Case Study of Analysis of Wind Load on U and V Shape High Rise Building: A Review

Ritik Chaudhary[1*], Astha Verma[2]

[1,2]*Department of Civil Engineering, College of Technology, Govind Ballabh Pant University of Agriculture & Technology, Uttarakhand, India.*

Abstract: *Conclusively, this case study provides an exhaustive and methodical investigation of wind load analysis on U and V-shaped high-rise structures, amalgamating conclusions from diverse scholars in the domain. The comprehensive examination and juxtaposition of current research techniques highlight the complex character of wind load assessment, utilizing a range of methods including wind tunnel testing, empirical observations, and Computational Fluid Dynamics (CFD) simulations. The research makes a substantial contribution to our comprehension of the complex relationships that exist between wind forces and the unique architectural arrangements of U and V shapes. By assimilating insights from numerous studies, the research successfully unveils commonalities, identifies divergences, and highlights emerging trends in the analysis of wind-induced effects on these high-rise structures. Professionals in high-rise building design, and architects and engineers. The goal of the research is to make a substantial contribution to the ongoing development of wind load analysis techniques. Understanding the building's response to wind rotation at different incident angles is a specific focus of this study. Validation against experimental data ensures the accuracy of CFD simulations, often involving model refinement based on real-world observations. Overall, this research contributes to a comprehensive understanding of building aerodynamics, offering insights into the intricate interplay between structures and wind forces.*

Keywords: *High-rise building, ANSYS CFX, CFD, wind tunnel, pressure coefficient.*

I.INTRODUCTION

The case study "Analysis of Wind Load on U and V Shape High-Rise Buildings" begins with a thorough summary of the literature, laying the groundwork for a detailed investigation of how wind forces affect particular architectural arrangements. The study's main components are deftly introduced in the opening section, which also emphasizes the use of sophisticated *Computational Fluid Dynamics (CFD)* simulations with ANSYS CFX in addition to conventional wind tunnel testing and the structural emphasis on U and V shapes.

The context of the study is first established in the introduction, which also highlights how important it is to comprehend how wind load affects high-rise structures. A distinctive element to the study is the selection of U and V shapes as particular structural configurations, which recognize the variety of architectural styles that can be found in urban environments. This first framing addresses an important issue in structural engineering and architecture, drawing the reader in. It is evident from the dual approach—using both wind tunnel testing and CFD simulations—that an intentional effort was made to offer a thorough analysis. This method acknowledges the complementary nature of these approaches in obtaining a comprehensive understanding of wind-induced forces, in addition to being in line with current research practices. The study's main goals—evaluating wind pressure, drag, lift, and streamline patterns at different heights inside U- and V-shaped high-rise buildings—are deftly stated in the introduction. These goals give the reader a clear overview of the particular areas that the research is trying to explore, acting as a kind of road map.

We have relatively little land accessible for habitation as a result of the industrial and corporate revolution and the current pattern of urban growth, and we also wish to protect the environment to support this growth. Therefore, high-rise building development is a very effective way to use land areas nowadays without negatively affecting the environment and to improve people's quality of life. However, a theoretical analysis is required before these structures can be built to ensure that the occupants can live in safety. We investigate using experimental research or a computer-based study. The computer-based analysis is carried out using ANSYS CFX, a program in the ANSYS suite. This research uses a domain generated for the wind flow in ANSYS CFX, and the building prototype is positioned in the domain to compute the parameters required to ensure the safety of the high-rise structure. The wind tunnel testing experiment uses the same phenomena, but instead of building an actual tunnel for study, we build a miniature prototype. Since we live in the computer age and the wind tunnel experiment is an outdated and more expensive method, we do not Favor it. Therefore, we now use ANSYS CFX in ANSYS software to perform wind load analyses of low-rise and high-rise buildings using the Computational Fluid Dynamics (CFD) method. High-rise building structural design is determined by wind loads. Generally, relevant wind loading guidelines and relevant literature are used to compute wind loads. Among the relevant wind load regulations are NZS: 1170.2-2021, ASCE: 7-02-2007, IS: 875-(Part-3)-2015, and NBC-(Part4)-2016. Nevertheless, the data on wind pressure or force coefficients offered by these codes is only useful for a restricted range of wind directions, even for a simple plan shape like a Y-SHAPE. Except for C-plan shapes, very little is known about wind loads for different building types.

Wind direction and speed are important factors to take into account when analysing the wind flow characteristics of high-

rise buildings. Tall building design frequently involves testing in wind tunnels. However, for a thorough evaluation at the early design stage in regions with low to moderate turbulence, a shift toward computational fluid dynamics analysis has been noted. The accuracy of both techniques varies from moderate to very high, although CFD analysis is more effective and less expensive than wind tunnel testing. Whereas wind tunnel testing yields abstract data, this method offers direct reporting. For direct resolution of unstable gusts and eddies, the wind tunnel is the preferred method in investigations involving extremely turbulent winds. CFD analysis also has the benefit of being able to get rid of any scaling and probes that could skew the results.

This paper discusses the challenges structural engineers face when designing wind-resistant skyscrapers, especially those with unique or asymmetrical plan shapes. Tall buildings are more wind-sensitive due to their greater structural flexibility and stronger wind pressure at higher altitudes. The Burj Khalifa and the Jeddah Tower are two prominent examples of "Y" plan buildings mentioned in the paper. It also draws attention to the dearth of international standards addressing wind effects on non-traditional plan shapes. Prior research on the impact of wind on buildings with irregular plans, such as computational and experimental fluid.

