

CFD INVESTIGATION ON HORIZONTAL AXIS WIND TURBINE USING DIFFERENT NACA SERIES AIRFOILS

Submitted in partial fulfilment of the requirements for the award of

**Bachelor of Engineering degree in
Aeronautical Engineering**

By

R BALASUBRAMANIAM (Reg. No. 38260005)

K GOKULAKRISHNAN (Reg. No. 38260012)



DEPARTMENT OF AERONAUTICAL ENGINEERING

SCHOOL OF MECHANICAL ENGINEERING

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited with grade "A" by NAAC

JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI – 600119

MARCH 2022



SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited with "A" grade by NAAC
Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai - 600 119
www.sathyabama.ac.in



DEPARTMENT OF AERONAUTICAL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the Bonafide work of
R BALASUBRAMANIAM (Reg. No. 38260005) and K GOKULAKRISHNAN
(Reg. No. 38260012) who carried out the project entitled **"CFD INVESTIGATION
ON HORIZONTAL AXIS WIND TURBINE BLADE USING DIFFERENT NACA
SERIES AIRFOILS"** under our supervision from September 2021 to April 2022.

Internal Guide

Mr. GUNASHEKAR, M.E.

Head of the Department

Dr. P. BOOMADEVI, ME., PhD.

Submitted for Viva voce Examination held on _____

Internal Examiner

External Examiner

DECLARATION

We R BALASUBRAMANIAM (Reg.No. 38260005) and K GOKULAKRISHNAN (Reg. No. 38260012) hereby declare that the Project Report entitled “**CFD INVESTIGATION ON HORIZONTAL AXIS WIND TURBINE BLADE USING DIFFERENT NACA SERIES AIRFOILS**” done by us under the guidance of **Mr. GUNASHEKAR, M.E.**, is submitted in partial fulfilment of the requirements for the award of Bachelor of Engineering degree in Aeronautical Engineering.

DATE: .04.2022

SIGNATURE OF THE CANDIDATE

PLACE: CHENNAI

ACKNOWLEDGEMENT

I am pleased to acknowledge my sincere thanks to Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and for completing it successfully. I am grateful to them.

I convey my sincere thanks to **Dr. S. Prakash, M.E., Ph.D.**, Dean, School of Mechanical Engineering and **Dr. P. Boomadevi, M.E., Ph.D.**, Head of the Department, Dept. of Aeronautical Engineering for providing me necessary support and details at the right time during the progressive reviews.

I would like to express my sincere and deep sense of gratitude to my Project Guide **Mr. Gunashekar, M.E.**, for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of my project work.

I also express my thanks to all Teaching and Non-teaching staff members of the Department of Mechanical Engineering who were helpful in many ways for the completion of the project.

ABSTRACT

The wind is an energy source that has the properties of a clean, free, and readily available energy source. However, the efficiency of the existing rotors used to harness wind power is still not satisfactory. Thus, in this current study, the development and aerodynamic performance investigation of horizontal axis wind turbine blade using 4 types of NACA airfoils comprising of one symmetrical, two 4-digit series and one 5 digit series airfoil have been analyzed through the computational fluids dynamic (CFD) simulation approach. The main motive of this study was to investigate the aerodynamic performance of NACA airfoils to be used on a horizontal axis wind turbine (HAWT) blade, which will assist in further understanding the physics of the interaction between airflow and the wind turbine blades. The simulation was performed using two-dimensional computational models based on an steady state and k-epsilon turbulence model to determine the aerodynamic efficiency of 4 different NACA airfoils. Comparing the results of these NACA airfoils to select the effective airfoil among these airfoils and calculate the power output of wind turbine.

TABLE OF CONTENTS

CHAPTER NO	TITLE	PAGE No
	ABSTRACT	5
	LIST OF FIGURES	8
	LIST OF TABLES	10
	LIST OF SYMBOLS AND ABBREVIATIONS	11
1	INTRODUCTION	14
	1.1 Wind energy	14
	1.2 Wind turbine	14
	1.3 Types of wind turbine	17
	1.4 Applications of wind turbines	18
	1.5 Difference between Horizontal and Vertical Axis Wind Turbine	20
	1.6 NACA Airfoils	20
	1.7 Types of NACA Airfoils	21
2	LITERATURE SURVEY	24
3	AIM AND SCOPE	32
	3.1 Objective and scope	32
	3.2 Methodology	32
	3.3 Overview of this Report	33
4	COMPUTATIONAL FLUID DYNAMICS	34

5	3D MODELING OF BLADES	35
	5.1 3D Modeling of Horizontal axis wind turbine blades using Symmetric, 4 Digit and 5 Digit Airfoils	35
	5.2 3D Modeling of NACA 4415 Airfoil blade (Reference blade)	36
	5.3 3D Modeling of NACA 0015 Airfoil blade :	36
	5.4 3D Modeling of NACA 2412 airfoil blade	37
	5.5 3D Modeling of NACA 63215 Airfoil blade	38
	5.6 Creating 3 Blades	40
6	COMUPUTATIONAL DOMAIN FOR SIMULATION	41
	6.1 General	41
	6.2 Boundary creation	42
	6.3 Meshing	43
	6.4 CFD Boundary	45
7	3D NUMERICAL SIMULATION IN ANSYS	48
	7.1 Overview of ANSYS Fluent	48
	7.2 Analysis	49
	7.3 Solver and Material Selection and Operating Condition Defining	49
	7.4 Boundary conditions	50
	7.5 Residual Monitoring	51

	7.6 Report Reference Values	52
	7.7 Iteration	52
8	RESULTS AND DISCUSSION	53
	8.1 Result sets	53
	8.2 Reference blade	
	(NACA 4415 Airfoil blade) Results	53
	8.3 NACA 0015 Airfoil blade results	57
	8.4 NACA 2412 Airfoil blade results	61
	8.5 NACA 63215 Airfoil blade results	65
	8.6 Comparison table for aerodynamic forces and aerodynamic efficiency values of various blades	70
9	WIND TUNNEL EXPERIMENT	71
	9.1 Coefficient of pressure (C_p) calculation	71
	9.2 Coefficient of Lift (C_L) and Drag (C_D) calculation	72
	9.3 Aerodynamic forces - Lift (L) and Drag (D) calculation	72
10	POWER CALCULATION	73
11	SUMMARY AND CONCLUSION	74
12	REFERENCES	76

LIST OF FIGURES

FIGURE No	TITLE	PAGE No.
Fig.1.1	Principle of wind turbine	15
Fig.1.2	Principles of wind turbine Aerodynamic lift	16
Fig.1.3	Horizontal axis wind turbine	17
Fig.1.4	Vertical axis wind turbine	18
Fig.1.5	Difference between Horizontal and Vertical Axis wind turbine	20
Fig.1.6	Types of NACA series airfoils	23
Fig.5.1	3D CAD model of NACA 4415 airfoil blade (Reference model)	36
Fig.5.2	3D CAD model of NACA 0015 airfoil blade	37
Fig.5.3	3D CAD model of NACA 2412 airfoil blade	38
Fig.5.4	3D CAD model of NACA 63215 airfoil blade	39
Fig.5.5	Creating 3 blades using circular pattern	40
Fig.6.1	Importing 3D STEP model into ANSYS Fluent	41
Fig.6.2	3D Turbine blade model in 3D boundary	42
Fig.6.3	3D Boundary specifications	43
Fig.6.4	Coarse mesh in Boundary and Fine mesh in turbine blade	44
Fig.6.5	Enlarged image of fine mesh on turbine blade	44
Fig.6.6	Velocity inlet of the computational domain	45
Fig.6.7	Pressure outlet of the computational domain	46
Fig.6.8	Walls of the computational domain	47

