EFFECT OF VARIOUS DESIGN PARAMETERS OF CHIMNEY UNDER WIND LOAD

A DISSERTATION

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS

FOR THE AWARD OF THE DEGREE

OF

MASTER OF TECHNOLOGY

IN

STRUCTURAL ENGINEERING

Submitted by:

MAYUR MADHUKAR NEMADE

(2K21/STE/13)

Under the supervision of

DR. PRADEEP K. GOYAL

Associate Professor

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi college of Engineering)

Bawana Road, Delhi-110042

MAY 2023

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi college of Engineering)

Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, MAYUR MADHUKAR NEMADE, 2K21/STE/13, student of M.Tech Structural Engineering, hereby declare that the project Dissertation titled "EFFECT OF VARIOUS DESIGN PARAMETERS OF CHIMNEY UNDER WIND LOAD " which is submitted by me to the Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi **MAYUR MADHUKAR NEMADE**

Date: **2K21/STE/13**

DEPARTMENT OF CIVIL ENGINEERING

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi college of Engineering)

Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the Project Dissertation titled "EFFECT OF VARIOUS DESIGN PARAMETERS OF CHIMNEY UNDER WIND LOAD" which is submitted by Mayur Madhukar Nemade, 2K21/STE/13 Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the students under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi
Date:

(**DR. P.K. GOYAL)**
Associate Professor
Department of Civil Engineering
Delhi Technological University

ABSTRACT

This thesis investigates the impact of wind on high-rise chimneys, focusing on predicting wind loads and analysing the effects of varying chimney dimensions. Using computational fluid dynamics (CFD) simulations, the study considers variables such as wind speed, direction, turbulence, chimney shape, size, and parameter variations.

Simulations were conducted using ANSYS CFX 2022 R2 and AutoCAD, covering wind incidence angles from 0 to 180 degrees. Pressure contours and coefficient of pressure (Cp) values were analysed to understand pressure distribution and wind-induced forces. Results showed that pressure values varied with wind angle, predominantly exhibiting suction forces. Cp values were consistently negative, indicating the presence of suction forces.

The findings highlight the influence of wind angle on chimney pressures, with a decrease in suction forces as the wind deviates from perpendicular. Wind velocity affects pressure variations, with higher velocities leading to intensified positive and negative pressures. Different wind zones result in varying pressure distributions, from laminar flow in low-velocity regions to turbulent flow in higher wind zones.

The study also explores the impact of chimney dimensions. Larger top diameters promote smoother flow transitions, reduced pressure drop, and improved flow efficiency. Smaller top diameters may cause increased turbulence and pressure fluctuations. Base diameter variations affect flow separation, windward flow deflection, and wind pressure distribution. Chimney height influences flow dynamics, with taller chimneys exhibiting more pronounced upward airflow. Chimney wall thickness affects streamlines' coherence, with thicker walls promoting laminar flow and thinner walls causing turbulence.

ACKNOWLEDGEMENT

I would like to express my sincere appreciation and gratitude to everyone who has contributed to the successful completion of this project. Although it may not be possible to name everyone who has helped me along the way, I would like to recognize some individuals for their exceptional support and guidance.

First and foremost, I would like to express my sincere gratitude to my project supervisor, Dr. P. K. Goyal, for their unwavering support and guidance throughout my project. Their expertise, feedback, and insightful comments have been invaluable in shaping the direction and scope of this study. I am grateful for their patience, dedication, and mentorship throughout my research journey.

Furthermore, I would like to extend my gratitude to the faculty and staff of the department for providing me with access to necessary resources and facilities. I am thankful for their support and encouragement.

Finally, I would like to express my heartfelt appreciation to my family, friends, and colleagues for their constant encouragement and motivation. Their unwavering support has been instrumental in helping me achieve my academic goals.

I am grateful to everyone who has contributed to the successful completion of this project. I am humbled and honored by their support and guidance, and I will cherish these experiences for a lifetime.

MAYUR NEMADE

2K21/STE/13

TABLE OF CONTENTS

LIST OF FIGURES

CHAPTER 1: INTRODUCTION

1.1 GENERAL

The history of chimneys is intriguing and goes back thousands of years. These vertical constructions, which are often made of brick, stone, or metal, perform the crucial job of supplying ventilation and removing smoke and pollutants from heating systems inside of buildings. Chimneys have changed over time in terms of their appearance and function, becoming crucial parts of buildings.

Ancient civilizations from the Indus Valley, Mesopotamia, and Egypt are where chimneys first appeared. These early chimneys were crude in design and frequently consisted of small apertures in walls or roofs that let smoke out of enclosed places. These chimneys, which were made of materials like mud, reeds, or stone, assisted in reducing the risks associated with the buildup of smoke indoors.

Improvements in chimney technology started to appear throughout the Roman era. The sophisticated central heating system known as the hypocaust was created by the Romans, who are renowned for their engineering brilliance. In order to transport warm air throughout buildings, this method made use of a network of underground flues. Smoke could rise and leave the buildings through these flues, which were joined to smoke chambers, which served as the forerunners of chimneys.

Following the fall of the Roman Empire, chimney design advancement in Europe was somewhat gradual. In the mediaeval era, chimneys were frequently included into the design of structures built around substantial hearths. Smoke was gathered in smoke bays or hoods on the hearths before being released via apertures in the roof.

With the advent of the Renaissance, architectural styles evolved, leading to the increased prevalence of fireplaces within households. This development necessitated more elaborate chimney systems. Chimneys were constructed as separate structures, often with decorative elements, and were connected to fireplaces through flues or smoke ducts. Chimneys became status symbols and prominent features in grand houses and castles, showcasing the wealth and social standing of the occupants.

The Industrial Revolution in the 18th century brought about significant advancements in chimney technology. The demand for efficient heating systems, driven by innovations such as coal-burning stoves and the steam engine, required chimneys capable of managing larger volumes of smoke and gases. Chimneys were designed taller and wider, accommodating multiple flues to handle various appliances and their exhaust.

Throughout the 19th and 20th centuries, chimney technology continued to progress. The widespread use of fossil fuels and the rise of internal combustion engines necessitated chimneys capable of handling different types of exhaust gases. Metal chimneys, including those made from cast iron or stainless steel, gained popularity due to their durability and flexibility in construction.

In modern times, chimneys still play a crucial role in buildings, although their purpose has evolved alongside advancements in heating systems and cleaner energy sources. Traditional masonry chimneys are still common, but prefabricated metal chimneys and factory-built chimney systems have become popular due to their ease of installation and cost-effectiveness.

Safety and maintenance are paramount in chimney design today. Regular cleaning and inspection are necessary to prevent chimney fires and ensure proper ventilation. Chimney caps and liners are often installed to enhance safety and improve the overall efficiency of the chimney system.

Chimneys are essential components of various industrial and residential structures, serving as conduits for the safe and efficient discharge of combustion gases. However, their exposure to wind loads poses significant challenges to their structural integrity. Understanding the complex wind-structure interaction phenomena and accurately predicting the response of chimneys is crucial for ensuring their safe design and operation. This thesis aims to investigate the effects of wind loads on chimneys, employing advanced analytical and computational techniques to optimize their structural performance.

In conclusion, the history of chimneys showcases their evolution from basic openings to complex architectural features. Despite changes in technology and energy sources, chimneys continue to serve a vital role in providing ventilation, preserving indoor air quality, and adding aesthetic value to buildings.

1.2 WIND EFFECT ON CHIMNEY

The impact of wind on industrial chimneys is a topic of significant importance in various engineering and environmental contexts. The behavior of wind around these structures can have diverse effects, influencing their stability, performance, and the dispersion of emissions. This article delves into the intricate relationship between wind and industrial chimneys, examining the consequences and implications of this interaction.

Primarily, wind can exert mechanical forces on industrial chimneys, posing challenges to their structural integrity. The magnitude and direction of wind flow can subject chimneys to varying amounts of pressure, known as wind loading. Strong winds perpendicular to the chimney's axis can induce significant lateral forces, potentially leading to structural oscillations and vibrations. Such vibrations, if left uncontrolled, can compromise the stability of the chimney and even result in catastrophic failures.

To mitigate the adverse effects of wind loading, engineers employ sophisticated design techniques. These include employing appropriate materials, determining the optimal chimney height and diameter, and implementing structural reinforcements such as guy wires or dampers. By considering wind patterns and velocities specific to the installation site, engineers can ensure that industrial chimneys are adequately designed to withstand wind-induced stresses.

Furthermore, wind plays a crucial role in the dispersion of pollutants emitted from industrial chimneys. The plume, or the column of exhaust gases released from the chimney, is subject to the prevailing wind patterns. Wind speed and direction influence the dispersion and dilution of pollutants in the surrounding environment. A strong, steady wind can aid in dispersing pollutants over a larger area, reducing their concentration in the immediate vicinity of the chimney and potentially minimizing the impact on local air quality.

Conversely, if wind conditions are stagnant or if the wind blows towards populated areas, pollutants may accumulate in specific regions, leading to potential health and environmental concerns. It is crucial for industries to consider wind patterns and their potential effects when selecting suitable locations for industrial chimneys. Computer simulations and modelling techniques are often employed to predict the dispersion patterns of pollutants, allowing for informed decision-making and regulatory compliance.

Moreover, wind-induced aerodynamic effects can affect the performance of industrial chimneys. Wind passing over the chimney can create areas of low pressure, resulting in the suction of air from the surrounding vicinity into the chimney. This phenomenon, known as wind-induced draft, can enhance the chimney's natural ventilation capabilities, facilitating the efficient exhaust of combustion gases or other byproducts. Properly harnessing wind-induced draft can enhance the overall performance and energy efficiency of industrial processes.

Lastly, wind has a multifaceted impact on industrial chimneys. While wind loading poses structural challenges, careful engineering and design considerations can mitigate these risks. The dispersion of pollutants emitted from chimneys is heavily influenced by wind patterns, necessitating thorough analysis and environmental planning. Additionally, wind-induced draft can enhance chimney performance, promoting efficient ventilation. By comprehensively understanding and accounting for the effects of wind, industries can ensure the safe and sustainable operation of their industrial chimneys.

1.3 SCOPE OF STUDY

Accurately estimating wind loads is of paramount importance when designing structures to ensure safety and cost-effectiveness. Different structures with varying aerodynamic characteristics experience distinct wind loads, which are influenced by factors such as terrain category, wind speed, topography, and the geometry of the structure. The wind flow approaching a structure is shaped by geographical features, including the roughness of the surrounding terrain and the presence of obstructions nearby.

As chimneys increase in height, they become more susceptible to wind damage due to the higher wind speeds experienced at greater heights. Additionally, the interference of other structures is a crucial consideration during the design process. Interference can have both positive and negative effects on chimneys, but it is the negative effects that can lead to severe damage or even structural collapse. The significance of conducting interference analysis was highlighted following the collapse of the Ferrybridge cooling tower in

England. The interference from other structures played a major role in the collapse of the cooling tower.

To ensure structural integrity and mitigate the risks associated with wind loads, engineers and designers must accurately assess the wind conditions specific to the location of the structure. This involves considering the surrounding terrain, performing wind speed measurements, and evaluating the impact of nearby obstructions. With a thorough understanding of the wind loads and potential interference effects, appropriate design measures can be implemented to reinforce the structure and prevent failure.

In summary, precise estimation of wind loads is crucial for the safe and cost-effective design of structures. Factors such as terrain category, wind speed, topography, and geometry influence the wind loads experienced by a structure. Interference from nearby structures can have both positive and negative effects, with negative effects posing significant risks. By conducting thorough interference analysis and considering the specific wind conditions, engineers can ensure the structural integrity and mitigate potential wind-induced failures.

From a structural point of view, the scope of study for the effects of wind on industrial chimneys focuses on understanding and mitigating the challenges posed by wind loading.

Wind Load Analysis is done for the mechanical forces exerted by wind on industrial chimneys. They study wind speed profiles, turbulence characteristics, and the resulting pressure distribution on the chimney's surface. This involves examining wind pressure coefficients, which provide insight into the variations in pressure along the height of the chimney. Advanced techniques such as wind tunnel testing, numerical simulations, and empirical formulas are utilized to quantify wind loads accurately.

The scope includes developing design methodologies to ensure the structural integrity of industrial chimneys under wind loading. Researchers consider various factors such as chimney height, diameter, and shape, along with the wind characteristics specific to the installation site. They employ structural analysis techniques to determine the optimal dimensions and configurations of chimneys to withstand wind-induced stresses.

The choice of materials for industrial chimneys is also within the scope of study. Researchers explore the properties and characteristics of different construction materials, such as steel, concrete, or composite materials, to determine their suitability for withstanding wind loading. They evaluate factors such as strength, stiffness, ductility, and resistance to corrosion and fatigue. The aim is to select materials that provide the necessary structural robustness while considering economic and sustainability considerations.

The response of industrial chimneys to wind loading is thoroughly investigated, taking into account their dynamic behaviour. The focus lies in understanding and including phenomena such as vortex shedding and galloping, which can lead to excessive stresses and fatigue damage. Additionally, the stability of chimneys against overturning and sliding is evaluated under wind loads to ensure safe and reliable operation.

By delving into these structural aspects, researchers aim to enhance the understanding of wind effects on industrial chimneys and provide valuable design guidelines. These guidelines aim to ensure the structural integrity, stability, and longevity of chimneys, thereby ensuring the safety and operational continuity of industrial facilities in the face of wind-induced stresses.

1.4 OBJECTIVES OF STUDY

- To analyse the effect of wind load on a stand-alone chimney with introducing wind on different wind angles.
- To analyse the chimney with Different Velocities that is in different wind zones.
- To analyse the chimney with Varying Top Diameter of chimney, Bottom Diameter of chimney, Thickness of chimney and Height of chimney.

CHAPTER 2: LITERATURE REVIEW

This literature review offers a comprehensive overview of the research conducted on wind CFD simulations for industrial chimneys. It focuses on the methodologies employed, challenges encountered, and advancements achieved in understanding flow characteristics and pressure distribution. By critically analysing existing literature, gaps in knowledge and potential research areas can be identified.

The review begins by exploring fundamental principles and theories of wind flow and its interaction with chimney structures. Key topics covered include boundary layer development, wind turbulence modelling, and the influence of wind direction and speed on flow patterns. This section provides a solid foundation for understanding subsequent research findings in wind CFD simulations of industrial chimneys.

B. Prathyusha Yadav, et.al (2021) focuses on assessing the outcomes of seismic analysis and wind analysis conducted on a 100 m Reinforced Cement Concrete Chimney. The chimney was modelled using STAAD. Pro V8i ss6, utilizing the lumped mass modelling approach. The findings from this study reveal an increase in shear force and bending moment values as the zone factor value increases. This paper aims to provide a comprehensive evaluation of the analyses and their implications on the structural behaviour of the chimney.

A.K. Roy, et.al (2016) focus lies on the pressure coefficients (Cp) associated with chimneys, which are limited to certain aspect ratios. The current body of knowledge lacks information regarding the variation of wind pressure coefficients (Cp) on the chimney's surface where high pressure or suction is anticipated. Both computational fluid dynamics (CFD) simulations and experimental findings align closely with previous experimental studies. The accuracy of the obtained results is contingent upon several factors, including accurate scaling of the model, appropriate meshing of the model geometry, and precise definition of physical property values to reflect realistic environmental conditions.

Dalal and Savsani (2016) undertook a significant endeavour by employing Computational Fluid Dynamics (CFD) analysis to investigate the effects of height and diameter variations. Their findings revealed valuable insights into the interplay between these dimensions and critical parameters. The study demonstrated that increasing chimney height resulted in enhanced dispersion of pollutants, suggesting that taller chimneys facilitate more effective pollutant dispersal and potentially reduce environmental impact. Furthermore, the analysis indicated that increasing chimney diameter led to a decrease in wind-induced pressure drop, highlighting the importance of diameter in mitigating pressure differentials between the interior and exterior of the chimney when subjected to wind forces.

H. J. Cui and Q. Yan (2017) conducted a CFD simulation study to explore the performance of a passive solar chimney. The authors presented Cp values, indicating the pressure coefficient, for various wind speeds and directions. Notably, the study revealed a maximum Cp value of 0.8 at the chimney's uppermost section. These findings offer valuable insights into the effectiveness of passive solar chimneys as a means of natural ventilation in buildings, emphasizing their potential for enhancing indoor air quality and thermal comfort.

J. Revuz et al. (2012) conducted a study focusing on the appropriate size of the flow domain surrounding the buildings. The researchers identified that existing guidelines, which were primarily designed for low to mid-rise buildings, often result in excessively large computational domains when applied to high aspect ratio structures (i.e., slender and tall buildings). Recognizing this discrepancy, the study aimed to address the need for more accurate guidelines specifically tailored to tall buildings. By considering the height-to-width ratio of the buildings, the authors provided details and recommendations for determining the optimal size of the flow domain.

Y. K. Park, K. H. Kwon, and K. Y. Kim (2015) conducted a study focusing on the evaluation of pressure coefficient (Cp) values at various locations along a circular chimney under different wind directions and wind speeds. Notably, the study revealed a maximum Cp value of 0.6 at the chimney's topmost section. These findings underscore the significance of accounting for wind effects when analysing circular chimneys, emphasizing the potential impact of wind on their performance. By considering Cp values, this study provides insights into the pressure distribution and flow characteristics around circular chimneys.

El Hassan et al. (2010) explored the wind flow characteristics around an industrial chimney through the application of Computational Fluid Dynamics (CFD) simulations. The primary focus was to investigate the impact of wind speed and chimney height on flow patterns and pressure distribution. The findings of the study revealed a notable relationship between wind speed and wind-induced pressures on the chimney. Specifically, as wind speed increased, higher pressures were observed acting upon the chimney.

Alok David John et al.(2011), special emphasis was placed on investigating the bending moment resulting from across-wind vibration, particularly when interference occurs at a 45-degree angle of wind incidence. The research revealed that across-wind vibration is more prevalent in this specific scenario. The bending moment induced by across-wind vibration in the presence of interference was found to be approximately twice as large as that observed in the standalone condition. To mitigate across-wind vibration, strakes were installed over the top one-third height of the chimney. The findings indicated a significant reduction in both the magnitude of across-wind bending moment and the resultant bending moment in the presence of strakes.

Gupta and Singh (2015) investigated the wind effects on industrial chimneys and analysed the pressure distribution using CFD simulations. Their results revealed that changes in wind speed, wind direction, and chimney dimensions directly influenced the pressure distribution along the chimney surface. The study demonstrated that varying these parameters led to variations in Cp values and pressure patterns, highlighting the need for a comprehensive understanding of wind effects on chimneys in order to optimize their design.

In the study conducted by **Wang and Zhang (2017),** numerical simulations were performed on circular chimneys to explore wind effects. The results showed variations in Cp and pressure distribution along the chimney surface, providing insights into the influence of wind speed and direction. The findings emphasized the significance of considering these factors during the design process to ensure optimal performance and structural integrity of circular chimneys.

Kumar and Sharma (2018) focused on tall chimneys and conducted a detailed analysis of wind effects using CFD simulations. Their research revealed that changes in chimney height, diameter, and wind speed resulted in variations in pressure along the chimney surface. The study provided valuable insights into the pressure distribution patterns and emphasized the importance of considering these parameters for accurate modelling and analysis of tall chimneys subjected to wind loads.

Patel and Desai (2016) conducted a computational analysis of wind effects on chimney structures, examining the pressure distribution using CFD simulations. Their results highlighted the influence of wind speed, wind direction, and chimney dimensions on Cp values and pressure distribution along the chimney surface. The study emphasized the importance of accurate modelling and analysis to ensure the structural integrity of chimneys under wind loads.

C.V. Siva Rama Prasad et al (2018), focus was on the dynamic response of chimneys to earthquake and wind loads, considering the changes in chimney geometry. The research conducted by C.V. Siva Rama Prasad et al. emphasized the importance of analysing the structural response of chimneys to these loads. It was observed that shear force and bending moment increased with higher zone factor values, indicating the increased vulnerability of chimneys to seismic forces. Additionally, the study highlighted the significant impact of wind forces on chimneys, particularly for a 100m RCC Chimney.

CHAPTER 3: METHODOLOGY

3.1 CFD METHOD FOR ANALYSIS

CFD simulation using ANSYS involves the utilization of ANSYS software suite's Computational Fluid Dynamics capabilities to analyze and predict air flow behavior in diverse engineering applications. The process comprises several steps, including preprocessing, meshing, solver setup, solution, and postprocessing. In the preprocessing stage, the model is prepared by defining boundary conditions, fluid properties, and numerical parameters. Meshing involves dividing the geometry into discrete elements, and ANSYS offers various meshing methods. Solver setup entails selecting appropriate solver options to solve the governing equations of air flow. The solution process involves iterative calculations until convergence is reached, followed by postprocessing to visualize and analyze results. The iterative nature of CFD simulations allows for validation and refinement of the model. ANSYS CFD simulations find extensive application in industries such as aerospace, automotive, energy, and chemical engineering, enabling the study and optimization of aerodynamics, heat transfer, multiphase flows, and fluid-structure interactions. Expertise in fluid dynamics, numerical methods, and ANSYS software is crucial to ensure accurate and reliable results through proper validation and verification procedures.Fig.3.1 show the proposed methodology.

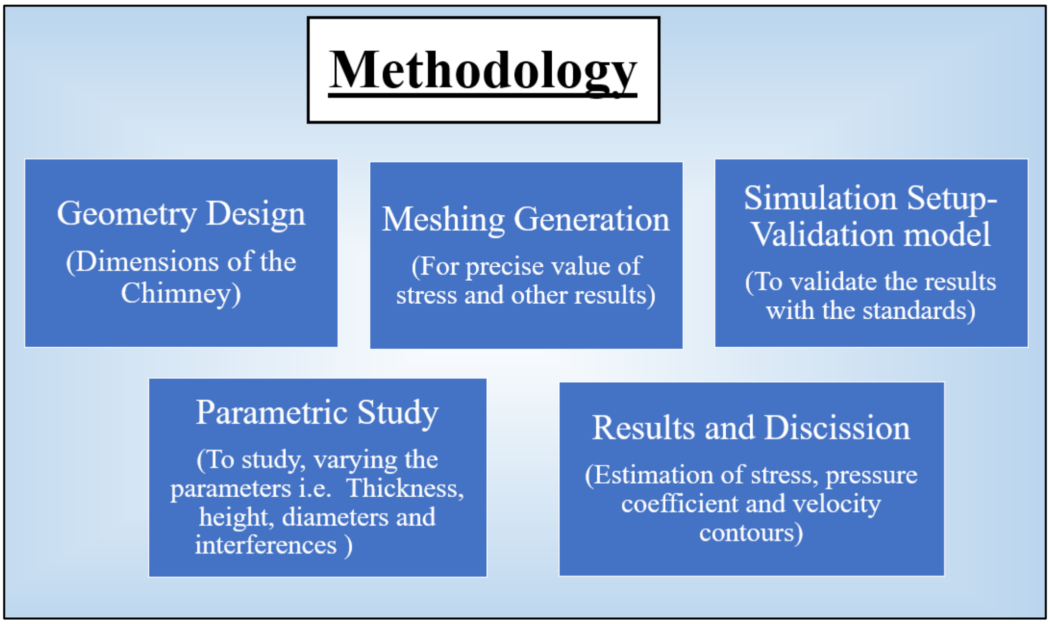


Fig.3.1: Methodology for Project

CFD simulation using ANSYS involves a step-by-step process to analyze the airflow over an airfoil as an example. In the preprocessing stage, the geometry of the airfoil is defined, including its shape and dimensions, along with setting the boundary conditions and specifying the fluid properties. Then, a mesh is generated to discretize the airfoil geometry into smaller elements using an unstructured mesh with triangular cells, with increased mesh density near the airfoil surface for improved accuracy. Moving on to the solver setup, an appropriate solver, such as the Reynolds-Averaged Navier-Stokes (RANS) solver, is selected, and a turbulence model like the Spalart-Allmaras model is chosen to capture turbulence effects. Numerical parameters are set, including convergence criteria and time step size for unsteady flows. The simulation is then run, with the solver iteratively solving the governing equations until convergence is achieved. Postprocessing involves visualizing the results through contour plots of velocity, pressure distribution, and extracting quantitative data such as lift and drag coefficients. The accuracy of the results can be validated by comparing them with experimental data or theoretical solutions. Overall, the CFD simulation process in ANSYS allows engineers to gain insights into the airflow over the airfoil, understand the forces acting on it, and optimize its aerodynamic performance.

3.2 GEOMETRY

During this stage, the geometry of the model is established, utilizing the specified dimensions outlined in the study (Figure). This can be accomplished by employing 3D CAD software or by importing an already existing model. It is crucial to define the geometry with utmost precision to minimize errors or inaccuracies that may have a substantial impact on the accuracy of the simulation outcomes.

In the geometry step of the CFD simulation for an industrial chimney in the context of your thesis, the accurate representation of the chimney's geometry is of utmost importance. This step involves creating or importing a model that precisely depicts the dimensions, shape, and overall structure of the chimney. Ensuring a high level of accuracy in defining the chimney's geometry is crucial as any errors or discrepancies can significantly impact the reliability and validity of the simulation results obtained. Key parameters such as the chimney's height, diameter, and the locations of inlet and outlet points need to be precisely defined to effectively capture the flow behavior within the chimney. Additionally, any additional features or components present in the chimney, such as baffles or flue gas outlets, should be included in the geometry model. A meticulous and precise representation of the chimney's geometry in the simulation facilitates a more realistic and reliable analysis, enabling valuable insights into flow patterns, pressure distribution, and other pertinent parameters that contribute to the understanding and optimization of the chimney's performance.

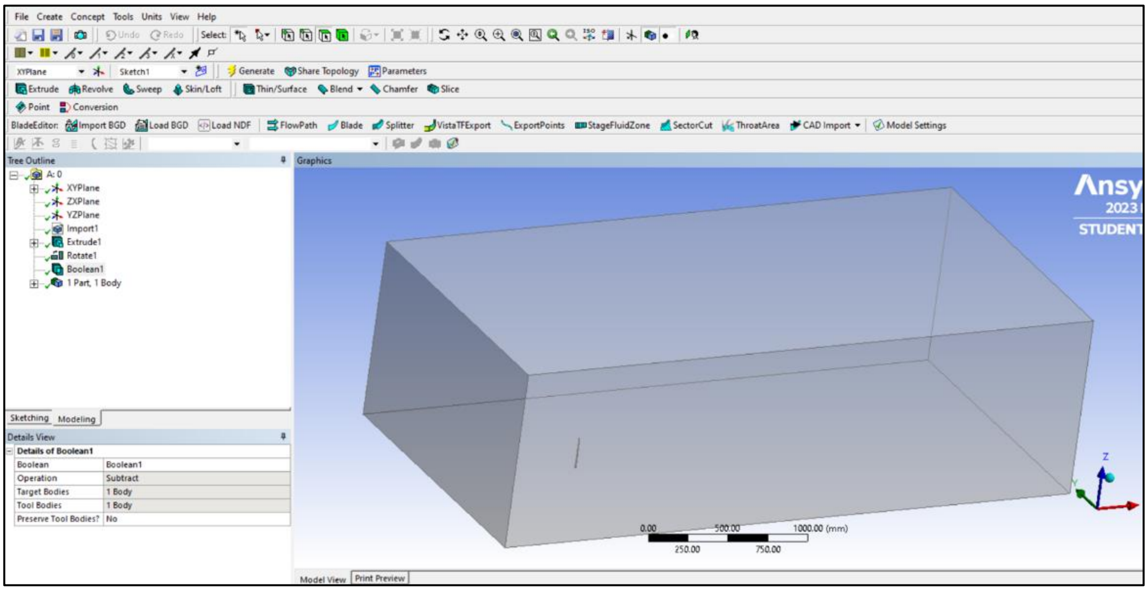


Fig.3.2: Geometry of Chimney and its Domain

As shown in Fig.3.2, dimensions of the wind domain are adopted based on the research paper, specifically 5H, 5H, and 5H for the width, length, and depth, respectively. The distance from the centre of the chimney to the inlet wall is set at 5H, while the distance to the outlet wall is 15H. Additionally, the dimensions of 5H and 5H are assigned to the walls on either side of the chimney, considering the centre as the reference point. The height of the wind domain is determined as 6H. It is important to note that in the given context, H represents the height of the chimney.