The study of wind effects on buildings is a crucial aspect of architectural and structural engineering, particularly in ensuring the safety and performance of structures in various environmental conditions. Two primary methodologies employed for such analyses are wind tunnels and Computational Fluid Dynamics (CFD). This research aims to comprehensively understand and differentiate between these techniques, focusing on the comparison of wind tunnel experiments and the use of ANSYS CFX, a CFD software, to assess the wind-induced effects on buildings of different shapes and heights. Wind tunnel testing has long been a traditional and widely accepted method for studying the aerodynamic behavior of structures. In a wind tunnel, physical models of buildings are subjected to controlled wind flows, allowing researchers to measure forces like wind pressure, drag, and lift and visualize streamlines. This empirical approach provides tangible data and insights into the real-world interaction between structures and wind, aiding in the design and optimization of buildings.

On the other hand, Computational Fluid Dynamics (CFD) offers a complementary approach to wind tunnel testing by leveraging advanced computer simulations. CFD involves the numerical solution of the governing fluid dynamics equations, such as the Navier-Stokes equations, to predict the behavior of fluid flow around complex geometries. This paper delves into the methodology of CFD, detailing the mathematical formulation, grid discretization, and numerical solution processes that enable the simulation of wind effects on buildings. The specific objectives of the study include determining wind pressures, drag forces, lift forces, and streamlines for buildings with various shapes and heights. Moreover, the research explores the impact of wind incident angles on the rotation of buildings during simulation setup, contributing to a comprehensive understanding of the dynamic interaction between structures and wind. It is essential to recognize that CFD, while a powerful tool, requires validation against experimental data to ensure its accuracy and reliability. The paper emphasizes the iterative nature of the CFD methodology, wherein the model is refined based on comparisons with real-world observations obtained from wind tunnel experiments or other validated sources.

By investigating both wind tunnel testing and CFD simulations, this study aims to provide valuable insights into the advantages, limitations, and complementarity of these two approaches in analyzing the aerodynamic behavior of buildings, thereby contributing to the advancement of structural design practices.

Computational fluid dynamics is the study of complex situations involving fluid-liquid, fluid-solid, and/or fluid-gas interaction. In this study, pressure coefficients and streamlines are generated in the ANSYS: CFX mode to analyse the solid-air interaction using numerical computation. The steps involved are usually as follows: formulating the flow problem, modelling it, defining beginning and boundary conditions, mesh creation, choosing a simulation strategy, entering parameters and files, executing the simulation, and determining when it is finished.

The paper discusses the need for estimating wind loads on unsymmetrical and tall buildings, as they are more susceptible to wind forces.

While there is ample information available on wind load for symmetrical structures, there is a lack of study on the effect of wind forces on unsymmetrical structures.The paper presents experimental and numerical studies on the wind effect on C-shaped buildings with varying aspect ratios and angles of incidence.

Computational Fluid Dynamics (CFD) techniques are used to evaluate surface pressure on the model, and the results are compared with experimental data, suggesting the feasibility of using CFD for predicting wind pressures on buildings accurately. The paper also mentions previous studies on wind pressure on irregular-shaped buildings and the use of CFD techniques for predicting wind loads. In this study, the G+64 tall building in Figure 1 with its two cross-sectional kinds, square and, and its dimensions in Figure 2 are analyzed using Computational Fluid Dynamics. The findings are explained in terms of the velocity streamlines and pressure coefficient values (from pressure contours) that were obtained for the proposed design model (Model A). Model X, a reference isolated model of 192 m height with only "square" as its cross-section throughout, is used to help validate the model. This model is illustrated in Figure 1. As a result of the fact that IS 875 (Part III) - 2015 specifies a range of deflection for the pressure coefficient for basic cross-sectional shapes, Therefore, as a result of Model X complies with the code, Model A will do the same as a result [3].
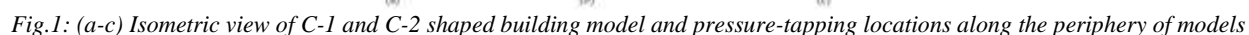
Presents the results of wind tunnel tests on rectangular building models having the same plan area and height but different side ratios of 1, 1.56, 2.25, 3.06, and 4. For wind directions of 0° to 90° at intervals of 15°, the effectiveness of the side ratio of buildings in altering the surface pressure distribution and mean responses of prototype buildings is evaluated. In contrast to the wind pressure on the windward wall, it has been found that the side ratio of buildings has a considerable impact on the wind pressures on the leeward and sidewalls. Additionally, the buildings' side ratio and wind incidence angles have a big impact on their mean displacements and torque [1].

The buildings used for the study have an octagonal plan. To investigate the impact of different wind incidence angles, specifically 0°, 15°, and 30° on wind pressure distribution, mean area-weighted average wind pressures on the faces of building

models were measured [4].

A series of wind tunnel studies were conducted for numerous structures, including the Pentagon, Y, T, C, and L. Finding the aerodynamic coefficient, which comprises the drag, lift, and torsional moment coefficients, is the main topic of this work. The building models are evaluated for several wind angles, including 0°, 45°, 90°, 135°, and 180°, while the wind tunnel is run at a wind speed of 10 m/s [2].