FIGURE No	TITLE	PAGE No.
Fig.7.1	Grid check in Fluent software	49
Fig.8.1	Print console of Lift force and Coefficient of lift values of NACA 4415 airfoil blade	54
Fig.8.2	Print console of Drag force and Coefficient of drag values of NACA 4415 airfoil blade	54
Fig.8.3	3D Contour of Velocity pathlines of Reference blade model (NACA 4415 airfoil blade)	55
Fig.8.4	3D Contour of pressure pathlines of Reference blade model (NACA 4415 airfoil blade)	56
Fig.8.5	3D Contour of Turbulence intensity pathlines of Reference blade model (NACA 4415)	57
Fig.8.6	Print console of Lift force and Coefficient of lift values of NACA 0015 airfoil blade	57
Fig.8.7	Print console of Drag force and Coefficient of drag values of NACA 0015 airfoil blade	58
Fig.8.8	3D Contour of Velocity pathlines of NACA 0015 airfoil blade	59
Fig.8.9	3D Contour of pressure pathlines of NACA 0015 airfoil blade	60
Fig.8.10	3D Contour of Turbulence intensity pathlines of NACA 0015 airfoil blade	61
Fig.8.11	Print console of Lift force and Coefficient of lift values of NACA 2412 airfoil blade	61
Fig.8.12	Print console of Drag force and Coefficient of drag values of NACA 2412 airfoil blade	62

FIGURE No	TITLE	PAGE No.
Fig.8.13	3D Contour of Velocity pathlines of NACA 2412 airfoil blade	63
Fig.8.14	3D Contour of pressure pathlines of NACA 2412 airfoil blade	64
Fig.8.15	3D Contour of Turbulence intensity pathlines of NACA 2412 airfoil blade	65
Fig.8.16	Print console of Lift force and Coefficient of lift values of NACA 63215 airfoil blade	66
Fig.8.17	Print console of Drag force and Coefficient of drag values of NACA 63215 airfoil blade	66
Fig.8.18	3D Contour of Velocity pathlines of NACA 63215 airfoil blade	67
Fig.8.19	3D Contour of pressure pathlines of NACA 63215 airfoil blade	68
Fig.8.20	3D Contour of Turbulence intensity pathlines of NACA 63215 airfoil blade	69

LIST OF TABLES

TABLE No	TITLE	PAGE No.
1	Solver specifications	50
2	Material Properties	50
3	Velocity inlet conditions & Pressure Outlet conditions	51
4	Reference Values	52
5	Iteration Values	52
6	Aerodynamic forces and aerodynamic efficiency comparison of various blade designs	70
7	Wind tunnel readings of slot holes with leading and trailing edge serration blade model	71
8	Coefficient of Lift and Drag calculation	71

LIST OF SYMBOLS AND ABBEREVIATIONS

C_L	Coefficient of Lift (No unit)
C_D	Coefficient of Drag (No unit)
C_p	Coefficient of Pressure (No unit)
L	Lift force (N)
D	Drag force (N)
V_{avg}	Average velocity (m/s)
V_∞	Free stream velocity (mm)
P_0	Static pressure (Pa)
P	Total pressure (Pa)
ρ	Density (kg/m ³)
C_P	Betz coefficient (No unit)
A	Surface Area (m ²)
P	Power output (W)

CHAPTER - 1

INTRODUCTION

1.1) Wind energy :

Wind energy has recently become the world's fastest growing source of renewable energy. They are emissions-free and wind is renewable and cost-free.

The terms "wind energy" and "wind power" both describe the process by which the wind is used to generate mechanical power or electricity. This mechanical power can be used for specific tasks (such as grinding grain or pumping water) or a generator can convert this mechanical power into electricity.

1.2) Wind turbine :

A wind turbine turns wind energy into electricity using the aerodynamic force from the rotor blades, which work like an airplane wing or helicopter rotor blade. When wind flows across the blade, the air pressure on one side of the blade decreases. The difference in air pressure across the two sides of the blade creates both lift and drag. The force of the lift is stronger than the drag and this causes the rotor to spin. The rotor connects to the generator, either directly (if it's a direct drive turbine) or through a shaft and a series of gears (a gearbox) that speed up the rotation and allow for a physically smaller generator. This translation of aerodynamic force to rotation of a generator creates electricity.

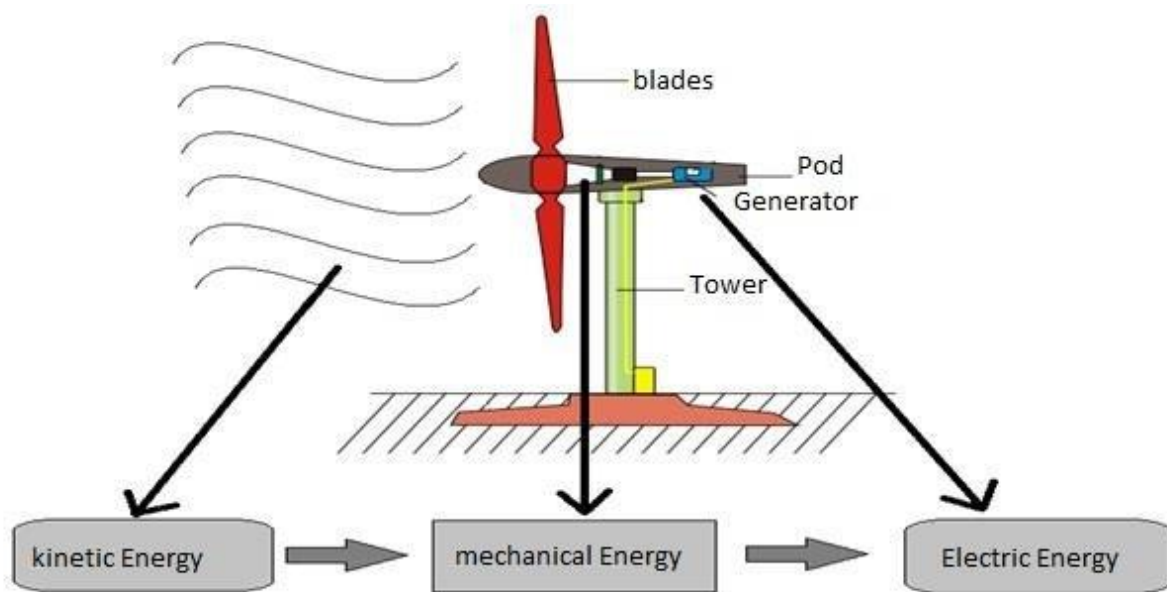


Fig.1.1 : Principle of wind turbine

In above figure represents, wind energy(kinetic energy) converted into mechanical energy and the mechanical energy converted into electrical energy (power output)

