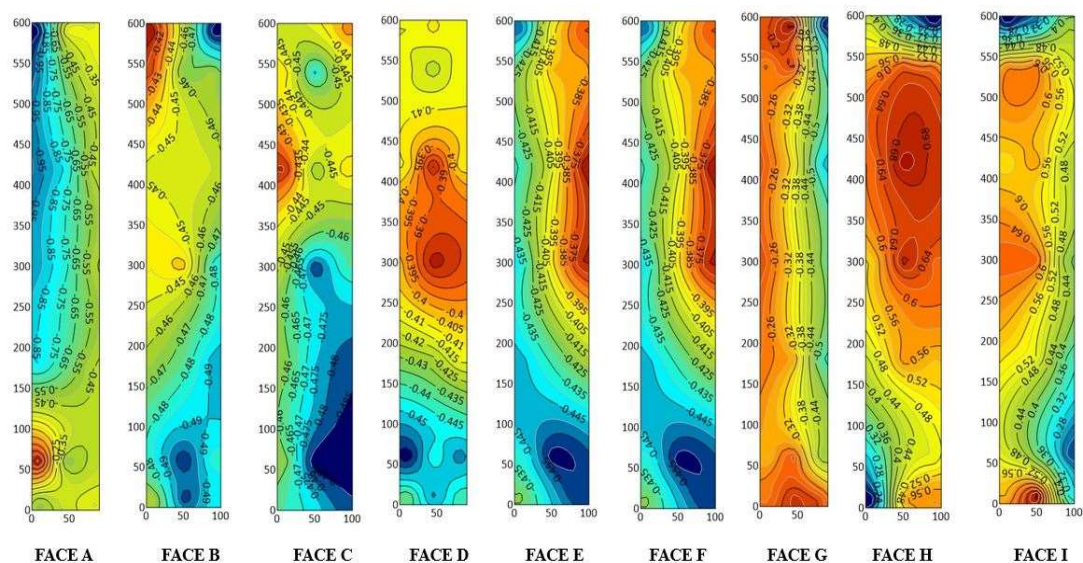
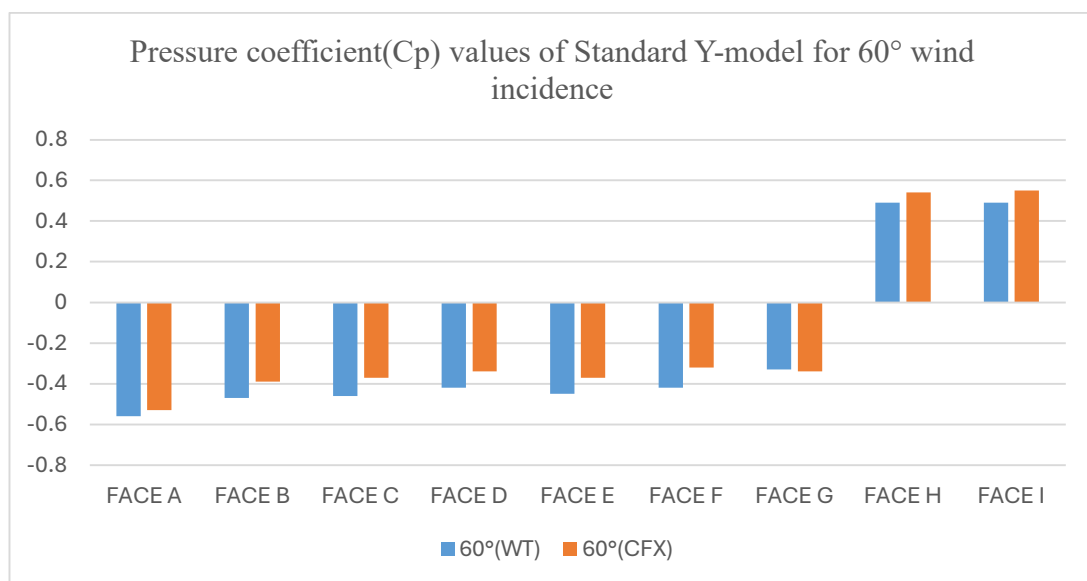
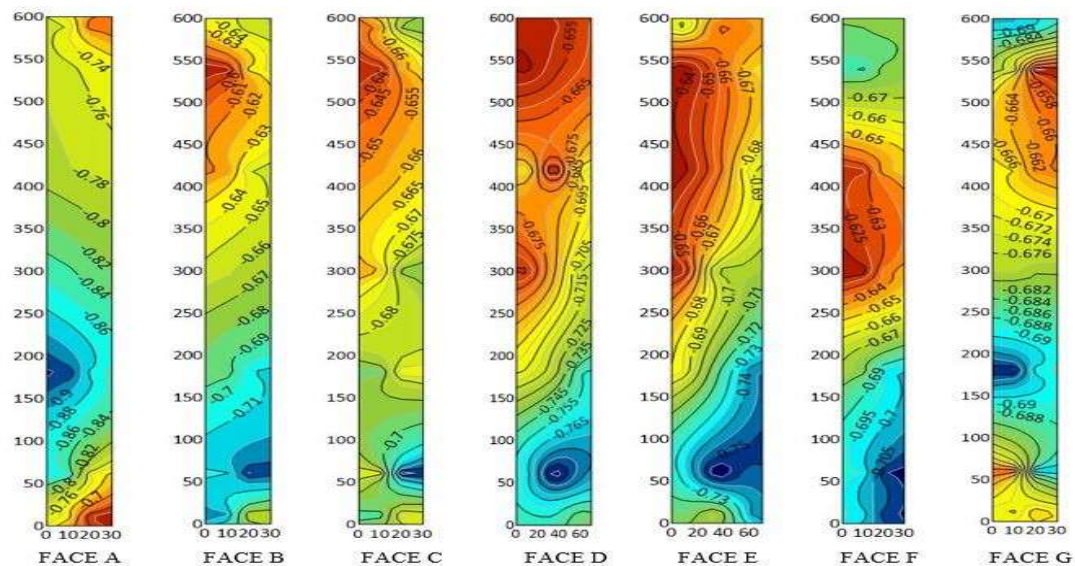
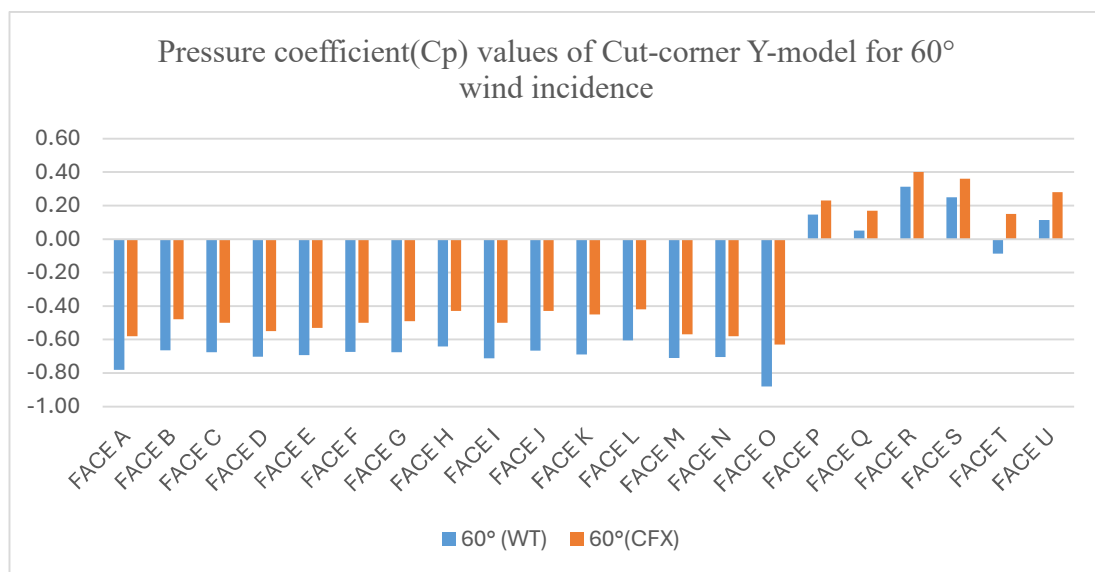


Fig.5.6: Pressure contours of Standard Y-model at 60°-wind incidence

Cut-Corner Y-Model:

- The modification is able to break down the air trapped volume.
- The positive pressure forces are sustained by the modified extremities (Faces P, Q, R, S) with pressure values ranging between 0.05 and 0.31 in WT, and between 0.15 and 0.40 in CFX.
- The extreme negative suction forces are distributed evenly on the leeward side surfaces which shows the effectiveness of the cut-corner to disperse the vortex shedding (visible in the pressure contours below).



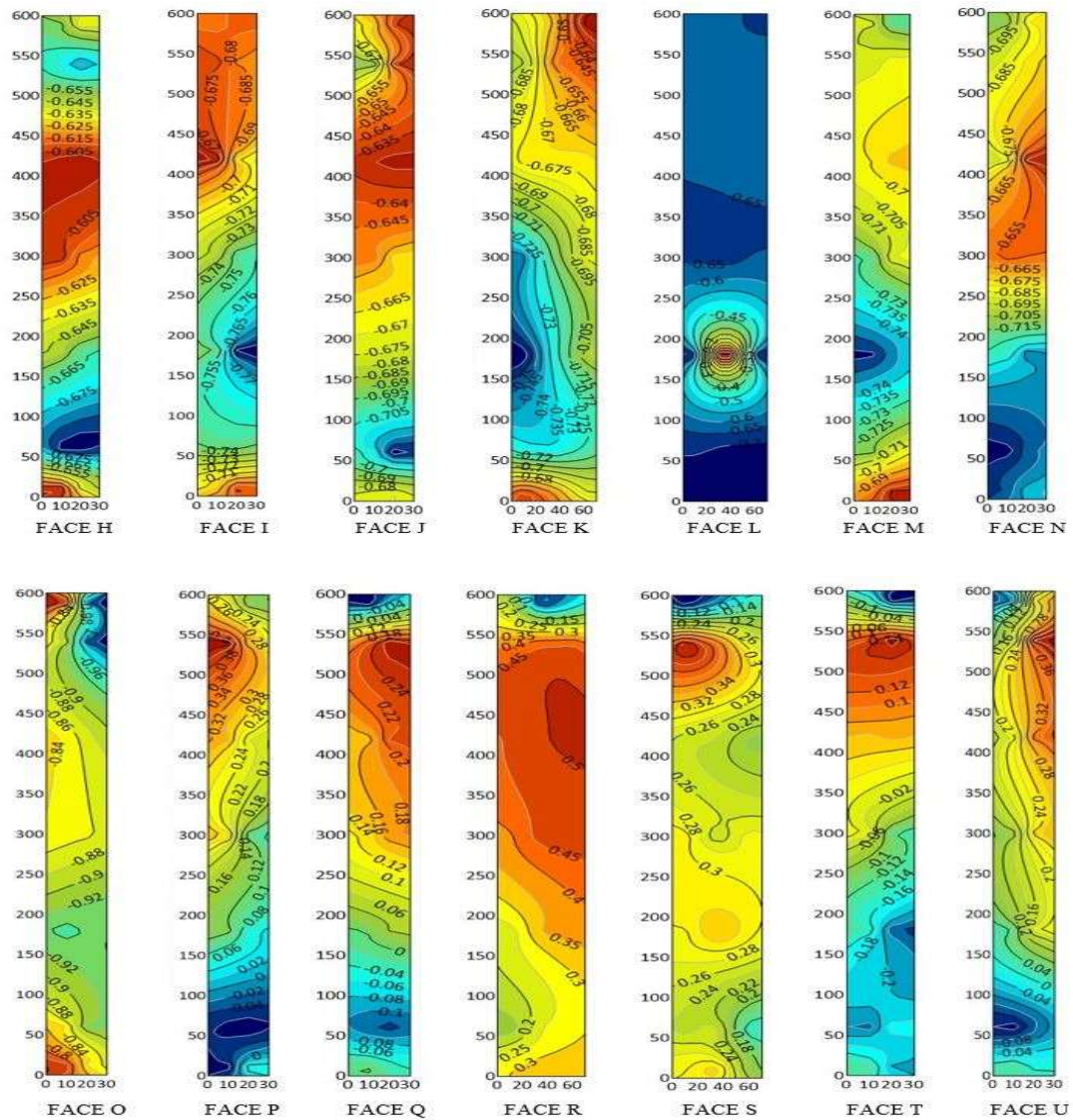


Fig.5.7: Pressure contours of Cut-Corner Y-model at 60°-wind incidence

Wind Incidence Angle: 90° (Cross-Wind Vulnerability)

At 90°, the wind flow is perpendicular to the vertical axis of one of the trailing limbs.