With the continuous developments in design methods and construction technologies and the context of huge urban growth, buildings are becoming more flexible, slender, and taller day by day and it poses new design challenges for structural engineers. In addition, there is a need to make the building lighter in order to control the development of inertial forces due to earthquakes. This further increases the wind-induced forces and motion in a building. Thus, wind-induced loads and motions generally govern the design of a tall building. This load and response directly depend on the outer shape of the building model and it can be significantly reduced by some outer shape modifications (Shiraishi et al. 1986, Amano 1995, Kawai 1998, Cooper et al 1997, Kim and You 2002, Kim et al. 2008, Kim and Kanda 2010a, 2010b, Bairagi and Dalui 2018). The Y-plan-shaped building is a triaxial building with three separate wings connected to a central core. Y shape plan is very common for residential, corporate, and hotel buildings as it allows the maximum views outward without overlooking a neighbouring unit. This type of building is also recommended given its ventilation efficiency and faster constructability. The current tallest structure, the Burj Khalifa, and the soon-to-be tallest tower, Jeddah, both are shape-modified Y plan-shaped tall buildings (Baker et al. 2007). [9]

II.WORK SCOPE

A 1:300 length scale model of the buildings has been created [25]. The internal angle between the limbs for Model B (Figure 2) is 120°. Moreover, Models A, C, and D result from changing the internal angle by 30° between two adjacent limbs while maintaining the same plan area.

The Model B's overall height is 500 mm, and each of its limb dimensions is 100 mm in length and 50 mm in width (Figure 6). The model's total plan area is 16082 mm^2.

The lengths of the limbs on Models A, C, and D had to be slightly increased or decreased in order to maintain the plan area almost unchanged. The specific measurements of the Model A, C, and D plans.[6]

Fig.1: (a-c) Isometric view of C-1 and C-2 shaped building model and pressure-tapping locations along the periphery of models

Fig.1: Isometric view of Model (a, b) Model B (in mm)

III. QUANTITATIVE ANALYSIS

Models of C-shaped buildings with varying aspect ratios have been generated in ANSYS for numerical analysis, and the

$k - \varepsilon$ *model* of turbulence has been employed for analysis. The governing equations are solved by ANSYS (Fluent) using the finite element method, which divides the region of interest into a finite number of cells (the mesh or grid). [6]

Information analyzed numerically:
(i) Fluid types: air
(ii) Air density: 1.225 kg/m3
(iii) Air viscosity: 1.789 x 10^{-5} kg/m·
(iv) Model of Turbulence: $k - \varepsilon$ model
(v)Pressure-based solution

The numerical analysis of C-shaped buildings using ANSYS Fluent, with a focus on the modeling of fluid flow and turbulence. Here is a breakdown of the key details:

3.1 Geometry and Aspect Ratios:
C-shaped buildings with varying aspect ratios have been created in ANSYS for numerical analysis. The aspect ratio typically refers to the ratio of the building's length to its width or height, and it can affect the flow patterns and aerodynamics around the building.

3.2 Fluid Type:
The fluid being analyzed is air. This is a common choice for simulations involving buildings, as it represents the airflow around and within the structure.

3.3 Air Density:
The density of air is given as 1.225 kg/m³. This parameter is crucial for accurately modelling fluid flow, especially in simulations involving external aerodynamics and natural convection.

3.4 Air Viscosity:
The dynamic viscosity of air is provided as 1.789 x 10^-5 kg/m·s. Viscosity is a critical property that affects how air flows and interacts with surfaces. It is used in the calculation of the Reynolds number and the modelling of turbulent flows.

3.5 Model of Turbulence:
The $k - \varepsilon$ model of turbulence is being employed for the analysis. The $k - \varepsilon$ model is a widely used turbulence model in CFD (Computational Fluid Dynamics) simulations, which helps to capture the effects of turbulence on the flow field. It's used to predict quantities like turbulence kinetic energy (k) and the rate of dissipation of turbulence (ε).

3.6 Pressure-Based Solution:
The governing equations for fluid flow, which include the Navier-Stokes equations and the continuity equation, are solved using a pressure-based solution method. This method is common in CFD and is used to calculate pressure, velocity, and other flow variables in the domain.

In summary, the information provided outlines the key parameters and methods used for the numerical analysis of C-shaped buildings in ANSYS Fluent. These details are essential for accurately simulating the flow of air around and within the buildings and for understanding the aerodynamic behavior of different aspect ratios.

IV.EXPERIMENTAL PROGRAM

The Indian Institute of Technology Roorkee's Department of Civil Engineering's Closed Circuit Wind Tunnel served as the experimental study's location in Roorkee, India. It is a single-fan, closed test section (closed jet type), closed circuit, continuous flow wind tunnel with a 65 HP motor. The test section has a length of 8.2 meters and a cross-section size of 1.3 meters by 0.85 meters. Models for the current study are created at a 1:300 length scale. The comparison of prototype and model dimensions is provided in Table 1. [10]

The wind speed at the inlet is assumed to be 10 m/s when modelling the domain with a scale factor of 1:300. The numerical simulation is mostly performed using the k-turbulence model. This model uses the gradient diffusion hypothesis to relate Reynolds stress to mean velocity gradient and turbulent viscosity. The product of turbulent velocity and length scale is the deafened turbulent viscosity.