3.3 MESHING

The meshing step in CFD simulation for a chimney plays a vital role in ensuring accurate and reliable results. It involves the division of the computational domain surrounding the chimney into a network of interconnected elements. The primary objective is to discretize the geometry in a manner that effectively captures the complex flow physics occurring within the system. A well-designed mesh is crucial as it needs to strike a balance between resolving important flow features and maintaining computational efficiency.

In the context of simulating an industrial chimney, a combination of structured and unstructured meshing techniques is commonly employed. Structured meshes are often used in areas where the geometry exhibits regularity and symmetry, allowing for a more efficient representation of the flow. On the other hand, unstructured meshes are suitable for capturing complex geometries and irregular shapes.

To accurately capture boundary layer effects and gradients near the chimney walls, the mesh in these regions is typically refined. This refinement ensures that the mesh is fine enough to capture the small-scale variations in velocity, temperature, and other flow parameters that occur near the wall. Additionally, specific considerations are given to the height and thickness of the chimney when determining the appropriate mesh resolution.

Obtaining a high-quality mesh is of utmost importance as it directly impacts the accuracy and reliability of the subsequent simulation stages. A well-designed mesh enables the simulation to capture the intricate flow phenomena occurring within and around the chimney, leading to more insightful and meaningful results. Therefore, meticulous attention is given to the meshing process to ensure that it adequately represents the geometry and facilitates an accurate analysis of the air flow dynamics in the industrial chimney. As shown in Fig.3.3, the tetrahedron meshing has been done.

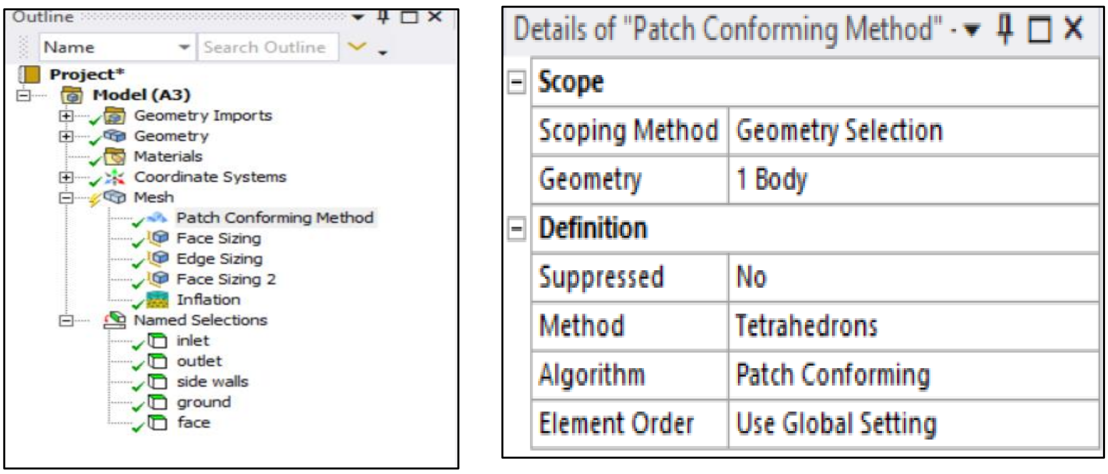


Fig.3.3: Name of faces and Mesh type

During the meshing step of the analysis, specific considerations are given to the meshing parameters for the chimney. The face sizing, representing the mesh size, is set at 10mm for the chimney's faces, while the edge sizing is determined to be 20mm along the chimney's edges. This mesh refinement allows for capturing finer details and resolving boundary layer effects near the chimney's surface. Furthermore, a larger mesh size of 40mm, denoted as face sizing 2, is utilized for the ground part of the model. This differentiation in mesh size ensures an appropriate balance between accuracy and computational efficiency in different regions of the model. The chosen mesh type for the simulation is Tetrahedron Mesh, which is commonly used for its ability to handle complex geometries and provide reliable results. By employing this meshing approach, the analysis aims to accurately represent the chimney's geometry, capturing the flow physics and facilitating a comprehensive understanding of the airflow characteristics around and within the industrial chimney.

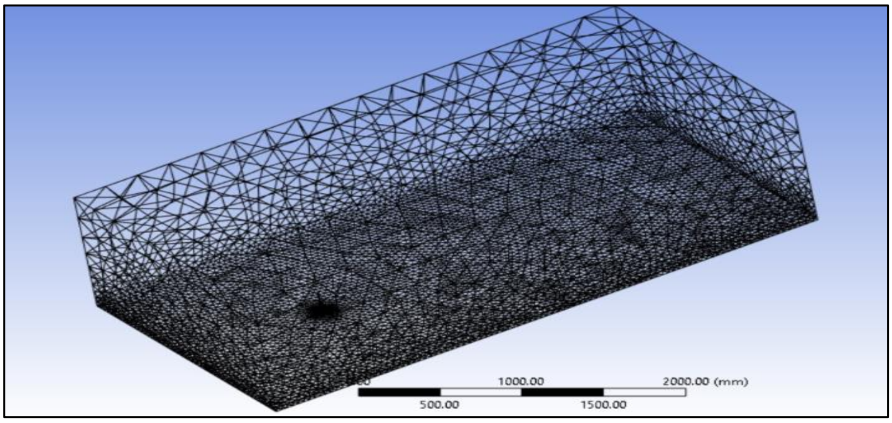


Fig.3.4: Meshing of Geometry

The meshing of a geometry is shown in Fig.3.4.Tetrahedral meshing is a widely used technique in computational fluid dynamics (CFD) simulations for representing complex geometries. This meshing approach involves dividing the computational domain into tetrahedral elements, which are four-sided polyhedral with triangular faces. Each tetrahedron is defined by four vertices that are interconnected to form a mesh. Tetrahedral meshes are particularly well-suited for irregular or intricate geometries as they can conform to complex shapes and capture the details of the flow field accurately. The ability to handle non-uniform and unstructured meshes makes tetrahedral meshing highly flexible and versatile in various engineering applications.

One of the advantages of tetrahedral meshes is their ability to capture complex boundary layers and regions of high gradient accurately. The small volume of tetrahedral elements near the boundaries allows for better resolution of flow phenomena occurring in these regions. Additionally, tetrahedral meshes can efficiently represent curved surfaces and handle the presence of geometric features such as corners, edges, and protrusions.

3.4 SETUP

The setup procedure in CFD simulation plays a critical role in ensuring accurate and reliable results. It involves several key steps that are essential for capturing the behaviour of the flow field effectively. The accuracy of the simulation outcomes is heavily dependent on the quality of the setup, as it directly influences the representation of the physical phenomena involved.

To begin the setup procedure, boundary conditions are defined, which establish the interactions between the model and the external environment. These boundary conditions can include physical parameters such as inflow velocity, pressure, and temperature, as well as numerical specifications like wall functions and symmetry planes. As shown in Fig.3.5, the setup of simulation is similar to Figure.

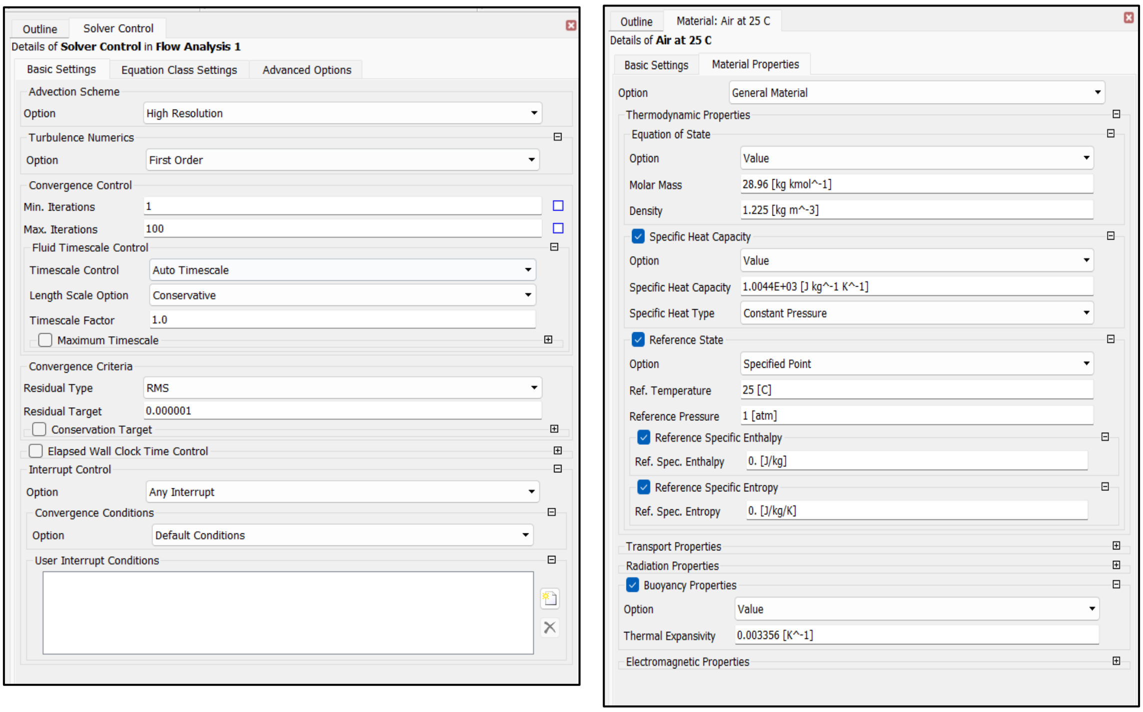


Fig.3.5: Setup and Material Properties

In CFD simulations, the turbulence model is employed to characterize the behaviour of turbulent kinetic energy and its dissipation rate. The selection of an appropriate turbulence model depends on factors such as the Reynolds number and flow characteristics. Commonly used turbulence models include the k-epsilon, k-omega, and SST models. Once the boundary conditions, fluid properties, and turbulence model have been determined, the numerical solver can be configured.

The numerical solver plays a crucial role in discretizing the governing equations of the flow field and iteratively solving them over a specified time domain. The accuracy of the solver is influenced by parameters such as the time step, convergence criteria, and numerical scheme employed. Careful consideration of these parameters is necessary to ensure accurate and reliable results.

Once the numerical solver has been set up, the simulation can be executed. The output of the simulation provides valuable data that can be visualized and analysed to gain insights into the flow field. It is important to emphasize that the quality of the simulation results is heavily dependent on the accuracy and precision of the setup procedure. Therefore, it is crucial to perform the setup process with meticulous attention to detail and careful consideration of the specific requirements of the problem at hand. By adhering to a rigorous setup procedure, engineers can enhance the accuracy and reliability of the simulation results and make informed decisions based on the insights obtained.

3.5 BOUNDARY CONDITIONS

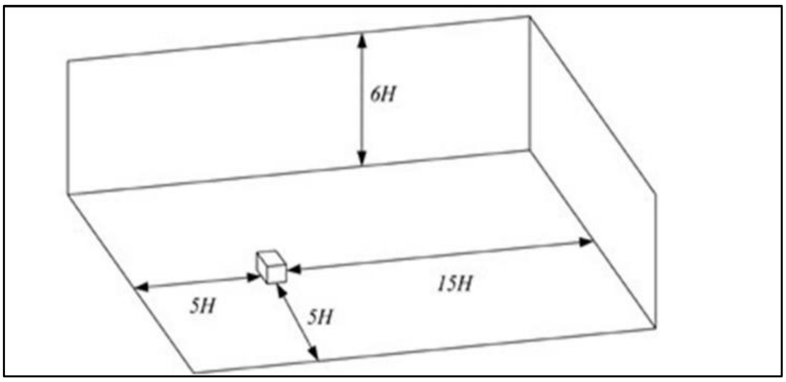


Fig.3.6: Wind Domain

In numerical simulations, it is crucial to define accurate boundary conditions for solving various problems. To ensure the accuracy of the simulation, large dimensions are utilized in the applied domain of the virtual wind tunnel to prevent any back-formation. In this context, the building model is positioned on the virtual wind tunnel's ground with specific dimensions. The pressure coefficients and velocity fields are highly influenced by the size of the computational domain, aligning with observations from wind tunnel studies conducted over many years.

For optimal results, the desirable domain dimensions are as shown in Fig.3.6. It is desirable to assume infinitely distant boundaries, which would be achieved with a large domain. This allows for reliable and high-quality simulation outcomes. On the other hand, the small domain is insufficient in providing accurate results due to its limited size. Therefore, careful consideration is given to selecting an appropriate domain size to ensure the reliability and accuracy of the numerical simulation for the given problem.

3.6 SOLUTION

The solution phase in CFD simulation of a chimney involves iteratively solving the governing equations of air flow to obtain numerical solutions. The solution process starts with initializing the flow field with initial conditions and then solving the equations using a suitable numerical solver. During each iteration, the solver calculates the values of velocity, pressure, and other flow properties based on the discretized domain and the defined boundary conditions.

The solution process continues until convergence is achieved, which means that the solution no longer undergoes significant changes with each iteration. Convergence is typically determined by monitoring key parameters, such as residual errors or flow variables, and comparing them against predefined convergence criteria.

Once convergence is reached, the solution is considered stable and can be postprocessed for further analysis. Postprocessing involves visualizing and analyzing the obtained results, such as flow patterns, pressure distribution, temperature profiles, and other relevant quantities of interest. These results provide valuable insights into the flow behavior and performance of the chimney.

During the solution phase, it is important to ensure that the numerical solver settings, such as the time step, convergence criteria, and numerical schemes, are appropriately chosen to achieve accurate and reliable results. Additionally, validation and verification procedures should be conducted to assess the accuracy of the simulation by comparing the results with available experimental or analytical data.

3.7 RESULTS

In the final stage of the CFD simulation, the obtained results are meticulously analyzed and interpreted to assess the simulation's performance and validate the model against any available experimental data. This involves conducting post-processing on the simulation results to generate visual representations of the flow field and calculating essential metrics like pressure drop, velocity profiles, and other relevant parameters of interest. The analysis of the results is conducted with utmost care to ensure their accuracy, reliability, and suitability for drawing meaningful conclusions. By carefully scrutinizing the simulation results, engineers can gain valuable insights, validate the model's accuracy, and make informed decisions based on the findings.

3.8 VALIDATION

To validate the software, a circular building with a base diameter of 200mm and a height of 600mm was chosen. The boundary conditions and meshing were set up according to the specifications mentioned earlier. In the validation model, the coefficient of pressure (Cp) was determined to be 0.998. Comparing this value with the one specified in IS 875 Part 3, which states that Cp should be 1 for the same height-to-diameter (H/D) ratio, we can observe that the Cp value obtained from the simulation is approximately the same. This suggests that the software's results align with the expected values and provide a valid representation of the flow behavior around the circular building.

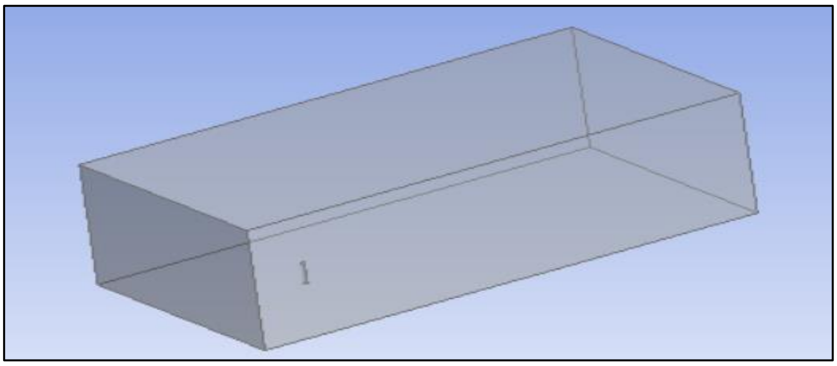
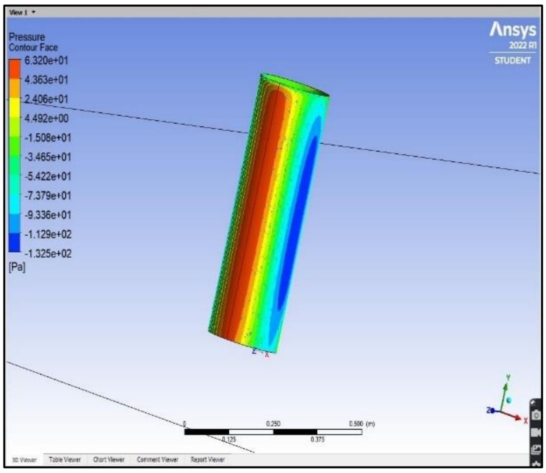


Fig.3.7: Cylindrical Geometry for Validation



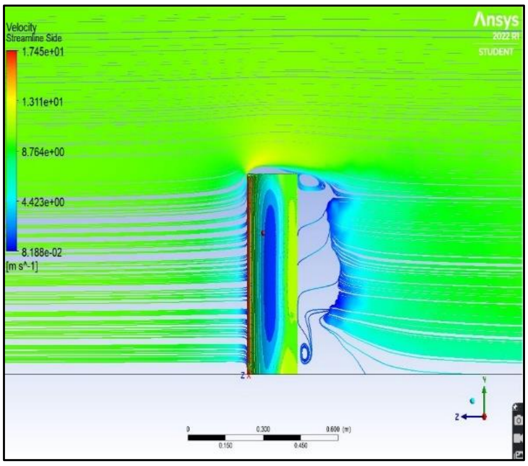


Fig.3.8: Pressure Contour and Velocity Streamline

As shown in Fig.3.7 and Fig.3.8, the geometry and results for the validation model.

The coefficient of pressure (Cp) is a dimensionless parameter used to quantify the local pressure variation on a surface relative to the free-stream pressure. It is a fundamental concept in fluid dynamics for assessing the pressure distribution around an object.

The formula to calculate the coefficient of pressure (Cp) is given as:

$$Cp = \frac{(P - P0)}{(0.5 * \rho * V2)}$$

Here, Cp represents the coefficient of pressure, P denotes the local pressure on the surface, P0 represents the free-stream pressure, ρ symbolizes the fluid density, and V signifies the velocity of the fluid relative to the object.

The application of this formula enables the determination of the Cp value, providing valuable insights into the pressure distribution and aerodynamic characteristics of the flow around the object.

CHAPTER 4: STAND-ALONE CHIMNEY SIMULATION

4.1 PROBLEM STATEMENT

In the CFD simulations, a chimney model has been chosen and scaled down at a ratio of 1mm to 1m (1:1000) to ensure a manageable computational domain. The chimney model has a height of 200mm, with a top diameter of 8mm and a bottom diameter of 12mm. It is designed with a uniform thickness of 0.4mm throughout its structure.

These dimensions and scaling factors are crucial in accurately representing the chimney in the CFD simulations. By maintaining the appropriate proportions and scale, the simulation can capture the flow behavior and characteristics around the chimney with greater precision. This enables engineers and researchers to analyze and evaluate the chimney's performance, including factors such as aerodynamic efficiency and flow patterns, while ensuring that the simulation results are reliable and applicable to real-world scenarios.

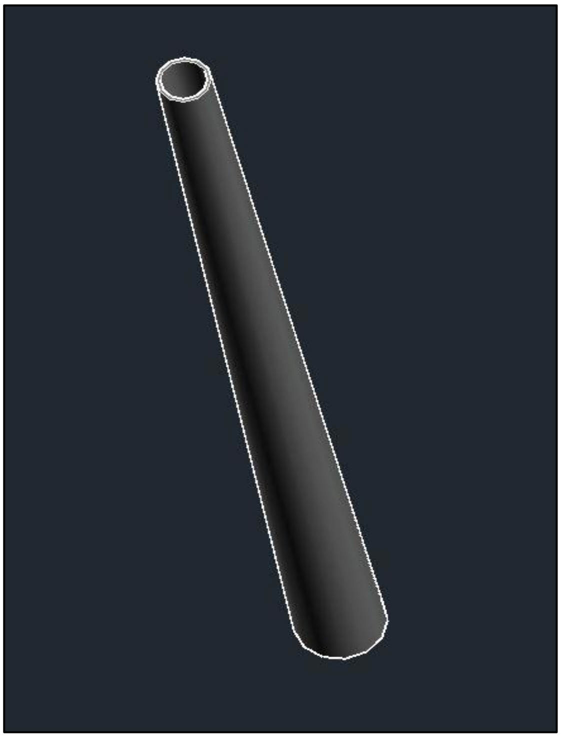


Fig.4.1: Auto-CAD model of Chimney

To examine the chimney's response to different wind directions, the incident wind angle is initially set at 0 degrees. However, to capture a comprehensive understanding of the chimney's behavior under varying wind conditions, the wind tunnel domain is rotated in increments of 15 degrees up to 180 degrees. This allows for a thorough investigation of the chimney's aerodynamic characteristics and how it interacts with the incident airflow from different angles.

By systematically analyzing the chimney's response at different wind angles can gain valuable insights into its performance, including the effects of wind direction on airflow patterns, pressure distribution, and potential turbulence. This information is essential for optimizing the chimney's design and ensuring its structural integrity in real-world scenarios.

4.2 GEOMETRY

The chimney's geometry aligns with the specifications outlined in the provided problem statement (4.1). The wind domain dimensions, as described in section 3.2, are employed to define the computational domain for the simulation. The chimney model is subjected to incident wind angles ranging from 0 degrees to 180 degrees, with intervals of 15 degrees.

By incorporating the specified geometry and wind conditions, the simulation can accurately capture the behavior of the airflow around the chimney from various incident angles. This comprehensive analysis enables engineers to assess the chimney's response to different wind directions and understand how it influences factors such as aerodynamic performance, pressure distribution, and flow patterns. The fig 4.1 shows the geometry with different wind angles.

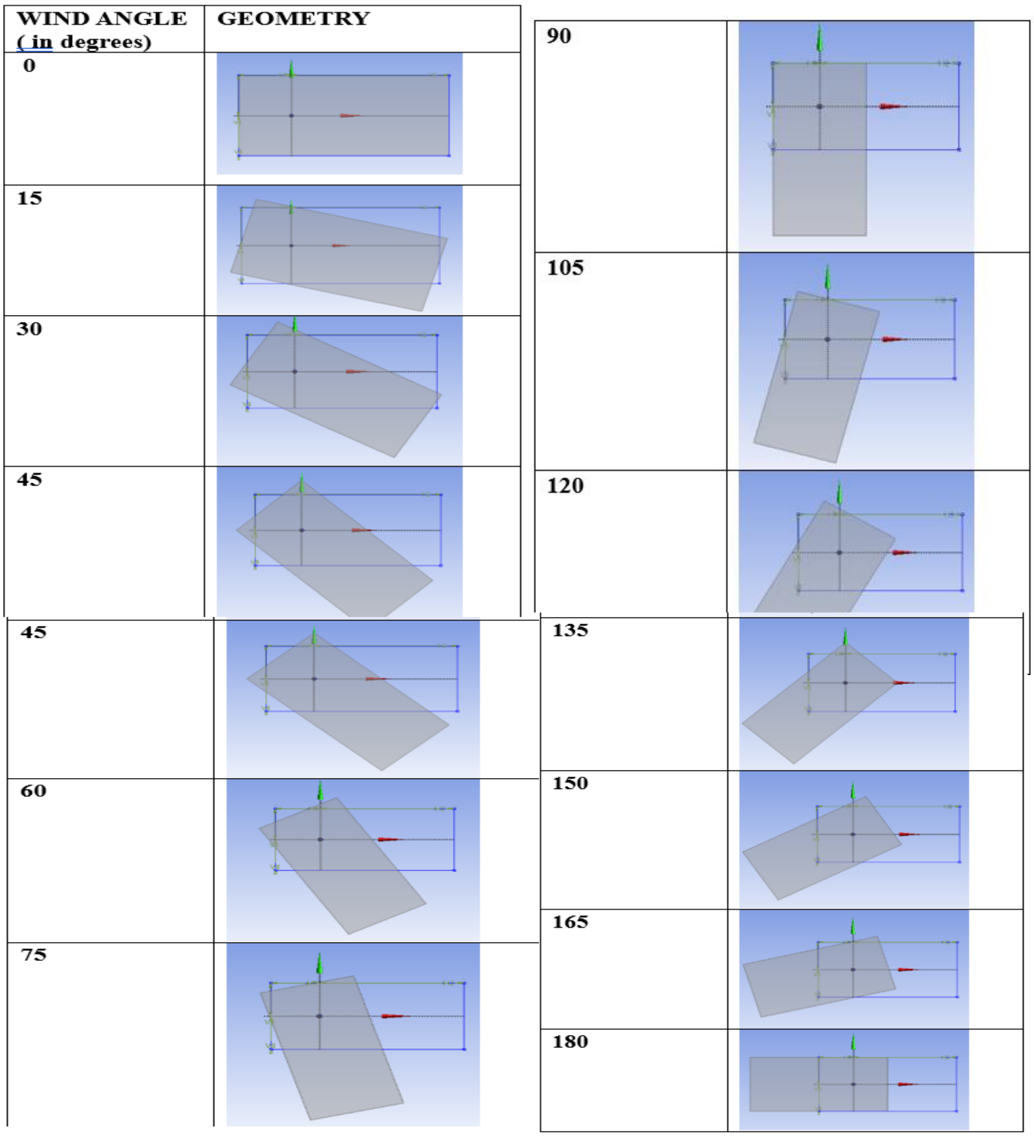


Fig.4.2: Geometry of Chimney with different wind angle

4.3 MESHING

The meshing of the chimney and its associated wind domain is presented in figure 4.3, as described in the provided documentation. The figure illustrates the discretization of the computational domain into interconnected elements, forming a mesh. The meshing process is crucial as it determines the accuracy and reliability of the simulation results.

By dividing the geometry into smaller elements, the mesh captures the intricate details and flow physics around the chimney. It allows for the accurate representation of boundary layer effects, gradients, and other important flow features. The mesh near the chimney walls may be refined to ensure precise capturing of boundary layer phenomena.

The meshing technique employed in this simulation provides a balance between computational efficiency and capturing the required level of detail. It ensures that the mesh resolution is appropriate to accurately simulate the flow behavior and capture the important aspects of the chimney and its surrounding wind domain.

The presented meshing scheme serves as a foundation for the subsequent analysis and enables the simulation to provide reliable and insightful results regarding the air flow characteristics and aerodynamic performance around the chimney.