The power can be defined as

$$P = \frac{1}{2} \rho A V^3 \text{ (Watts)}$$

A German physicist Albert Betz concluded in 1919 that no wind turbine can convert more than 16/27 (59.3%) of the kinetic energy of the wind into mechanical energy turning a rotor. To this day, this is known as the Betz Limit or Betz' Law. The theoretical maximum power efficiency of any design of wind turbine is 0.59 (i.e. no more than 59% of the energy carried by the wind can be extracted by a wind turbine). This is called the “power coefficient” and is defined as: $C_p = 0.59$. Also, wind turbines cannot operate at this maximum limit. The C_p value is unique to each turbine type and is a function of wind speed that the turbine is operating in. Once we incorporate various engineering requirements of a wind turbine - strength and durability in particular - the real world limit is well below the Betz Limit with values of 0.35-0.45 common even in the best designed wind turbines. By the time we take into account the other factors in a complete wind turbine system - e.g. the gearbox, bearings, generator and so on - only 10-30% of the

power of the wind is ever actually converted into usable electricity. Hence, the power coefficient needs to be factored in equation and the extractable power from the wind is given by

$$P = \frac{1}{2} \rho A V^3 C_p \text{ (Watts)}$$

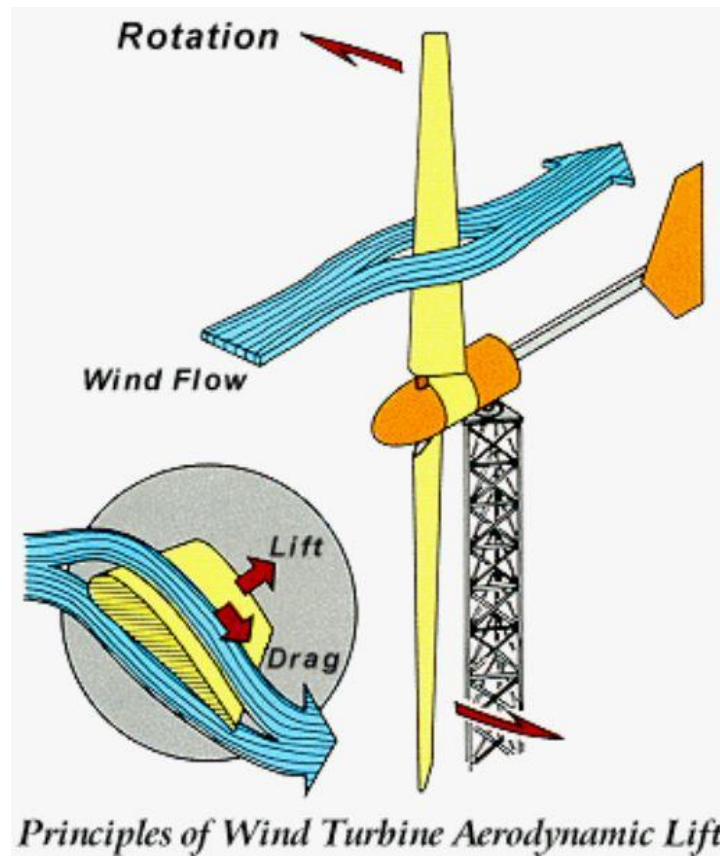


Fig.1.2 : Principles of wind turbine Aerodynamic lift

In figure 2, the lift and drag forces are generated when wind flow acts on the blade. Based on aerodynamic shape of blade, the blade is rotated when those forces acts on a blade. Speed of rotation is completely depends on wind speed and aerodynamic shape of blade to produce lift and drag forces

1.3) Types of Wind Turbines :

The two main types of turbines are Horizontal-axis Turbines (HAWT) and Vertical-axis turbines (VAWT). HAWT have the rotating axis oriented horizontally. They typically feature 3-blades and are designed to face to the wind. VAWT have the rotating axis aligned vertically and are designed to harnesses kinetic energy in the opposite direction.

(a) HORIZONTAL-AXIS TURBINES

The most common type of wind turbine is the 'Horizontal Axis Wind Turbine' (HAWT). It is referred to as a horizontal axis as the rotating axis lies horizontally. Horizontal-axis wind turbines have three blades and operate "upwind," with the turbine pivoting at the top of the tower so the blades face into the wind.

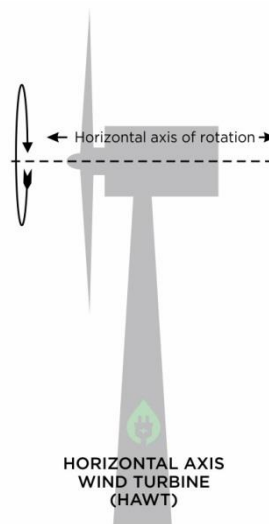


Fig.1.3 : Horizontal axis wind turbine

A HAWT needs to point directly into the wind to operate at maximum efficiency, and the whole head is designed to turn to face the wind. As the wind changes direction, so the head must turn (or 'yaw') to stay pointing into the wind.

HAWTs are chosen for off-shore wind farms and on-shore wind farms where the land is largely flat and open, because they work more efficiently than VAWTs in areas where the wind is not turbulent.

(b) VERTICAL-AXIS TURBINES

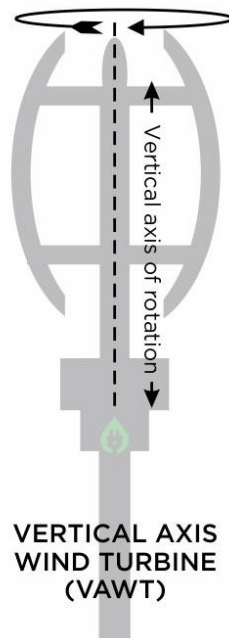


Fig.1.4 : Vertical axis wind turbine

These turbines are omnidirectional, meaning they don't need to be adjusted to point into the wind to operate. It is referred to as vertical axis as the rotating axis is aligned vertically upwards.

Wind turbines can be built on land or offshore in large bodies of water like oceans and lakes

1.4) Applications of Wind Turbines :

Modern wind turbines can be categorized by where they are installed and how they are connected to the grid:

(a) LAND-BASED WIND

Land-based wind turbines range in size from 100 kilowatts to as large as several megawatts.

Larger wind turbines are more cost effective and are grouped together into wind plants, which provide bulk power to the electrical grid.

(b) OFFSHORE WIND

Offshore wind turbines tend to be massive. They do not have the same transportation challenges of land-based wind installations, as the large components can be transported on ships instead of on roads.

These turbines are able to capture powerful ocean winds and generate vast amounts of energy.

(c) DISTRIBUTED WIND

Many turbines used in distributed applications are small wind turbines. Single small wind turbines below 100 kilowatts are typically used for residential, agricultural, and small commercial and industrial applications.

Small turbines can be used in hybrid energy systems with other distributed energy resources, such as microgrids powered by diesel generators, batteries, and photovoltaics.

These systems are called hybrid wind systems and are typically used in remote, off-grid locations(where a connection to the utility grid is not available) and are becoming more common in grid-connected applications for resiliency.