Standard Y-Model:

- Strong suction areas occur on the upper and lower limbs, on Face E (-0.65 WT) and Face F (-0.66 WT).
- Pressure contours show maximum positive pressure concentration on Face H, I (0.36, 0.34 in WT and 0.42, 0.44 in CFX), this shows moderate values implying high turbulence in the physical test for which RANS predicts higher values.

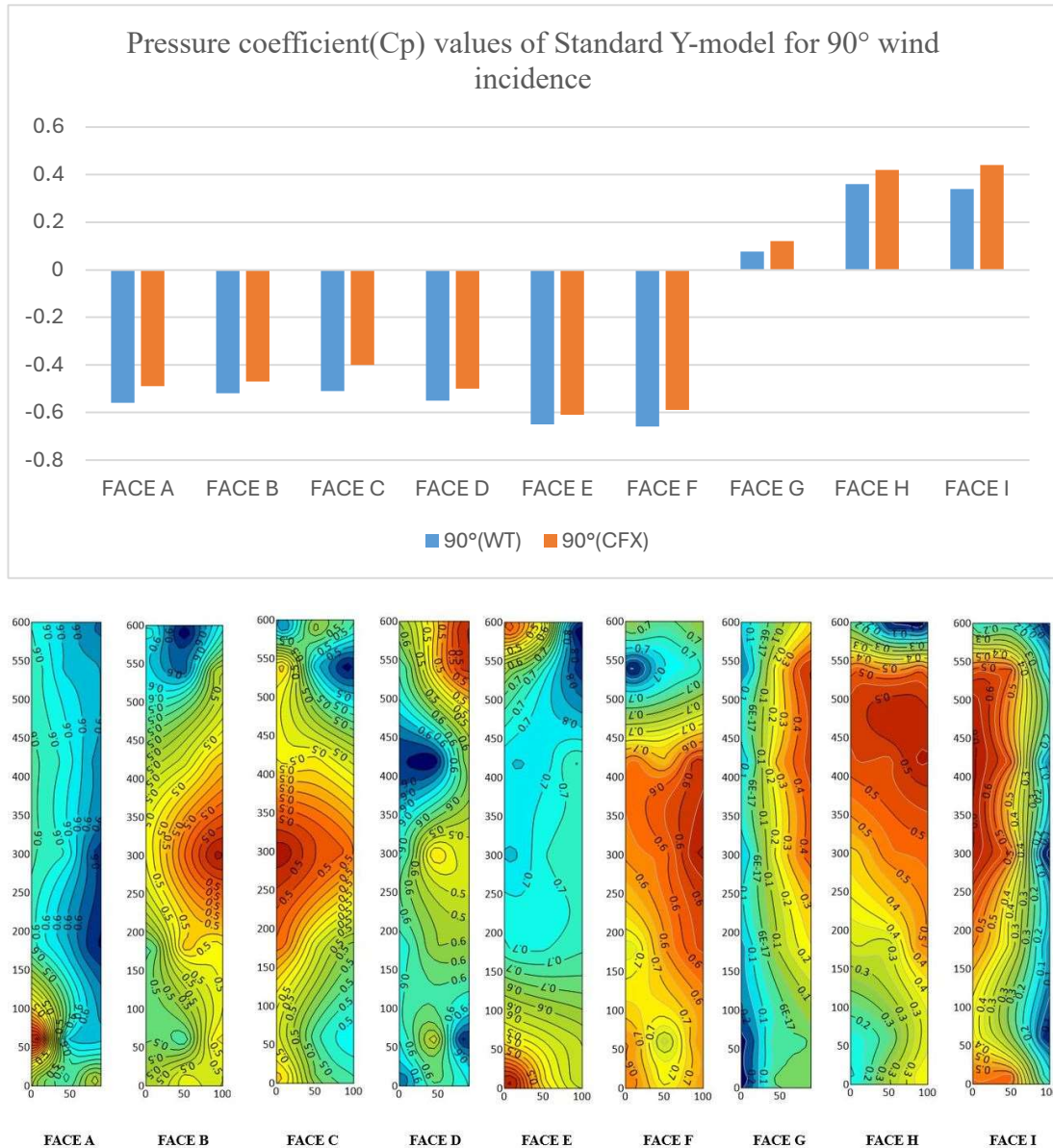
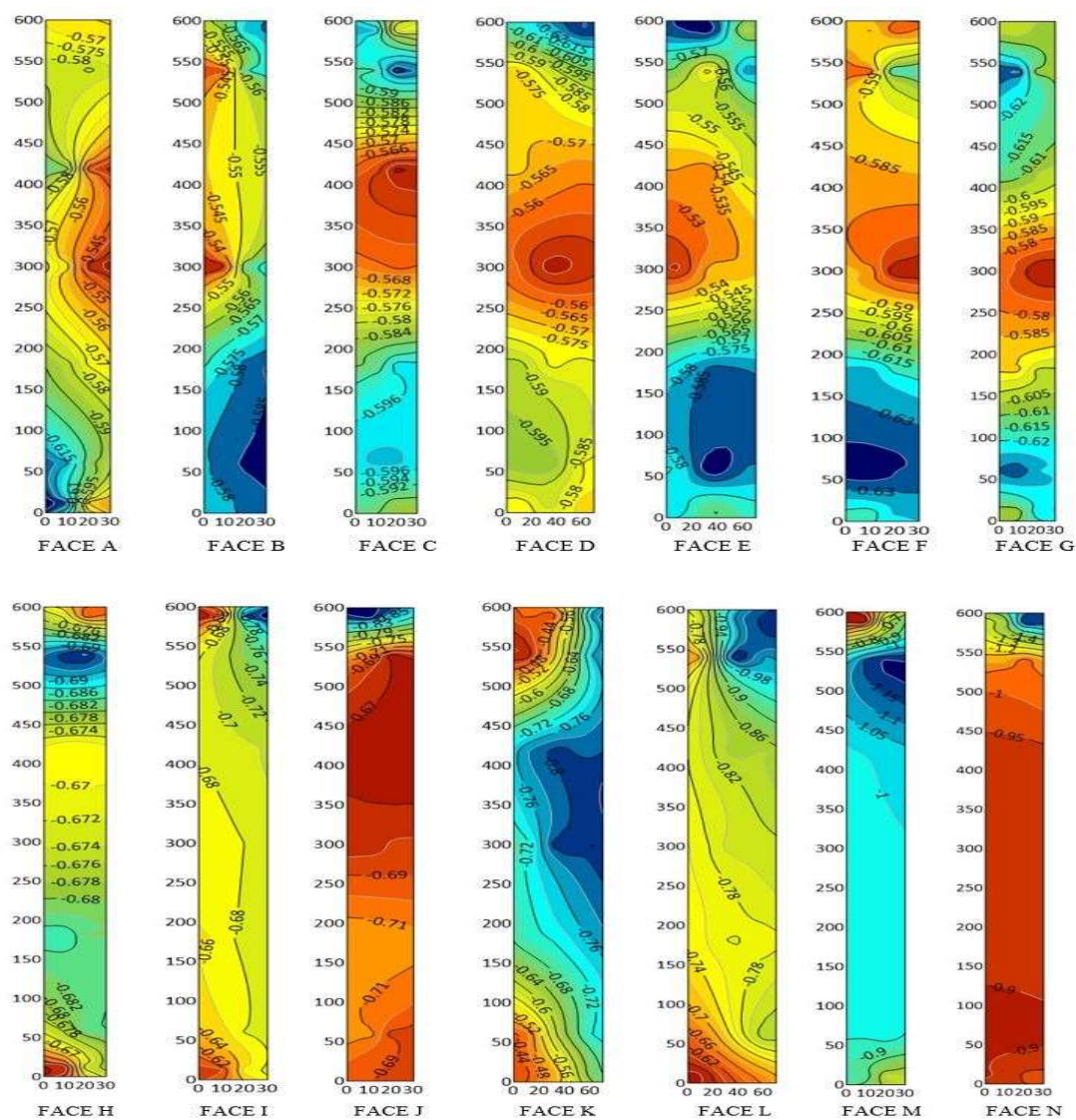
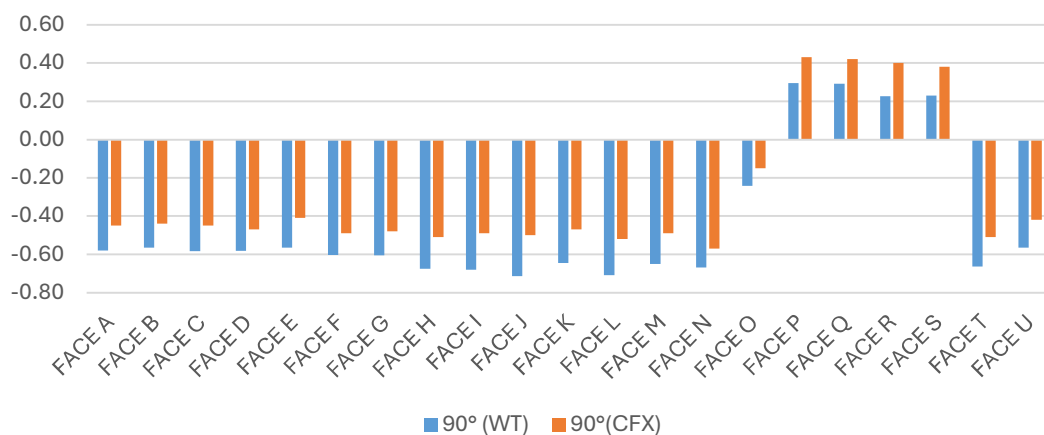


Fig.5.8: Pressure contours of Standard Y-model at 90°-wind incidence

Cut-Corner Y-Model:

- The recessed modifications (Faces P, Q, R and S) trap the cross-wind with positive C_p values of 0.23 to 0.30 in WT.
- However, a large suction region develops on the Faces H to M (mean C_p values ranging from -0.65 to -0.71 in WT, from -0.47 to -0.57 in CFX).

Pressure coefficient(C_p) values of Cut-corner Y-model for 90° wind incidence



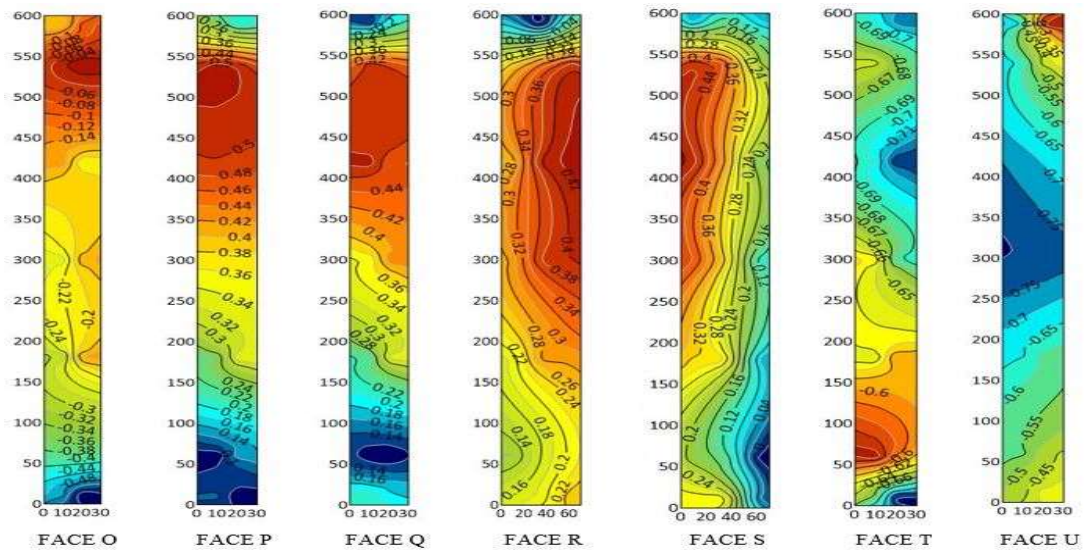


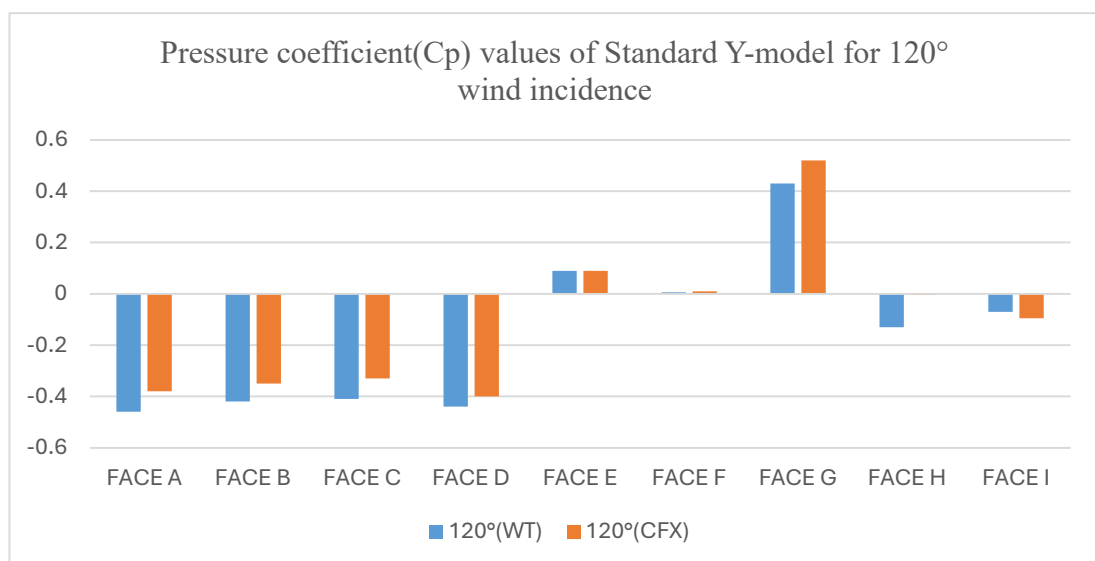
Fig.5.9: Pressure contours of Cut-Corner Y-model at 90°-wind incidence