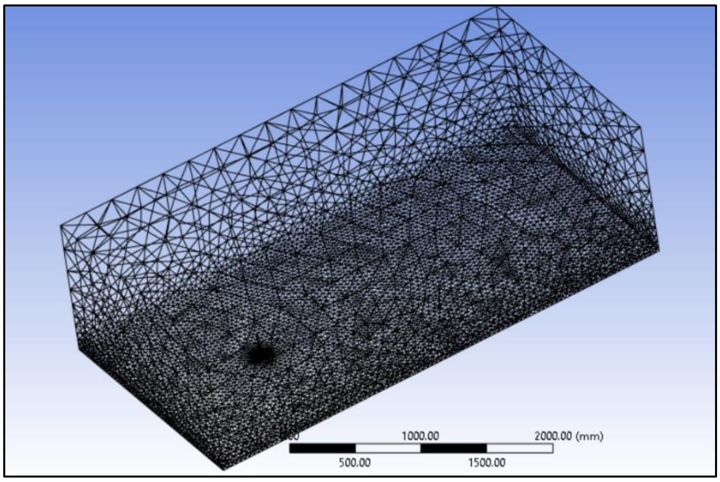


Fig.4.3: Meshing of model.

4.4 SETUP

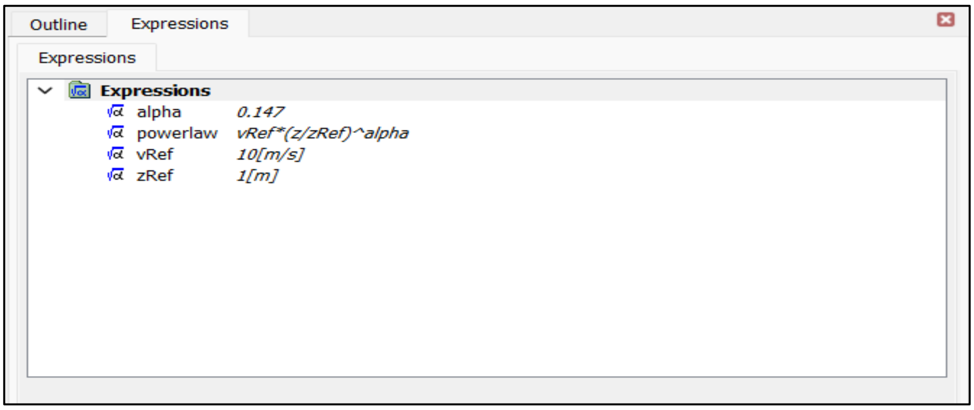


Fig.4.4: Power-law

The wind speed chosen for a CFD simulation is a critical decision that depends on the specific requirements and objectives of the study. While a commonly used value is 10 m/s, it may vary based on the application and the desired simulation conditions. The selected wind speed should represent the typical or relevant wind conditions for the problem at hand, such as the average or design wind speed for a specific location. Additionally, it should be sufficient to induce significant flow effects and turbulence in the simulation domain, capturing important phenomena like vortices and flow separation. The accuracy and validity of the simulation results heavily rely on the appropriate selection of the wind speed, ensuring that it aligns with the study's goals and is supported by empirical data or industry standards when available. Striking the right balance between realistic conditions and generating meaningful results is crucial when determining the wind speed for CFD simulations. The setup of power law is as in Fig.4.4.

The wind speed profile within the atmospheric boundary layer can be calculated using the "**Power Law equation**." This equation relates the wind speed at a specific height, denoted as **vRef,** to the reference height **zRef,** the ground roughness parameter **α**, and a constant value of 1.0 for **zRef.** In this case, the reference wind speed **vRef** is determined as 10 m/s, while the ground roughness α can vary depending on the specific conditions of the location or terrain.

$$\frac{v}{v_{Ref}} = \left(\frac{z}{z_{Ref}}\right)^{\alpha}$$

4.5 RESULTS AND DISCUSSIONS

4.5.1 PRESSURE CONTOURS

Case 1: When the incident wind angle is **0°.**

The pressure values observed in the chimney simulation range from -59.44 Pa to 50.85 Pa, with an average pressure of -24.6691 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.97 to 0.83, with an average Cp value of -0.41. The negative average Cp value further supports the presence of suction forces acting on the chimney.

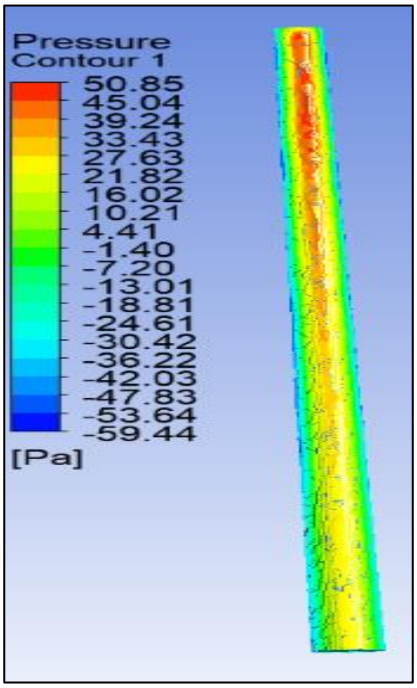


Fig.4.5: Pressure Contour at 0°

Case 2: When the incident wind angle is **15°**.

The pressure values observed in the chimney simulation range from -55.75 Pa to 49.30 Pa, with an average pressure of -20.04 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.91 to 0.7, with an average Cp value of -0.33. The negative average Cp value further supports the presence of suction forces acting on the chimney.

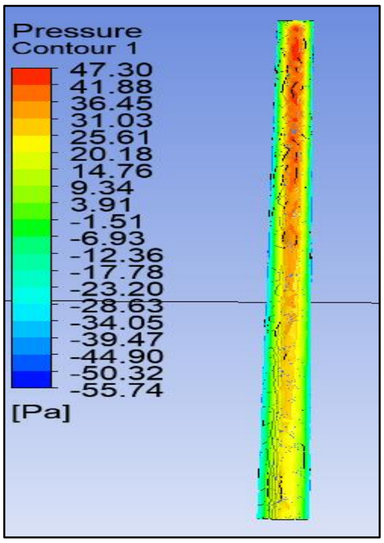


Fig.4.6: Pressure Contour at 15°

Case 3: When the incident wind angle is **30°**.

The pressure values observed in the chimney simulation range from -58.43Pa to 47.36 Pa, with an average pressure of -18.42 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.95 to 0.77, with an average Cp value of -0.30. The negative average Cp value further supports the presence of suction forces acting on the chimney.

Case 4: When the incident wind angle is **45°**.

The pressure values observed in the chimney simulation range from -53.40 Pa to 51.62 Pa, with an average pressure of -17.12 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.87 to 0.84, with an average Cp value of -0.28. The negative average Cp value further supports the presence of suction forces acting on the chimney.

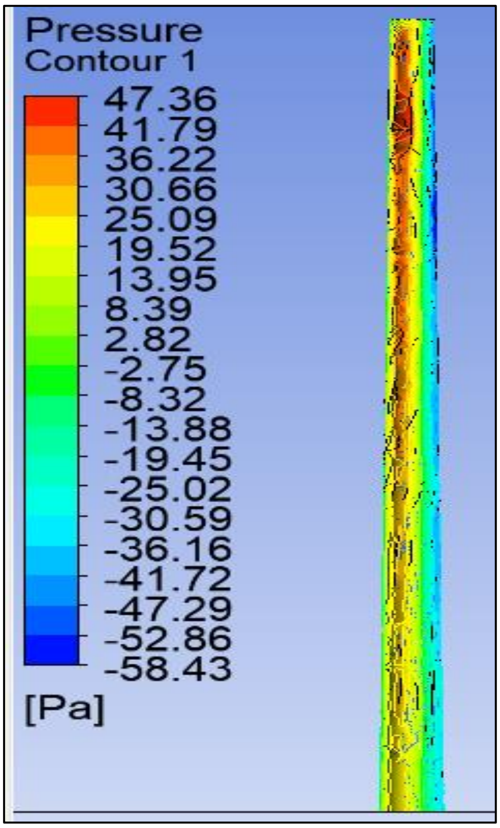


Fig.4.7: Pressure Contour at 30°

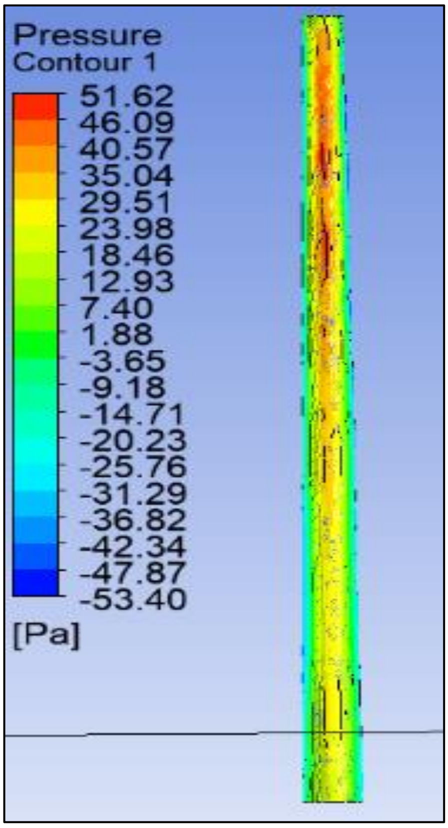


Fig.4.8: Pressure Contour at 45°

Case 5: When the incident wind angle is **60º**.

The pressure values observed in the chimney simulation range from -51.32Pa to 48.98Pa, with an average pressure of -18.32 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.84 to 0.8, with an average Cp value of -0.31. The negative average Cp value further supports the presence of suction forces acting on the chimney.

Case 6: When the incident wind angle is **75º**.

The pressure values observed in the chimney simulation range from -53.66Pa to 44.84Pa, with an average pressure of -18.94 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.88 to 0.73, with an average Cp value of -0.30. The negative average Cp value further supports the presence of suction forces acting on the chimney.

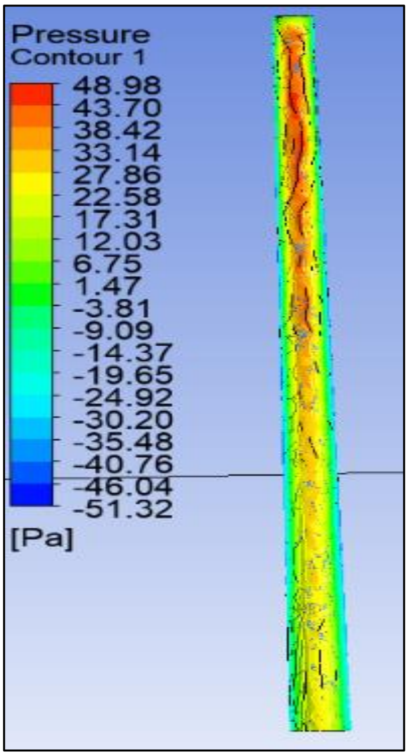


Fig.4.9: Pressure Contour at 60°

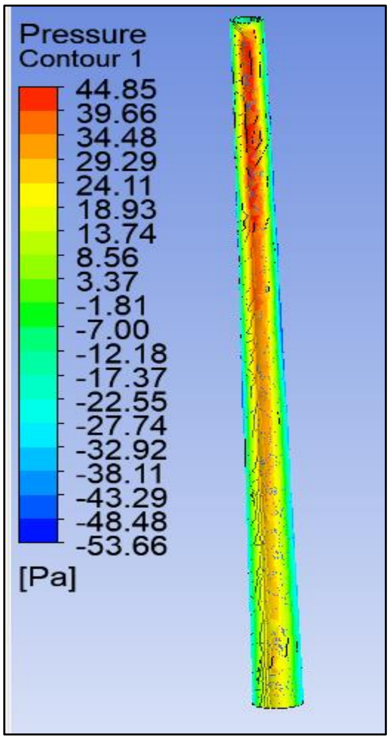


Fig.4.10: Pressure Contour at 75°

Case 7: When the incident wind angle is **90º**.

The pressure values observed in the chimney simulation range from -63.35Pa to 50.15Pa, with an average pressure of -24.47Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -1.03 to 0.82, with an average Cp value of -0.41. The negative average Cp value further supports the presence of suction forces acting on the chimney.

Case 8: When the incident wind angle is **105º**

The pressure values observed in the chimney simulation range from -55.51Pa to 47.59Pa, with an average pressure of -18.8219Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.91 to 0.78, with an average Cp value of -0.31. The negative average Cp value further supports the presence of suction forces acting on the chimney.

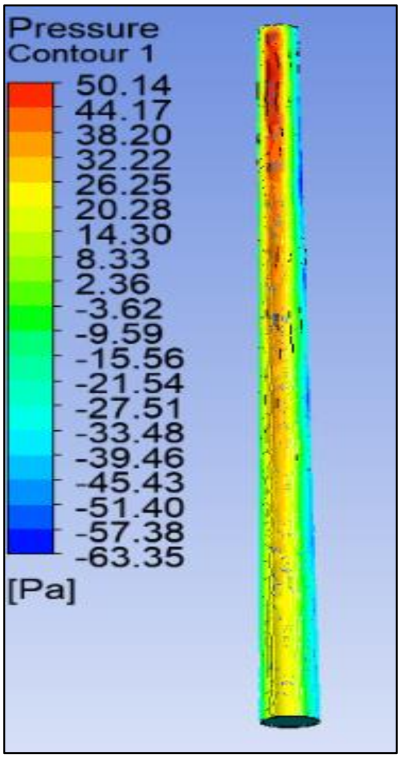


Fig.4.11: Pressure Contour at 90°

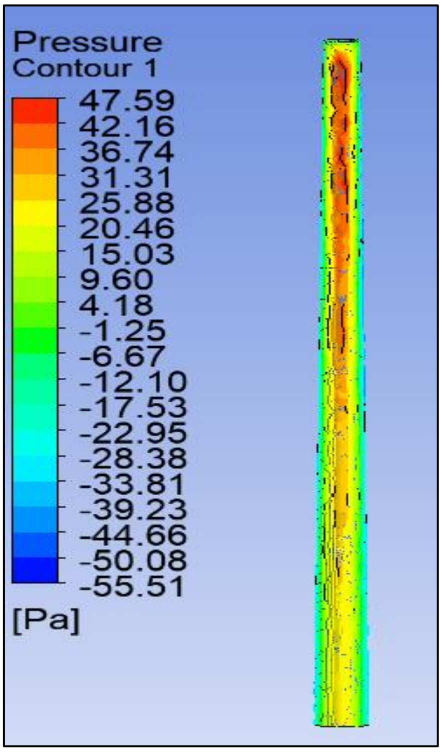


Fig.4.12: Pressure Contour at 105°

Case 9: When the incident wind angle is **120º**.

The pressure values observed in the chimney simulation range from -53.39Pa to 46.40Pa, with an average pressure of -17.542Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.87 to 0.76, with an average Cp value of -0.29. The negative average Cp value further supports the presence of suction forces acting on the chimney.

Case 10: When the incident wind angle is **135º**.

The pressure values observed in the chimney simulation range from -53.40 Pa to 51.62 Pa, with an average pressure of -17.12 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.87 to 0.84, with an average Cp value of -0.28. The negative average Cp value further supports the presence of suction forces acting on the chimney.

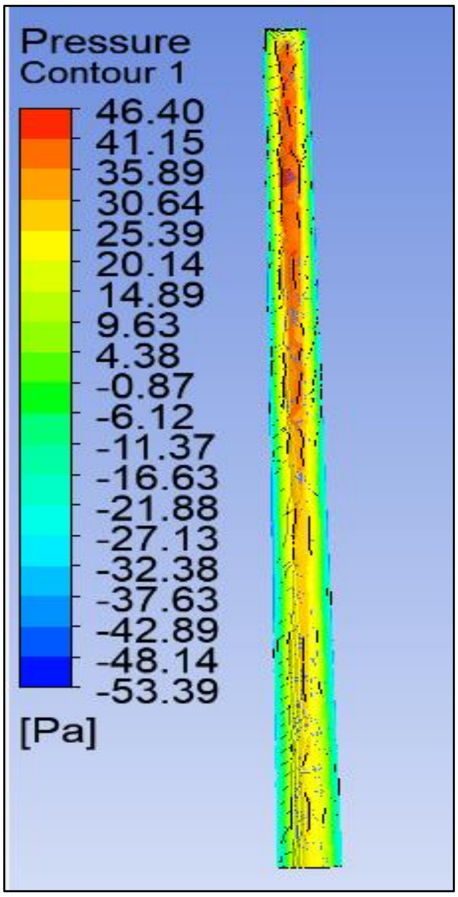


Fig.4.13: Pressure Contour at 120°

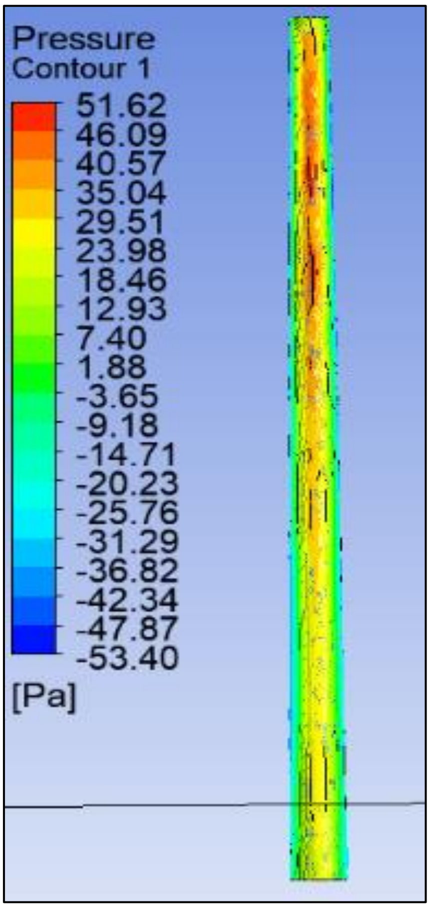


Fig.4.14: Pressure Contour at 135°

Case 11: When the incident wind angle is **150º**.

The pressure values observed in the chimney simulation range from -50.33 Pa to 49.85 Pa, with an average pressure of -17.51 Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.82 to 0.81, with an average Cp value of -0.29. The negative average Cp value further supports the presence of suction forces acting on the chimney.

Case 12: When the incident wind angle is **165º**.

The pressure values observed in the chimney simulation range from -50.25 Pa to 47.53Pa, with an average pressure of -19.47Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -0.82 to 0.78, with an average Cp value of -0.32. The negative average Cp value further supports the presence of suction forces acting on the chimney.

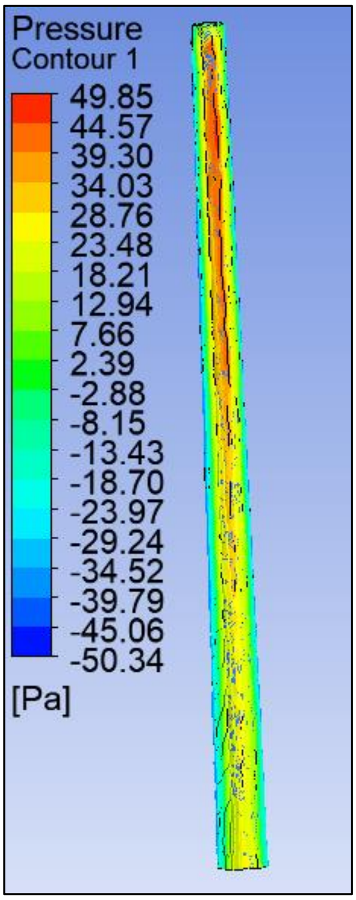


Fig.4.15: Pressure Contour at 150°

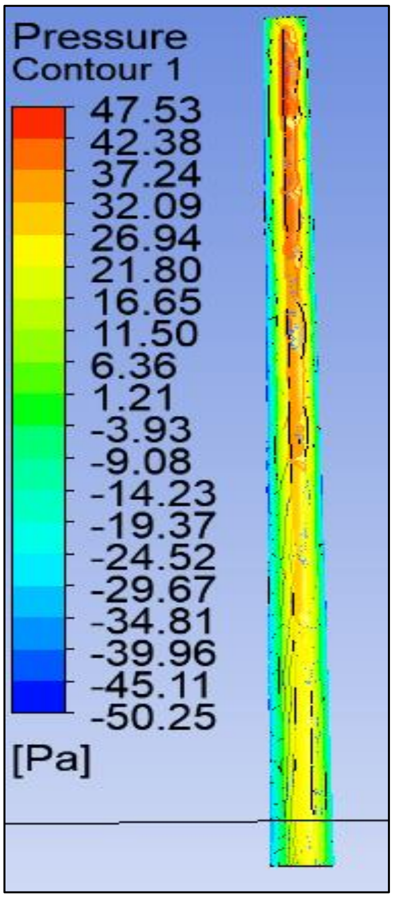


Fig.4.16: Pressure Contour at 165°

Case 13: When the incident wind angle is **180º**.

The pressure values observed in the chimney simulation range from -69.26 Pa to 53.27Pa, with an average pressure of -24.24Pa. The predominantly negative pressure values indicate a higher suction force on the chimney overall. The coefficient of pressure (Cp) varies from -1.13 to 0.87, with an average Cp value of -0.41. The negative average Cp value further supports the presence of suction forces acting on the chimney.

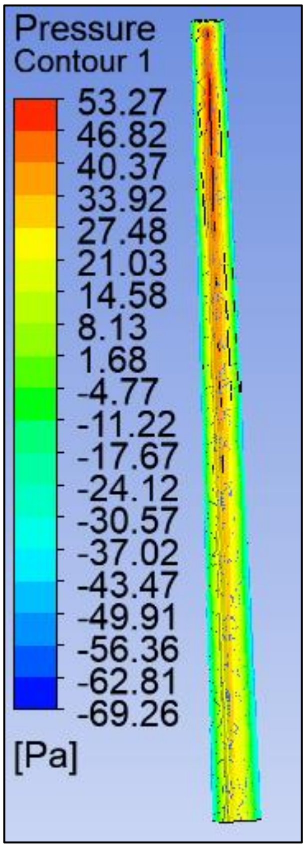


Fig.4.17: Pressure Contour at 180°

4.5.2 HORIZONTAL STREAMLINES

Horizontal streamlines are employed for visualizing the movement of air across a flat surface, enabling the identification of regions with varying levels of pressure. Utilizing this data, one can enhance the design of objects by modifying their form, dimensions, or surface characteristics in order to minimize high-pressure zones and enhance low-pressure areas. The incident wind angle and their horizontal streamlines as shown in Fig.4.18.

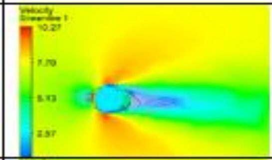
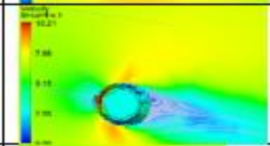
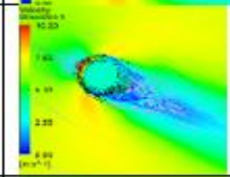
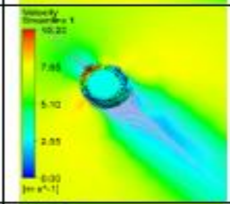
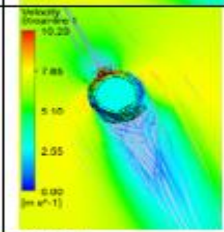
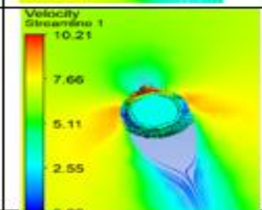
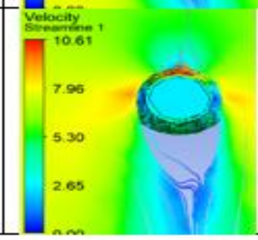
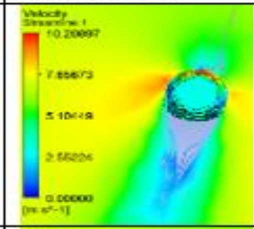
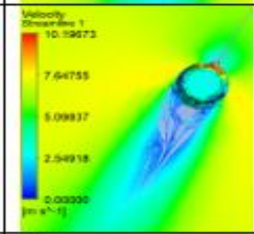
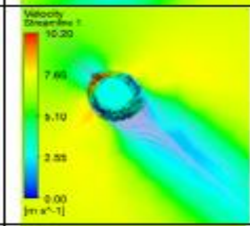
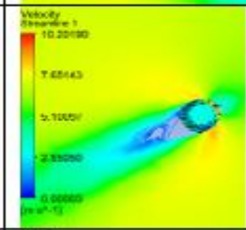
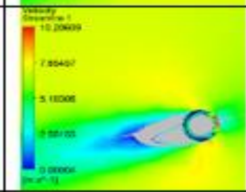
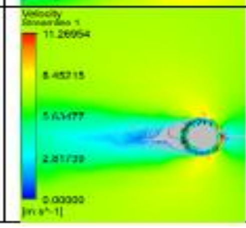
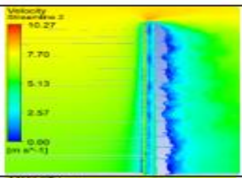
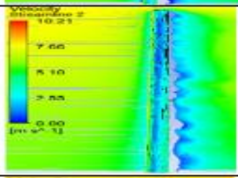
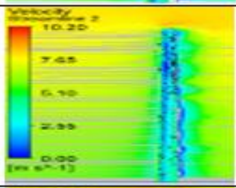
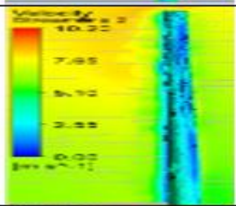
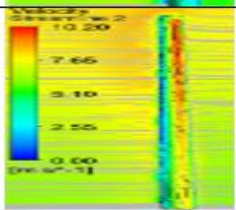
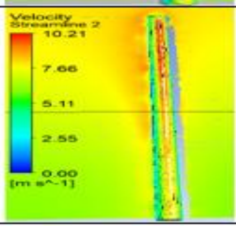
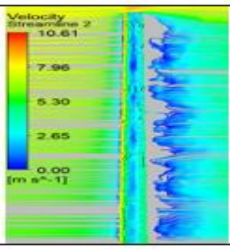
Incident Wind Angle	Horizontal Velocity Streamline
0	
15	
30	
45	
60	
75	
90	
105	
120	
135	
150	
165	
180	

Fig.4.18: Horizontal Streamlines at different wind angles

4.5.3 VERTICAL STREAMLINES

Vertical streamlines in a computational fluid dynamics (CFD) simulation of a chimney depict the flow characteristics of air or fluid in the vertical direction. They provide valuable insights into the vertical distribution and movement of the flow within the chimney and as shown in Fig.4.19.

Incident Wind Angle	Vertical Velocity Streamline
0	
15	
30	
45	
60	
75	
90	

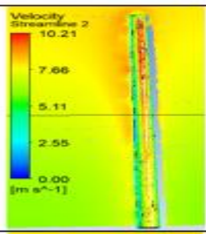
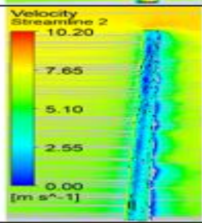
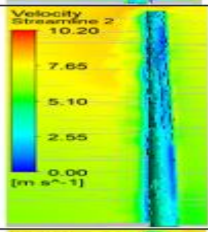
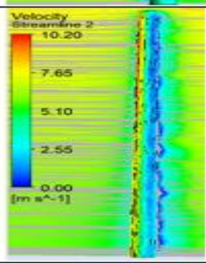
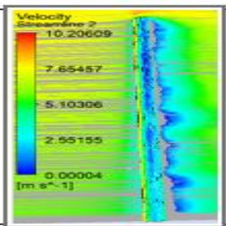
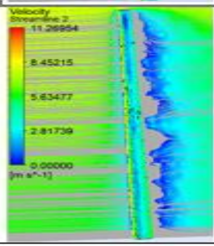
Incident Wind Angle	Vertical Velocity Streamline
105	
120	
135	
150	
165	
180	

Fig.4.19: Vertical Streamlines at different wind angles

A vertical streamline refers to an imaginary path followed by a fluid particle as it moves vertically within a fluid medium. It is a line perpendicular to the Earth's surface, with its direction determined by the velocity and flow direction of the fluid. By tracing the trajectory of a vertical streamline from ground level to the height of a structure, one can observe how the presence of the building influences the surrounding airflow and its interaction with the structure.