1.5) Difference between Horizontal and Vertical Axis Wind Turbine :

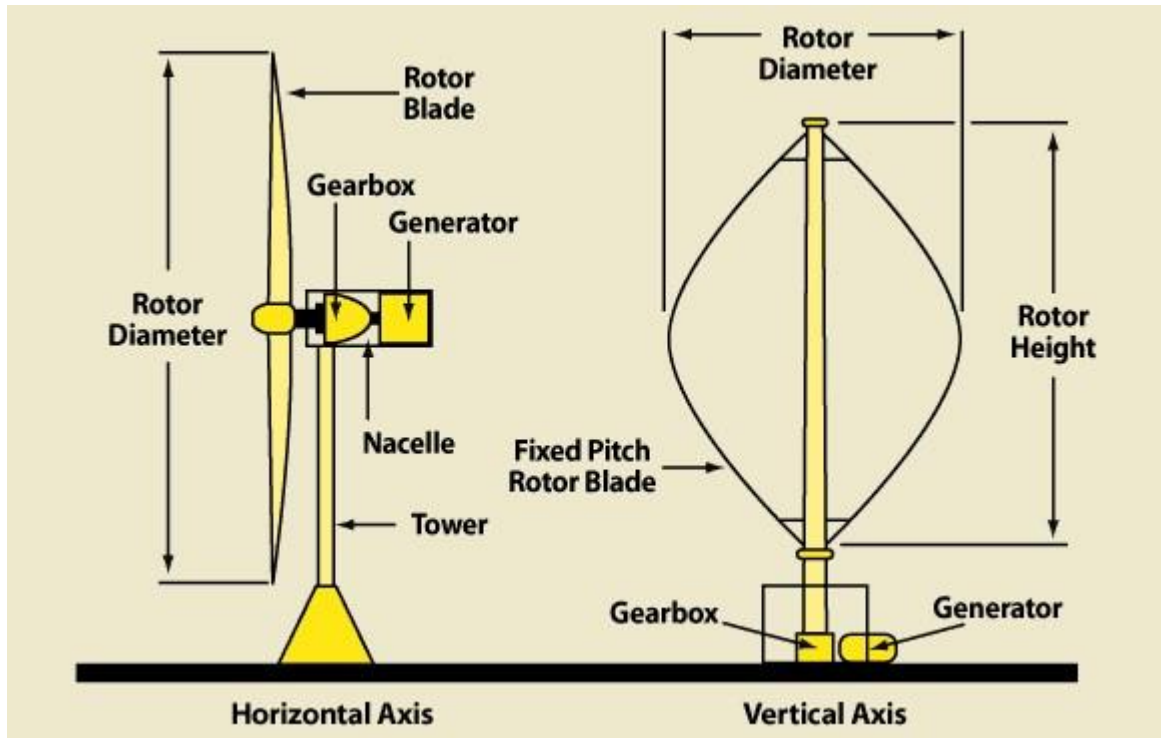


Fig.1.5 : Difference between Horizontal and Vertical Axis wind turbine

Larger difference between horizontal and vertical axis wind turbines. Vertical axis wind turbines are wind turbines whose rotors rotate around a vertical shaft with vertically oriented blades. They produce electricity by utilizing wind power the same way horizontal axis wind turbines do: Wind drives the rotor to turn, the rotation connects to the generator, and the generator converts the mechanical energy into electricity.

1.6) NACA Airfoils :

The NACA airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of the NACA airfoils is described using a series of digits following the word "NACA". The parameters in the numerical code can be entered into equations to precisely generate the cross-section of the airfoil and calculate its properties.

Why NACA Airfoils are used in wind turbines ?

An airfoil is the foundation of wind turbine blade design, and accordingly, optimizing its design plays a key role in improving aerodynamic performance, noise control, and structural robustness of a rotor blade. Throughout the 40-year history of wind industry development, airfoil optimization has had the following three primary objectives: (1) improving blade aerodynamic performance, (2) reducing blade aerodynamic noise, and (3) increasing blade structural robustness.

In general, aviation airfoils (such as the NACA series airfoils) were widely utilized in blade design due to their advantages of high lift coefficients and low drag coefficients, both of which significantly enhanced the early development of wind power industry.

1.7) Types of NACA Airfoils :

The early NACA airfoil series, the 4-digit, 5-digit, and modified 4-/5-digit, were generated using analytical equations that describe the camber (curvature) of the mean-line (geometric centerline) of the airfoil section as well as the section's thickness distribution along the length of the airfoil. Later families, including the 6-Series, are more complicated shapes derived using theoretical rather than geometrical methods. Before the National Advisory Committee for Aeronautics (NACA) developed these series, airfoil design was rather arbitrary with nothing to guide the designer except past experience with known shapes and experimentation with modifications to those shapes.

(a) NACA Four-Digit Series:

The first family of airfoils designed using this approach became known as the NACA Four-Digit Series. The first digit specifies the maximum camber (m) in percentage of the chord (airfoil length), the second indicates the position of the maximum camber (p) in tenths of chord, and the last two numbers provide the maximum thickness (t) of the airfoil in percentage of chord. For example, the NACA 2415 airfoil has a maximum thickness of 15% with a camber of 2% located 40% back from the airfoil leading edge (or 0.4c).

(b) NACA Five-Digit Series:

The NACA Five-Digit Series uses the same thickness forms as the Four-Digit Series but the mean camber line is defined differently and the naming convention is a bit more complex. The first digit, when multiplied by $3/2$, yields the design lift coefficient (c_l) in tenths. The next two digits, when divided by 2, give the position of the maximum camber (p) in tenths of chord. The final two digits again indicate the maximum thickness (t) in percentage of chord. For example, the NACA 23012 has a maximum thickness of 12%, a design lift coefficient of 0.3, and a maximum camber located 15% back from the leading edge

(c) NACA 6-Series:

The 6-Series was derived using an improved theoretical method that, like the 1-Series, relied on specifying the desired pressure distribution and employed advanced mathematics to derive the required geometrical shape. The goal of this approach was to design airfoils that maximized the region over which the airflow remains laminar. In so doing, the drag over a small range of lift coefficients can be substantially reduced. The naming convention of the 6-Series is by far the most confusing of any of the families discussed thus far, especially since many different variations exist. One of the more common examples is the NACA 641-212, $a=0.6$. In this example, 6 denotes the series and indicates that this family is designed for greater laminar flow than the Four- or Five-Digit Series. The second digit, 4, is the location of the minimum pressure in tenths of chord (0.4c). The subscript 1 indicates that low drag is maintained at lift coefficients 0.1 above and below the design lift coefficient (0.2) specified by the first digit after the dash in tenths. The final two digits specify the thickness in percentage of chord, 12%.