The rate at which the fluctuation of velocity dissipates is known as the turbulent eddy dissipation rate, or ~. Turbulent kinetic energy, or k, is the variability of velocity fluctuation. L2 T -2 is the dimension of k, and ~ is T. [9]

Table 1: Prototype and model dimensions

Parameter	Prototype (m)	Model (mm)	Scale
Length	30	100	
Width	30	100	1:300
Total Height	150	500	

4.1 Turbulent Eddy Dissipation Rate (ε):

The turbulent eddy dissipation rate (ε) represents the rate at which turbulence kinetic energy (k) is converted into internal energy through viscous dissipation in turbulent flows.

It is a measure of the rate at which turbulent energy is being dissipated into heat due to the effects of viscosity within the flow. ε is a crucial parameter in turbulence models, such as the k-ε model, which is used to predict the turbulent behavior of fluid flows. In these models, ε is a source term in the transport equation for k.

It is typically expressed in units of (m²/s³) or (ft²/s³) depending on the system of units used.

$$\varepsilon = C\varepsilon 1 * (k^{1.5}) / (l)$$

4.2 Turbulent Kinetic Energy (k):

Turbulent kinetic energy (k) represents the energy associated with the turbulent fluctuations in velocity within a fluid flow.

It measures the intensity of turbulence in the flow and is related to the variance of velocity fluctuations.

The units of turbulent kinetic energy depend on the system of units but are typically expressed in (m²/s²) or (ft²/s²).

V.ANALYSIS OF ANSYSCFX

The ANSYS analysis process consists of five steps: geometry, mesh, setup, solution, and result. Indian Standards were used in the creation of the domain [3].

5.1 Geometry

Geometry is constructed in any 3D modeling software and then imported into the ANSYS CFX. Model A is designed in 3D modelling as shown in Figure 1. Model A is a G+64 building, 192 m in height with two typesof cross – sections – square and plus havingheight of 96 m each as shown in Figure 2, and is compared with a reference isolated model A [6].
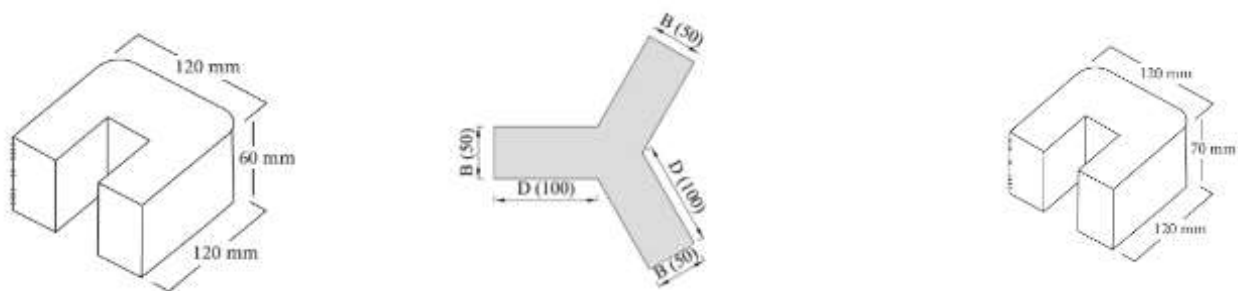

Fig.2: Dimension model A and model B

5.2 Meshing

The geometry meshing in ANSYS is displayed in Figure 2. When discretization is carried out using finer cell elements, CFD produces good results. However, as the number of cells increases, so does the computational duration, necessitating the use

of more computational power. [9].

The final meshing of the domain volume was implemented using the automated
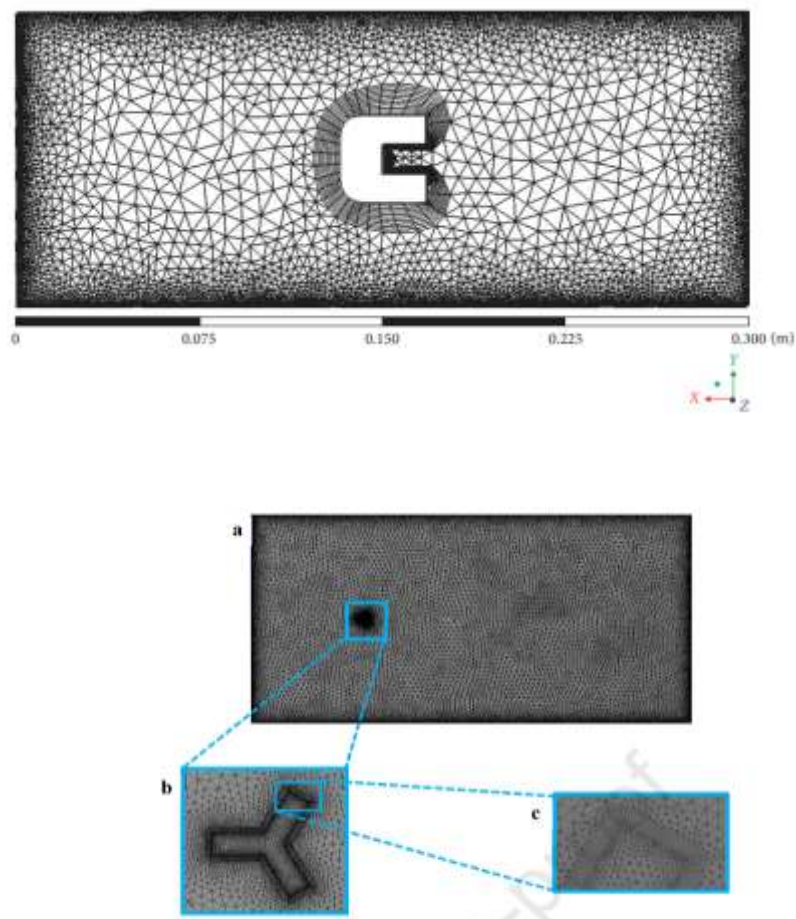
(a)Typical mesh pattern in the computational domain (b) meshing around the building model (c) detail of mesh near edge
Fig.3: The meshing of model C-shape and Y-shape