Vertical streamlines play a crucial role in understanding and visualizing complex three-dimensional flows in fluid dynamics. They find practical applications in designing structures like buildings, bridges, and other constructions exposed to intense wind flows. The objective is to optimize their performance and minimize potential wind-induced damage. Vertical streamlines help illustrate the downward deviation of airflow over the top of a building, resulting in additional pressure on the leeward side. This phenomenon can lead to increased wind loads exerted on the structure.

4.5.4 VARIATION OF PRESSURE WITH WIND ANGLE

Changes in the incident wind angle can affect the pressure distribution around an object or structure. The direction and velocity of the wind striking the surfaces of the object are influenced by the wind angle. Consequently, the pressure distribution on the object can change. Modifying the wind angle can result in increased or decreased pressure on different areas of the object. For instance, an oblique wind angle may cause flow separation, leading to high-pressure areas and potential turbulence, resulting in increased wind loads. Conversely, changing the wind angle may reduce pressure on specific surfaces, particularly if the object is streamlined and aligned with the wind direction.

Table 1- Maximum, Average and Minimum Pressure

Incident Wind Angle	Pressure in Pa		
	Min	Max	AVG
0	-59.44	50.85	-24.6691
15	-55.75	49.3	-20.0478
30	-58.43	47.36	-18.4239
45	-53.4	51.62	-17.1236
60	-51.32	48.98	-18.3161
75	-53.66	44.584	-18.9466

90	-63.35	50.15	-24.4447
105	-55.51	47.59	-18.8219
120	-53.39	46.4	-17.542
135	-53.4	51.62	-17.1236
150	-50.33	49.85	-17.510
165	-50.25	47.53	-19.47
180	-69.26	53.27	-24.2338

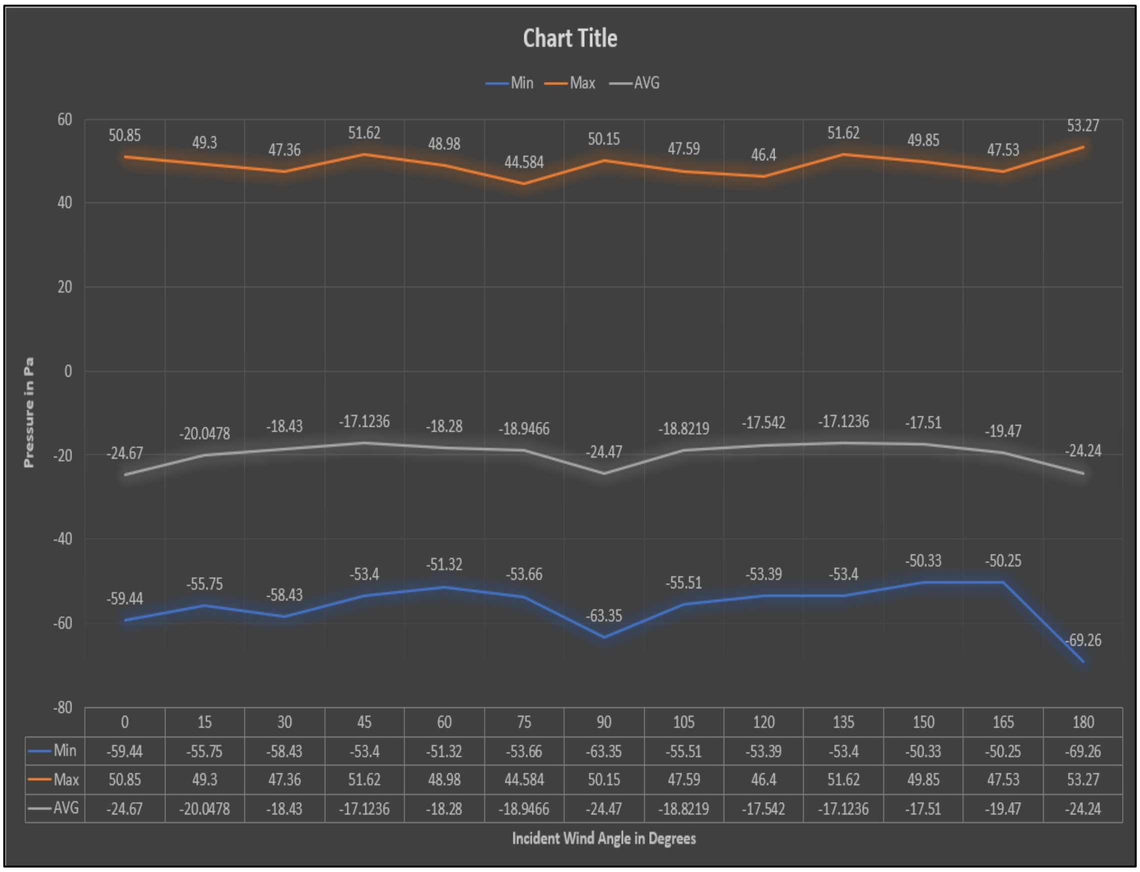


Fig.4.20: Avg., Max., Min. Pressure vs different wind angles

The above Fig.4.20 shows the variation of Pressure with different wind angles. When the wind angle increases from 0 to 90 degrees, the average pressure experiences a decreasing trend, going from -24.67 Pa to -17.12 Pa up to 45 degrees. However, beyond 45 degrees, the average pressure starts to increase, reaching -24.47 Pa at 90 degrees. This same pattern repeats in the wind angle range of 90 to 180 degrees.

Regarding the maximum pressure, it varies from 50.85 Pa to 44.58 Pa as the angle increases from 0 to 90 degrees, and then increases further to 53.27 Pa as the angle reaches 90 to 180 degrees.

In terms of negative pressure, the values initially decrease from -59.44 Pa to -51.32 Pa as the angle varies from 0 to 45 degrees. However, as the angle reaches 90 degrees, the negative pressure increases to -63.35 Pa. The same pattern persists in the wind angle range of 90 to 180 degrees.

4.5.5 VARIATION OF Cp WITH WIND ANGLE

Analysing the variation of the coefficient of pressure (Cp) with incident wind angle in a CFD simulation of an industrial chimney provides insights into the pressure distribution on the chimney surfaces at different wind angles. The above Fig.4.21 shows the Coefficient of Pressure vs different wind angles

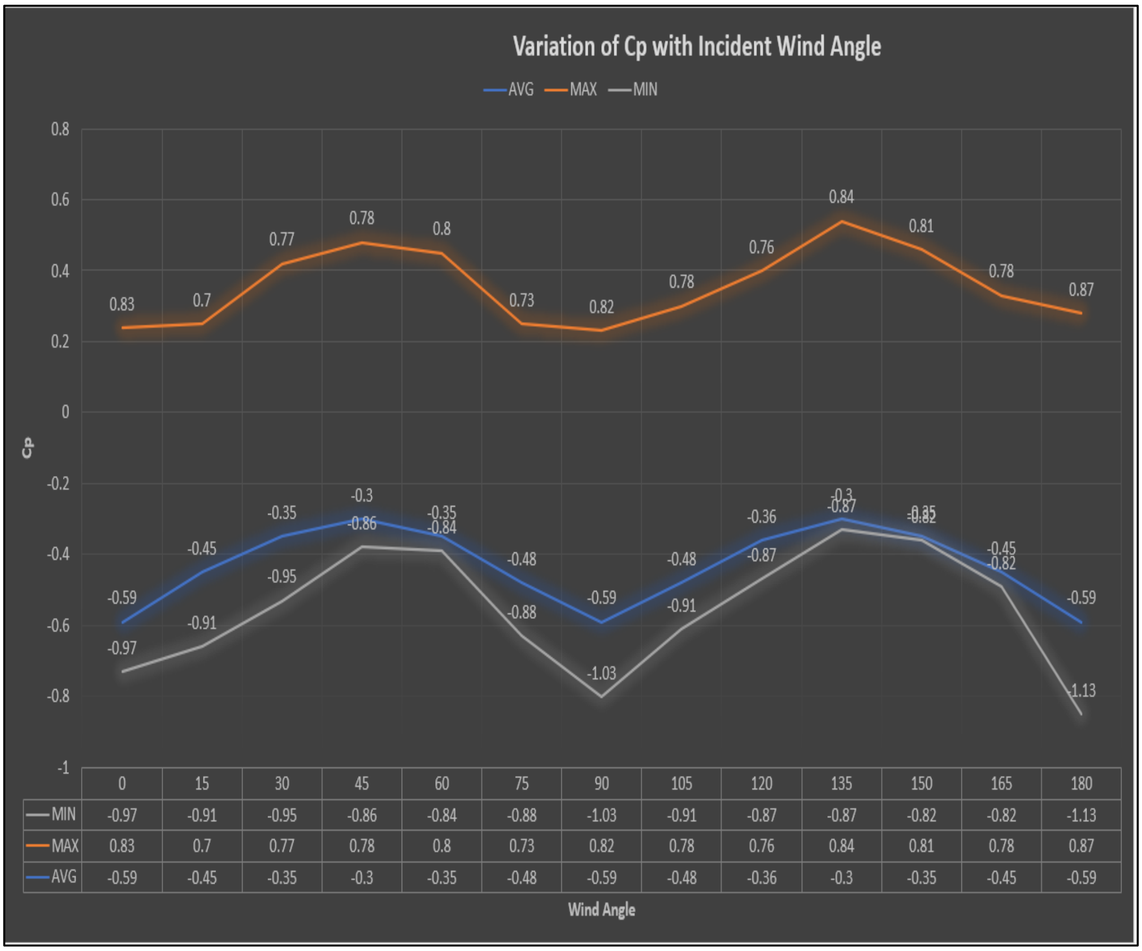


Fig.4.21: Coefficient of Pressure vs different wind angles

Cp represents the difference between local pressure and a reference pressure normalized by the dynamic pressure. Changes in incident wind angle affect the Cp distribution on the chimney surfaces. Higher Cp values indicate increased pressure regions, while lower Cp values suggest reduced pressure areas. The relationship between Cp and incident wind angle depends on factors like chimney geometry, wind speed, and flow conditions.

The Cp variation with incident wind angle is depicted in the graph, particularly concerning the maximum pressure. As the wind angle increases from 0 to 90 degrees, the Cp value fluctuates from 0.83 to 0.78, and subsequently rises to 0.82 at 135 degrees, and further to 0.84 and 0.87 at 180 degrees.

It is observed that the graph exhibits similar variations for all angles. The nature of the curve remains consistent throughout the range of wind angles.

CHAPTER 5: CHIMNEY SIMULATION FOR DIFFERENT WIND ZONES

5.1 WIND EFFECT ON CHIMNEY IN DIFFERENT ZONES

The wind velocity and its classification into different zones, as outlined by IS 875 or relevant wind load codes, significantly influence the design and performance of industrial chimneys. Industrial chimneys, being tall structures, are exposed to varying wind loads. The wind velocity associated with a specific zone determines the strength of the wind forces acting on the chimney. Higher wind velocities in a zone indicate more substantial loads that the chimney must withstand.

The design of an industrial chimney takes into account the wind velocity within its corresponding zone to ensure its structural integrity against the anticipated wind loads. Considerations such as the chimney's shape, height, and overall structural stability are crucial in this process, ensuring it can endure the expected wind forces without failure or excessive deformation. The wind velocity also impacts the stability and dynamic behavior of the chimney. Elevated wind velocities can induce phenomena such as vortex shedding, wind-induced vibrations, and resonance. These effects may lead to structural fatigue and compromise the chimney's integrity. Hence, the design and construction of industrial chimneys incorporate the specific wind velocities within the designated zone to mitigate such risks.

Adhering to wind load codes, such as IS 875, facilitates accurate assessment of the wind forces acting on industrial chimneys. By considering the wind velocity zones and their associated loads, engineers and designers can ensure that chimneys are designed and constructed to withstand the expected wind conditions within a given region. This approach prioritizes safety, structural robustness, and optimal operational performance. In the analysis of industrial chimneys, the wind velocity plays a crucial role in determining their structural design and performance. By considering wind load codes, such as IS 875, specific wind velocity zones are defined, each associated with a corresponding basic wind speed.

For instance, let's consider zone 1, where the basic wind speed is set to 10 m/s. This means that for the analysis of chimneys located in zone 1, a wind speed of 10 m/s is considered as the reference value. The wind loads and structural requirements are then calculated based on this basic wind speed.

Similarly, for other zones, corresponding wind speeds are identified and utilized for analysis. These wind speeds vary depending on the specific zone and its classification within the wind load code. The actual values for wind speeds in different zones can be found in the corresponding table provided by the code. In our current scale model, we are utilizing a reference wind speed of 10 m/s for the purpose of conducting a CFD simulation. Specifically, this reference wind speed corresponds to a wind speed of 33 m/s in wind zone I. To ensure consistent and accurate analysis, a conversion factor of 3.3 is employed to relate other wind velocities. According to the codal provision IS 875 Part 3, specific wind speeds are specified for different wind zones. These wind speeds are presented in the table below. To facilitate ease of use and maintain a consistent conversion ratio of 1:1000, the wind speeds are divided by a factor of 3.3.

To effectively investigate the wind analysis in different zones, it is essential to employ these adjusted wind speeds during the CFD simulation of the chimney. By incorporating the appropriate wind speeds for the respective zones, a smoother and more reliable analysis can be conducted to assess the impact of wind on the chimney structure.

Table 2: Recommended and Taken velocity in different Wind Zones.

Wind Zones	Recommended Velocity (m/s)	Velocity taken for the analysis (m/s)
Zone I	33	10
Zone II	39	11.82
Zone III	44	13.33
Zone IV	47	14.24
Zone V	50	15.15
Zone VI	55	16.67

5.2 ANALYSIS FOR DIFFERENT WIND SPEED

In the analysis of a chimney, the problem statement is addressed by considering the guidelines outlined in section 4.1. The steps involved in the analysis follow a similar approach as discussed previously in Chapter 3. These steps encompass various aspects such as geometry definition, meshing, setup configuration, and establishing boundary conditions.

Firstly, the geometry of the chimney is constructed and defined. This involves accurately representing the dimensions, shape, and features of the chimney in the simulation model. Careful attention is given to capturing all relevant details that may affect the flow behavior and performance of the chimney.

Next, the meshing process takes place. The computational domain surrounding the chimney is discretized into small computational cells or elements to facilitate the numerical calculations. The mesh should be fine enough to capture the flow characteristics and ensure accurate results while maintaining computational efficiency.

Once the meshing is completed, the setup phase begins. This includes selecting appropriate numerical methods and solver settings for the simulation. It involves defining the fluid properties, turbulence models, and other relevant parameters necessary to accurately represent the flow physics.

Furthermore, establishing boundary conditions is a crucial step in the analysis. This involves specifying the inflow conditions, such as the incident wind velocity and direction, and the outflow conditions, which may include atmospheric pressure or other relevant factors. Additionally, boundary conditions on the chimney surfaces are defined to account for the interactions between the air flow and the chimney structure.

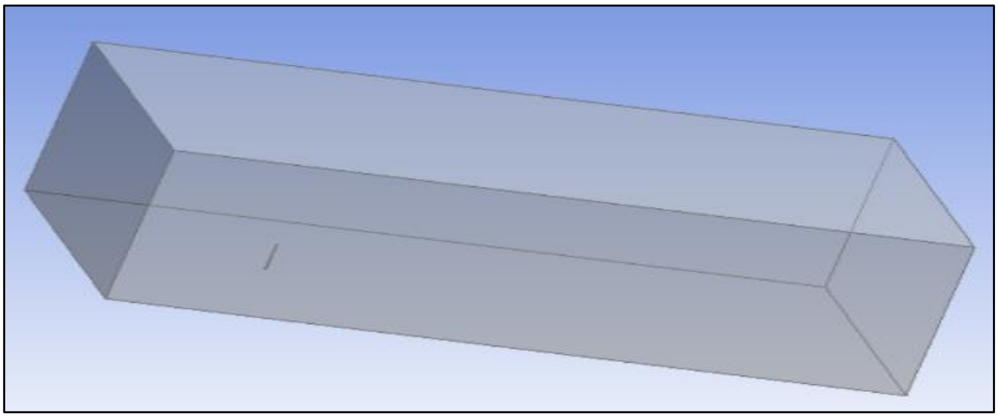


Fig.5.2: Geometry

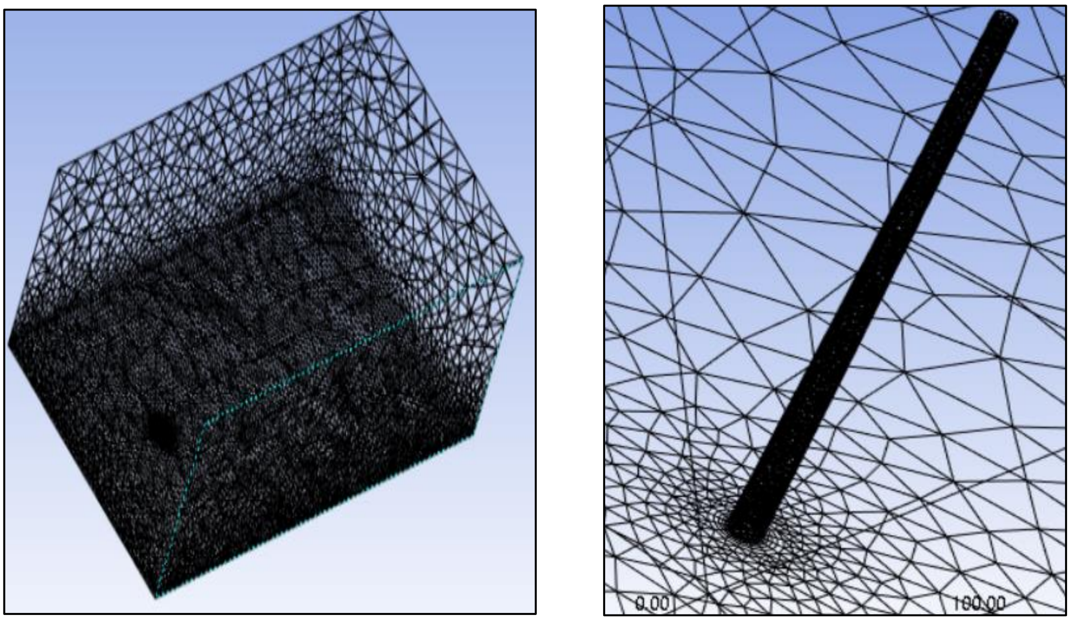


Fig.5.3: Meshing of Model

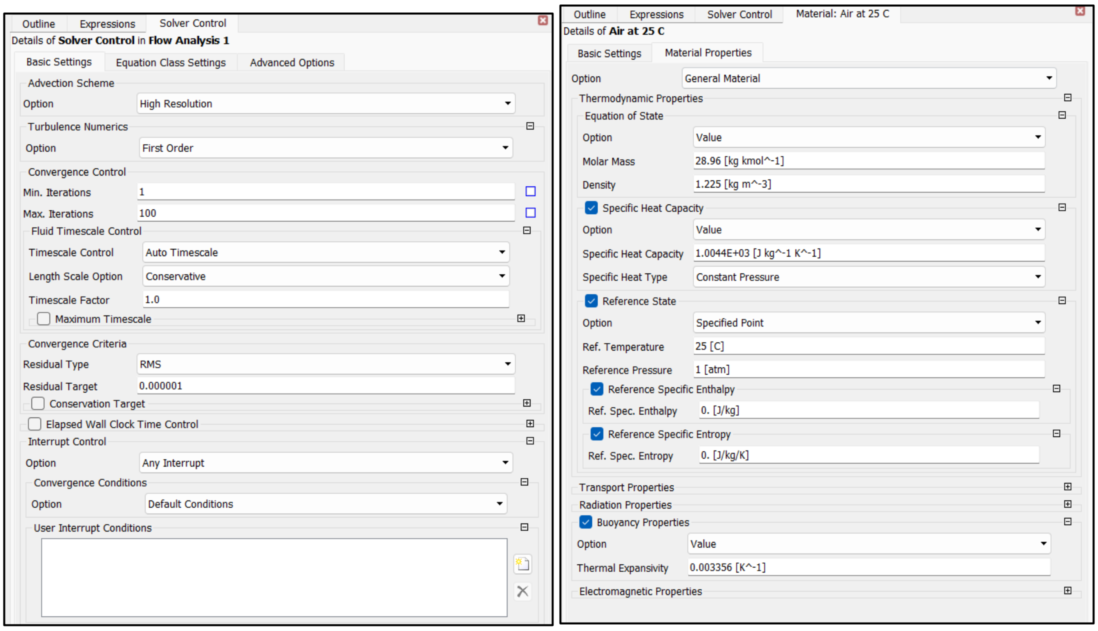


Fig.5.4: Setup and Material

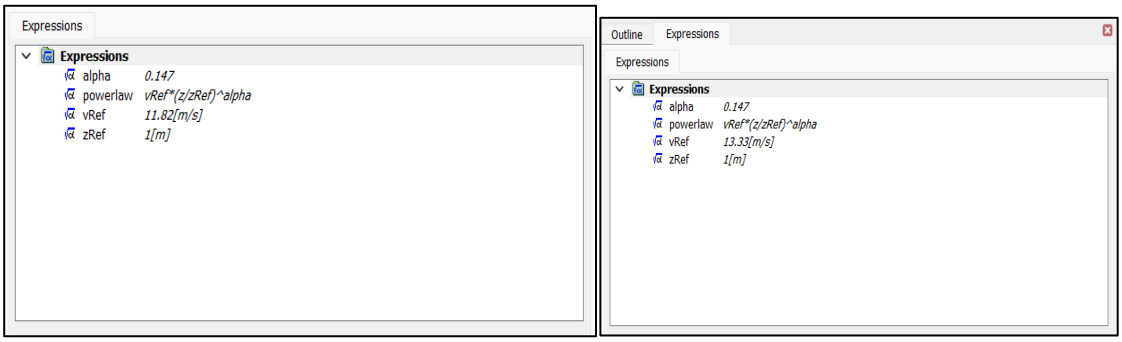
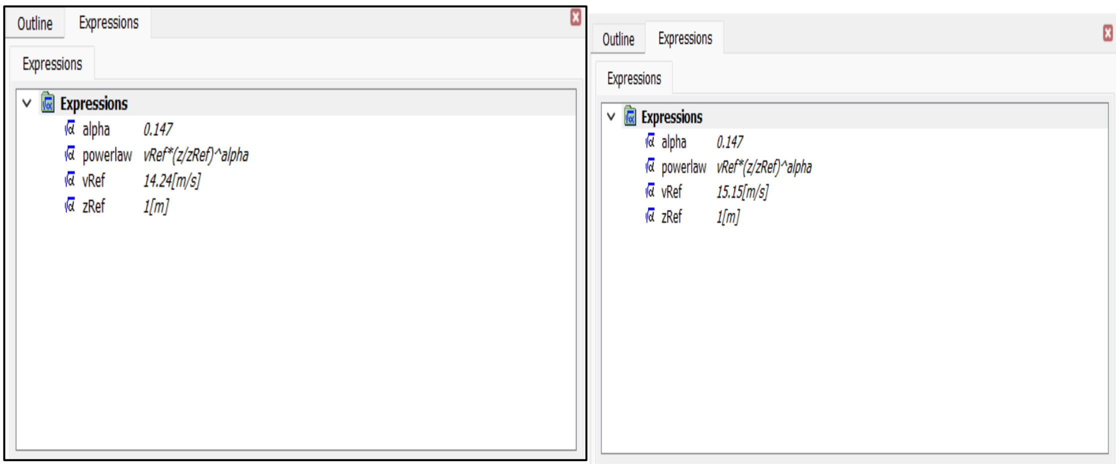


Fig.5.5: Velocity equation in Zone II & III



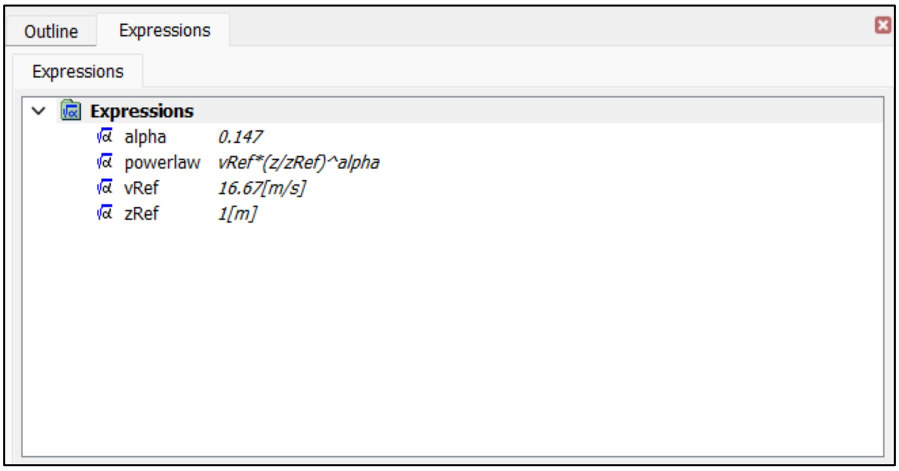
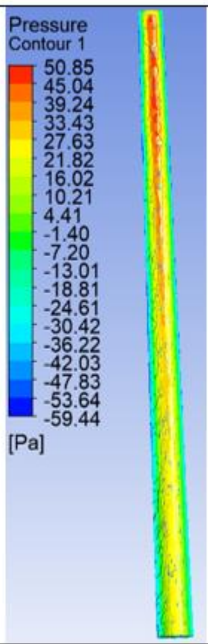
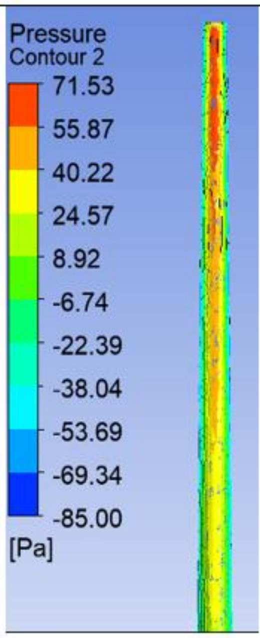
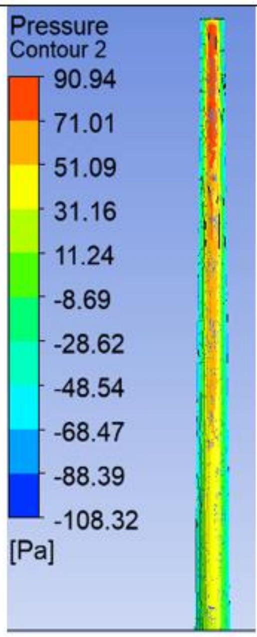


Fig.5.6: Velocity equation in Zone IV, V & VI

5.3 RESULT AND DISCUSSION

5.3.1 PRESSURE CONTOUR

Wind Speed	10 m/s	11.82 m/s	13.33 m/s
Pressure Contours			

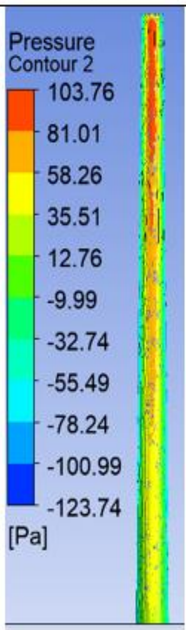
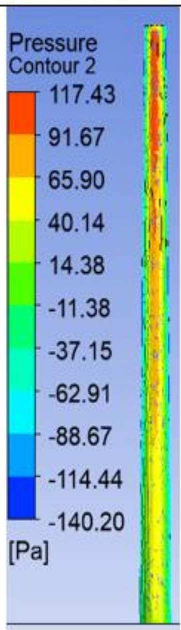
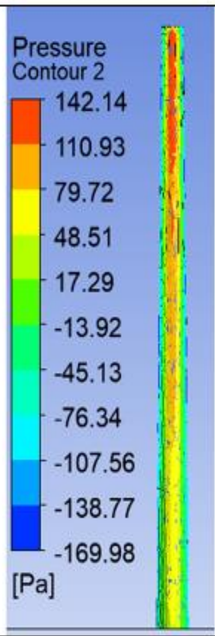
<table>
<tr><th>Wind Speed</th><th>14.24 m/s</th><th>15.15 m/s</th><th>16.67 m/s</th></tr>
<tr><td>Pressure Contours</td><td>
</td><td>
</td><td>
</td></tr>
</table>

Fig.5.7: Pressure Contours for Different Wind speed

As the wind velocity increases, the pressure exerted on the chimney also increases. Let's explore this relationship further by examining additional cases with varying wind velocities. The pressure contours for different velocities are given in Fig.5.7.