Wind Incident Angle: 120° (Tri-axial symmetrical shift)

Due to the 120° symmetry of the Y-shape, the wind flow dynamics at 120 degrees behave in the same way as at 0 degrees, but shifted onto another limb.

Standard Y-Model:

- Face G experiences pressure stagnation (having C_p values = 0.43 WT, 0.52 CFX). This is as expected, since all other faces are in the suction wake area, clearly proving the geometric symmetry of the physical model.
- Negative pressure ranges between -0.17 to -0.46 in WT and between -0.004 to -0.40 in CFX.



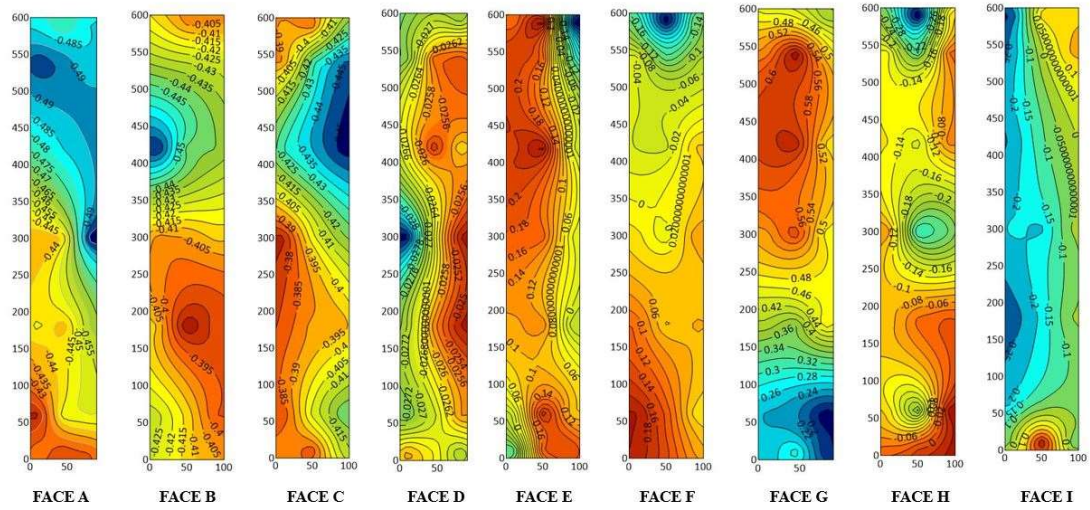
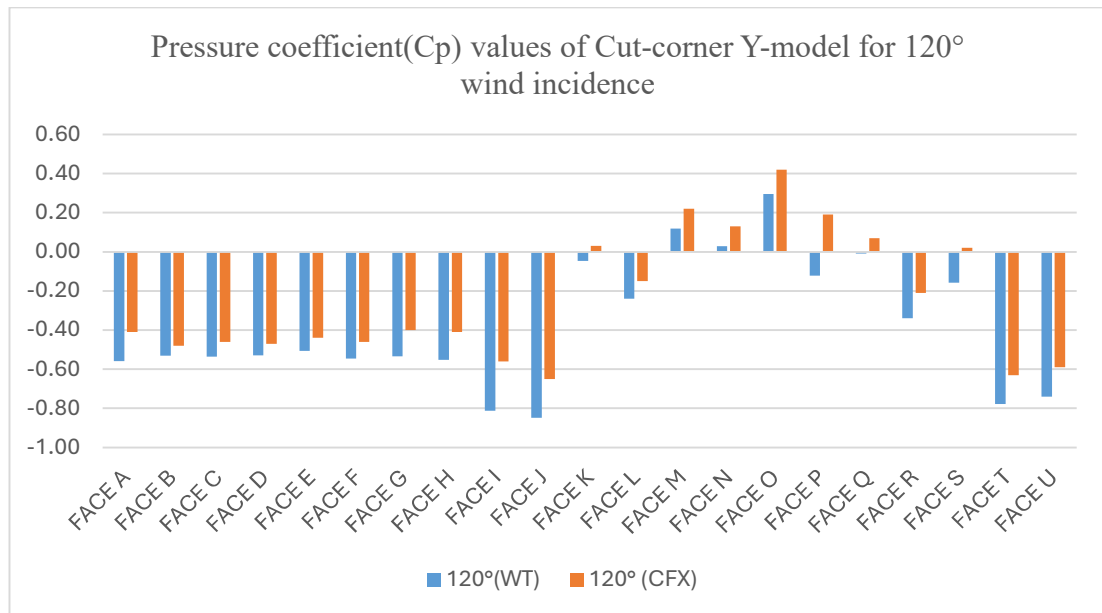


Fig.5.10: Pressure contours of Standard Y-model at 120°-wind incidence

Cut Corner Y-Model:

- Symmetry is maintained in the flow dynamics.
- The recessed faces (M, N, O) are stagnation areas with positive pressure, while the remaining part of the building develops suction.
- The consistency between CFX and WT simulation results in this case is extremely high, with the errors well within acceptable limits for engineering purposes.



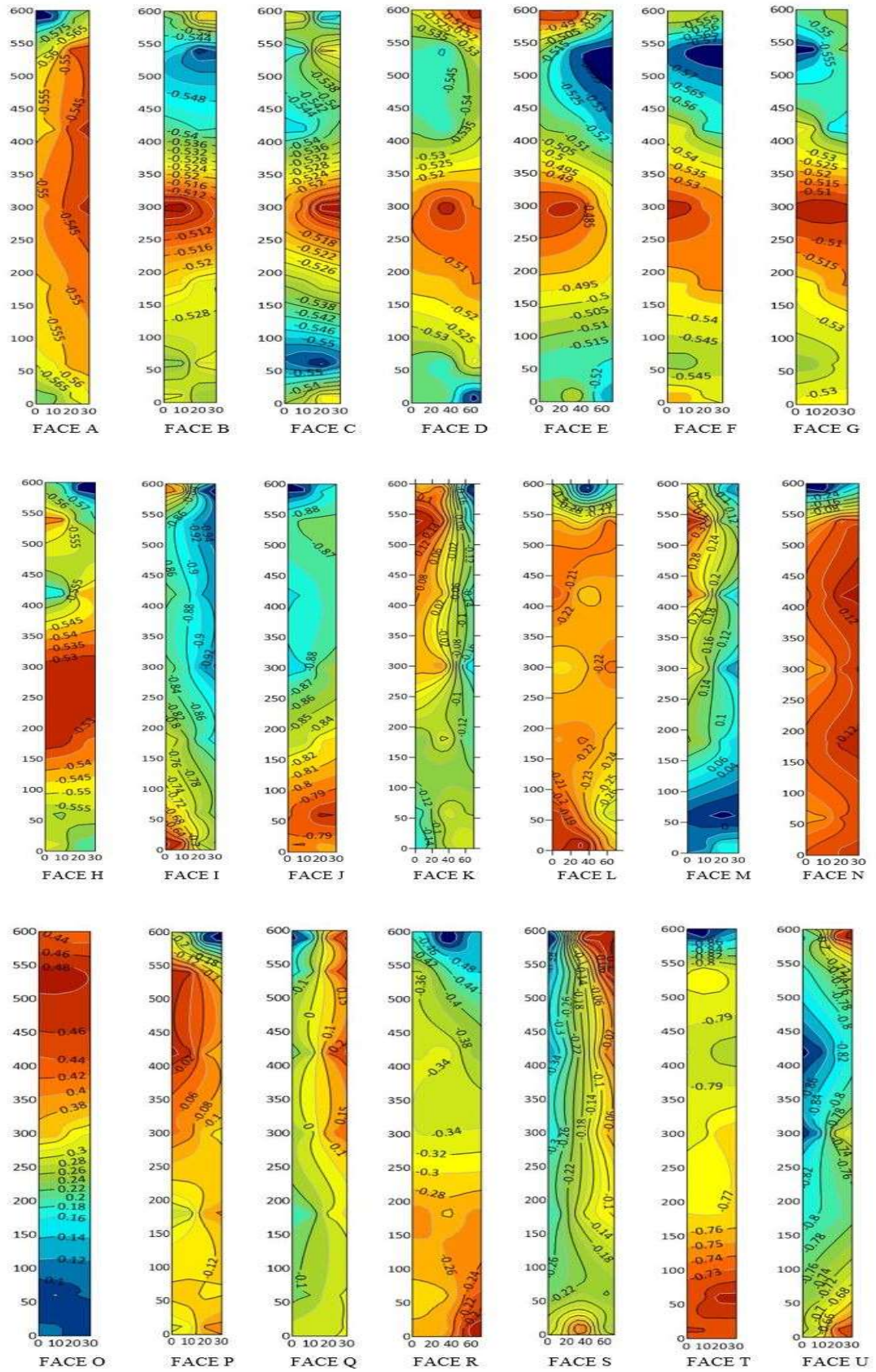


Fig.5.11: Pressure contours of Cut-Corner Y-model at 120° -wind incidence

Wind Incident Angle: 150° (Asymmetrical mirror of 30°)

This wind incidence angle mirrors the phenomena occurring at 30 degrees.

Standard Y-Model:

- Positive pressure concentration occurs on Faces E and F (0.42 and 0.53 WT).
- Strong negative pressure occurs on the re-entrant faces H and I (-0.68 and -0.59 WT).