5.3 Setup in Domain

Figure 3 shows the prototype placed in the domain for the testing.
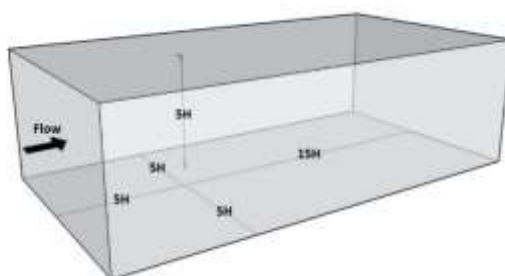

Fig.4: Domain set up

The domain size adheres to the methodology outlined in Franke et al. [24] and is depicted in Figures 4(a) and 4(b). Five times the building's height (5H) from the building's face is the upstream side, and fifteen times the building's height (15H) from the building's face is the downstream side. The domain's two side clearances are spaced five times the building's height (5H) from the building's face. Furthermore, the top clearance is fixed five times. [6]
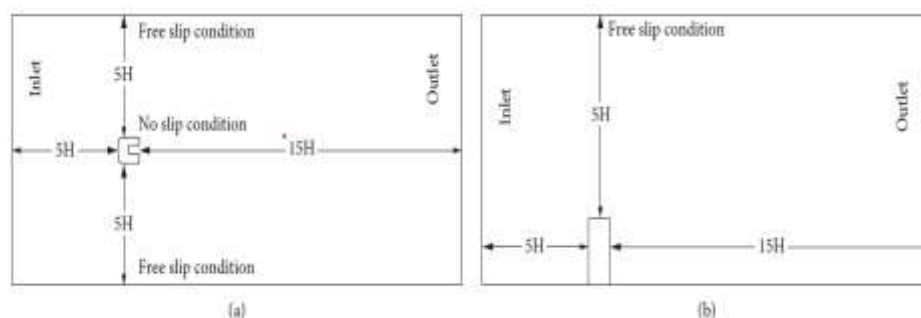
Fig.5: (a, b) Domain for the plan and elevation

5.4 Solution

Frequently, the answers are steady-state answers. Second-order differencing was used for the pressure, momentum, and turbulence equations as well as the "coupled" pressure-velocity coupling technique because of its robustness for steady-state, single-phase flow problems. The residuals fell short of the commonly accepted threshold of falling below 10-4 of their initial levels after several hundred cycles. To ascertain whether the simulations had converged, there were other approaches besides this one. Throughout the simulation, the moments acting on the building as well as the drag, lift, side, and moment forces were monitored.

VI.RESULTS

The study's findings led to the following broad conclusions. A good alternative to traditional methods of predicting wind-related phenomena on buildings and other types of structures is Computational Fluid Dynamics (CFD). With the aid of CFD simulation, boundary layer separation can be properly shown.

CFD analysis can be used to determine the pressure values, flow streamline, velocity vector, and other associated parameter variables through the model surface.

The meshing of the geometry model and precisely defining the physical property values as the prerequisites of a realistic environment are necessary for accurate outcomes.

CFD simulations make it simple to examine boundary layer separation and wake creation, and they also allow for the use of appropriate mitigation strategies to mitigate the impacts of vortex shedding.

In the present study, the rectangular plan-shaped building with an opening and inner courtyard is analyzed by the CFD package, namely ANSYS CFX (version 16.0). The boundary layer wind profile is governed by the power law equation:
$U(z) = U0 (Z/Z0)$ Where $U(z)$ is the velocity at some particular height Z, U0 is boundary layer velocity, Z0 is the boundary layer depth and is power law exponent and its value is taken as 0.133 which satisfies terrain category II, mentioned in IS 875-part 3 (2015). [3]

The CFD analysis conducted on the rectangular plan-shaped building with an opening and inner courtyard using ANSYS CFX yielded valuable insights into the wind-induced phenomena on the structure. The findings from this study contribute to the growing body of evidence supporting Computational Fluid Dynamics as a robust alternative to traditional methods for predicting and understanding wind-related effects on buildings and other structures. Through the CFD simulation, the researchers were able to precisely visualize and analyze the occurrence of boundary layer separation on the building's surfaces. This capability is a significant advantage of CFD over some traditional methods, as it allows for a detailed examination of flow patterns and pressure distribution.[5]

The CFD analysis further provided comprehensive data on pressure values, flow streamlines, velocity vectors, and other associated parameter variables across the model surface. This information is essential for understanding the distribution of forces acting on the building, aiding in the identification of areas prone to high wind pressure or structural vulnerability. The ability to obtain such detailed data through numerical simulations enhances the efficiency and accuracy of structural design processes, allowing for better-informed decisions and optimized building performance

The meshing of the geometry model and accurately defining the physical property values were identified as crucial prerequisites for obtaining realistic and accurate outcomes in CFD simulations.

The study underscores the importance of meticulous attention to these steps in the simulation setup to ensure the reliability of the results. This emphasizes the need for researchers and engineers to adopt best practices in mesh generation and parameter specification to achieve meaningful and trustworthy simulations.