In the case of a wind velocity of 11.82 m/s, the pressure on the chimney ranges from 50.85 Pa (maximum) to -59.44 Pa (minimum). This indicates that certain areas of the chimney experience an increase in pressure, reaching a maximum value of 50.85 Pa, while other areas experience a decrease in pressure compared to the ambient conditions, reaching a minimum value of -59.44 Pa.

When the wind velocity increases to 13.33 m/s, the pressure variation on the chimney expands further. The pressure ranges from 90.94 Pa (maximum) to -85 Pa (minimum). With the higher wind velocity, the pressure exerted on the chimney surfaces increases, reaching a maximum value of 90.94 Pa. Conversely, there are areas where the pressure drops to a minimum value of -85 Pa, indicating a significant decrease in pressure compared to the surrounding atmosphere.

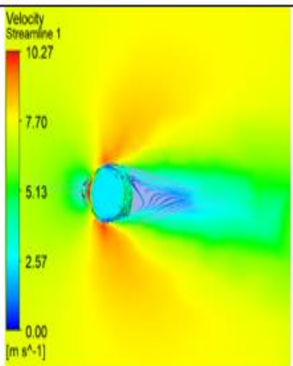
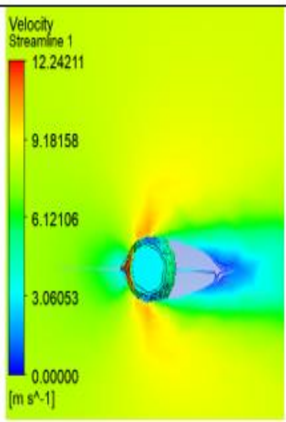
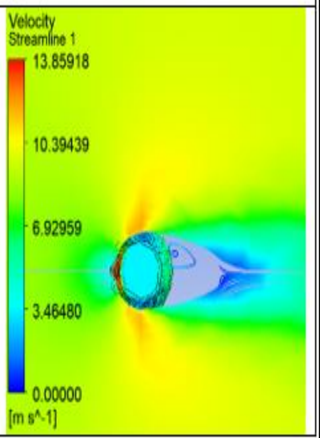
In the case of a wind velocity of 14.24 m/s, the pressure range on the chimney expands even more. It varies from 103.76 Pa (maximum) to -123.74 Pa (minimum). This indicates a further increase in pressure on certain areas of the chimney, reaching a maximum value of 103.76 Pa, and a corresponding decrease in pressure to a minimum value of -123.74 Pa.

When the wind velocity increases to 15.15 m/s, the pressure variation on the chimney continues to widen. The pressure ranges from 117.43 Pa (maximum) to -140.20 Pa (minimum). The higher wind velocity leads to an increase in pressure on the chimney surfaces, reaching a maximum value of 117.43 Pa. Conversely, certain areas experience a decrease in pressure, dropping to a minimum value of -140.20 Pa.

Finally, for a wind velocity of 16.67 m/s, the pressure range on the chimney expands even further. It varies from 142.14 Pa (maximum) to -169.98 Pa (minimum). This indicates a significant increase in pressure on certain areas of the chimney, reaching a maximum value of 142.14 Pa, and a corresponding decrease in pressure to a minimum value of -169.98 Pa.

5.3.2 HORIZONTAL AND VERTICSL STREAMLINES

The Horizontal Streamlines as shown in Fig.5.8.

Wind Speed	10 m/s	11.82 m/s	13.33 m/s
Horizontal Streamlines			

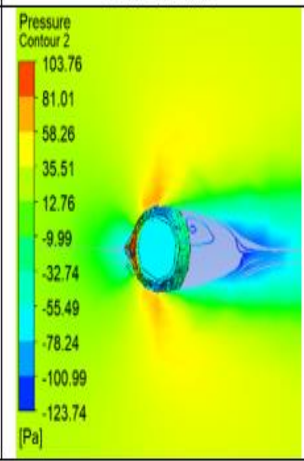
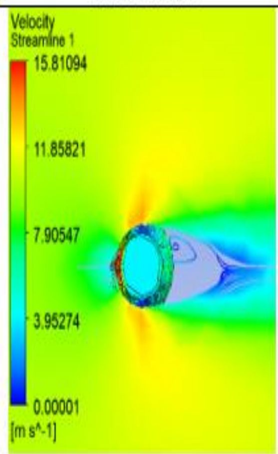
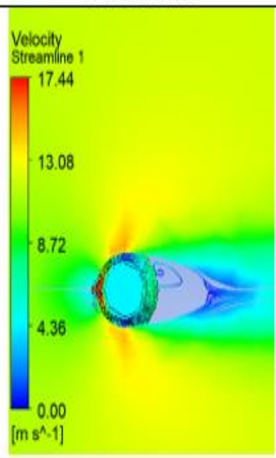
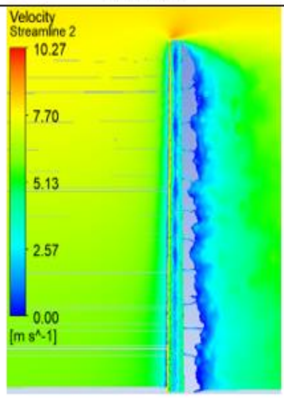
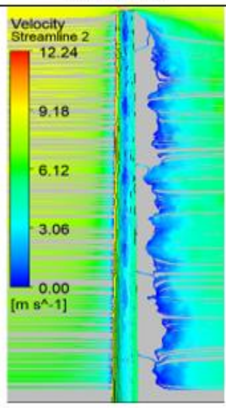
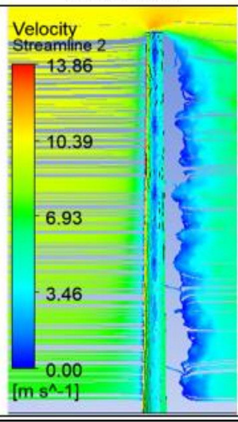
Wind Speed	14.24 m/s	15.15 m/s	16.67 m/s
Horizontal Streamlines			

Fig.5.8: Horizontal Streamlines for Different Wind speed

- Horizontal streamlines represent the paths taken by fluid particles as they travel horizontally within a fluid medium. These streamlines are influenced by the velocity of the air flow, which determines their direction and spacing. Let's examine the characteristics of horizontal streamlines at different velocities.
- When the air flow has a low velocity, the horizontal streamlines appear closely spaced and parallel to each other. This indicates a smooth and orderly flow known as laminar flow. In laminar flow, the fluid particles move in a well-organized manner with minimal mixing or turbulence. The streamlines maintain their initial spacing and follow a predictable trajectory.
- As the velocity of the air flow increases, the horizontal streamlines begin to separate and display more pronounced curvature. This signifies the presence of greater fluid shear and mixing, leading to a transition from laminar to turbulent flow. Turbulent flow is characterized by chaotic movement, frequent changes in velocity and direction, and increased mixing of the fluid. Consequently, the streamlines become more irregular, intersecting each other and forming vortices and eddies.
- At even higher velocities, the horizontal streamlines become further dispersed and fragmented. The intensity of turbulent flow increases, resulting in enhanced mixing and swirling motion of the fluid. The streamlines exhibit intricate patterns, with regions of high and low velocity intertwining and forming complex flow structures.

Wind Speed	10 m/s	11.82 m/s	13.33 m/s
Vertical Streamlines			

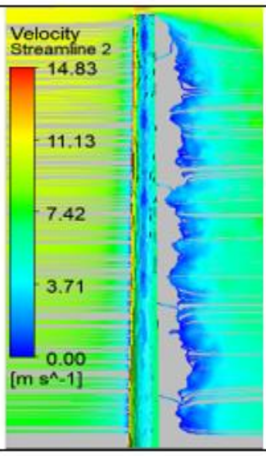
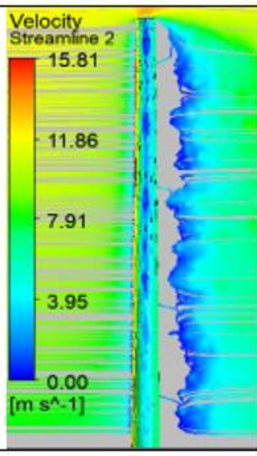
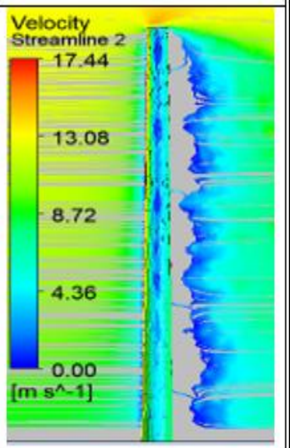
Wind Speed	14.24 m/s	15.15 m/s	16.67 m/s
Vertical Streamlines			

Fig.5.9: Vertical Streamlines for Different Wind speed

- We can clearly see fig.5.9, as the velocity of the airflow increases, a region of negative pressure is observed on the leeward side of the chimney. This negative pressure creates a favourable condition for the formation of vortices behind the chimney. Vortices are swirling patterns of air flow characterized by rotating motion.
- As the velocity further increases, the intensity of the vortices behind the chimney becomes more pronounced. The higher velocity amplifies the negative pressure zone, enhancing the conditions necessary for vortex formation. This leads to an increased number and strength of the vortices.
- The formation of vortices can have important implications for the chimney and its surrounding environment. Vortices can induce additional pressure on the leeward side of the chimney, which may result in increased wind loads on the structure.

5.3.3 VARIATION OF PRESSURE WITH DIFFERENT VELOCITIES

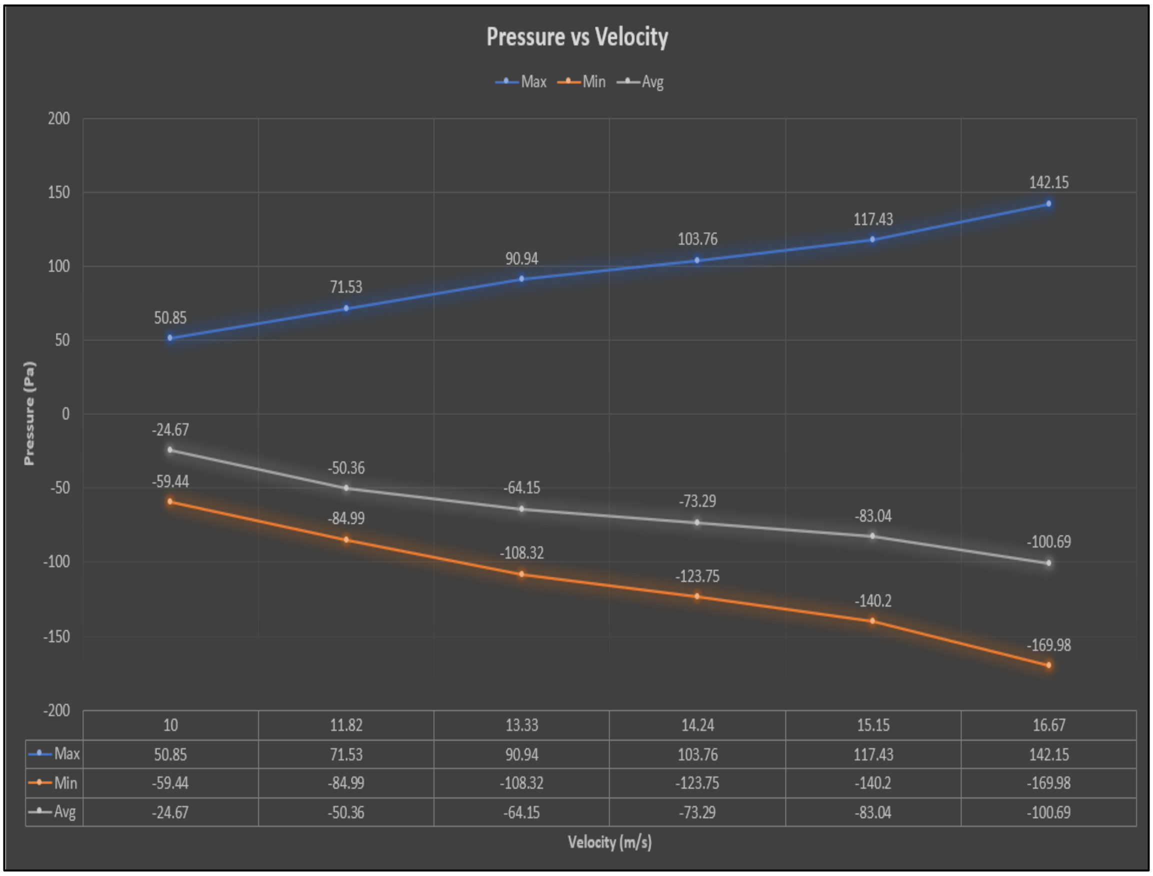


Fig.5.10: Avg., Max., Min. Pressure vs Different Wind speed

As wind speed increases, the pressure on structures also increases as shown in fig.5.10. Wind zone I, characterized by lower wind speeds, exhibits relatively lower pressure values compared to higher wind zones. The pressure values vary based on the location within each wind zone and factors such as local topography and surrounding structures.

5.3.4 Cp VS VELOCITY

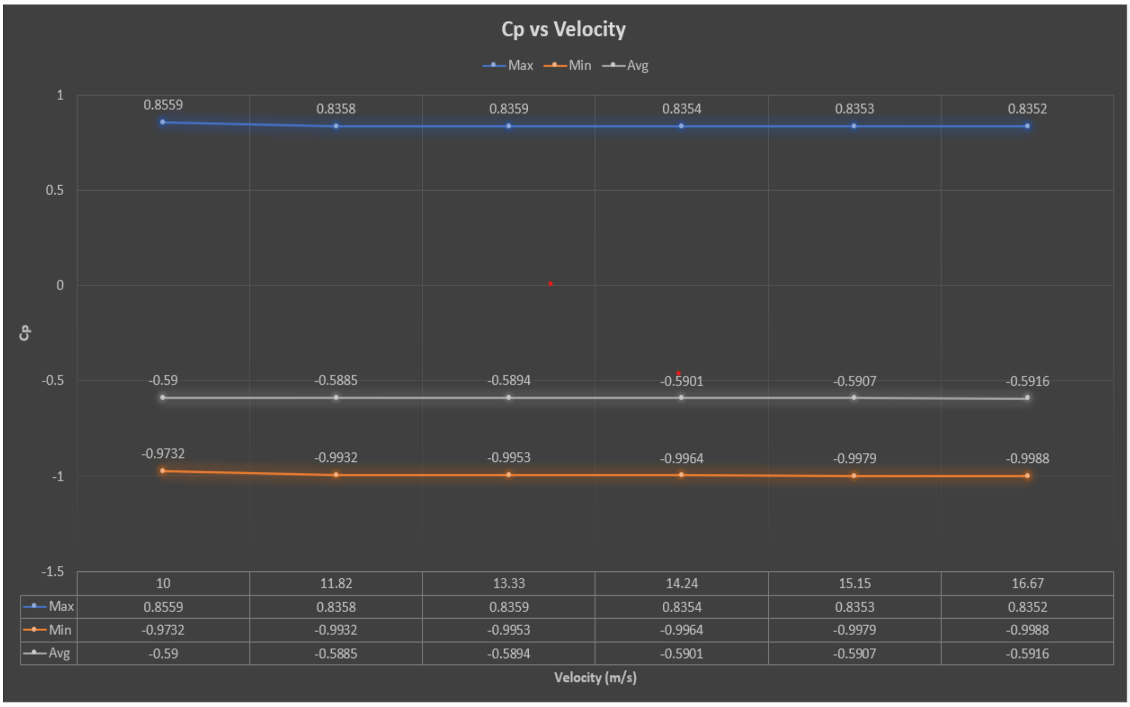


Fig.5.11: Coefficient of Pressure vs Different Wind speed

As the wind velocity increases, the Cp values can deviate from 1 as shown in fig.5.11, indicating variations in the pressure distribution. Higher wind velocities can lead to regions of lower pressure, resulting in Cp values less than 1. These areas of negative Cp values indicate suction or a lower pressure on the surface of the chimney.

The specific relationship between Cp and wind velocity in the CFD simulation of an industrial chimney depends on various factors such as the chimney's geometry, orientation, and the flow conditions surrounding it. The shape and design of the chimney can also influence the Cp values.

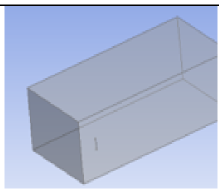
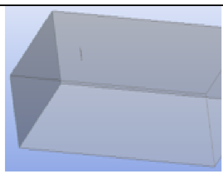
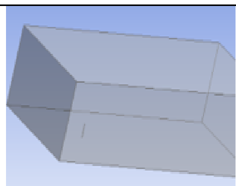
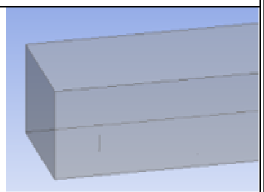
CHAPTER 6: EFFECT OF WIND ON CHIMNEY BY VARYING ITS PARAMETERS

6.1 WITH VARIATION OF TOP DIAMETER

The top diameter of a chimney holds significant importance in its design and performance. In this particular study, the researchers have focused on analyzing the effects of different top diameters on chimney behavior. The range of top diameters considered for comparison spans from 7 meters to 10 meters, resulting in a total of four chimney models.

By varying the top diameter, researchers can examine how it influences various aspects of chimney performance. For instance, changes in the top diameter can affect the flow patterns within the chimney and the surrounding environment. A larger top diameter may lead to smoother flow transition and reduced pressure drop, potentially optimizing the chimney's efficiency. On the other hand, a smaller top diameter may result in increased velocities and turbulence, impacting the chimney's overall performance.

6.1.1 NUMERICAL SETUP

TOP DIAMETE R	7m	8m	9m	10m
GEOMET RY				

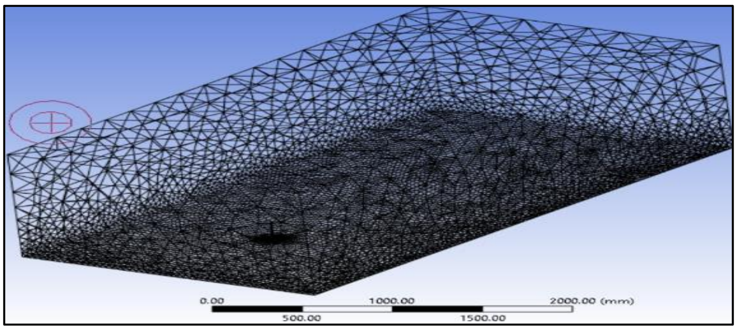


Fig.6.1: Geometry and Meshing of Chimney with different Top Diameter

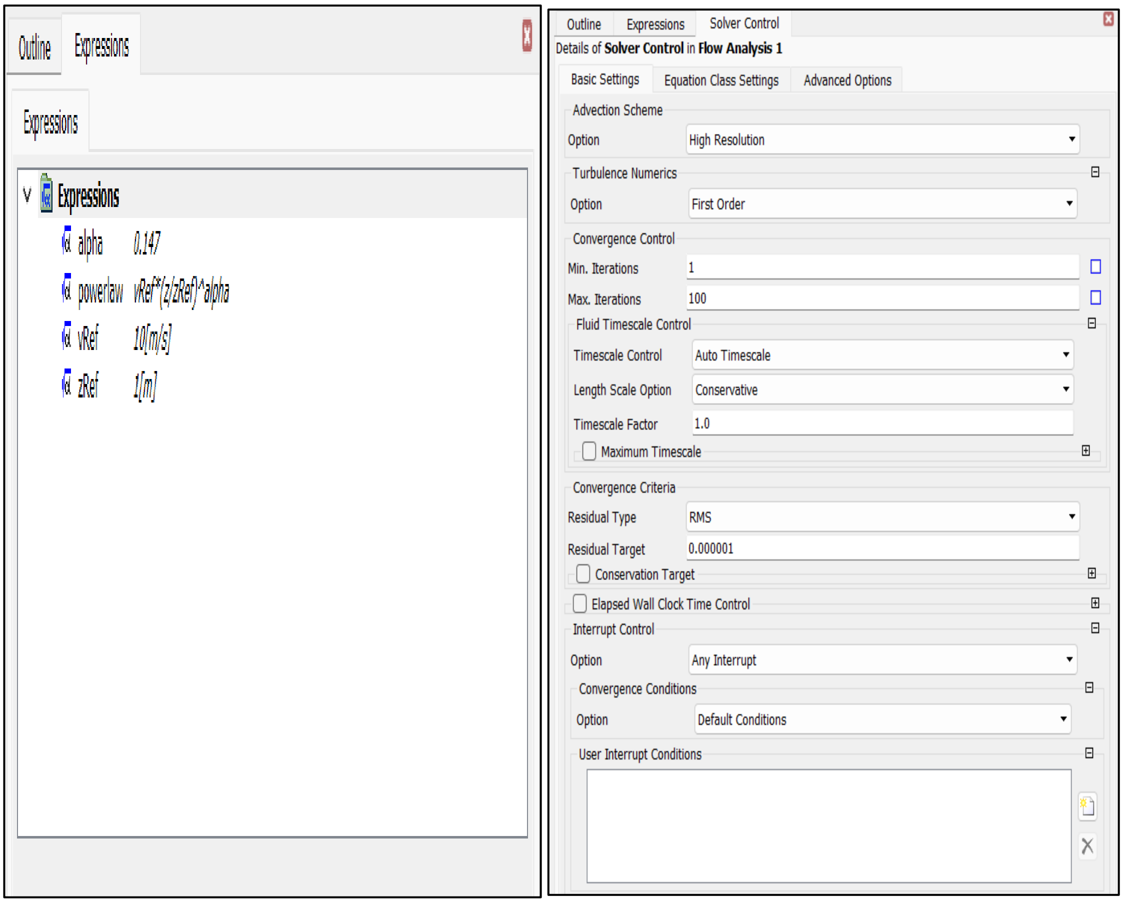


Fig.6.2: Numerical Setup for Chimney with different Top Diameter

To ensure a fair comparison, a consistent numerical setup was maintained for all the chimney models with varying top diameters as the geometry and meshing is shown in fig.6.1. This means that the parameters and conditions used in the computational fluid dynamics (CFD) simulations were identical across all the models. By keeping the numerical setup the same in fig.6.2, any differences observed in the results can be attributed solely to the variations in top diameter. This approach allows for a focused analysis of how the top diameter influences the performance of the chimneys, without interference from other factors. The uniform numerical setup ensures reliable and meaningful comparisons, facilitating accurate insights into the effects of top diameter on factors such as flow patterns, pressure distribution.

6.1.2 RESULTS

6.1.2.1 PRESSURE CONTOUR

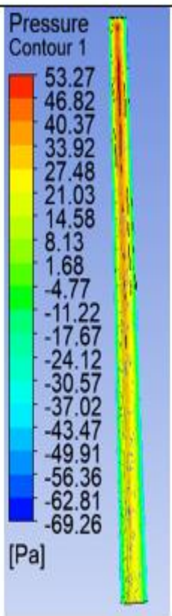
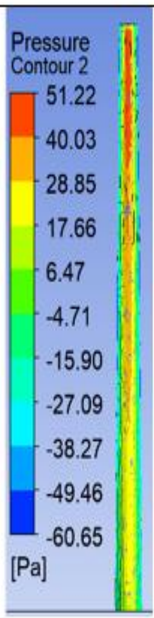
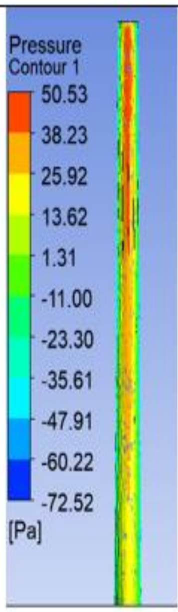
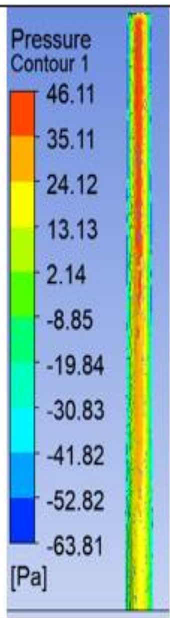
<table>
<tr><th>TOP DIAMETER</th><th>7m</th><th>8m</th><th>9m</th><th>10m</th></tr>
<tr><td>PRESSURE CONTOURS</td>
<td>
</td>
<td>
</td>
<td>
</td>
<td>
</td></tr>
</table>

Fig.6.3: Pressure Contours for Chimney with different Top Diameter

As the top diameter of the chimney increases, there is a noticeable decrease in the pressure values. The maximum pressure values range from 53.27 Pa to 46.11 Pa, indicating a decrease in the overall pressure exerted on the chimney. Surprisingly, the average pressure values show less variation than anticipated, suggesting that the change in top diameter has a relatively minor impact on the average pressure experienced by the chimney.

6.1.2.2 VELOCITY STREAMLINES

Based on the results obtained in this fig.6.4, it is evident that the top diameter of the chimney has a significant influence on the flow patterns and velocity distribution at the outlet. The experiments conducted show that when the top diameter is larger, the flow experiences a smoother transition as it exits the chimney. This smooth flow transition leads to a reduction in the pressure drop, indicating improved flow efficiency. Additionally, the larger top diameter promotes a more uniform velocity distribution throughout the chimney, ensuring a consistent and controlled flow.

Conversely, a smaller top diameter tends to have contrasting effects on the flow behavior. The experiments demonstrate that a smaller top diameter results in higher velocities at the outlet, suggesting a more concentrated and accelerated flow. This higher velocity flow can induce increased turbulence and pressure fluctuations, which may have implications for the overall performance of the chimney.