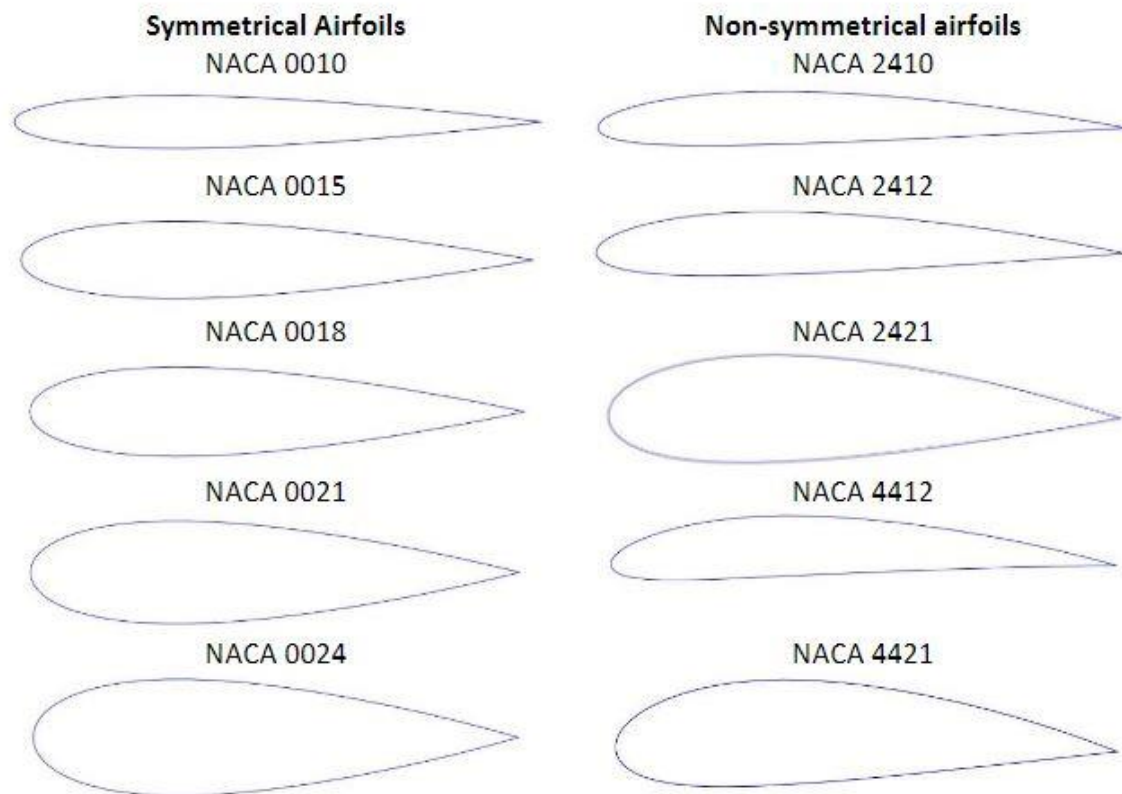


Fig.1.6 : Types of NACA series airfoils

CHAPTER - 2

LITERATURE REVIEW

1) Effect of Airfoil Concavity on Wind Turbine Blade Performances

Authors : Jianlong Ma, Yafan Duan, Ming Zhao, Wenchun Lv, Jianwen Wang,
Qilao Meng Ke, and Yongfeng Ren

Year : 2019

Although the optimization of wind turbine blade aerodynamic performance has achieved fruitful results, whether airfoil concavity, an important method for preventing flow separation, is also feasible for improving the aerodynamic performance has not been confirmed scientifically. Thus, we selected the blade of a small horizontal-axis wind turbine as a research model and proposed an optimization method based on airfoil concavity near the trailing edge of the blade suction surface. The experimental results showed that airfoil concavity improved blade aerodynamic performance by 3–15%. Subsequently, its effects on the sound pressure level within the wake flow field were investigated using an acoustic array, and the results suggested that the sound pressure level was reduced by 9.6–15.8%. Lastly, a modal test of the rotor blade was conducted. Although the natural frequencies of the 1st and 2nd order vibrations had hardly changed, their vibrational stiffness were increased by 7 and 4.9%, respectively, which indicated that airfoil concavity significantly improved structural robustness.

2) Design of Naca63215 Airfoil for a Wind Turbine

Authors : N Manikandan, Stalin Balasubramaniam

Year : 2017

The ultimate objective of the work is to increase the reliability of wind turbine blades through the development of the airfoil structure and also to reduce the noise produced during the running period of the wind turbine blades. The blade plays a pivotal role, because it is the most important part of the energy absorption system. Consequently, the blade has to be designed carefully to enable to absorb energy with its greatest efficiency. In this work, Pro/E, Hypermesh software has been used to design blades effectively. NACA 63-215 airfoil profile is considered for analysis of wind turbine blade. The wind turbine blade is modeled and several sections are created from root to tip with the variation from the standard design for improving the efficiency. For the further improvement required in the efficiency of the wind turbine the winglet is to be included at the tip of the blade which would help in increasing the efficiency and reducing the noise produced from the blades in working condition. The existing turbine blade and the modified blade with the winglet are compared for their results.

3) Development of High Performance Airfoils for Application in Small Wind Turbine Power Generation

Authors : Emmanuel Yeboah Osei, Richard Opoku, Albert K. Sunnu, and Muiyiwa S. Adaramola

Year : 2020

Small wind turbine power generation systems have the potential to meet the electricity demand of the residential sector in developing countries. However, due to their exposure to low Reynolds number (Re) flow conditions and associated problems, specific airfoils are required for the design of their blades. In this research, XFOIL was used to develop and test three high performance airfoils (EYO7-8, EYO8-8, and EYO9-8) for small wind turbine application. The airfoils were subsequently used in conjunction with Blade Element Momentum Theory to develop and test 3-bladed 6 m diameter wind turbine rotors. The aerodynamic performance parameters of the airfoils tested were lift, drag, lift-to-drag ratio, and stall angle. At , EYO7-8, EYO8-8, and EYO9-8 had maximum lift-to-drag ratios of 134, 131, and 127, respectively, and maximum lift coefficients of

1.77, 1.81, and 1.81, respectively. The stall angles were 12° for EYO7-8, 14° for EYO8-8, and 15° for EYO9-8. Together, the new airfoils compared favourably with other existing low Re airfoils and are suitable for the design of small wind turbine blades. Analysis of the results showed that the performance improvement of the EYO-Series airfoils is as a result of the design optimization that employed an optimal thickness-to-camber ratio () in the range of 0.85–1.50. Preliminary wind turbine rotor analysis also showed that the EYO7-8, EYO8-8, and EYO9-8 rotors had maximum power coefficients of 0.371, 0.366, and 0.358, respectively.

4) Simulation Analysis on the Blade Airfoil of Small Wind Turbine

Authors : Hong Zongheng , Yu Tao , Chen Guanyu ,Li Xiangrui, Hong Yu

Year : 2019

The selection of the blade airfoil of wind turbine has a crucial impact on the overall efficiency of wind turbine and wind energy utilization. Aiming at the small 300W household wind turbine on the market, based on its known design parameters and using the software of Profili and SolidWorks, the 3D solid modeling of the blade has been achieved in three common airfoils (NACA4412, NACA0009 and NACA 23012). Then, by using the Fluent module of the finite element analysis software ANSYS, the actual working environment of the blade is simulated. The results show that the blades under the NACA0009 airfoil can produce a relatively maximum axial torque of 3.7808 Nm. Finally, based on the above research method, the new airfoil Icarus was discovered by improving the existing airfoil, which has a relatively low manufacturing cost, and the average pressure difference per unit area of the blade is 4.95% higher than that of the NACA0009 airfoil. This research can provide reference for the improvement of airfoil of small wind turbines, which has certain practical significance and practical value.