The CFD simulations facilitated the examination of boundary layer separation and wake creation. Understanding these aspects is vital for assessing the aerodynamic stability of the structure and implementing appropriate mitigation strategies to counteract the impacts of vortex shedding. The ability to identify potential issues related to wake formation and vortex shedding through CFD simulations allows for proactive design modifications to enhance structural resilience and safety.

Both CFD and wind tunnel tests can provide information on the distribution of wind pressures across different surfaces of the building. This information is crucial for understanding how the wind interacts with the building and where the maximum pressures occur. The analyses can quantify the drag and lift forces acting on the building.

CFD simulations can provide detailed velocity profiles, showing how wind speeds vary at different locations around the building. This information is valuable for evaluating potential wind-induced structural vibrations.

Understanding the impact of wind on the building's rotation for varying incident angles is important, especially for tall structures. This information can guide design decisions to minimize the building's vulnerability to wind-induced torsional effects. Both methods can yield pressure coefficients, which are dimensionless numbers expressing the pressure at a certain point relative to the free-stream atmospheric pressure. These coefficients are crucial for designing structures that can withstand wind loads. The results can be used to optimize the building's shape and height to minimize wind-induced effects, enhancing the overall performance and safety of the structure.

It's important to note that while wind tunnel testing and CFD simulations can provide valuable information, validation against experimental data is essential to ensure the accuracy of the results. The combination of both methods can offer a comprehensive understanding of the building's behaviour under various wind conditions.

Table 2. Comparison of C_{pe} on Faces of Rectangular Model

As per	Wind Angle	C_{Pe} Face A	C_{Pe} Face B	C_{Pe} Face C	C_{Pe} Face D
ANSYX (CFX)	0°	+0.68	-0.28	-0.48	-0.48
	90°	-0.43	-0.44	0.71	-0.23
IS: 875 (Part 3): 2015	0°	+0.8	-0.25	-0.8	-0.8
	90°	-0.8	-0.8	0.8	-0.25
ASCE/SEI 7-10	0°	+0.8	-0.5	-0.7	-0.7
	90°	-0.7	-0.7	0.8	-0.5
AS/NZS- 1170.2 (2002)	0°	+0.8	-0.5	-0.65	-0.65
	90°	-0.65	-0.65	0.8	-0.5
EN: 1991-1-4	0°	0.8	-0.55	-0.8	-0.8
	90°	-0.8	-08	0.8	-0.55
BS: 6399-2	0°	0.76	-0.5	-0.8	-0.8
	90°	-0.8	-0.8	0.76	-0.5
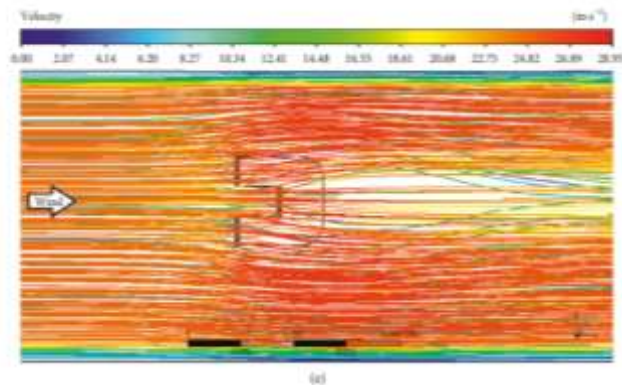
Fig.6 Wind flow patten for different wind incidence angles (a) 0^0 (b) 30^0 (c) 60^0 (d) 90^0 (e) 180^0
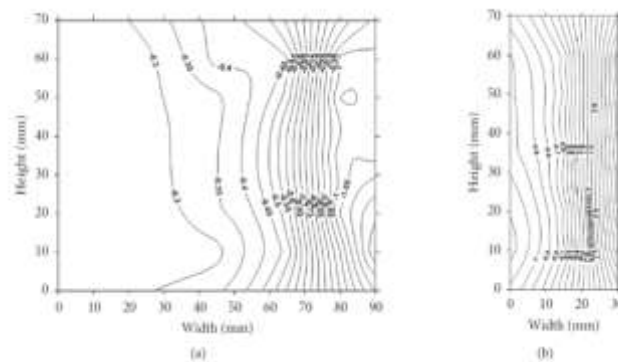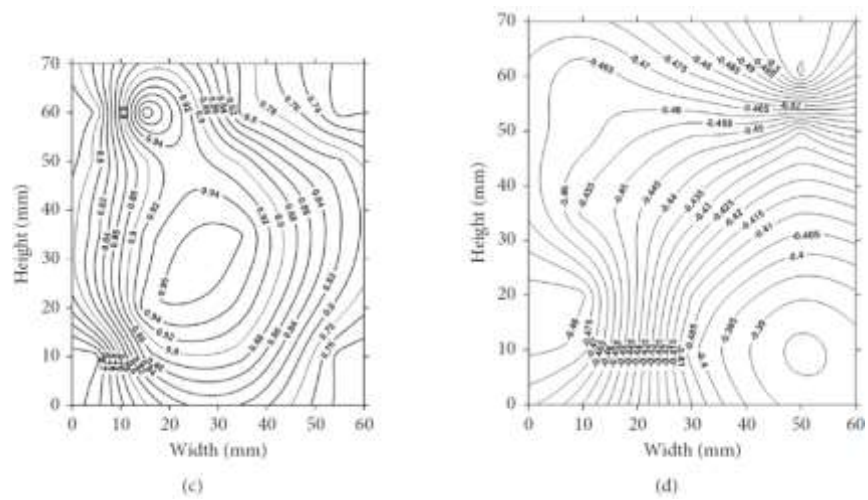
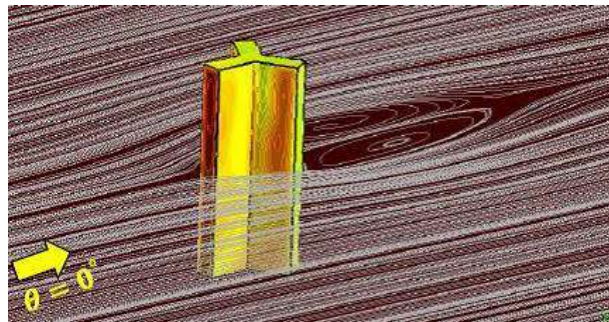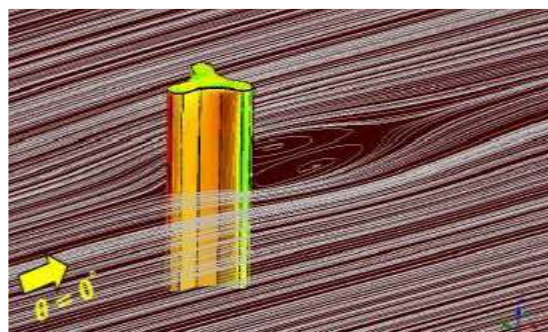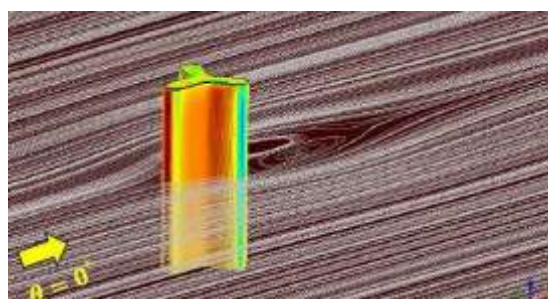*Fig.7: Pressure contour on different faces of the C-1 model at different wind angles.*
(a)   Face A 90^0 (b) Face B 45^0 (c) Face C 0^0 (d) Face G 90^0
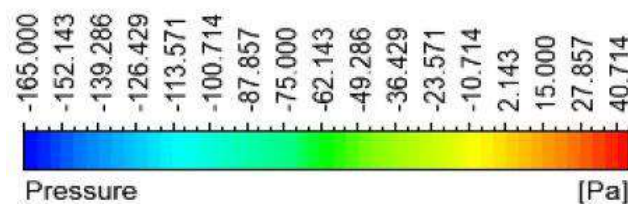