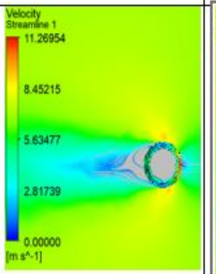
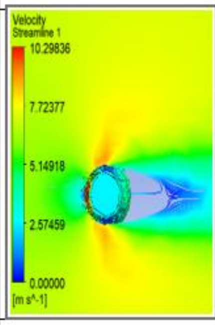
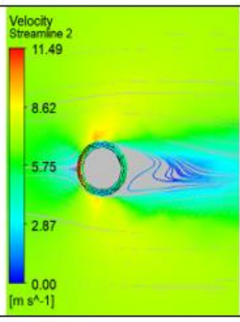
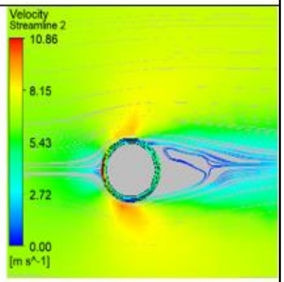
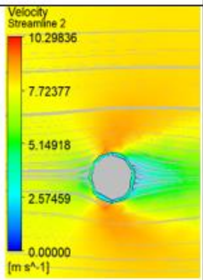
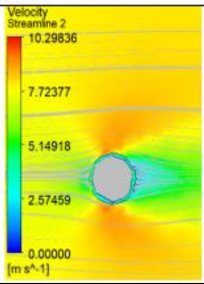
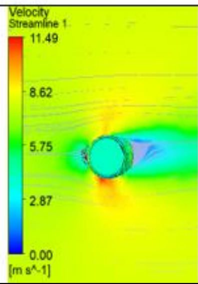
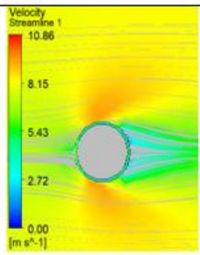
TOP DIAMETER	**7m**	**8m**	**9m**	**10m**
HORIZONTAL STREAMLINES				
TOP DIAMETER	**7m**	**8m**	**9m**	**10m**
HORIZONTAL STREAMLINES (at top of Chimney)				

Fig.6.4: Horizontal Streamlines for Chimney with different Top Diameter

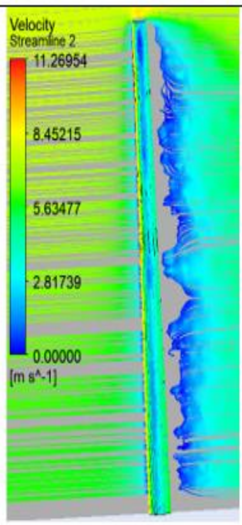
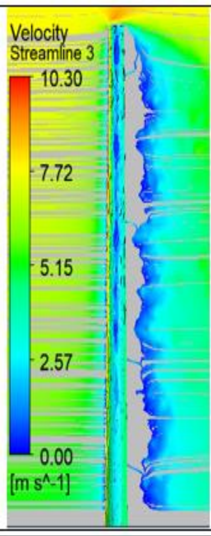
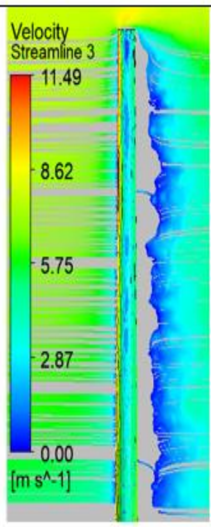
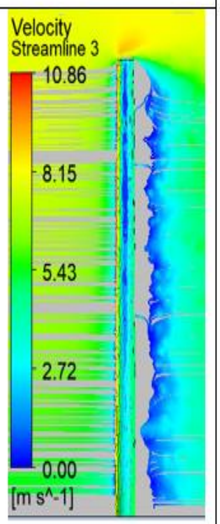
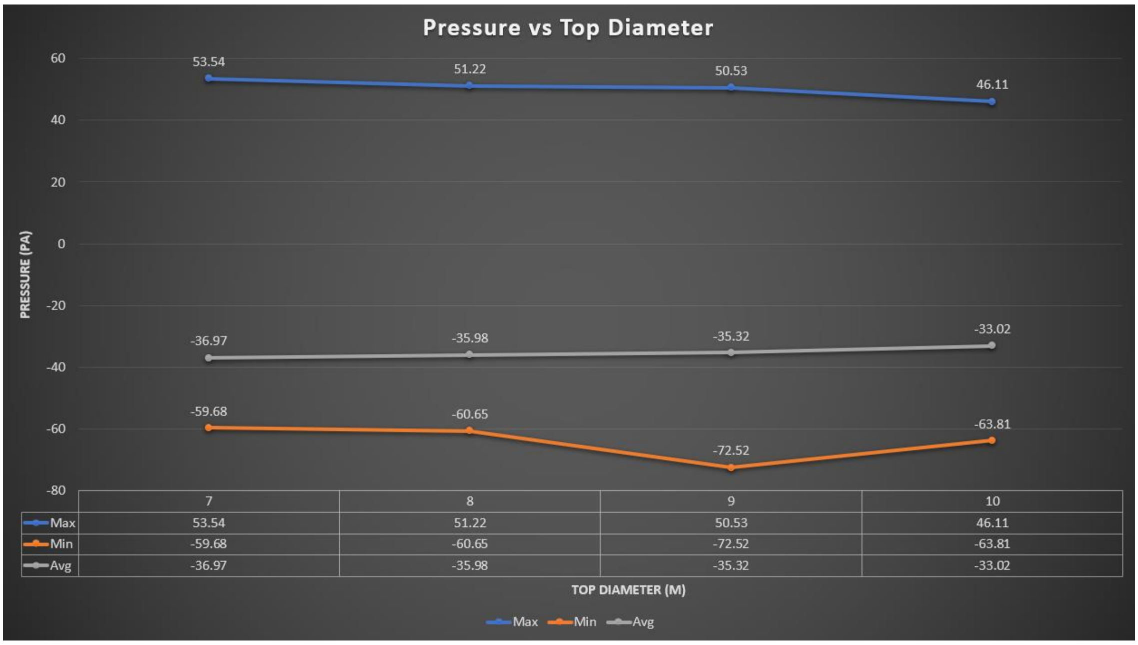
TOP DIAMETER	**7m**	**8m**	**9m**	**10m**
VERTICAL STREAMLINE				

Fig.6.5: Vertical Streamlines for Chimney with different Top Diameter

In this thesis, the investigation of the top diameter's impact on vertical streamlines in chimneys reveals significant findings. The Vertical streamlines are as shown in fig.6.5. An increase in the top diameter brings about noticeable changes in the shape and distribution of the streamlines. The observed effect is a more orderly and consistent flow pattern, indicating smoother upward airflow. This improvement can be attributed to the broader channel created by the larger diameter, which reduces disturbances and facilitates laminar airflow.

Conversely, a decrease in the top diameter results in distinct modifications to the vertical streamlines. The narrower opening constrains the flow path, leading to heightened turbulence and the formation of vortices. These vortices introduce irregularities and disrupt the flow, potentially causing pressure fluctuations that can impact the chimney's overall performance and stability.

6.1.2.3 VARIATION OF PRESSURE



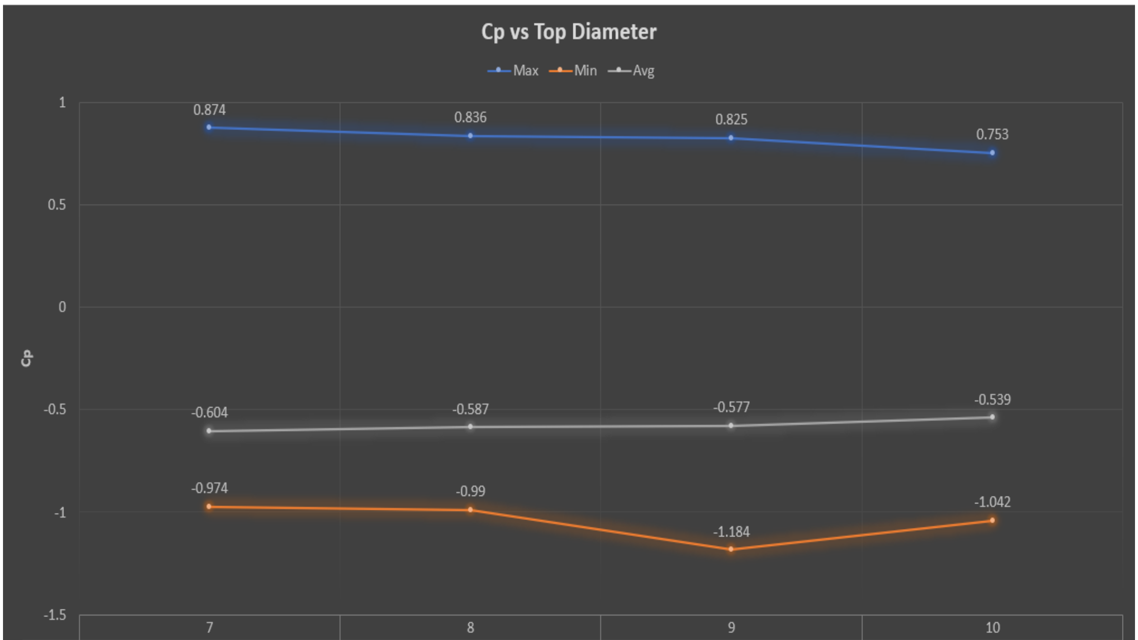
	7	8	9	10
Max	53.54	51.22	50.53	46.11
Min	-59.68	-60.65	-72.52	-63.81
Avg	-36.97	-35.98	-35.32	-33.02

Fig.6.6: Avg., Max., Min. Pressure vs Top Diameter

The relationship between pressure and variation in top diameter of the chimney was investigated as part of this thesis and plotted in fig.6.6. The results revealed a clear correlation between these two factors. As the top diameter of the chimney increased, there was a consistent decrease in pressure. This can be attributed to the larger cross-sectional area allowing for smoother and more efficient airflow, resulting in reduced pressure drop along the chimney.

On the other hand, when the top diameter of the chimney decreased, the pressure exhibited a corresponding increase. The narrowing of the chimney opening created flow restrictions, leading to higher velocities and increased flow resistance. As a result, the pressure within the chimney increased.

6.1.2.4 VARIATION OF Cp



	7	8	9	10
Max	0.874	0.836	0.825	0.753
Min	-0.974	-0.99	-1.184	-1.042
Avg	-0.604	-0.587	-0.577	-0.539

Fig.6.7: Cp vs Top Diameter

In the investigation of the variation in top diameter of the chimney, the coefficient of pressure (Cp) was analysed at two specific points. The Cp values were obtained to understand the impact of changing the top diameter on the pressure distribution and shown in fig.6.7.

As the top diameter increased, the Cp value showed a slight decrease. This suggests that a larger top diameter contributes to a more favorable pressure distribution, potentially reducing pressure fluctuations and improving the overall performance of the chimney.

A similar trend was observed. As the top diameter increased, the Cp value exhibited a gradual decrease. This indicates that a larger top diameter can contribute to a more stable pressure distribution, reducing the potential for high-pressure zones and minimizing the risk of structural issues.

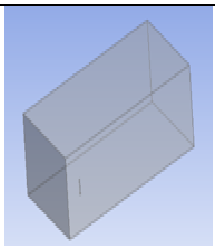
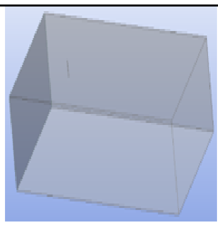
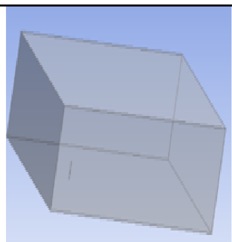
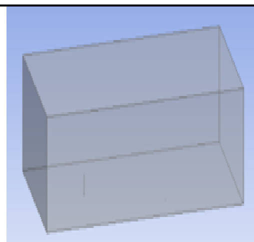
6.2 WITH VARIATION IN HEIGHT

In this study, the height of the chimneys was examined as a crucial parameter. Four different models were considered, with chimney heights ranging from 150m to 300m, in 50m increments. The objective was to investigate the impact of chimney height on various aspects related to the chimney's performance and behavior.

The height of a chimney plays a significant role in determining its overall efficiency, stability, and effectiveness in transporting exhaust gases or facilitating airflow. By varying the height across different models, it allows for a comparative analysis of key factors such as pressure distribution, velocity profiles, and dispersion characteristics.

The results obtained from the simulation provide valuable insights into the influence of chimney height on these factors. It enables a better understanding of the optimal height range for achieving desirable flow patterns, minimizing pressure losses.

6.2.1 NUMERICAL SETUP

HEIGHT	150m	200m	250m	300m
GEOMETRY				

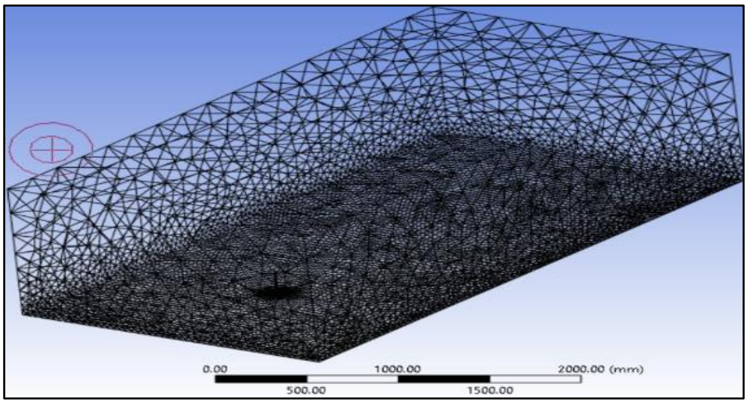


Fig.6.8: Geometry and Meshing of Chimney with different Height.

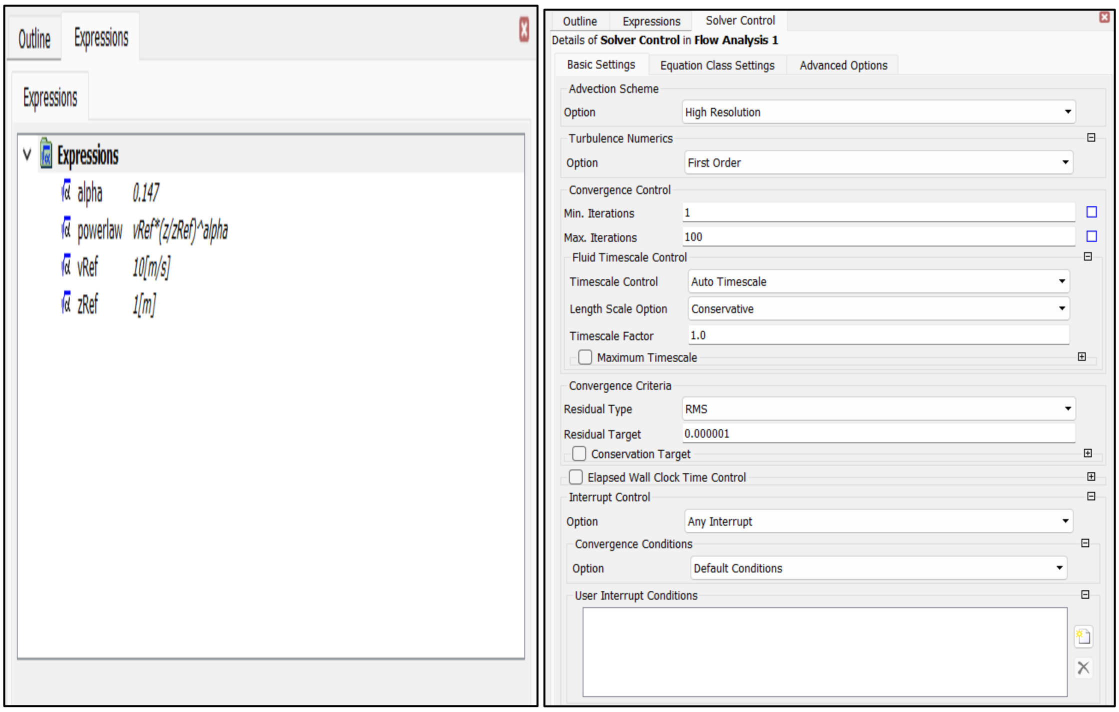


Fig.6.9: Numerical Setup of Chimney with different Height

By maintaining a consistent numerical setup, it becomes possible to isolate and assess the specific impact of height on various parameters, such as pressure distribution, airflow patterns. This approach allows for a clearer understanding of how height influences these factors independently of other variables. Fig.6.8 shows the geometry and meshing of chimney with different height.

Keeping the numerical setup the same for all models (fig.6.9) with varying heights ensures that any observed differences can be attributed solely to the change in height, eliminating potential confounding factors. This controlled approach enhances the reliability and validity of the results, enabling more accurate conclusions to be drawn about the relationship between chimney height and its effects on airflow.

6.2.2 RESULTS

6.2.2.1 PRESSURE CONTOUR

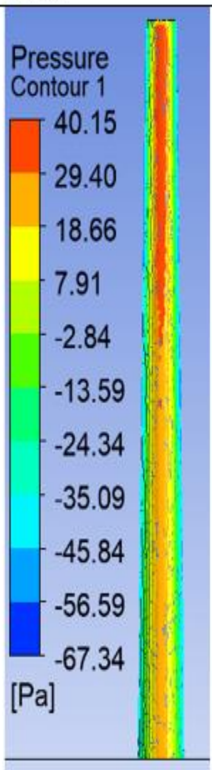
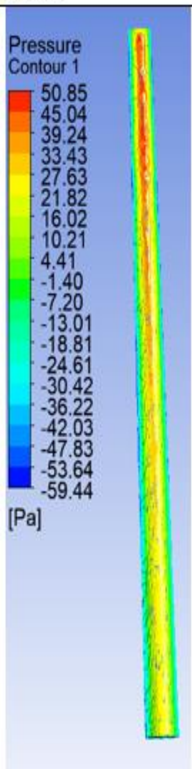
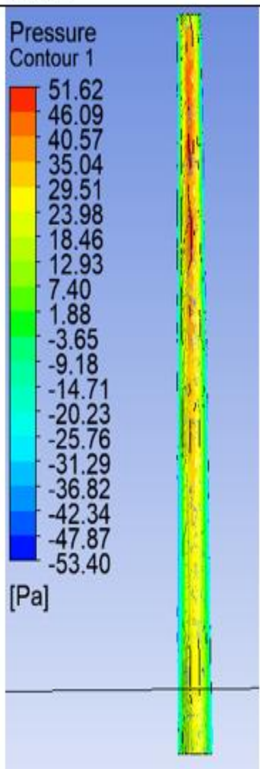
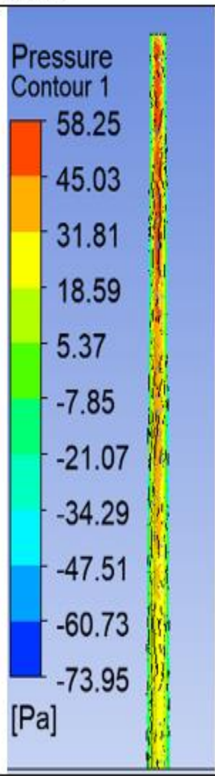
HEIGHT	150m	200m	250m	300m
PRESSURE CONTOUR				

Fig.6.10: Pressure Contours for Chimney with different Height

In the analysis of chimney height variations ranging from 150m to 300m, it was observed that increasing the chimney height resulted in a decrease in the overall average pressure (fig.6.10). The maximum pressure recorded was 58.25 Pa, while the minimum pressure reached -73.95 Pa. These findings indicate that taller chimneys can lead to lower average pressures within the system.

6.2.2.2 VELOCITY STREAMLINES

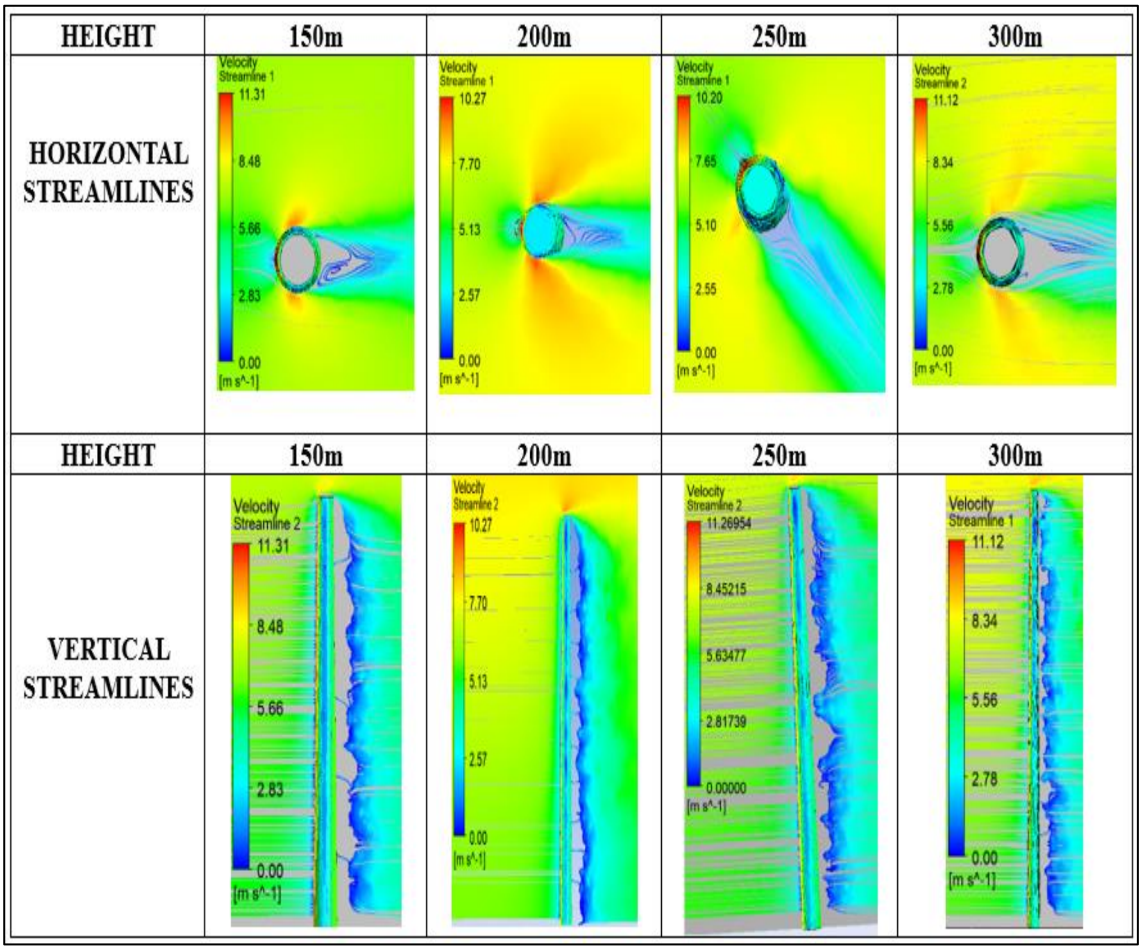


Fig.6.11: Horizontal & Vertical Streamlines for Chimney with different Height

The investigation into the impact of chimney height on velocity streamlines revealed noteworthy findings and as shown in fig. 6.11. Notably, a substantial correlation was observed between chimney height and the configuration of velocity streamlines. As the chimney height increased, the streamlines exhibited an elongated and vertically aligned pattern, indicating a more pronounced upward airflow. This can be attributed to the provision of a larger vertical channel within the chimney, facilitating a streamlined and organized flow of air. Conversely, shorter chimney heights resulted in more compact and inclined streamlines, indicative of a less prominent upward flow. These outcomes emphasize the influence of chimney height on the overall flow dynamics and velocity distribution within the chimney system.

6.2.2.3 VARIATION OF PRESSURE

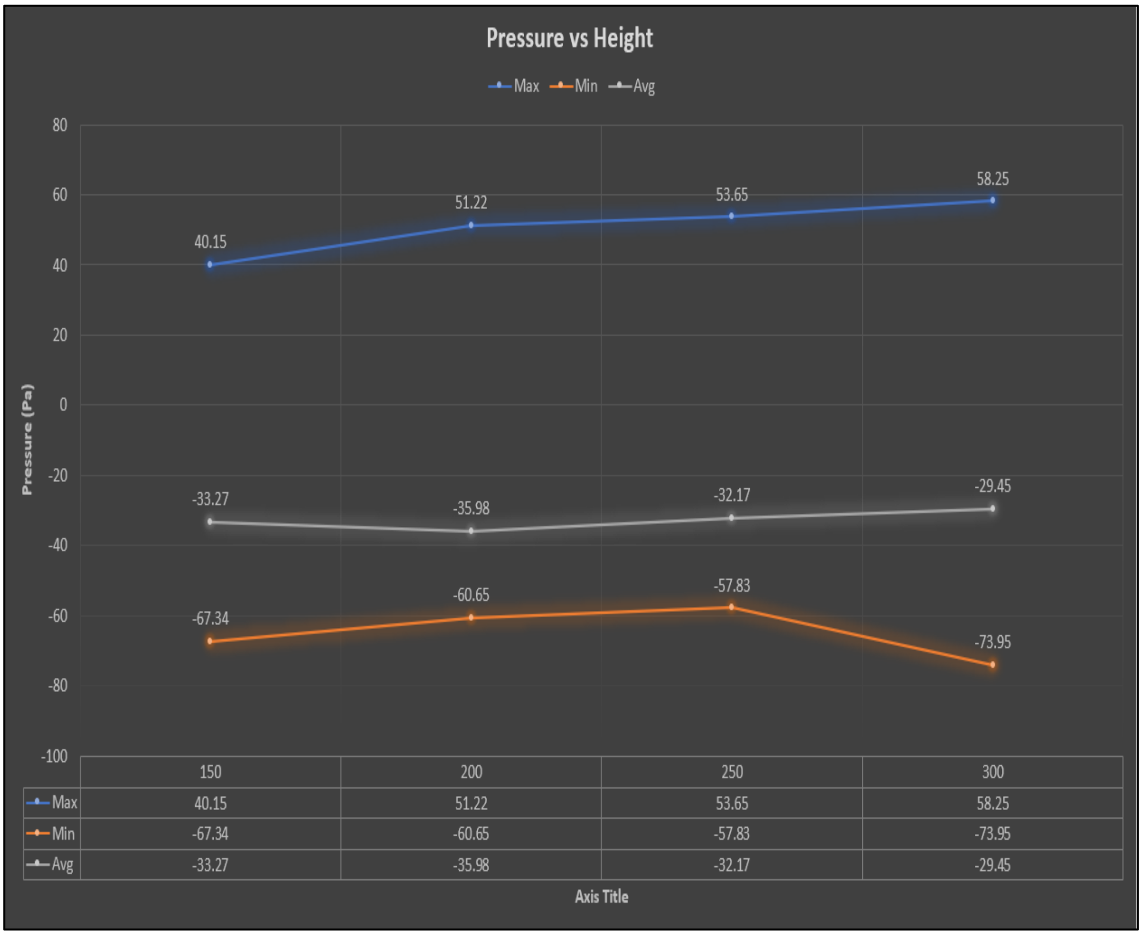


Fig.6.12: Avg., Max., Min. Pressure vs Height

- Average Pressure: As the height of the chimney increases, the average pressure within the chimney tends to decrease. This is due to the reduction in the weight of the gas column above, resulting in a decrease in average pressure along the chimney's height.
- Maximum Pressure: The maximum pressure within the chimney tends to increase with height. Factors such as increased gas velocity and potential flow disturbances contribute to localized regions of high pressure as the gases rise within the chimney.