5) Effects of Design Parameters on Aerodynamic Performance of New Profile Small Wind Turbine Blades

Authors : Mahasidha Birajdar, Sandip A Kale, Shivalingappa N Sapali

Year : 2015

Wind is a one of the clean resources of energy and has the ability to contribute a considerable share in growing world energy consumption. The small wind turbine plays a vital role in fulfillment of energy needs preferably for household purpose. In order to unleash the budding of applicability of small wind turbine, it is necessary to improve its performance. The performance of a small wind turbine can be distinguished by the manners in which power, thrust and torque vary with the wind speed. The wind power indicates the amount of energy captured by the wind turbine rotor. It is convenient to express the performance of small wind turbine by means of non-dimensional performance curves, therefore in this paper the most graphs are drawn to power, thrust and torque coefficients as a function of the tip speed ratio. This paper presents the effect of design parameters such as the tip speed ratio, angle of attack, wind speed, solidity, number of blades, etc. on the aerodynamic performance of small wind turbine and proposes the optimum values of these parameters for the newly designed blade. The new designed blade consists of two new airfoils and named as IND 15045 and IND 09848. This new profile blade is designed for a wind turbine of 1 kW rated power. The blade is divided into ten sections. The designed length of blade is 1.5 m and it is made using IND 15045 airfoils at three root sections and IND 09848 airfoils for remaining seven sections. Q-Blade is used for the numerical simulation of wind turbine airfoils and blade. It is integrated tool of XFOIL and blade element momentum theory of wind turbine blade design. Also the effect of constant rotational speed operation, effect of stall regulation effect of rotational speed change and the effect of solidity on the performance of wind turbine is discussed. This paper delivers a broad view of perception for design of small wind turbine and parameter selection for the new wind turbine blade. Also in this paper the effect of different losses viz. tip losses, drag losses, stall losses and hub losses on the small wind turbine are discussed. The efficiency of the small wind turbine varies significantly with wind speed, but it would be designed such a way that maximized efficiencies are achieved at the wind speed where the maximum energy is available.

6) Blade design and performance testing of a small wind turbine rotor for low wind speed applications

Authors : R. K. Singh and M. R. Ahmed

Year : 2013

Small wind turbines operating at low wind speeds regularly face the problem of poor performance due to laminar separation and laminar separation bubbles on the blades. This is due to the low Reynolds number (Re) resulting from low wind speeds and small rotor size. The use of specially designed low Re airfoils permits start up at lower wind speeds, increasing the startup torque and thus improving the overall performance of the turbine. A new airfoil was designed and the performance of a 2-bladed rotor designed for low Re application fitted to an Air-X marine 400 W wind turbine was tested at a wind speed range of 3–6 m/s. The low Re rotor incorporated taper and twist to the low Re AF300 airfoil section. The pitch of the blades was varied over a range of 15° , 18° and 20° to study the performance and the startup wind speed. It was found that the turbine performed best at 18° pitch angle. On an average, the wind turbine yielded a power coefficient (C_p) of 0.255 at a height of 8.22 m at a wind speed of 6 m/s at 18° pitch angle. Maximum C_p based on 10 s data at the freestream velocity of 6 m/s was 0.291. The cut-in wind speed based on 10 s averaged data at the optimum pitch angle was 3.24 m/s whereas the instantaneous cut-in wind speed was 2.34 m/s. In comparison with the baseline 3-bladed rotor, the new 2-bladed rotor produced more electrical power at the same freestream velocity.

7) A novel low Reynolds number airfoil design for small horizontal axis wind turbines

Authors : H. Shah, S. Mathew, and Chee Ming Lim

Year : 2014

In order to improve the performance of small horizontal axis wind turbines at low wind speed, this study designs a novel airfoil section with an optimum transition ramp through multipoint inverse design method. A viscous analysis code is used to close the design loop. Further, Shear stress transport-transition model in ANSYS-Fluent is employed with

modified constants to analyze the flow and aerodynamic performance of the airfoil at Reynolds numbers 60 000, 100 000, 200 000, 300 000 and 500 000. Next we consider the designing of a 2m rotor from the new airfoil section using an evolutionary algorithm for optimization. The power coefficient and self starting of a small 2m wind turbine is improved significantly with the chosen generator resistive torque of 0.5Nm, the new airfoil section and the optimization technique for finding the optimal values of the parameters.

This paper describes the design of a new airfoil profile “UBD6166” that can improve the self starting and power capture of small HAWTs at low wind speed. This was accomplished by designing the airfoil with optimum transition ramp via multipoint inverse design method and the optimization technique for the rotor design. The goal of improving the aerodynamic performance by controlling the LSB via optimum transition ramp was accomplished. Airfoil UBD6166, owing largely to the higher aerodynamic performance with moderate thickness, shall benefit small HAWTs by reducing the rotor inertia and increasing the starting torque. From two dimensional optimization, fast starting blades were found to have increased chord at hub region along with slight increase in twist whereas the tip regions mostly remain the same. Moreover, the two dimensional optimization is based upon the tradeoff between maximizing one objective function while reducing the other; the desired optimal design can be chosen from the Pareto front.

8) An airfoil parameterization method for the representation and optimization of wind turbine special airfoil

Authors : Y. Liu, C. Yang, and X. Song

Year : 2015

A new airfoil shape parameterization method is developed, which extended the Bezier curve to the generalized form with adjustable shape parameters. The local control parameters at airfoil leading and trailing edge regions are enhanced, where have significant effect on the aerodynamic performance of wind turbine. The results show this improved parameterization method has advantages in the fitting characteristics of

geometry shape and aerodynamic performance comparing with other three common airfoil parameterization methods. The new parameterization method is then applied to airfoil shape optimization for wind turbine using Genetic Algorithm (GA), and the wind turbine special airfoil, DU93-W-210, is optimized to achieve the favorable Cl/Cd at specified flow conditions. The aerodynamic characteristic of the optimum airfoil is obtained by solving the RANS equations in computational fluid dynamics (CFD) method, and the optimization convergence curves show that the new parameterization method has good convergence rate in less number of generations comparing with other methods. It is concluded that the new method not only has well controllability and completeness in airfoil shape representation and provides more flexibility in expressing the airfoil geometry shape, but also is capable to find efficient and optimal wind turbine airfoil. Additionally, it is shown that a suitable parameterization method is helpful for improving the convergence rate of the optimization algorithm.

9) A solution-based stall delay model for horizontal-axis wind turbines

Authors : J. L. Dowler and S. Schmitz

Year : 2014

This work proposes a new solution-based stall delay model to predict rotational effects on horizontal-axis wind turbines. In contrast to conventional stall delay models that correct sectional airfoil data prior to the solution to account for three-dimensional and rotational effects, a novel approach is proposed that corrects sectional airfoil data during a blade element momentum solution algorithm by investigating solution-dependent parameters such as the spanwise circulation distribution and the local flow velocity acting at a section of blade. An iterative process is employed that successively modifies sectional lift and drag data until the blade circulation distribution is converged. Results obtained with the solution-based stall delay model show consistent good agreement with measured data along the National Renewable Energy Laboratory Phase VI and Model Experiments in Controlled Conditions rotor blades at low and high wind speeds.