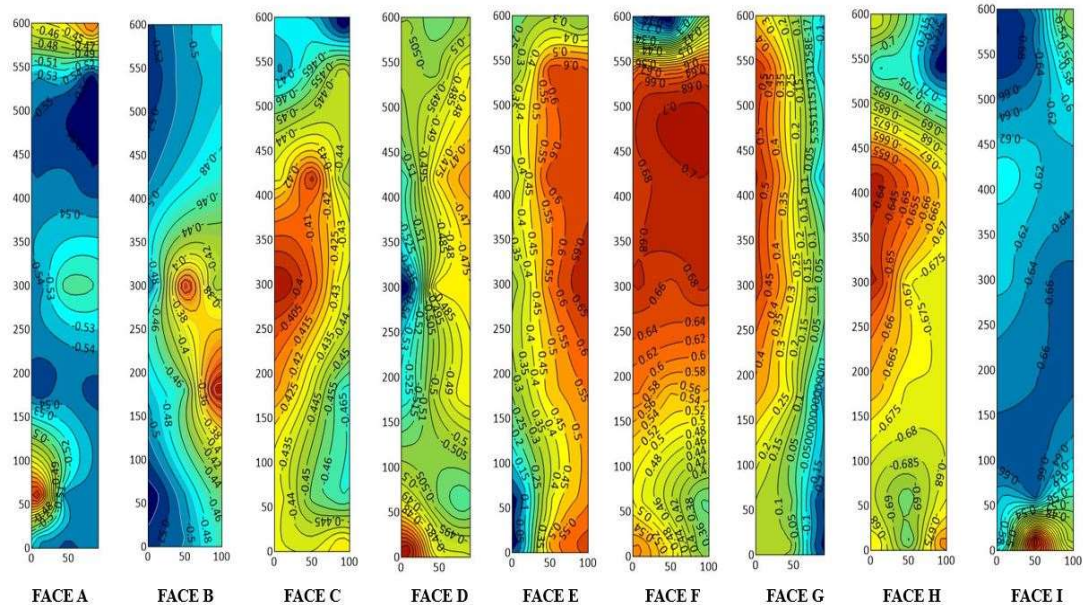
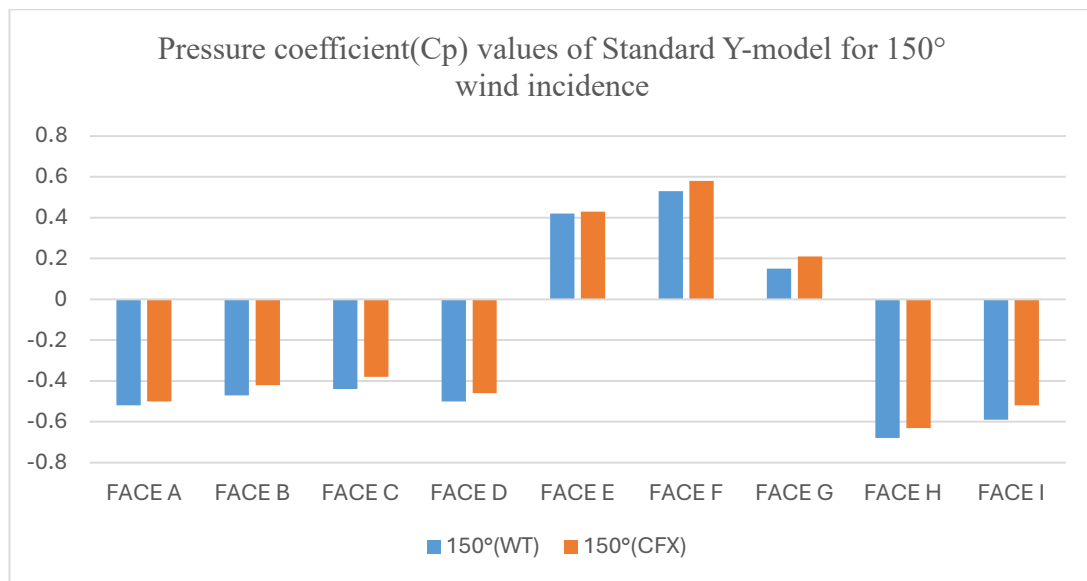
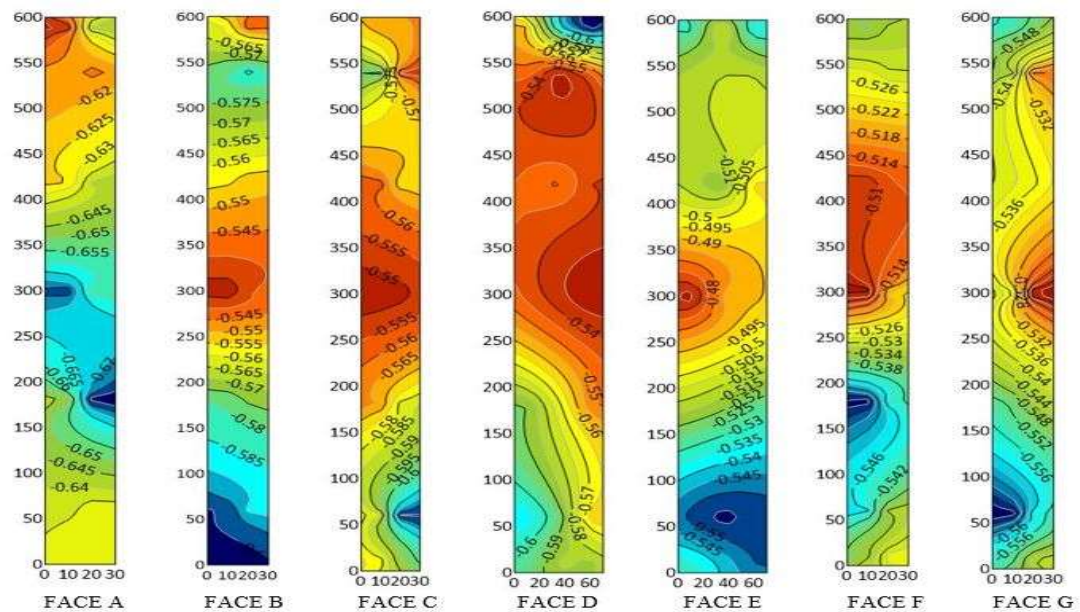
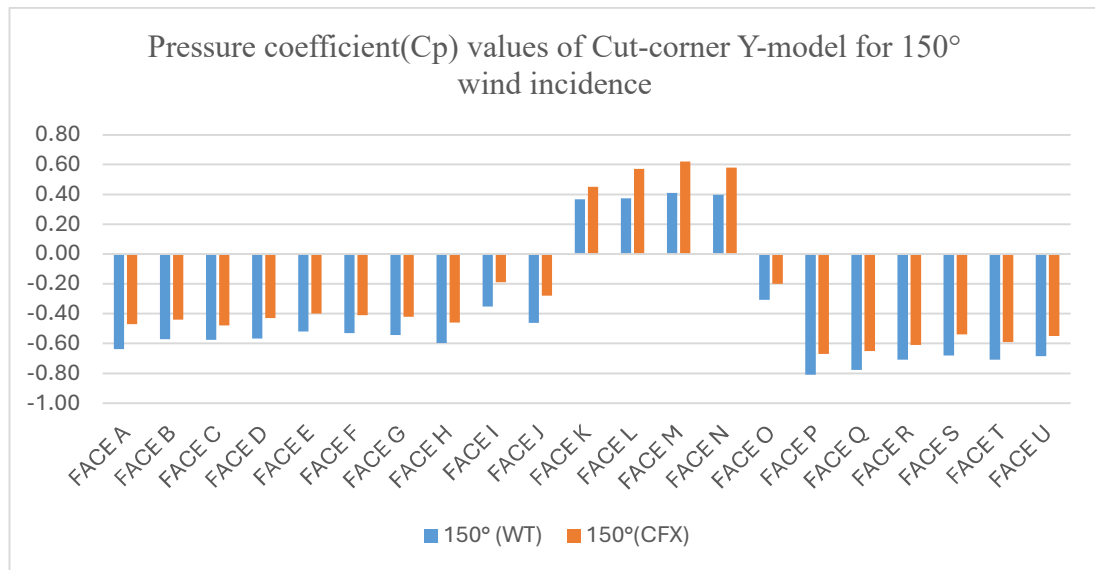


Fig.5.12: Pressure contours of Standard Y-model at 150°-wind incidence

Cut-Corner Y-Model:

- Faces K, L, M, and N experience the maximum positive pressure (C_p values ranging between 0.37 to 0.41).
- In similar manner to the 30° -wind incidence angle, the cut corners lead to detachment of the shear layer and peak suction occurs at the surrounding faces from Face P to Face U (ranging between -0.68 and -0.81 in WT).



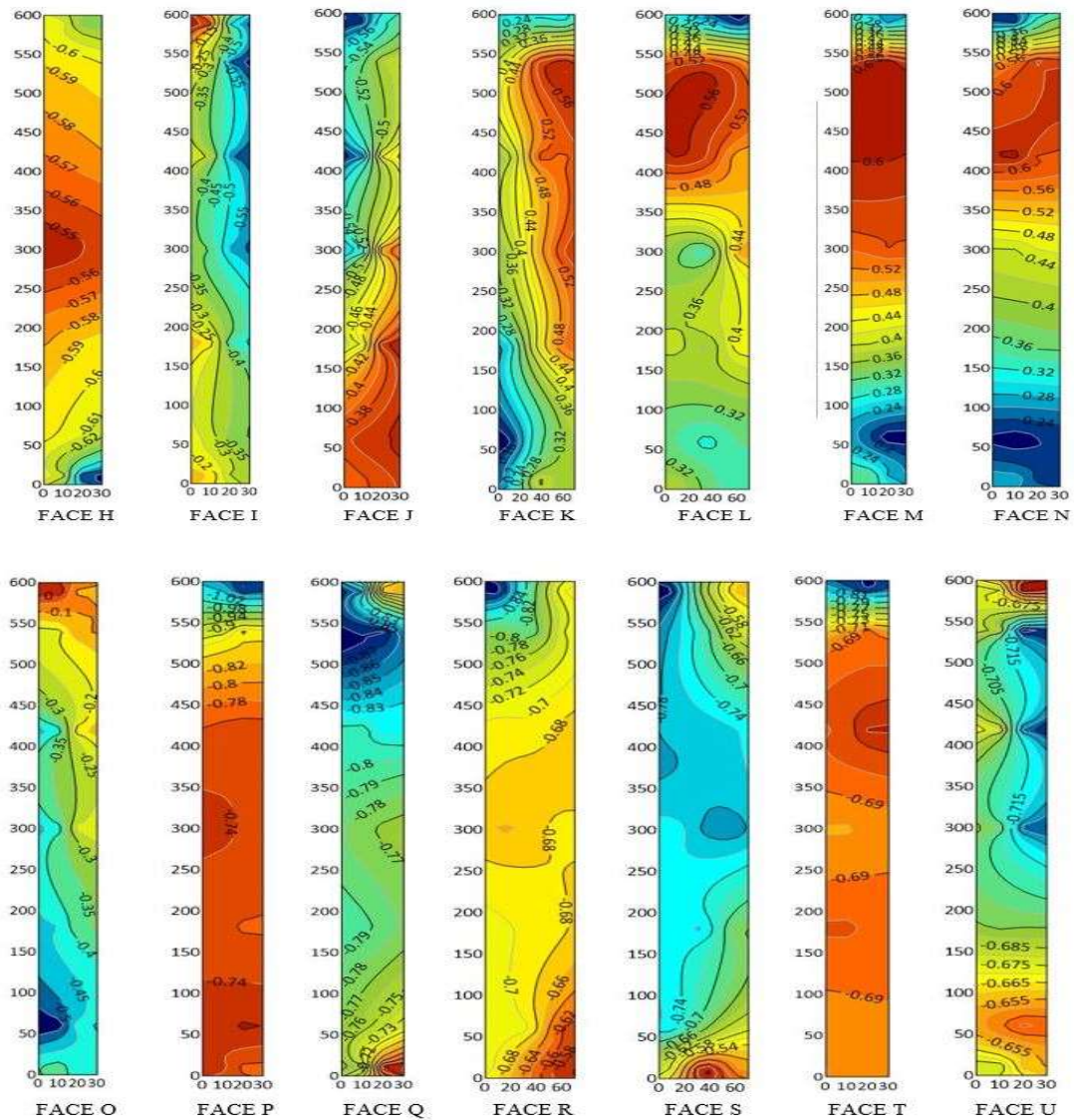


Fig.5.13: Pressure contours of Cut-Corner Y-model at 150°-wind incidence

Wind Incidence Angle: 180° (Inner Cavity V-Trap)

At 180° wind incidence, the wind hits the building perfectly symmetrical but right into the V-shape re-entrant corner created by two limbs, trapping the air.

Standard Y-Model:

- The wind is trapped by Faces E and F (C_p values 0.49 and 0.47 in WT; 0.53 in CFX).
- The flow then is directed outward due to the sharp extremities, resulting in strong suction of the outer trailing faces G (-0.64 WT), H (-0.53 WT), and I (-0.52 WT).