- Minimum Pressure: Initially, as the height increases, the minimum pressure within the chimney decreases. This is caused by increased gas flow rate and velocity near the chimney inlet. However, as the height further increases, the minimum pressure begins to rise due to factors such as flow patterns, turbulence, or interactions between gas flow and chimney structure.

6.2.2.4 VARIATION OF Cp

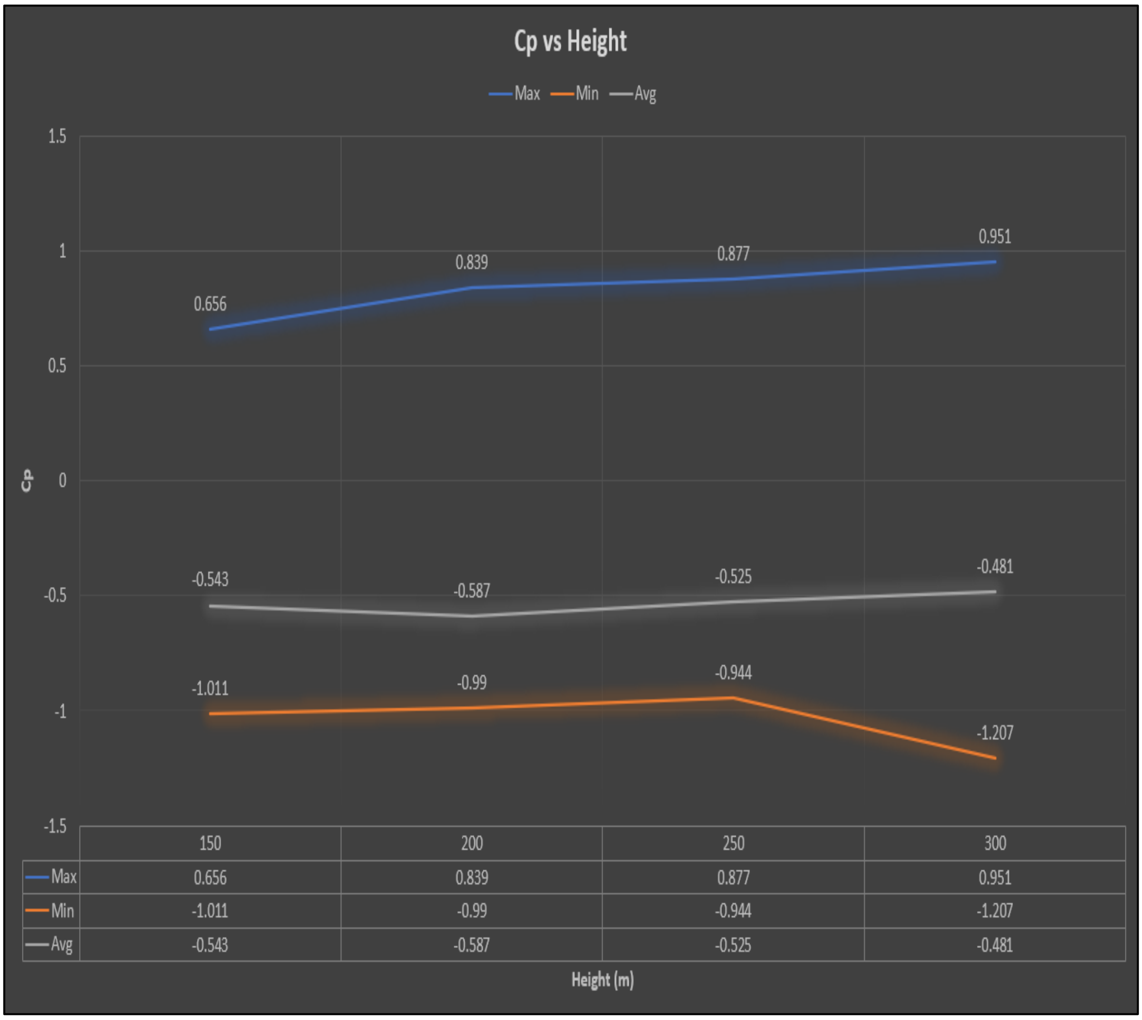


Fig.6.13: Cp vs Height

- Cp Reduction: Generally, the Cp values decrease with increasing height. This reduction is primarily due to the decrease in gas density as the gases rise within the chimney. As the density decreases, the pressure exerted by the gas column diminishes, leading to lower Cp values.

- Maximum Cp: The maximum Cp value typically occurs at the base or lower region of the chimney. This is because the gas flow near the chimney inlet often encounters obstructions or changes in geometry, resulting in localized regions of higher pressure.

- Minimum Cp: Conversely, the minimum Cp value is usually observed at the upper portion of the chimney. As the gases ascend, the flow tends to accelerate and streamline, resulting in lower pressure compared to the ambient conditions. This leads to a minimum Cp value.

6.3 WITH VARIATION OF THICKNESS

The thickness of an industrial chimney is a critical parameter that can significantly impact its structural integrity and performance. In this particular study, the chimney thickness was investigated, considering variations ranging from 0.3m to 0.6m, with intervals of 0.1m. As a result, a total of four chimney models were created to facilitate a comprehensive comparison.

The chosen thickness intervals provide a range of options commonly encountered in industrial applications. By including multiple thickness models, a comparative analysis can be conducted to identify any significant differences in terms of performance and behavior. Such a comparison helps in making informed decisions regarding the optimal thickness selection for industrial chimneys, considering factors like cost-effectiveness, structural requirements, and operational considerations.

6.3.1 NUMERICAL SETUP

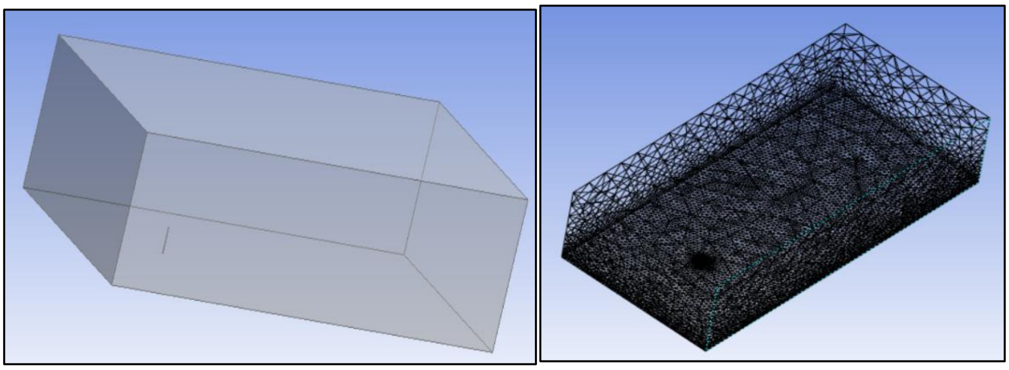


Fig.6.14: Geometry and Meshing of Chimney with different Thickness.

The numerical setup for analyzing the variation of chimney thickness follows a similar approach as described in Chapter 3. The primary difference lies in the problem statement, where the focus is now on investigating the effects of different chimney thicknesses.The geometry and meshing is shown in fig.6.14.

To begin, the geometric parameters of the chimney, such as the base diameter, top diameter, and height, are defined. The thickness of the chimney is then systematically varied while keeping other parameters constant. This allows for a controlled analysis of how thickness impacts the chimney's behavior.

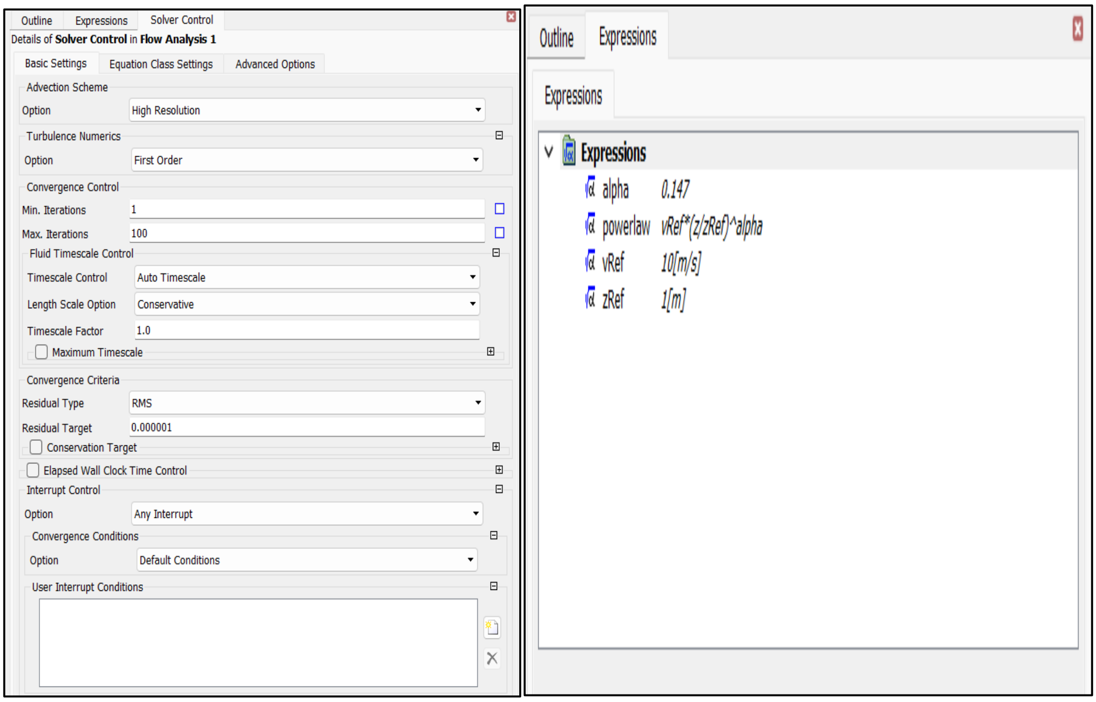


Fig.6.15: Numerical Setup of Chimney with different Thickness

Once the geometry is established, a meshing process is conducted to discretize the computational domain. The mesh ensures accurate representation of the chimney's shape and captures the flow features appropriately. Boundary conditions are applied to simulate realistic scenarios, including inlet and outlet conditions, as well as any additional constraints or specifications.

With the numerical setup in place and as per fig.6.15, CFD simulations are performed to study the flow characteristics and pressure distribution within the chimneys of varying thickness. The simulations generate valuable data, which is subsequently analyzed and compared to draw meaningful conclusions about the relationship between chimney thickness and performance.

6.3.2 RESULTS

6.3.2.1 PRESSURE CONTOUR

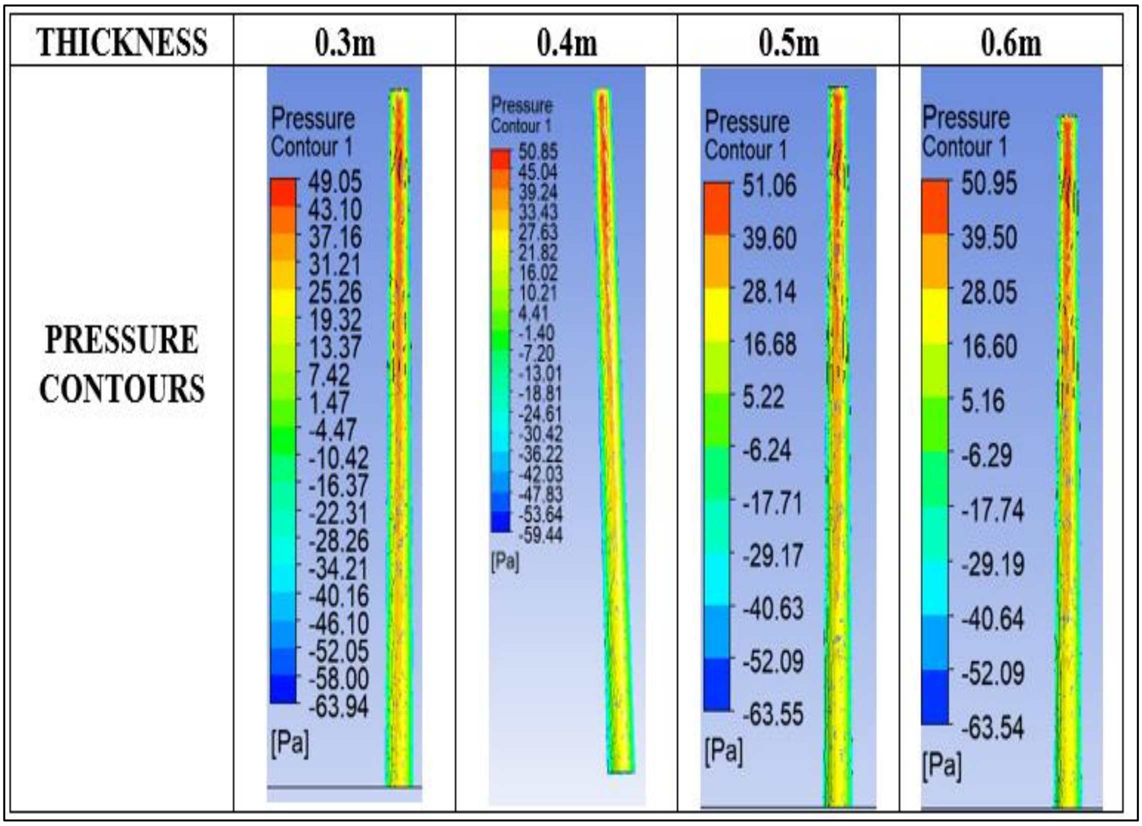


Fig.6.16: Pressure Contours of Chimney with different Thickness

It was observed that with thicker chimney walls, the pressure values tended to decrease. This can be attributed to the increased structural integrity provided by thicker walls, which resulted in reduced deflection and deformation under the influence of wind forces. Consequently, the pressure on the chimney surfaces was alleviated, leading to lower pressure values.

On the other hand, thinner chimney walls exhibited higher pressure values. The reduced thickness resulted in decreased structural stability, leading to greater deflection and deformation. This, in turn, increased the pressure on the chimney surfaces.

6.3.2.2 VELOCITY STREAMLINES

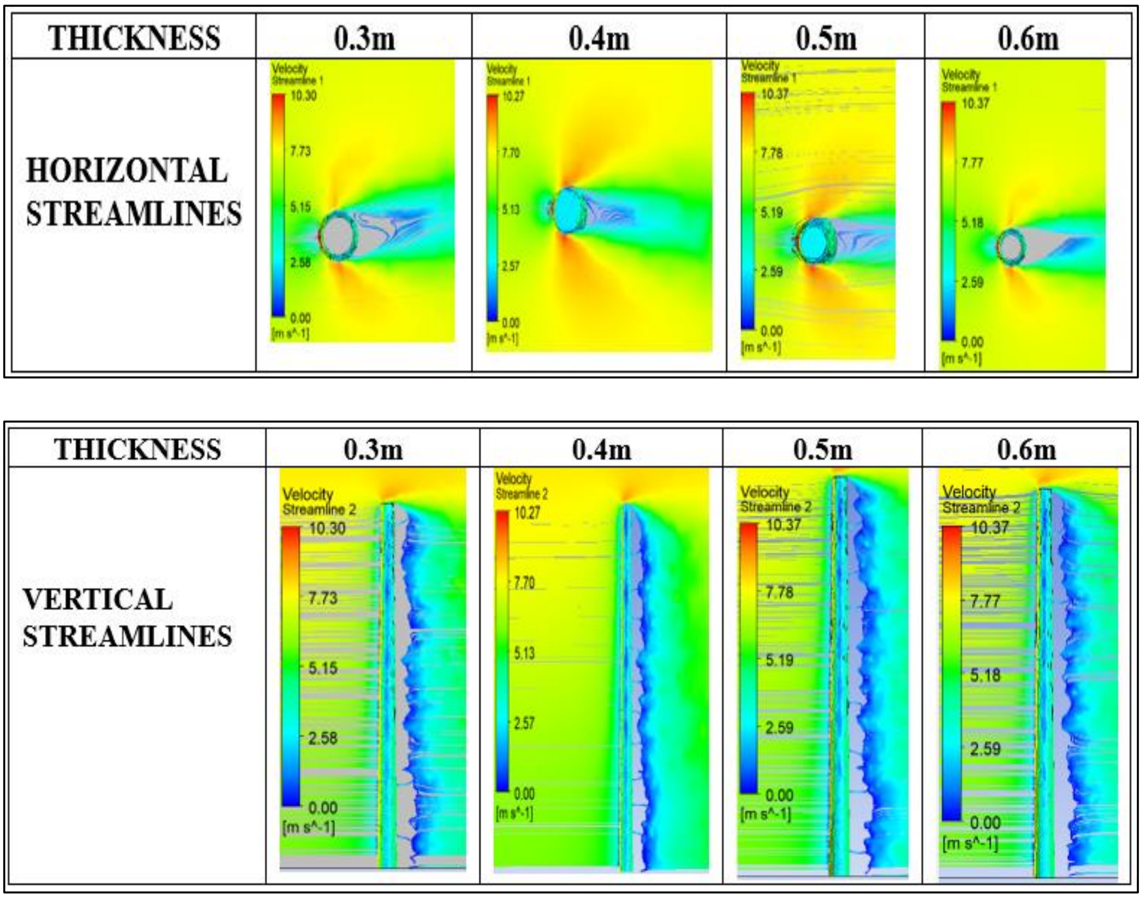
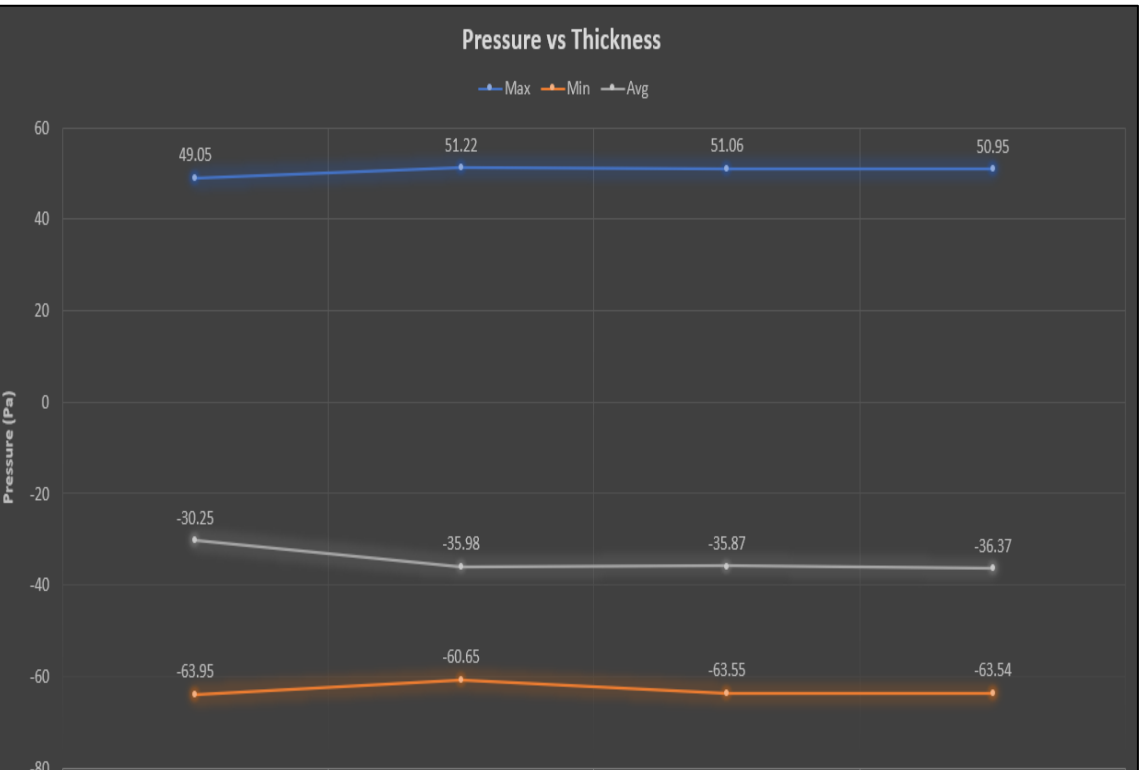


Fig.6.17: Horizontal & Vertical Streamlines for Chimney with different Thickness

It was observed that as the chimney thickness increased, the velocity streamlines exhibited a more streamlined and organized flow. The increased thickness provided a larger flow passage, allowing for smoother and more laminar airflow. The streamlines appeared to follow a more predictable and coherent path.

Conversely, with thinner chimney walls, the velocity streamlines became more disrupted and irregular. The reduced thickness caused increased turbulence and flow disturbances, leading to more chaotic and unpredictable flow patterns. The streamlines displayed greater deviations and fluctuations in their paths.

6.3.2.3 VARIATION OF PRESSURE



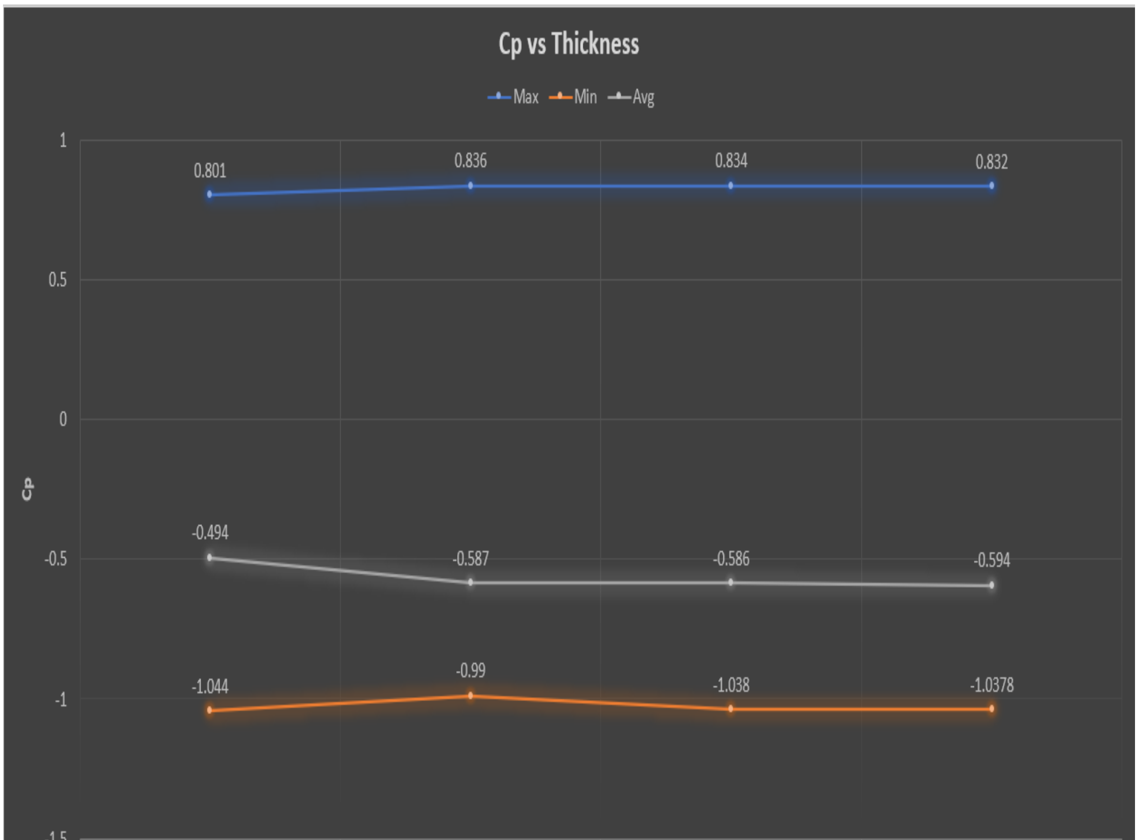
	0.3	0.4	0.5	0.6
Max	49.05	51.22	51.06	50.95
Min	-63.95	-60.65	-63.55	-63.54
Avg	-30.25	-35.98	-35.87	-36.37

Fig.6.18: Avg., Max., Min. Pressure vs Thickness

The relationship between chimney thickness and average pressure is evident in the analysis conducted. As the thickness of the chimney increases, there is a tendency for the average pressure to also increase. This can be attributed to the increased resistance offered by a thicker chimney, which results in higher pressure levels within the system.

However, it is worth noting that the variations in thickness did not have a significant impact on the maximum and minimum pressure values. The maximum and minimum pressure levels remained relatively stable regardless of the thickness changes. These findings suggest that while chimney thickness can influence the average pressure, it has a limited effect on the extreme pressure values experienced within the chimney system.

6.3.2.4 VARIATION IN Cp



	0.3	0.4	0.5	0.6
Max	0.801	0.836	0.834	0.832
Min	-1.044	-0.99	-1.038	-1.0378
Avg	-0.494	-0.587	-0.586	-0.594

Fig.6.19: Cp vs Thickness

The results indicate that as the thickness of the chimney increases, the Cp values tend to decrease. This suggests that a thicker chimney tends to experience lower pressure coefficients. The relationship between thickness and Cp is inverse, meaning that thicker chimneys tend to have a more favorable pressure distribution.

However, it is important to note that the variation in thickness did not result in a significant change in the overall Cp trend. The Cp values remained relatively consistent across different thicknesses, indicating that other factors, such as wind velocity and chimney geometry, may have a more dominant influence on Cp.

6.4 WITH VARIATION OF BOTTOM DIAMETER

This study focused on the variation of the chimney dimensions, specifically the bottom diameter. The dimensions of the chimney, including its height, top diameter, and thickness, remained constant while only the bottom diameter was changed. The purpose of this investigation was to understand how the variation in bottom diameter affects the flow behavior and performance of the chimney system.

In this particular study, the chimney's dimensions were kept constant, except for the bottom diameter. The bottom diameter was carefully varied within a range of 10m to 16m, with an interval of 2m. As a result, a total of four different models were analyzed to investigate the impact of the bottom diameter on the chimney's behavior and characteristics. This systematic variation allowed for a comprehensive understanding of how changes in the bottom diameter influence the overall performance and flow dynamics of the chimney.

6.4.1 NUMERICAL SETUP

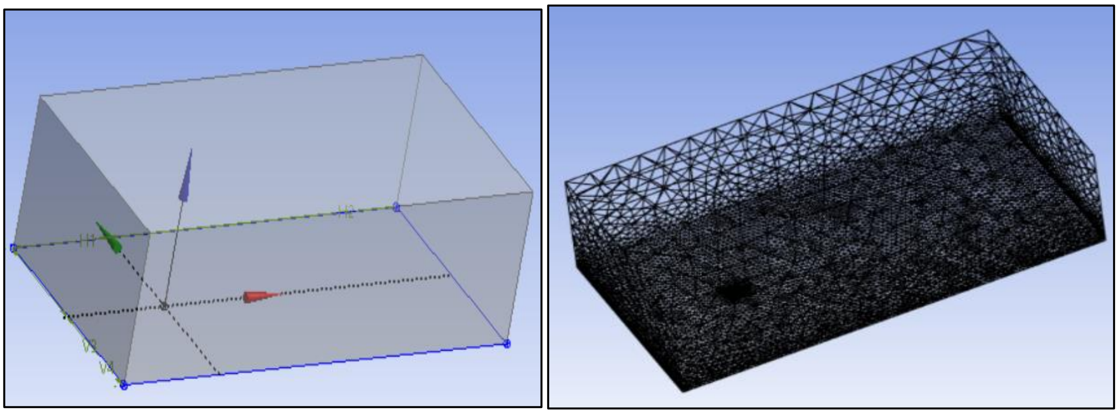


Fig.6.20: Geometry and Meshing of Chimney with different Bottom Diameter

The numerical setup for the analysis remains consistent with the methodology outlined in Chapter 3. However, a key parameter that is being varied in this study is the bottom diameter of the chimney. The fig 6.20 shows the geometry and meshing of one of the chimney model with changes in bottom diameter.