(a)

(b) Model B ($\gamma c=30\%$)

Fig.8: Pressure contours and streamlines of various model for 0^0 wind angl
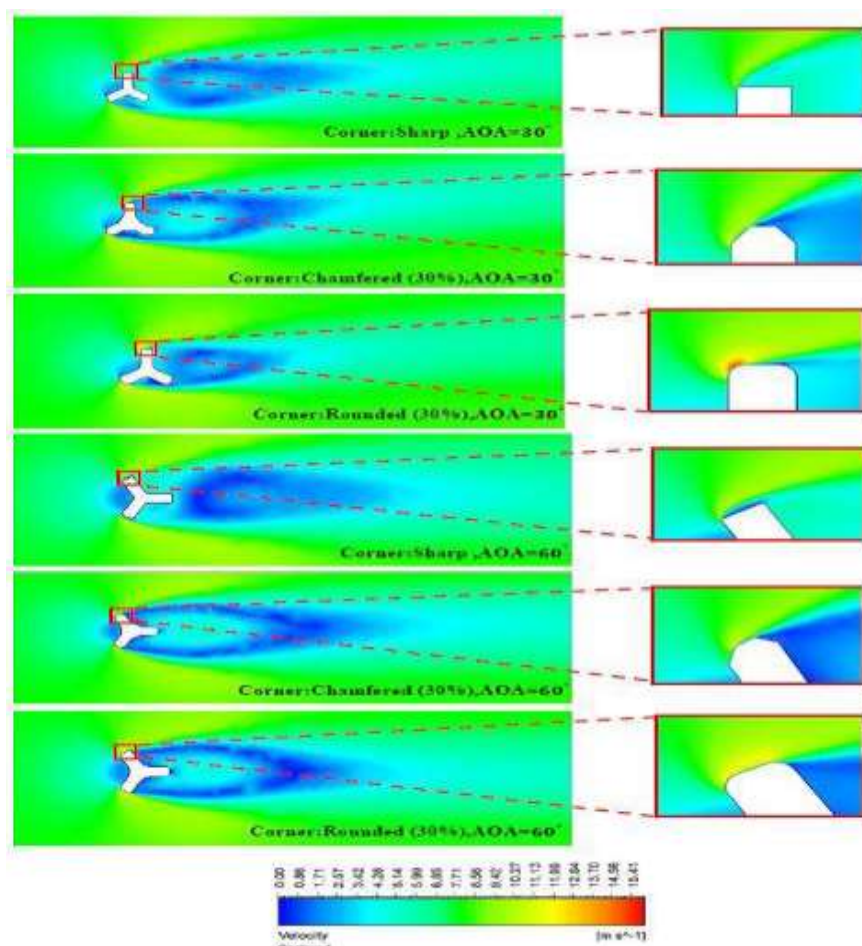