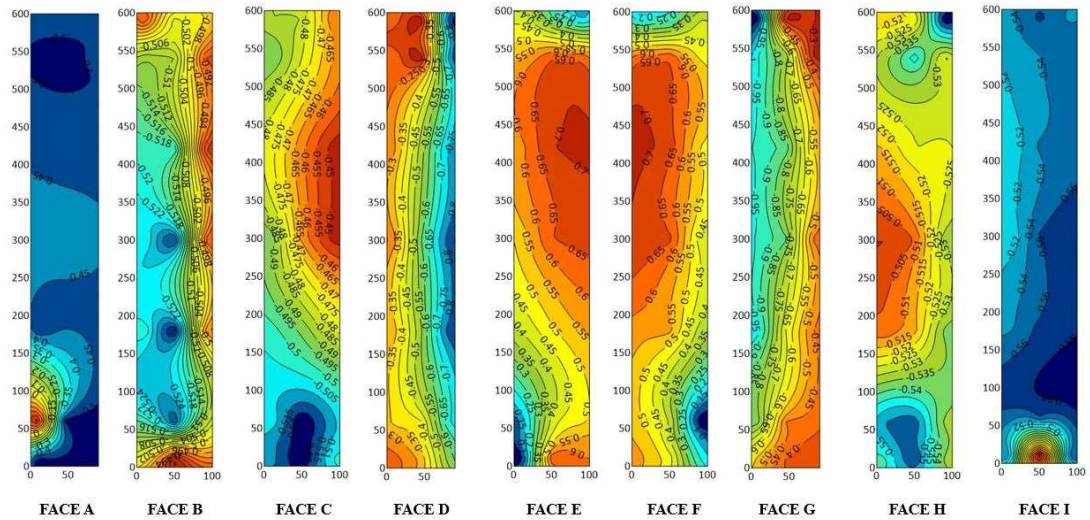
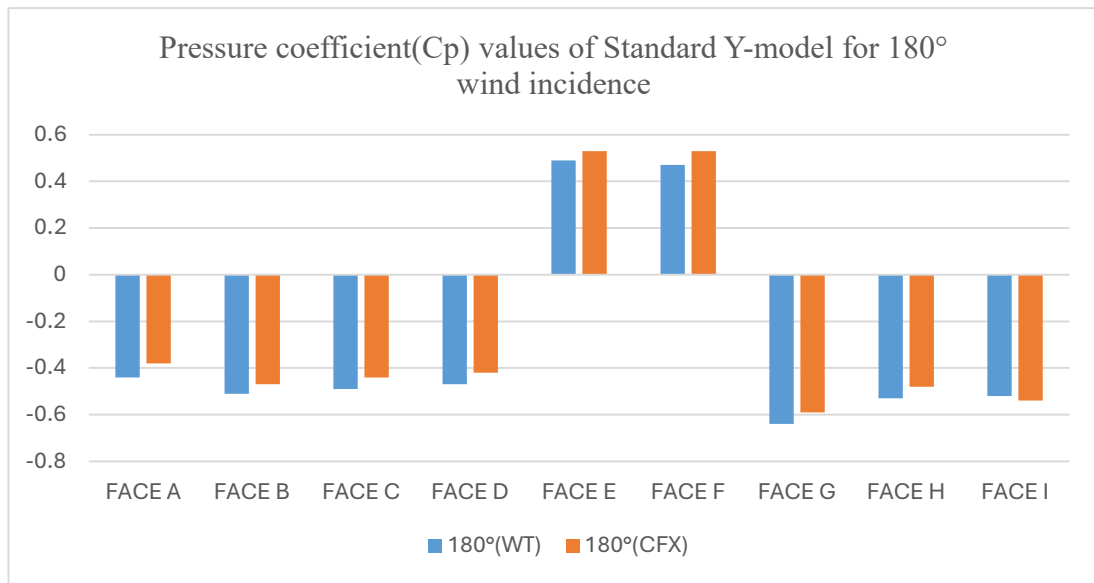
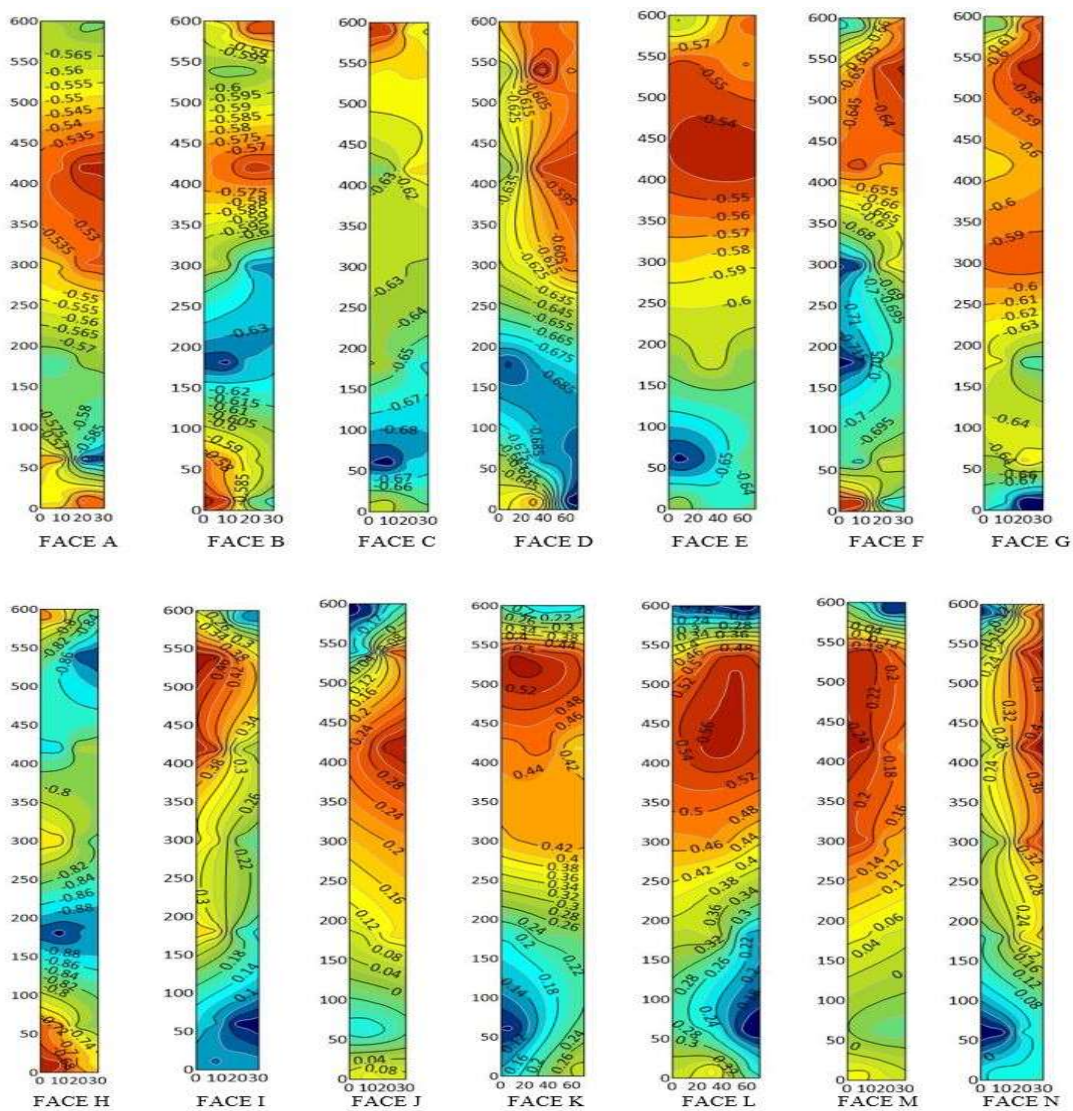
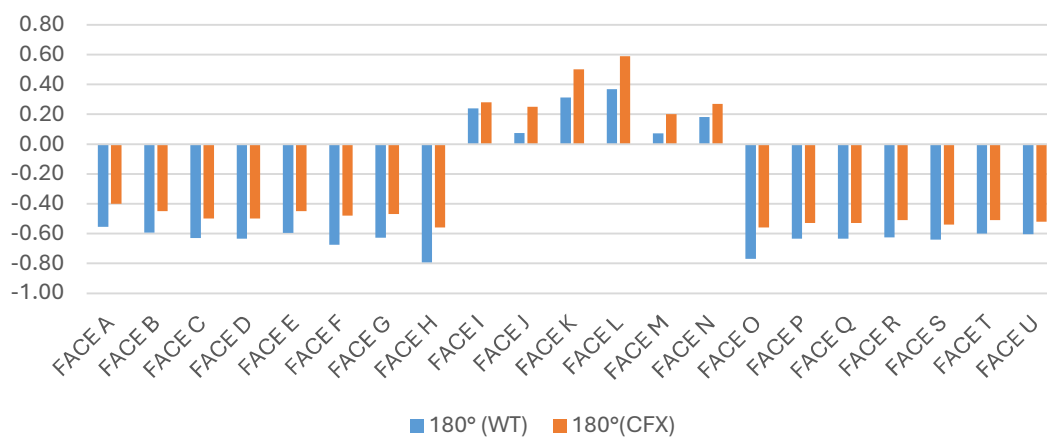


Fig.5.14: Pressure contours of Standard Y-model at 180° -wind incidence

Cut-Corner Y-Model:

- It is precisely at this incidence angle that the cut corners model demonstrates its true advantages.
- Eliminating the sharp trailing extremities, the model allows the trapped air to bleed off.
- Faces I, J, K, L, M and N are positively charged in the wind tunnel (ranging from 0.07 to 0.37), confirming the ability of the cut corners to break the pressure concentration zone.
- Negative pressure ranges between -0.60 to -0.79.

Pressure coefficient(C_p) values of Cut-corner Y-model for 180° wind incidence



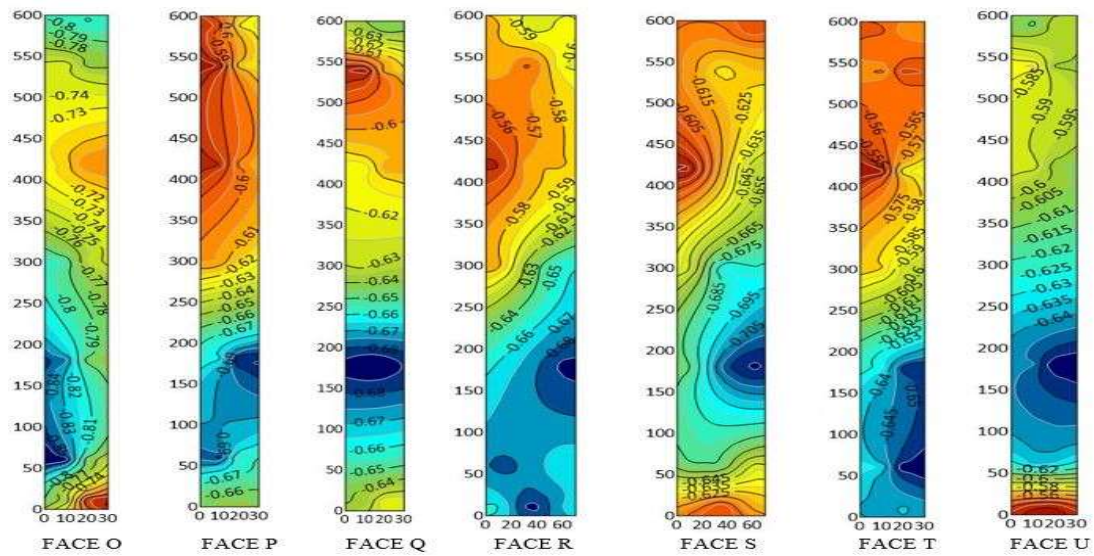


Fig.5.15: Pressure contours of Cut-Corner Y-model at 180°-wind incidence

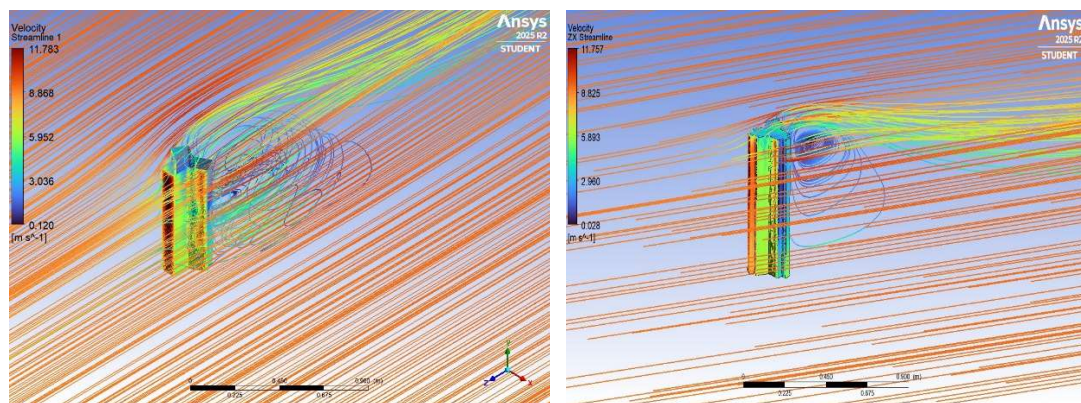
5.4 Velocity Streamlines

- Velocity streamlines are the best tool to depict wind flow around a structure in detail.
- Here, through comparing streamlines of standard and cut-corner model the wind flow mechanism can be understood.
- For each angle of attack the streamlines show less violent wind behavior in case of cut-corner model as compared to the standard y model.
- The cut-corner model successfully mitigates the large eddy formations behind the standard y model.
- The abrupt flow separation due to the sharp-edged extremities of the standard model is countered by the modified corners of the cut-corner model.