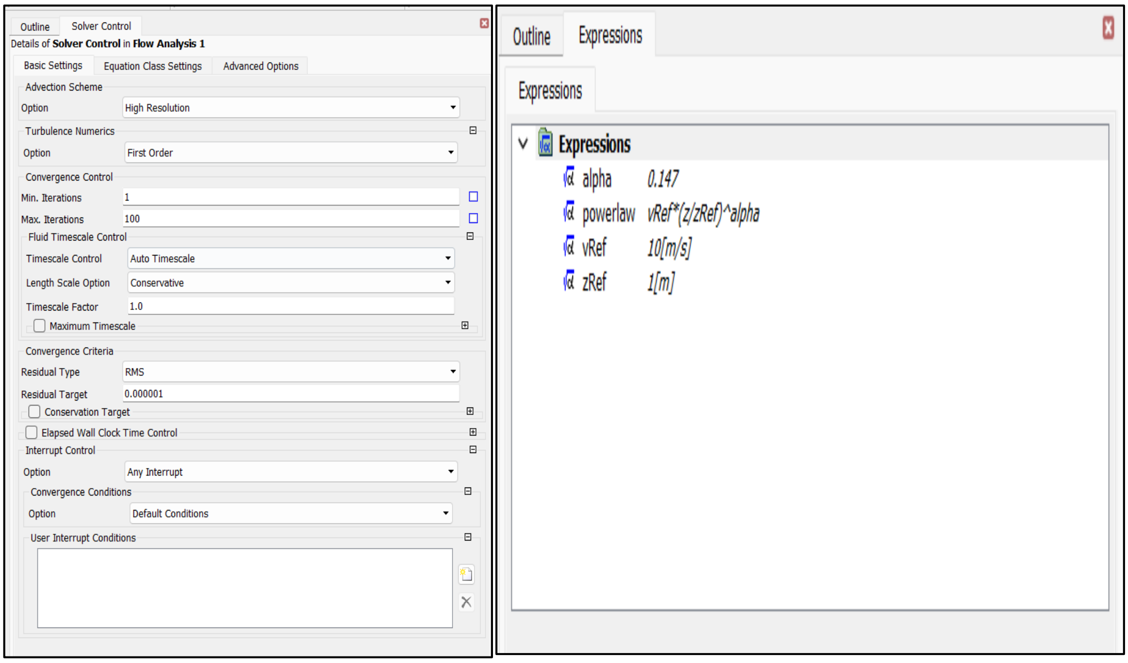


Fig.6.21: Numerical Setup of Chimney with different Bottom Diameter

While other aspects of the setup, such as geometry, meshing, and boundary conditions, remain the same, the bottom diameter is intentionally altered to examine its effects on the overall behavior and performance of the chimney. The numerical setup is shown in fig.6.21. By systematically varying this specific parameter, a comprehensive understanding of the relationship between the bottom diameter and various flow characteristics can be obtained.

6.4.2 RESULTS

6.4.2.1 PRESSURE CONTOUR

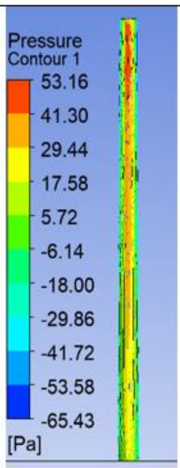
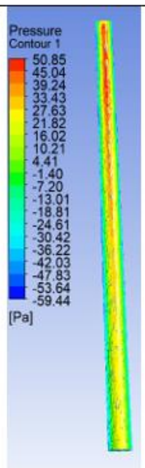
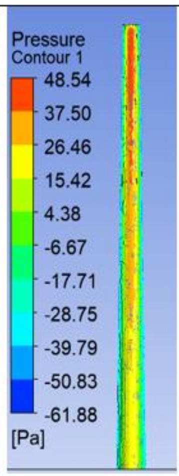
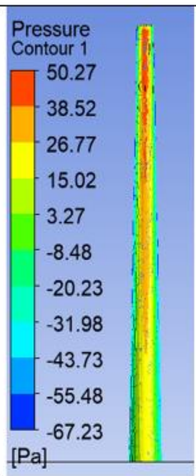
BOTTOM DIAMETER	10m	12m	14m	16m
PRESSURE CONTOUR				

Fig.6.22: Pressure Contours of Chimney with different Bottom Diameter

Altering the base diameter of the chimney can lead to changes in pressure distribution. In general, increasing the base diameter results in lower pressures, while reducing the base diameter leads to higher pressures. This relationship is due to the interaction between wind flow and the chimney geometry, which influences the velocity and pressure fields.

The location of maximum pressure within the chimney will depend on various factors such as wind direction, wind speed, and the specific chimney design. Typically, the maximum pressure occurs at the side of the chimney facing the incoming wind, and it may be influenced by the shape and configuration of the chimney's top.

The minimum pressure occurs on the side of the chimney opposite to the incoming wind direction. This is where the wind flow tends to separate and create areas of lower pressure, often referred to as the lee side or windward side.

6.4.2.2 VELOCITY STREAMLINES

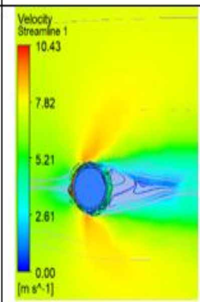
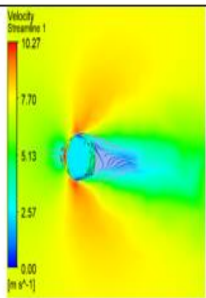
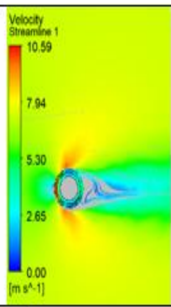
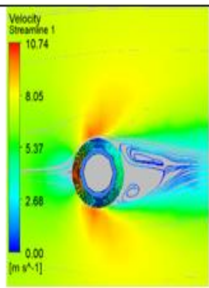
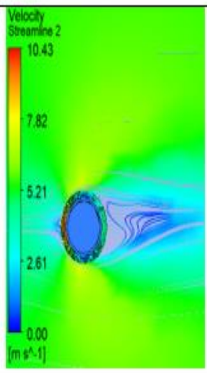
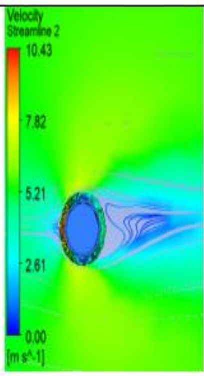
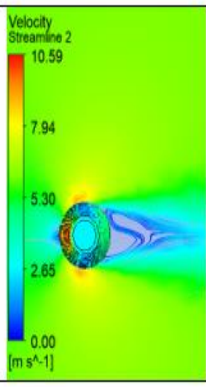
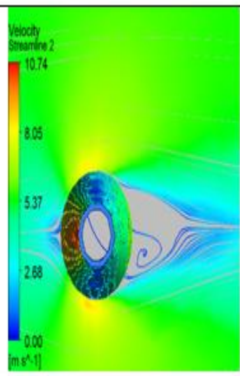
BOTTOM DIAMETER	10m	12m	14m	16m
HORIZONTAL STREAMLINES				
BOTTOM DIAMETER	10m	12m	14m	16m
HORIZONTAL STREAMLINES (at bottom)				

Fig.6.23: Horizontal Streamlines of Chimney with different Bottom Diameter

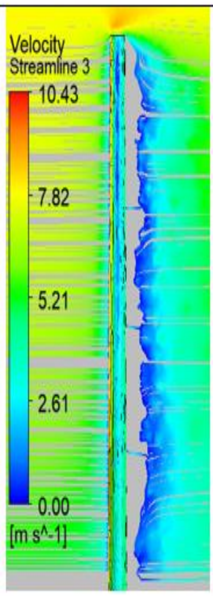
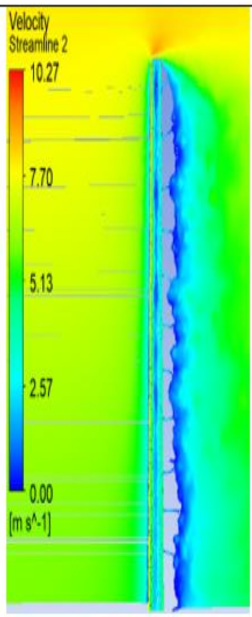
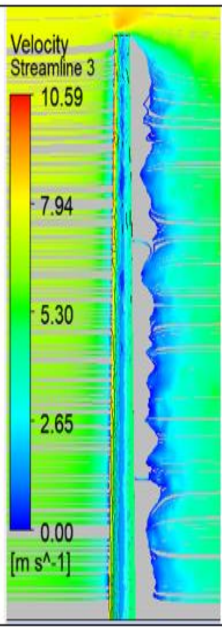
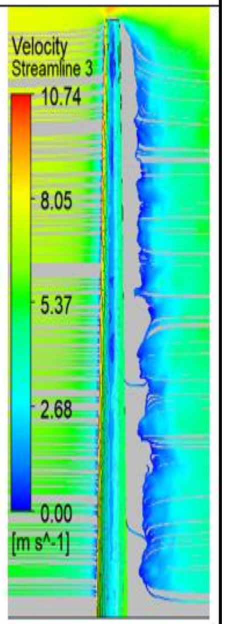
BOTTOM DIAMETER	10m	12m	14m	16m
VERTICAL STREAMLINES				

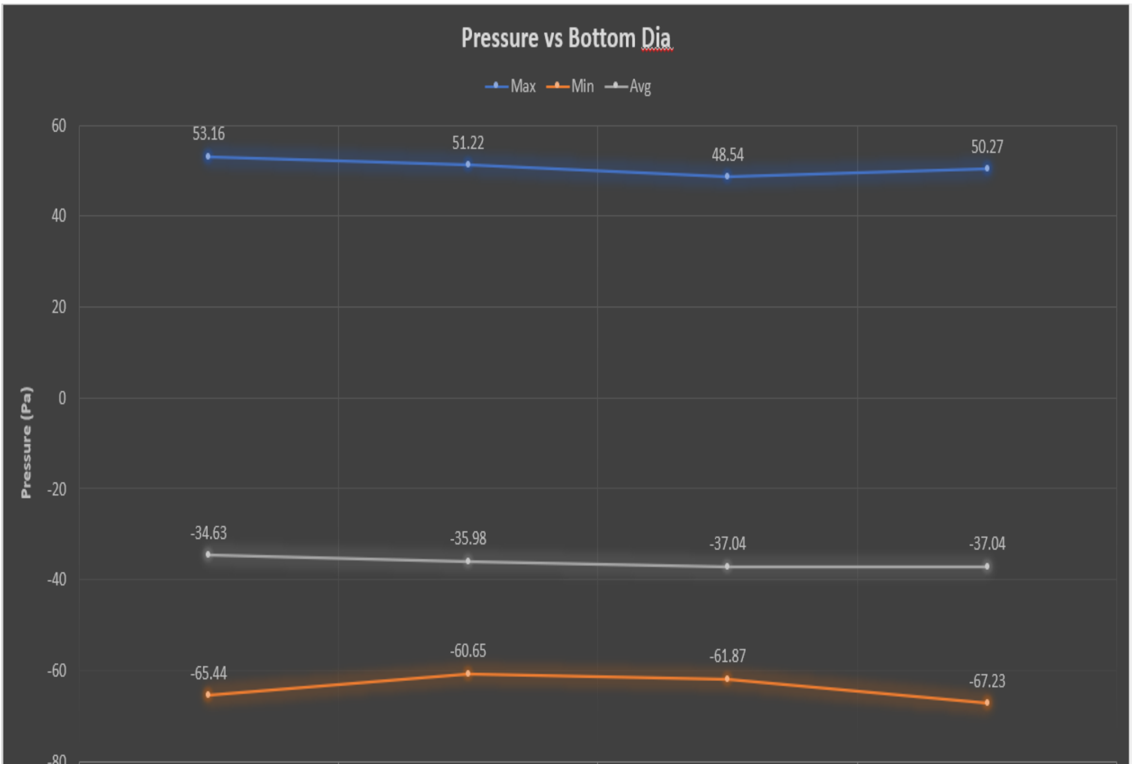
Fig.6.24: Vertical Streamlines of Chimney with different Bottom Diameter

Increasing the base diameter of the chimney can result in changes to the velocity streamlines. As the base diameter increases, the streamlines tend to spread out and become more dispersed. Conversely, decreasing the base diameter leads to tighter and more focused streamlines.

Varying the base diameter can influence the occurrence of flow separation and recirculation zones around the chimney. With a larger base diameter, the flow separation may happen at different locations, leading to the formation of recirculation zones behind the chimney. On the other hand, a smaller base diameter can reduce the extent of flow separation and minimize recirculation.

The base diameter affects the windward and leeward flow characteristics around the chimney. A larger base diameter can enhance windward flow deflection and increase the windward pressure zone. Conversely, a smaller base diameter can reduce windward flow deflection and alter the distribution of wind pressure around the chimney.

6.4.2.3 VARIATION OF PRESSURE



	10	12	14	16
Max	53.16	51.22	48.54	50.27
Min	-65.44	-60.65	-61.87	-67.23
Avg	-34.63	-35.98	-37.04	-37.04

Fig.6.25: Avg., Max., Min. Pressure vs Bottom Diameter

As the base diameter of the chimney varies, there is a corresponding change in the pressure distribution.

Increasing the base diameter of the chimney generally leads to a decrease in pressure. This is due to the larger surface area that the wind interacts with, resulting in a distributed force over a larger region and thus lower pressure. Conversely, decreasing the base diameter tends to increase the pressure as the wind is confined to a smaller area, leading to a higher localized force.

The relationship between pressure and base diameter may not be linear and can vary based on the specific wind conditions and chimney design.

Analyzing the pressure variations with base diameter in CFD wind simulations is crucial for designing chimneys that can withstand wind loads and ensure structural integrity.

6.4.2.4 VARIATION OF Cp

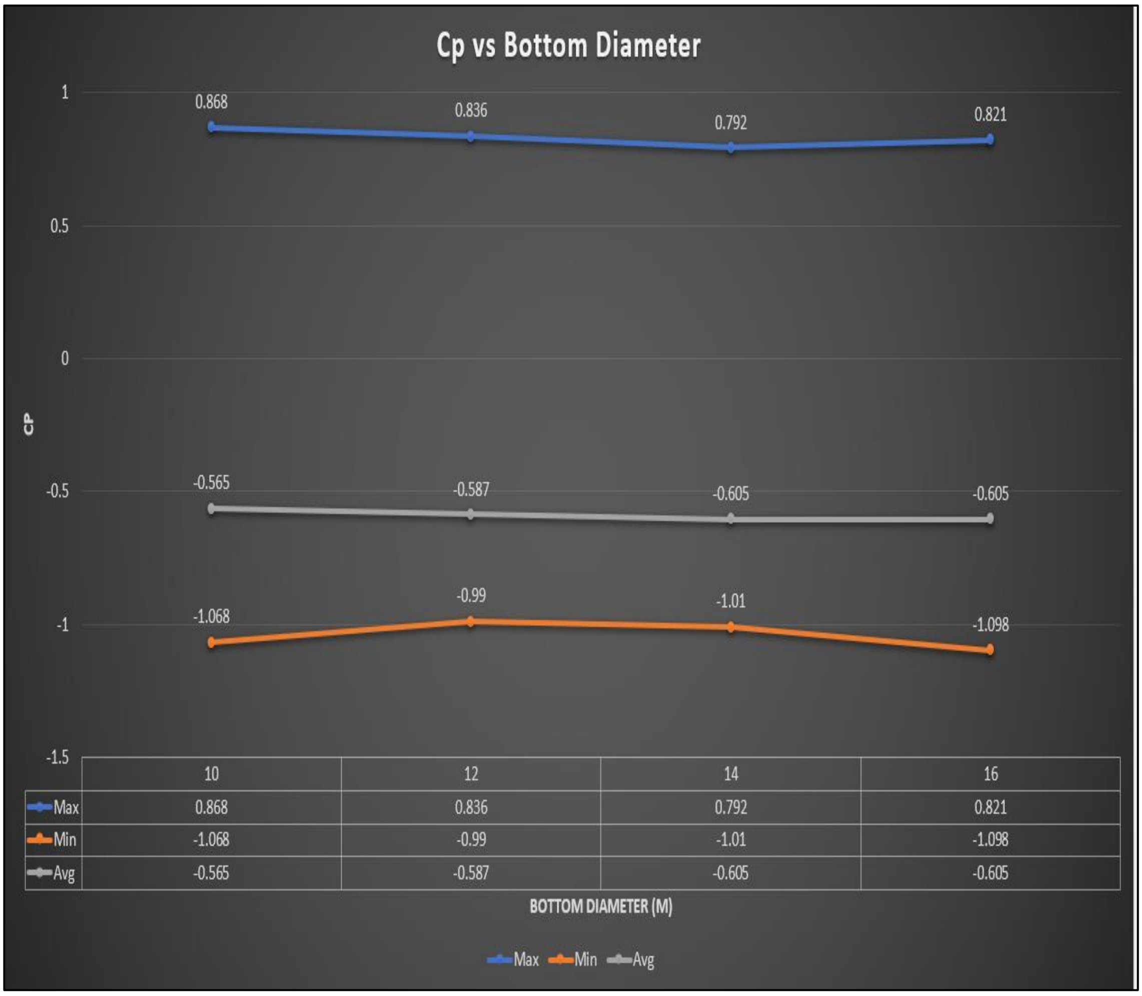


Fig.6.26: Cp vs Bottom Diameter

As the base diameter of the chimney varies, there are observable effects on Cp values. The relationship between Cp and base diameter is crucial in understanding the pressure distribution and airflow characteristics around the chimney structure.

Altering the base diameter influences Cp values. Increasing the base diameter often leads to a decrease in Cp values, indicating a reduced pressure difference between the chimneys. Conversely, decreasing the base diameter can result in higher Cp values, reflecting an increased pressure differential.

The specific relationship between Cp and base diameter may exhibit non-linear behavior, which is contingent upon factors such as wind speed, direction, and the chimney's geometrical features.

CHAPTER 7: CONCLUSIONS

Based on the results obtained from the chimney simulation, which examined the wind effects on a chimney at different incident wind angles, several key observations can be made. The purpose of this thesis was to investigate the impact of wind on chimneys, considering variations in parameters such as size and wind zone. The following conclusion summarizes the findings:

- The simulation results demonstrate that the pressure values observed in the chimney vary depending on the incident wind angle. In all cases examined, the pressure values ranged from negative to positive, indicating the presence of both suction and positive forces acting on the chimney. However, predominantly negative pressure values were observed, suggesting a higher suction force overall.

The coefficient of pressure (Cp) was calculated to quantify the wind pressure distribution on the chimney surface. The Cp values also varied with the incident wind angle, ranging from negative to positive. However, the average Cp values were consistently negative, further supporting the existence of suction forces acting on the chimney.

- These findings indicate that the wind effects on the chimney are influenced by the incident wind angle. As the wind angle deviates from 0 degrees (perpendicular to the chimney), the magnitude of the suction forces tends to decrease. This suggests that wind direction plays a significant role in the wind-induced pressures experienced by the chimney.

It is important to note that the size and wind zone parameters were specifically addressed in this particular conclusion. The obtained results can be used as a foundation for further analysis and investigation of the effects of these parameters on chimneys.

The results obtained from the examination of wind velocities reveal a clear relationship between the velocity of the air flow and the pressure exerted on the chimney. As the wind velocity increases, the pressure variation on the chimney becomes more pronounced, with both positive and negative pressures intensifying.

- In terms of wind zones, the effect of pressure on the chimney can vary depending on the specific zone. For example, in regions where the wind velocity is relatively low, the pressure exerted on the chimney may be moderate, with the streamlines appearing closely spaced and parallel to each other. This laminar flow indicates a smooth and orderly movement of the air particles, with minimal mixing or turbulence.

- However, as the wind velocity increases, particularly in higher wind zones, the pressure on the chimney becomes more significant. The horizontal streamlines begin to separate and display greater curvature, indicating the presence of increased fluid shear and mixing. This transition from laminar to turbulent flow is characterized by chaotic movement, frequent changes in velocity and direction, and enhanced fluid mixing. The streamlines become more irregular, intersecting each other and forming vortices and eddies, leading to a more complex flow structure.
- At even higher wind velocities, such as in extreme wind zones, the pressure variation on the chimney becomes more extreme. The horizontal streamlines become further dispersed and fragmented, reflecting the intensified turbulent flow and enhanced mixing of the fluid. The intricate patterns of the streamlines demonstrate regions of high and low velocity intertwining and forming complex flow structures.
- Overall, the results highlight the influence of wind velocity and wind zones on the pressure distribution and behavior of the streamlines around the chimney. As wind velocity increases, the pressure on the chimney surfaces becomes more pronounced, with a wider range of positive and negative pressures. Additionally, the flow pattern changes from laminar to turbulent, resulting in more irregular and complex streamlines.

The following Conclusions are based on Variation of Parameters.

- Top Diameter: Changing the top diameter of the chimney affects the pressure distribution. A larger top diameter results in a decrease in average pressure and maximum pressure values, but an increase in minimum pressure. Similarly, the behavior of the Coefficient of Pressure (Cp) follows a similar pattern. A larger top diameter facilitates a smoother flow transition as the airflow exits the chimney, leading to a reduction in pressure drop and improved flow efficiency. Moreover, a larger top diameter promotes a more uniform velocity distribution throughout the chimney, ensuring a consistent and controlled flow. Conversely, a smaller top diameter has contrasting effects on flow behavior. Experiments show that a smaller top diameter leads to higher velocities at the outlet, indicating a more concentrated and accelerated flow. This higher velocity flow can induce increased turbulence and pressure fluctuations, which can impact the overall performance of the chimney.

- Bottom Diameter: Modifying the bottom diameter of the chimney results in changes to the pressure distribution. Increasing the base diameter leads to an increase in average pressure and minimum pressure values, while there is a decrease in the maximum pressure. The Coefficient of Pressure (Cp) exhibits a similar variation as the pressure. An increase in the base diameter causes the velocity streamlines to spread out and become more dispersed. In contrast, a smaller base diameter reduces the extent of flow separation and minimizes recirculation. A larger base diameter enhances windward flow deflection and increases the windward pressure zone. Conversely, a smaller base diameter reduces windward flow deflection and alters the distribution of wind pressure around the chimney.
- Height: Altering the height of the chimney influences the pressure values. Decreasing the chimney height leads to a decrease in average pressure, while the maximum and minimum pressure values increase. The Coefficient of Pressure (Cp) follows a similar trend. As the chimney height increases, the streamlines exhibit an elongated and vertically aligned pattern, indicating a more pronounced upward airflow. On the other hand, shorter chimney heights result in more compact and inclined streamlines, indicating a less prominent upward flow. These findings highlight the significant influence of chimney height on the overall flow dynamics and velocity distribution within the chimney system.
- Thickness: Changes in the thickness of the chimney show no significant effect on the maximum and minimum pressure values, but there is a slight increase in the average pressure. The Coefficient of Pressure (Cp) exhibits a similar trend with respect to thickness. Increasing the chimney thickness results in velocity streamlines displaying a more streamlined and organized flow. The increased thickness provides a larger flow passage, allowing for smoother and more laminar airflow. The streamlines follow a more predictable and coherent path. Conversely, thinner chimney walls disrupt and irregularize the velocity streamlines. The

reduced thickness causes increased turbulence and flow disturbances, leading to more chaotic and unpredictable flow patterns. The streamlines display greater deviations and fluctuations in their paths.

REFERENCES

[1] B. Prathyusha Yadav, S. Anitha Reddy, J. Ganesh Yadav, C. Venkata Siva Rama Prasad, "Estimation of seismic and wind loads for design of a 100 m self-supported industrial RCC chimney", Materials Today: Proceedings ,2021.

[2] A.K. Roy, Mohd. Mohsin Khan, "CFD Simulation of Wind Effect on Industrial Chimney", Civil Engineering Conference-Innovation for Sustainability, 2016.

[3] A. N. Dalal, S. B. Savsani, "CFD Analysis of Industrial Chimney Height and Diameter for Optimum Performance", International Journal of Emerging Technology and Advanced Engineering,2016.

[4] H. J. Cui and Q. Yan., "CFD Simulation of the Performance of a Passive Solar Chimney for Natural Ventilation in Buildings", Journal of Wind Engineering and Industrial Aerodynamics,2017.

[5] J. Revuz, D.M. Hargreaves and J.S. Owen, "On the domain size for the steady-state CFD modelling of a tall building", Wind and Structures, 2012.

[6] Y. K. Park, K. H. Kwon, and K. Y. Kim, "CFD Simulation of Wind Effects on a Circular Chimney in a Turbulent Environment" Journal of Wind Engineering and Industrial Aerodynamics,2015.

[7] A. El Hassan, A. Ben Mnaouer, M. T. Chaieb, "CFD Analysis of Wind Flow Around Industrial Chimney", Applied Mechanics and Materials,2010.

[8]"IS 875 (Part 3)," Bureau of Indian Standards. IS 875 (Part 3): Wind loads on buildings and structures (3rd ed.), 2015.

[9] Alok David John, Ajay Gairola, Eshan Ganju, and Anant Gupta, "Design Wind Loads on Reinforced Concrete Chimney – An Experimental Case Study", The Twelfth East Asia-Pacific Conference on Structural Engineering and Construction, Procedia Engineering,2011.

[10] A. Gupta and V. Singh, "CFD simulation of wind effects on industrial chimney and analysis of pressure distribution", Wind and Structures, 2015.

[11] L. Wang and H. Zhang, “Numerical simulation of wind effects on circular chimneys and analysis of pressure distribution”, Journal of Wind Engineering and Industrial Aerodynamics ,2017.

[12] M. Kumar and S. Sharma, “CFD analysis of wind effects on tall chimneys and investigation of pressure distribution”, Materials Today: Proceedings ,2018.

[13] R. Patel and S. Desai, “Computational analysis of wind effects on chimney structures and evaluation of pressure distribution”, Journal of Wind Engineering and Industrial Aerodynamics ,2016.

[14] C.V. Siva Rama Prasad, Y. Vijaya Simha Reddy, J. Prashanth Kumar, A.Vijay Kumar and S.Sreevastav Reddy, “Earthquake and Wind Analysis of a 100m Industrial RCC Chimney”, International Journal of Technical Innovation in Modern Engineering & Science,2018.

[15] B. R. Jayalekshmi, S. V. Jisha, R. Shivashankar, “Wind load analysis of tall chimneys with piled raft foundation considering the flexibility of soil”, International Journal Advance Structural Engineering,2015.