Fig.9: Velocity field for different corner shapes

VII.CONCLUSION

Conclusively, this case study provides an exhaustive and methodical investigation of wind load analysis on U and V-shaped high-rise structures, amalgamating conclusions from diverse scholars in the domain. The comprehensive examination and juxtaposition of current research techniques highlight the complex character of wind load assessment, utilizing a range of methods including wind tunnel testing, empirical observations, and Computational Fluid Dynamics (CFD) simulations. The research makes a substantial contribution to our comprehension of the complex relationships that exist between wind forces and the unique architectural arrangements of U and V shapes. By assimilating insights from numerous studies, the research successfully unveils commonalities, identifies divergences, and highlights emerging trends in the analysis of wind-induced effects on these high-rise structures is a powerful tool for studying how wind interacts with buildings.

It gives us a clear picture of what's happening and helps us find ways to make structures more wind-resistant and safer. It's like using a high-tech microscope to examine wind patterns and make informed decisions to protect buildings and structures from wind-related issues.Computational Fluid Dynamics (CFD) in understanding wind-related phenomena around buildings and structures. CFD is a fantastic alternative to traditional methods for predicting how wind affects buildings and structures. It's like having a high-tech tool that helps us understand the behavior of wind around objects. CFD simulations can clearly show something called "boundary layer separation." This means we can see where the wind flow separates from the surface of a building. This is crucial for understanding how wind interacts with structures. Its analysis provides a lot of information, like

pressure values, the path of the wind (flow streamline), and how fast the air is moving (velocity).

We can look at all these things on the surface of the model. With CFD, we can easily see when the wind slows down and separates from the building's surface. This helps us understand how wind behaves around the structure and where potential problems might occur. This study examined how a tall, Y-shaped building's corner cuts affected the building's ability to reduce wind load. ANSYS CFX software has been used for CFD simulation.

The SST and k-ϵ models have been validated using wind tunnel data. Since k-ϵ models yield nearly identical outcomes, they are employed in subsequent numerical simulations. At the end is a comparison section for the SST turbulence model and k-ϵ. Areas for future research. analysis of wind loads on U-shape and V-shape buildings under isolated and interference conditions usingCFDUpon examining the search results, it was found that there were no particular papers that addressed the use of CFD for the analysis of wind loads on U-shaped and V-shaped buildings under isolated and interference conditions.

On the other hand, a few of the reviewed papers addressed the use of CFD analysis for wind loads on tall buildings in various scenarios. The review paper's principal conclusions are:

• Building height, shape, orientation, surrounding buildings, wind speed and direction, wind turbulence, and interference effects are some of the factors that affect wind loads on high-rise buildings.

• CFD analysis is a valuable tool for analysing wind loads on high-rise buildings. It has evolved from the validation of isolated buildings to more sophisticated modelling, including aerodynamic shape optimization. due to the presence of surrounding buildings or other structures, in wind loads on a high-rise building. Wake interference, shielding, and other factors can result in interference effects.

REFERENCES

1. *Kumar, A., & Raj, R. (2022). CFD Study of Flow Characteristics and Pressure Distribution on Re-Entrant Wing Faces of L-Shape Buildings. Civil Engineering and Architecture, 10(1), 289–304. https://doi.org/10.13189/cea.2022.100125*

2. *Mallick, M., Mohanta, A., Kumar, A., & Raj, V. (2018). Modeling of Wind Pressure Coefficients on C-Shaped Building Models. Modeling and Simulation in Engineering, 2018. https://doi.org/10.1155/2018/6524945*

3. *Paul, R., & Dalui, S. K. (2016). Wind effects on 'Z' plan-shaped tall building: a case study. International Journal of Advanced Structural Engineering, 8(3), 319–335. https://doi.org/10.1007/s40091-016-0134-9*

4. *Sanyal, P., & Dalui, S. K. (2018). Effects of courtyard and opening on a rectangular plan-shaped tall building under wind load. International Journal of Advanced Structural Engineering, 10(2), 169–188. https://doi.org/10.1007/s40091-018-0190-4*

5. *Sanyal, P., & Dalui, S. K. (2020). Effect of corner modifications on Y' plan-shaped tall building under wind load. Wind and Structures, An International Journal, 30(3), 245–260. https://doi.org/10.12989/was.2020.30.3.245*

6. *Sanyal, P., & Dalui, S. K. (2021). Effects of the internal angle between limbs of "Y" plan-shaped tall building under wind load. Journal of Building Engineering, 33. https://doi.org/10.1016/j.jobe.2020.101843*

7. *Verma, D. S. K., Roy, A. K., Lather, S., & Sood, M. (2015). CFD Simulation for Wind Load on Octagonal Tall Buildings. International Journal of Engineering Trends and Technology, 24(4), 211–216. https://doi.org/10.14445/22315381/IJETT-V24P239*

8. *Yuan, W., Wang, Z., Chen, H., & Fan, K. (2017). Numerical analyses of aerodynamic characteristics of integrated L-shaped high-rise building. Advances in Engineering Software, 114, 144–153. https://doi.org/10.1016/j.advengsoft.2017.06.018*

9. *Zheng, C. R., & Zhang, Y. C. (2012). Computational Fluid Dynamics study on the performance and mechanism of suction control over a high-rise building. Structural Design of Tall and Special Buildings, 21(7), 475–491. https://doi.org/10.1002/tal.622*