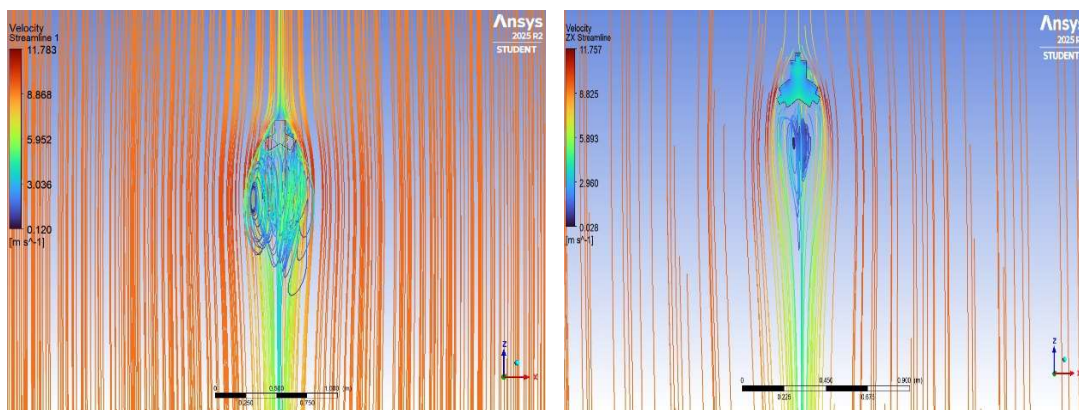
0° wind angle of attack

Standard Y-model

Cut-corner Y-model



Isometric View

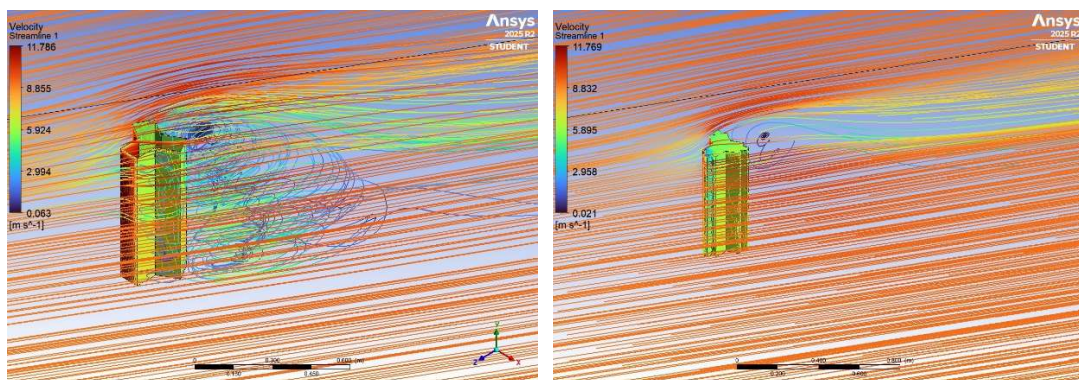


Bottom View

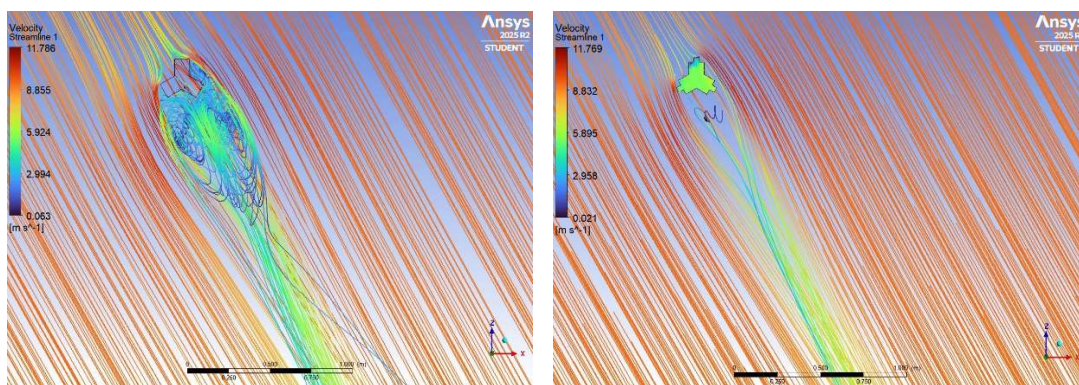
30° wind angle of attack

Standard Y-model

Cut-corner Y-model



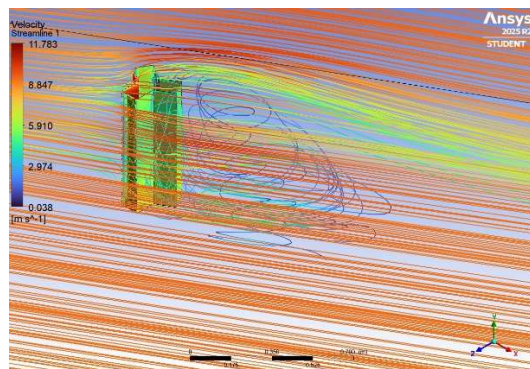
Isometric View



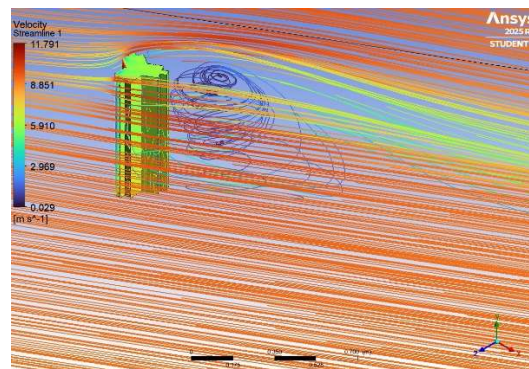
Bottom View

60° wind angle of attack

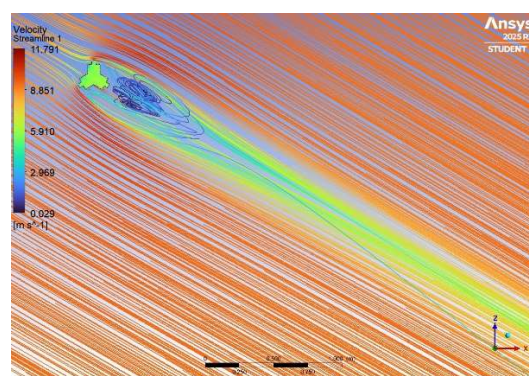
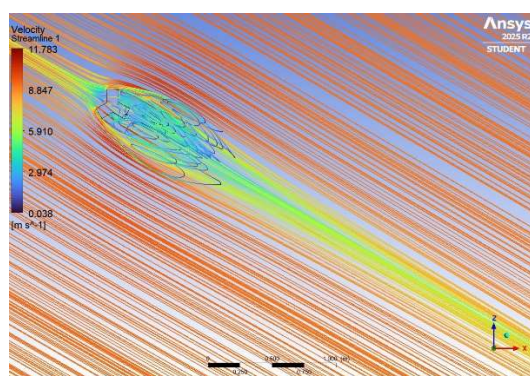
Standard Y-model



Cut-corner Y-model



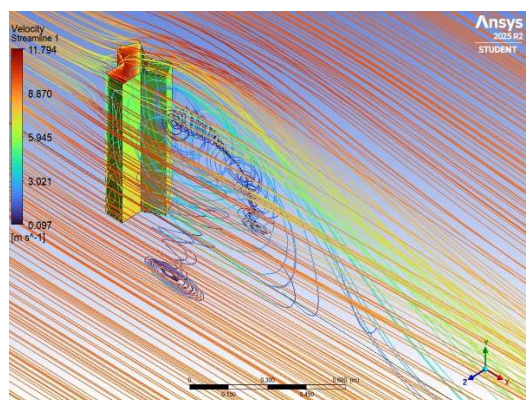
Isometric View



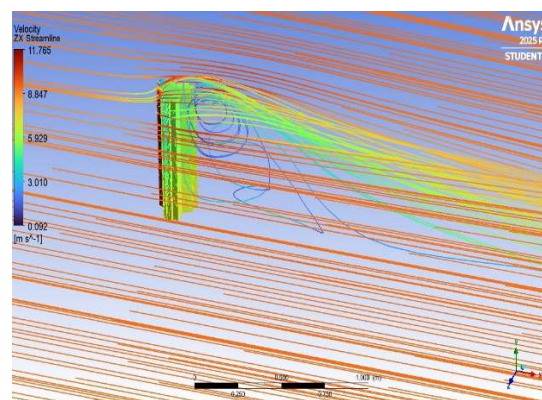
Bottom View

90° wind angle of attack

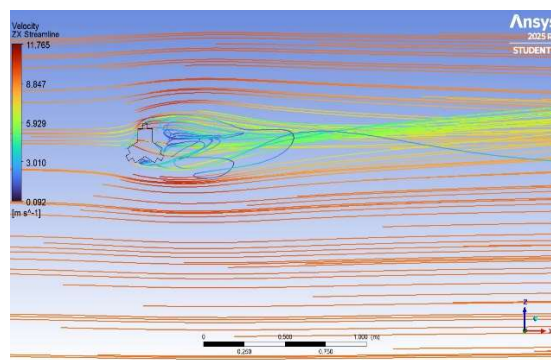
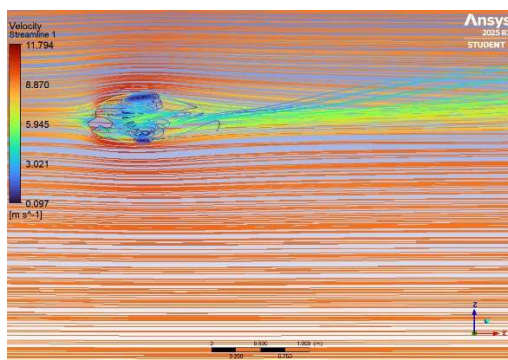
Standard Y-model



Cut-corner Y-model



Isometric View

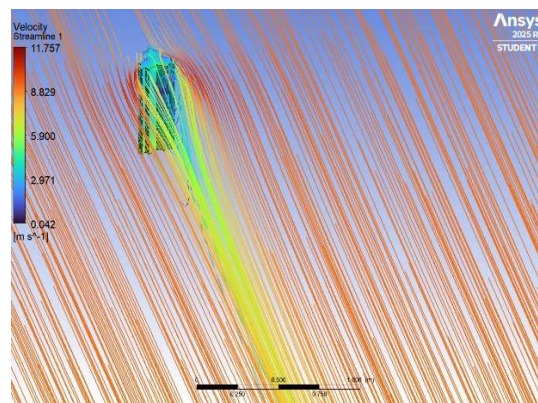
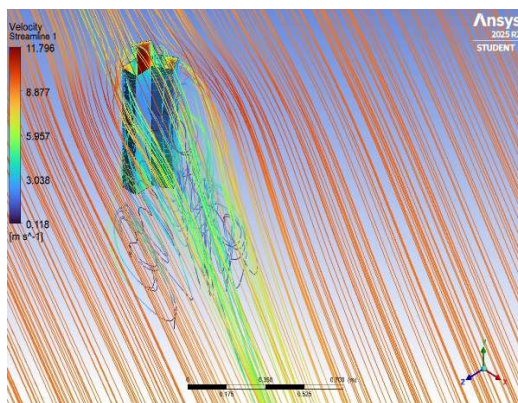


Bottom View

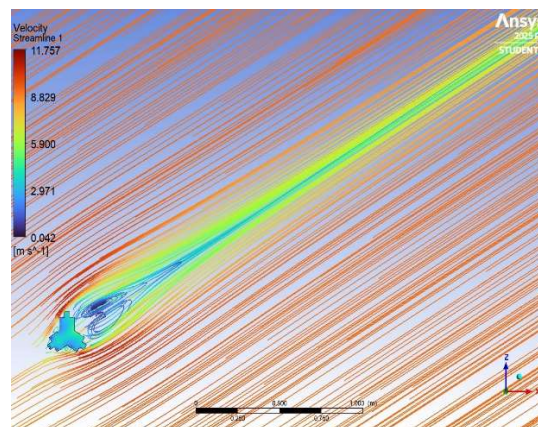
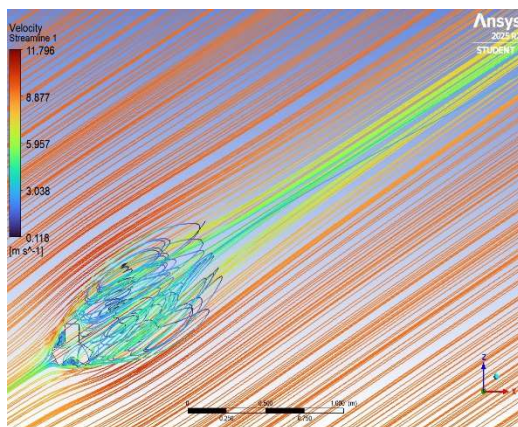
120° wind angle of attack