10) A new direct design method of wind turbine airfoils and wind tunnel experiment

Authors : J. Chen, Q. Wang, S. Zhang, P. Eecen, and F. Grasso

Year : 2015

A general parametric representation function (Shape Function) for wind turbine – dedicated airfoils based on Taylor high-order polynomial series is presented for the first time. The design space and shape control function of the airfoil have been studied. The objective of the high-performance WT (Wind Turbine) airfoils was to maximise the lift/drag ratio at the design angle of attack both in free and fixed transitions. The optimised mathematical model of the airfoils is built combining genetic algorithm and the flow solver RFOIL. The new airfoil family (ranging in thickness from 15 to 20%) with target characteristics were designed for variable-speed operation with pitch control of large megawatt-sized rotors. Wind tunnel experiments for the WT180 airfoil were carried out for both clean and rough conditions. The experimental results for the lift and drag coefficients agree well with the RFOIL predictions. The testing verified the high lift/drag ratio and the high maximum lift coefficient for the WT180 airfoil. The results indicate that this novel design method is feasible to optimise wind turbine airfoils.

CHAPTER – 3

AIM AND SCOPE OF THIS PROJECT

3.1) Objective and scope:

Aim:

The objective of the present study is to evaluate the influence of blade shape modification by using various NACA series airfoils on the enhancement of wind speeds and aerodynamic efficiency. In the present study only the wind turbine blade is considered without the wind turbine for simulation purpose.

Scope:

The scope of the present study includes analysis of the variation of the free stream velocity and aerodynamic efficiency determines an efficient shape modification by using four different NACA series airfoils set up of the wind turbine by establishing 3D CFD models of wind turbine blade with the varying as follows

Blade length as 60m, Rotor diameter as 140m and NACA 4412 airfoil blade which is initially fixed in the Reference model for the following four cases:

NACA 0015 : By applying NACA 0015 airfoil to the blade and maintaining all the other dimensions of the model constant.

NACA 2412 : By applying NACA 2412 airfoil to the blade and maintaining all the other dimensions of the model constant

NACA 63215 : By applying NACA 63215 airfoil to the blade and maintaining all the other dimensions of the model constant

3.2) Methodology:

This report deals with the study of flow around a wind turbine blade. Computational fluid analysis is carried out to determine the velocity variation and aerodynamic efficiency. Computational fluid analysis is carried using Ansys Fluent software

CATIA V5 is used to build the 3d model and and Fluent is used to mesh it and carry out the velocity, pressure and aerodynamic forces changes along the blade. This total analysis is known as Computational fluid dynamic analysis.

3.3) Overview of this Report:

A comprehensive study of flow around wind turbine blade model is done using 3D CFD simulation. The introduction is given in Chapter 1, In Chapter 2, the literature studies are compiled. The aims and objectives are given in Chapter 3. In Chapter 4, the theories of computational fluid dynamics are explained. 3D Modeling of wind turbine blade with modification is described in Chapter 5. Meshing of the blade and creation of computational domain using Ansys Fluent software are presented in Chapter 6, and three dimensional numerical simulation procedures using fluent are given in the Chapter 7. The velocity and aerodynamic forces are changes and efficient modification is considered and the results are given in Chapter 8. The windtunnel experiment and calculations are presented in Chapter 9. In Chapter 10, power calculations are presented. The conclusions are summarized in chapter 11

CHAPTER - 4

COMPUTATIONAL FLUID DYNAMICS

4.1) GENERAL

With the rapid advances in computing power, application of Computational Fluid Dynamics (CFD) has become practice for various wind flow related studies. Even though Direct Numerical Simulation (DNS) is limited to super computer user for studying and generating benchmark solutions for low Reynolds numbers of up to few thousands. Various turbulence models like Large Eddy Simulation (LES) and Reynolds Averaged Navier-Stokes Simulation (RANS) turbulence models are being used for numerically simulating turbulent flow around bluff bodies. At high Reynolds numbers, time varying flow quantity like velocity and pressure consists of a time averaged mean component, a periodic component and a random turbulent fluctuating component. The periodic component is usually due to the vortex shedding in the wake of a bluff body.

LES which is based on spatial averaging resolves not only the periodic component but also random turbulent fluctuating component with length scales more than the cell size, while modeling the turbulence with length scales less than the cell size. However, Unsteady RANS (URANS) which is based on ensemble averaging resolves only the periodic component while modeling the turbulence component. Even though LES predictions are more accurate than RANS predictions, RANS turbulence models especially two-equation turbulence models like $k-\epsilon$ have been very popular because they require less computer memory and CPU time than LES.

CHAPTER - 5

3D MODELING OF BLADES

5.1) 3D Modeling of Horizontal axis wind turbine blades using Symmetric, 4 Digit and 5 Digit Airfoils :

The coordinates are provided for the development of the airfoil wind turbine blade model from airfoil tools website.

- Symmetric airfoils - NACA 0015
- 4 digit airfoils - NACA 2412, NACA 4415 (Reference blade)
- 5 digit airfoils - NACA 63215

NACA 0015, NACA 2412, NACA 4415 and NACA 4415 airfoil coordinates and airfoil shapes are generated in CATIA by using view macros feature in Microsoft excel according to the geometry of the blade is created to complete the model.

The following parameters are used to design the horizontal axis wind turbine

Blade length : 60 meters

Rotor diameter : 20 meters

Rotor radius : 70 meters

Requirements:

Software Used : CATIA V5

Processor : Intel Core i3

RAM : 4GB

Operating System : Windows 10

Features Used:

MULTISECTION SOLIDS : This feature is used to connect the various cross sections

(modified airfoil shapes) of the blade in longitudinal direction

ROTATE : This option is used to set the wind turbine blade angle

CIRCULAR PATTERN : This feature is used to create (pattern) 3 similar blades by selecting

initial blade

5.2) 3D Modeling of NACA 4415 Airfoil blade (Reference blade) :