Standard Y-model

Cut-corner Y-model



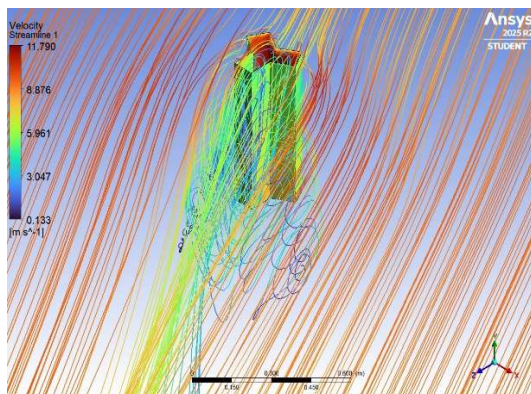
Isometric View



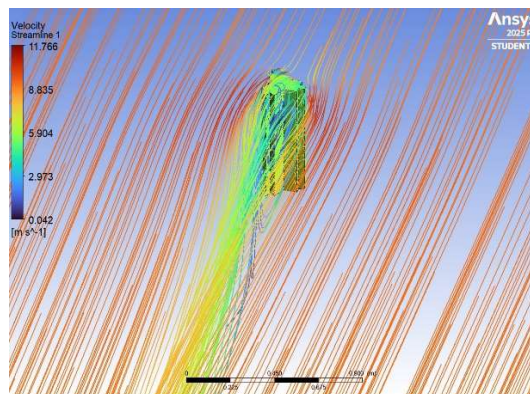
Bottom View

150° wind angle of attack

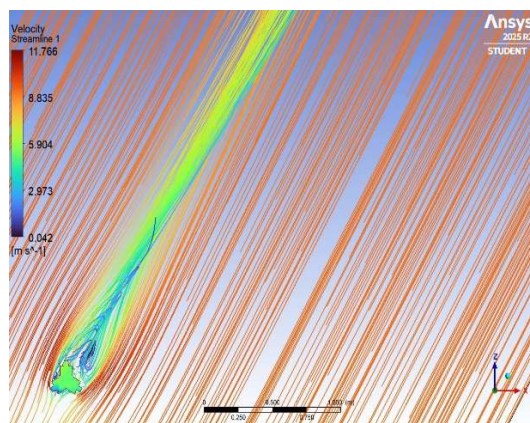
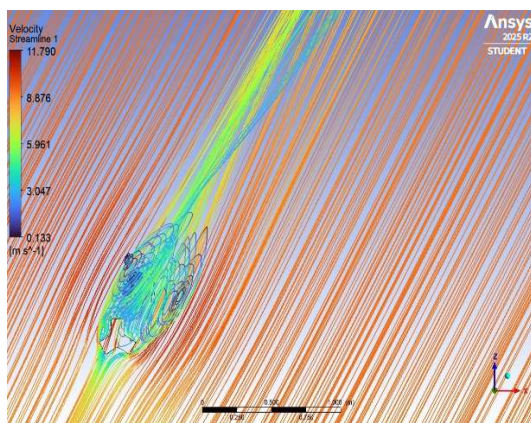
Standard Y-model



Cut-corner Y-model



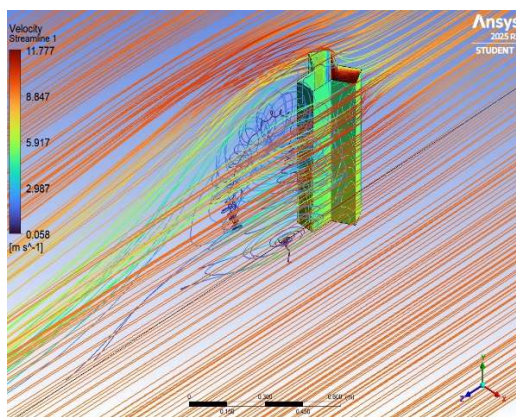
Isometric View



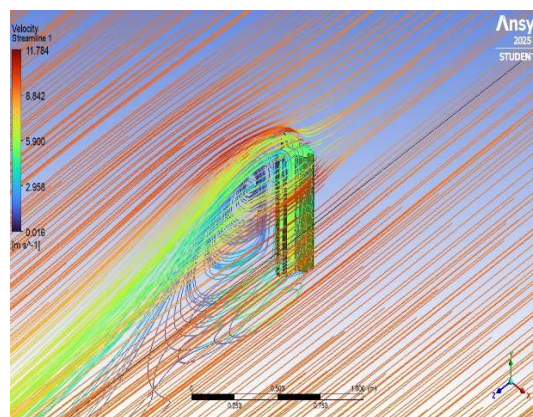
Bottom View

180° wind angle of attack

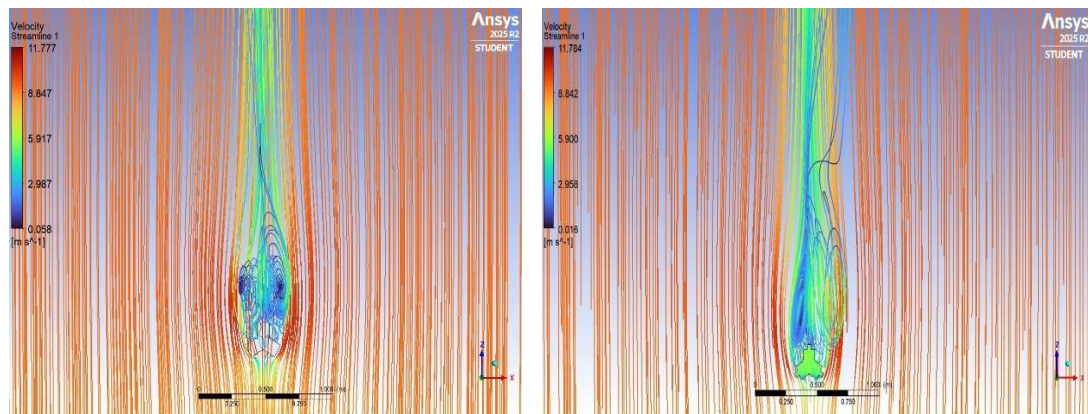
Standard Y-model



Cut-corner Y-model



Isometric View



Bottom View

5.5 Aerodynamic Mitigation Summary

- Mitigation of trapped flow: The cut-corner design was effective at every angle of attack (especially 60° and 180°). By developing local cushions aerodynamically at the ends of the three limbs, the recessed corners helped prevent the formation of destructive trapped flow within the re-entrant corners, resulting in distribution of positive pressure evenly along the surface of structure.
- Localized suction disadvantage: Even though the overall aerodynamics are stabilized, the data shows that at critical angles of asymmetry (30° and 150°), the cut corners create conditions for accelerated shear layer separation. As a result, these wind incidence angles developed increased localized negative pressures immediately downstream of the cut-corner. This is an important conclusion to make, since although the structural framework will benefit from corner cuts, the cladding design needs extra reinforcement in those particular areas to counter the higher suction developed.

CHAPTER 6

CONCLUSION AND FUTURE SCOPE

6.1 Research Summary

This study provides a detailed aerodynamic analysis of wind interaction with Y-shaped high-rise structures. The tri-axially symmetric shape of the Y-plan imparts many structural benefits; thus, this study focuses on its aerodynamic drawbacks the occurrence of severe flow separation in sharp extremities and formation of air trapping zones in inner re-entrant corner region. As a remedy for the above-mentioned extreme wind effects, a cut corner geometric design was developed and analyzed. A hybrid approach was adopted to investigate the aerodynamics of the Y-plan building, where experimental tests using the Boundary Layer Wind Tunnel (BLWT) of 1:150 scale rigid Perspex models were conducted along with CFD numerical simulations based on the Reynolds-Averaged Navier-Stokes (RANS) equations and employing the $k-\epsilon$ turbulence model which was implemented through ANSYS CFX.

6.2 Principal Findings

Based on the rich experimental evidence and numerically validated visualization, the following conclusions are stated about the aerodynamic performance and effectiveness of the cut-corner modification:

- High accuracy of computational flow dynamics: It was evident from the obtained CFD results that the steady-state computational simulation exhibited high accuracy when compared with wind tunnel data at all considered wind incidence angles. The standard $k-\epsilon$ turbulence model used in combination with the controlled meshing procedure for the inflation layer was able to reproduce accurately the flow separation regions, wake zones and pressure stagnation areas. Thus, the computational approach can be considered as a precise predictor of aerodynamic characteristics of bluff bodies.
- Mitigation of high positive surface pressures: The introduction of recessed corners successfully mitigated the concentration of positive pressure concentration on the windward faces of the building. By providing a geometric relief zone that diffuses the direct impact of airflow, the cut-corner Y-model demonstrated a quantifiable reduction in peak positive surface pressures by approx. 10% to 15% across the wind incidence angles.
- Reduction of wind-trapping in the cavity area: In certain wind incidence angles, especially at 180° (where the wind impacts into the V-cavity between two limbs), the standard sharp-edged Y-model showed profound pressure stagnation that caused an unbalanced suction effect on the trailing faces of the structure. The cut-corner modification has fundamentally solved this problem by switching the sharp

trailing edges with the recessed corner design that provides an aerodynamic cushion for efficient draining of the trapped air.

- Earlier controlled separation of shear layer and narrowing of wake width: Due to the inclusion of the cut corners, the detachment of the shear layer took place earlier and in a more controlled way, compared to the standard model with sharp corners having abrupt violent flow separation. Instead of one big separation, the cut-corner design created smaller separation areas due to the formation of secondary stagnation points on the cut faces. As a result, the width of the highly turbulent wake behind the structure was decreased, which led to the decrease in the total along-wind force.
- The amplification of suction effect: At angles of 90° and 150° , there appeared to be a major aerodynamic penalty for cut-corners in terms of the aerodynamic stability. Although the cut corner design was successful in reducing pressure concentrations, the flow separation created a strong suction effect on the sides of the building located immediately after the cut-corner.

6.3 Engineering and Design Implications

The results from this study provide very practical recommendations for engineers in designing unusual tall buildings:

- Global vs. Local Design Considerations: The use of cut-corner (recessed-corner) is recommended as a technique for mitigating severe pressure stagnation zones and overturning wind loads, thus improving the performance of the primary lateral load resisting system (like core walls).
- Special Cladding Design: Since the cut corner increases suction at oblique angles, facade design engineers should not employ uniform cladding thickness throughout the building. Glass and cladding fixings situated in the vicinity of the recessed corners, especially in the inner re-entrant limbs, need to be strengthened for extreme high negative pressure.

6.4 Recommendations for Future Research

This study has successfully mapped the pressure distribution and force mitigation for the standard and cut corner Y-shaped building, but the field of wind engineering is vast and ever-evolving. So, in order to enhance knowledge about this particular structure, the following areas of research are proposed:

- Aeroelastic Dynamic Wind Tunnel Testing: The present study employed rigid models in order to obtain the static pressure coefficients. It is recommended that future studies make use of flexible multi-degree of freedom models that will help understand the wind-structure interactions, especially across wind resonance, vortex induced vibration and aerodynamic damping in detail.

- **Interference Effects Evaluation:** The present study was conducted on an isolated condition. However, in the real-world scenario of metro cities, tall buildings are built in clusters. Therefore, future studies should investigate how interference effects such as shielding and amplification from other adjacent buildings affect the efficiency of cut corner modification.
- **Aerodynamic Optimization Using Machine Learning:** The current study focused on a single, fixed cut corner design. Future numerical simulations may integrate CFD modeling with ANN and genetic algorithms for a huge parametric optimization exercise, whereby different percentages of the cut corner (such as 5%, 10%, 15% of the limb width) would be varied to identify the precise mathematical solution that reduces drag and torsional forces.

REFERENCES

- [1] N. Singh and A. Verma, "Reducing wind load on tall buildings: A review," *International Journal of Structural Design and Engineering*, vol. 6, no. 1, 2025, doi: 10.22271/27078280.2025.v6.i1a.40.
- [2] N. Sharma, H. Yadav, and A. K. Roy, "Wind-driven oscillation and dynamic response of a Y-plan shaped tall building under interference," *Structures*, vol. 70, 2024, doi: 10.1016/j.istruc.2024.107619.
- [3] H. Verma and R. S. Sonparote, "Forecasting Aerodynamic Coefficients of Bi-Axial Symmetric C Plan-Shaped Tall Buildings Using ANFIS," *KSCE Journal of Civil Engineering*, vol. 28, no. 6, 2024, doi: 10.1007/s12205-024-0982-y.
- [4] N. T. Yu, W. H. Li, Y. Zheng, and W. B. Yuan, "Wind-Induced Aerodynamic Effects on Set-Back Tall Buildings Using LES," *Buildings*, vol. 14, no. 5, May 2024, doi: 10.3390/buildings14051252.
- [5] M. Aldoum, T. Stathopoulos, M. Chavez, and A. Baskaran, "Wind loading on a stepped roof building: Comparison of field measurements, wind tunnel data, and standard provisions," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 238, 2023, doi: 10.1016/j.jweia.2023.105441.
- [6] A. H. H. Al-Masoodi, Y. M. Abbas, F. Alkhatib, M. I. Khan, N. Shafiq, and M. ElGawady, "Aerodynamic optimization for corner modification of octagonal-shape tall buildings using computational approach," *Journal of Building Engineering*, vol. 76, 2023, doi: 10.1016/j.jobbe.2023.107017.
- [7] A. H. H. Al-Masoodi, F. H. Alkhatib, N. Shafiq, and M. M. A. Wahab, "The Aerodynamic Performance of Tall Buildings by Utilizing Aerodynamic Modifications - A Review Study," *IOP Conference Series: Earth and Environmental Science*, vol. 1022, no. 1, 2022, doi: 10.1088/1755-1315/1022/1/012046.
- [8] N. Gaur and R. Raj, "Aerodynamic mitigation by corner modification on square model under wind loads employing CFD and wind tunnel," *Ain Shams Engineering Journal*, vol. 13, no. 1, 2022, doi: 10.1016/j.asej.2021.06.007.
- [9] P. K. Goyal, S. Kumari, S. Singh, R. K. Saroj, R. K. Meena, and R. Raj, "Numerical Study of Wind Loads on Y Plan-Shaped Tall Building Using CFD," *Civil Engineering Journal*, vol. 8, no. 2, Feb. 2022, doi: 10.28991/CEJ-2022-08-02-06.

- [10] A. Kumar, R. Meena, and A. Verma, "Computational Fluid Dynamics (CFD) and Wind Tunnel Study for Wind Load on High Rise Buildings: A Review," *YMER*, vol. 21, no. 12, 2022, doi: 10.37896/YMER21.12/87.
- [11] R. K. Meena, R. Raj, and S. Anbu Kumar, "Wind Excited Action around Tall Building Having Different Corner Configurations," *Advances in Civil Engineering*, vol. 2022, Feb. 2022, doi: 10.1155/2022/1529416.
- [12] R. K. Meena, R. Raj, and S. Anbu Kumar, "Effect of wind load on irregular shape tall buildings having different corner configuration," *Sādhana*, vol. 47, 2022, doi: 10.1007/s12046-022-01895-2.
- [13] E. Rajabi, H. Sadeghi, and M. R. Hashemi, "Wind effect on building with Y-shaped plan," *Asian Journal of Civil Engineering*, vol. 23, no. 1, Jan. 2022, doi: 10.1007/s42107-022-00417-z.
- [14] P. Sanyal and S. K. Dalui, "Forecasting of aerodynamic coefficients of tri-axially symmetrical Y plan shaped tall building based on CFD data trained ANN," *Journal of Building Engineering*, vol. 47, Apr. 2022, doi: 10.1016/j.jobbe.2021.103889.
- [15] K. N. Srivastava and R. Raj, "Aerodynamic study of tall buildings under wind load," *International Journal of Health Sciences*, vol. 6, no. S3, 2022, doi: 10.53730/ijhs.v6nS3.6142.
- [16] R. K. Meena, R. Raj, and S. Anbu Kumar, "Analysis of Wind on Different Shape of High-Rise Structure," *International e-Conference on Sustainable Development & Recent Trends in Civil Engineering*, Jan. 2022.
- [17] A. K. Bairagi and S. K. Dalui, "Estimation of Wind Load on Stepped Tall Building Using CFD Simulation," *Iranian Journal of Science and Technology - Transactions of Civil Engineering*, vol. 45, no. 2, 2021, doi: 10.1007/s40996-020-00535-1.
- [18] P. Sanyal and S. K. Dalui, "Effects of internal angle between limbs of 'Y' plan shaped tall building under wind load," *Journal of Building Engineering*, vol. 33, 2021, doi: 10.1016/j.jobbe.2020.101843.
- [19] P. Sanyal and S. K. Dalui, "Effects of side ratio for 'Y' plan shaped tall building under wind load," *Building Simulation*, vol. 14, no. 4, 2021, doi: 10.1007/s12273-020-0731-1.
- [20] R. Raj, T. Rana, T. Anchalia, and U. Khola, "Numerical Study of Wind Excited Action on H Plan-Shaped Tall Building," *International Journal on Emerging Technologies*, vol. 11, no. 3, 2020.

- [21] P. Sanyal and S. K. Dalui, "Comparison of aerodynamic coefficients of various types of Y-plan-shaped tall buildings," *Asian Journal of Civil Engineering*, vol. 21, no. 7, 2020, doi: 10.1007/s42107-020-00265-9.
- [22] M. Mallick, A. Kumar, and K. C. Patra, "Experimental Investigation on the Wind-Induced Pressures on C-Shaped Buildings," *KSCE Journal of Civil Engineering*, vol. 23, no. 8, 2019, doi: 10.1007/s12205-019-1929-6.
- [23] G. Zu and K. M. Lam, "LES and wind tunnel test of flow around two tall buildings in staggered arrangement," *Computation*, vol. 6, no. 2, 2018, doi: 10.3390/computation6020028.
- [24] R. Ahlawat and A. K. Ahuja, "Wind Loads on 'T' Plan Shape Tall Buildings," *Journal of Academia and Industrial Research (JAIR)*, vol. 4, no. 1, 2015.
- [25] R. Ahlawat and A. K. Ahuja, "Wind loads on 'Y' plan shape tall building," *International Journal of Engineering and Applied Sciences (IJEAS)*, vol. 2, no. 4, Apr. 2015.
- [26] A. Elshaer, G. Bitsuamlak, and A. El Damatty, "Aerodynamic shape optimization for corners of tall buildings using CFD," *14th International Conference on Wind Engineering (ICWE14)*, Jun. 2015.
- [27] S. K. Dalui, A. K. Ahuja, and S. Mukherjee, "Wind induced pressure on Y-shaped tall building," *Wind and Structures*, vol. 19, no. 5, (2014), doi: <http://dx.org/10.12989/was.2014.19.5.523>.
- [28] R. Raj and A. K. Ahuja, "Wind Loads on Cross Shape Tall Buildings," *Journal of Academia and Industrial Research (JAIR)*, vol. 2, no. 2, Jul. 2013.
- [29] P. A. Irwin, "The Wind Engineering of the Burj Dubai Tower," *CTBUH 2005 7th World Congress*, 2005.
- [30] J. Franke, et al., "Recommendations on the use of CFD in wind engineering," *COST Action 732*, 2004.
- [31] Bureau of Indian Standards, *Design Loads (Other than Earthquake) for Buildings and Structures—Code of Practice, Part 3: Wind Loads*, IS 875 (Part 3): 2015, New Delhi, India, 2015.
- [32] American Society of Civil Engineers, *Minimum Design Loads and Associated Criteria for Buildings and Other Structures*, ASCE/SEI 7-16, Reston, VA, USA, 2017.
- [33] European Committee for Standardization, *Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions*, EN 1991-1-4:2005, Brussels, Belgium, 2005.

- [34] Standards Australia and Standards New Zealand, *Structural design actions - Part 2: Wind actions*, AS/NZS 1170.2:2021, Sydney, NSW, Australia and Wellington, New Zealand, 2021.
- [35] R. Raj, “Effects of cross-sectional shapes on response of tall buildings under wind loads,” PhD. Thesis, Department of Civil Engineering., Indian Institute of Technology Roorkee, Roorkee, India, 2015.
- [36] P. Krishna, K. Kumar, and N. M. Bhandari, “Explanatory Handbook on Indian Standard IS 875 (Part 3): Wind Loads on Buildings and Structures,” *Indian Institute of Technology Roorkee*, Roorkee, India.

LIST OF RESEARCH PAPERS

[1] Singh, S., Srivastava, K.N., Raj, R. (2026), “Wind Load Effects on Y-Shaped High-Rise Building: Comparative Study of Standard and Cut- Corner Modification with Interference using Ansys CFX”- National Conference on Innovative Construction Machineries, Materials and Methods for Infrastructure development (ICMMMID-2026) held on February 9th – 10th, 2026 at Dehradun, Uttarakhand.

National Conference

INNOVATIVE CONSTRUCTION MACHINERIES, MATERIALS AND METHODS FOR INFRASTRUCTURE DEVELOPMENT (ICMMMID-2026)

February 9-10, 2026

Hotel Fairfield by Marriott, Mussoorie Road, Dehradun (Uttarakhand)



CERTIFICATE OF PARTICIPATION

This is to certify that **Mr. Shreyansh Singh**, Deptt. of Civil Engg., Delhi Technological University, Delhi, participated in the Conference ICMMMID-2026 and presented the paper titled “**WIND LOAD EFFECTS ON Y-SHAPED HIGH-RISE BUILDING: COMPARATIVE STUDY OF STANDARD AND CUT-CORNER MODIFICATION WITH INTERFERENCE USING ANSYS CFX**”.

(Dr. Ashok K. Ahuja)
Convener

(Dr. R. Siva Chidambaram)
Organizing Secretary

[2] Singh, S., Srivastava, K.N., Raj, R. (2026), “Wind Tunnel and CFD Analysis of Y-Shaped Tall Building with Cut-Corner Modification”- 1st International Conference on Urban Built Environment (ICUBE-2026) held on March 27th – 28th, 2026 at NICMAR Delhi-NCR Campus.



NICMAR
Delhi-NCR
— APPROVED BY AICTE —

ICUBE-2026



1ST INTERNATIONAL CONFERENCE ON URBAN BUILT ENVIRONMENT
BRIDGING ACADEMIA, RESEARCH AND PRACTITIONERS FOR SUSTAINABLE CONSTRUCTION

142

Certificate of Paper Presentation

This is to certify that

Dr./Mr./Ms. Shreyansh Singh

of Delhi Technological University

has presented a paper entitled Wind tunnel and CFD analysis of y-shaped tall building with cut-corner modification

in the **1st International Conference on Urban Built Environment (ICUBE 2026)**, held on **27th – 28th March, 2026** at NICMAR Delhi-NCR Campus.

Academic Partners



Sponsored by

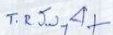


Knowledge Partners



Publication Partners





Dr. Praveenkumar T. R.
Co-Convenor, ICUBE-2026
Head, Innovation and Incubation



Dr. Nakul Gupta
Convener, ICUBE-2026
Dean-Research & Development



Prof. (Dr.) Rajesh Goyal
Chairperson, ICUBE-2026
Director and Professor

COMPARATIVE ANALYSIS OF CUT-CORNER MODIFICATION ON AERODYNAMIC PERFORMANCE OF Y-SHAPED TALL BUILDING: A COMBINED WIND TUNNEL AND CFD SIMULATION STUDY

ORIGINALITY REPORT



4%

INTERNET SOURCES

6%

PUBLICATIONS

2%

STUDENT PAPERS

PRIMARY SOURCES

1 Rajdip Paul, Sujit Dalui. "Shape Optimization to Reduce Wind Pressure on the Surfaces of a Rectangular Building with Horizontal Limbs", Periodica Polytechnica Civil Engineering, 2020 Publication 2%

2 researchtrend.net Internet Source 1%

3 ebin.pub Internet Source <1%

4 Submitted to Sharda University Student Paper <1%

5 Himanshoo Verma, Ranjan S. Sonparote. "Forecasting Aerodynamic Coefficients of Bi-Axial Symmetric C Plan-Shaped Tall Buildings Using ANFIS", KSCE Journal of Civil Engineering, 2024 Publication <1%

6 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%