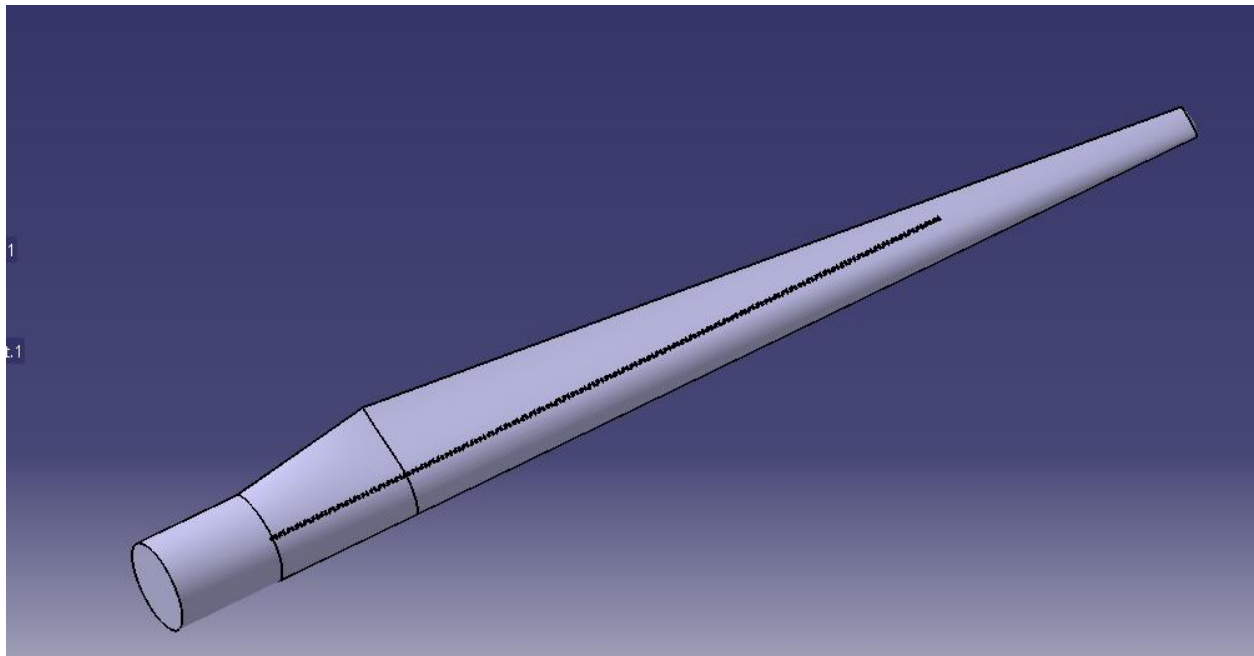


Fig.5.1 : 3D CAD model of NACA 4415 airfoil blade (Reference model)

- Initially NACA 4415 coordinates were collected from www.airfoiltool.com and imported into CATIA V5 software by using view macros option in Microsoft excel.
- Modifying the airfoil chord length and width to create different cross section of blade in longitudinal direction by using transform features and datum planes.
- Creating the full length blade by connecting the cross section airfoils by using Mutisection solid feature

5.3) 3D Modeling of NACA 0015 Airfoil blade :

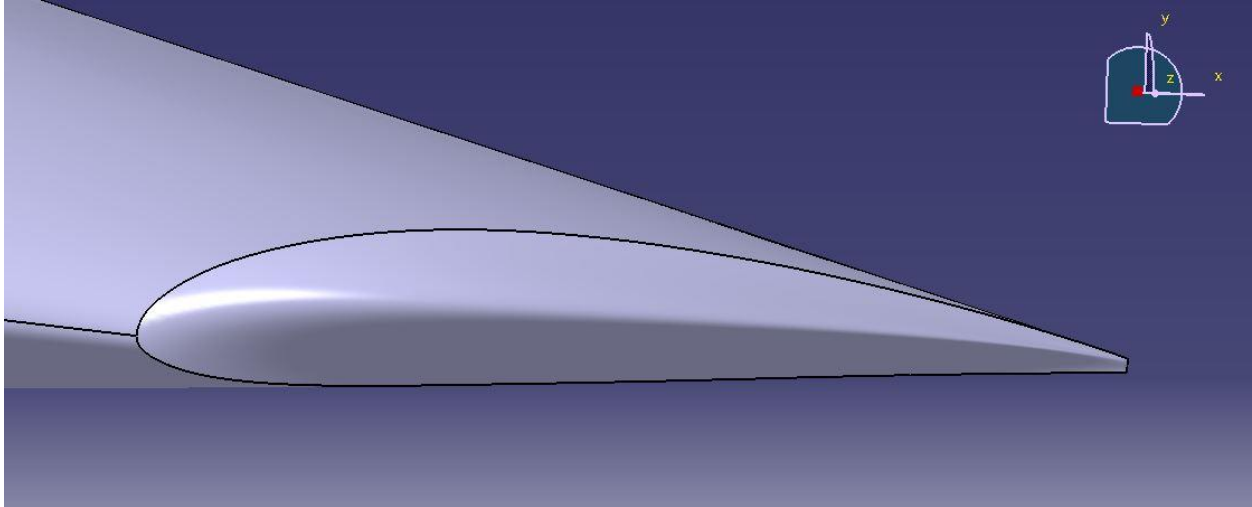


Fig.5.2 : 3D CAD model of NACA 0015 airfoil blade

- Initially NACA 0015 coordinates were collected from www.airfoiltool.com and imported into CATIA V5 software by using view macros option in Microsoft excel.
- Modifying the airfoil chord length and width to create different cross section of blade in longitudinal direction by using transform features and datum planes.
- Creating the full length blade by connecting the cross section airfoils by using Mutisection solid feature

5.4) 3D Modeling of NACA 2412 airfoil blade :

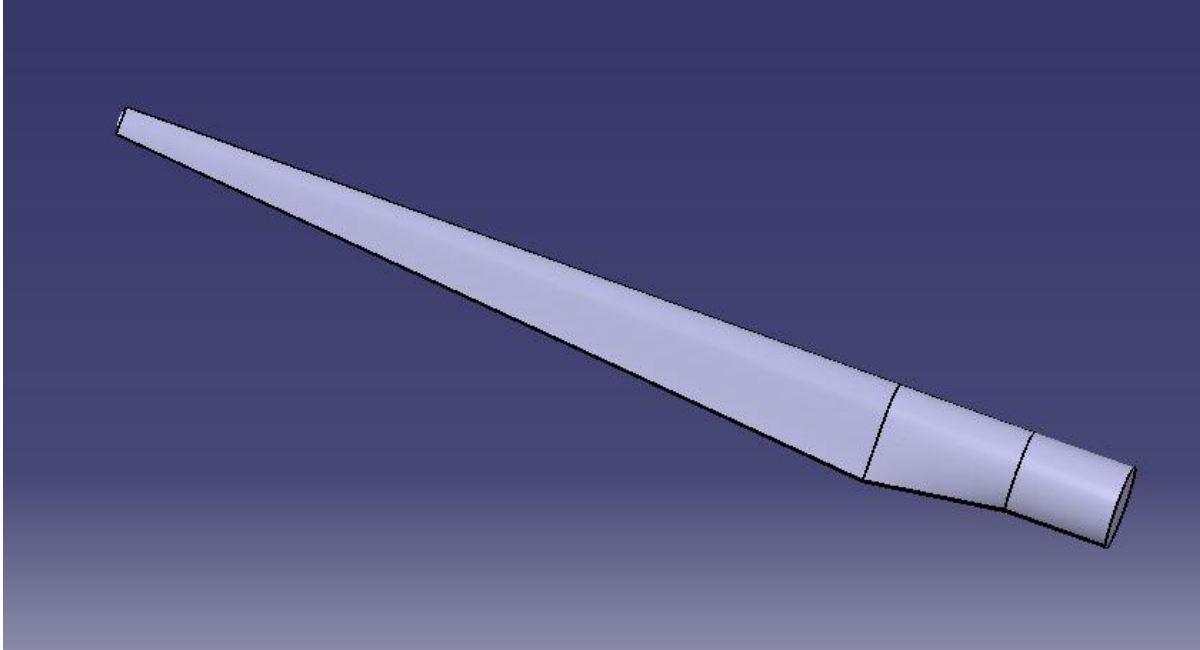


Fig.5.3 : 3D CAD model of NACA 2412 airfoil blade

- Initially NACA 2412 coordinates were collected from www.airfoiltool.com and imported into CATIA V5 software by using view macros option in Microsoft excel.
- Modifying the airfoil chord length and width to create different cross section of blade in longitudinal direction by using transform features and datum planes.
- Creating the full length blade by connecting the cross section airfoils by using Mutisection solid feature

5.5) 3D Modeling of NACA 63215 Airfoil blade :

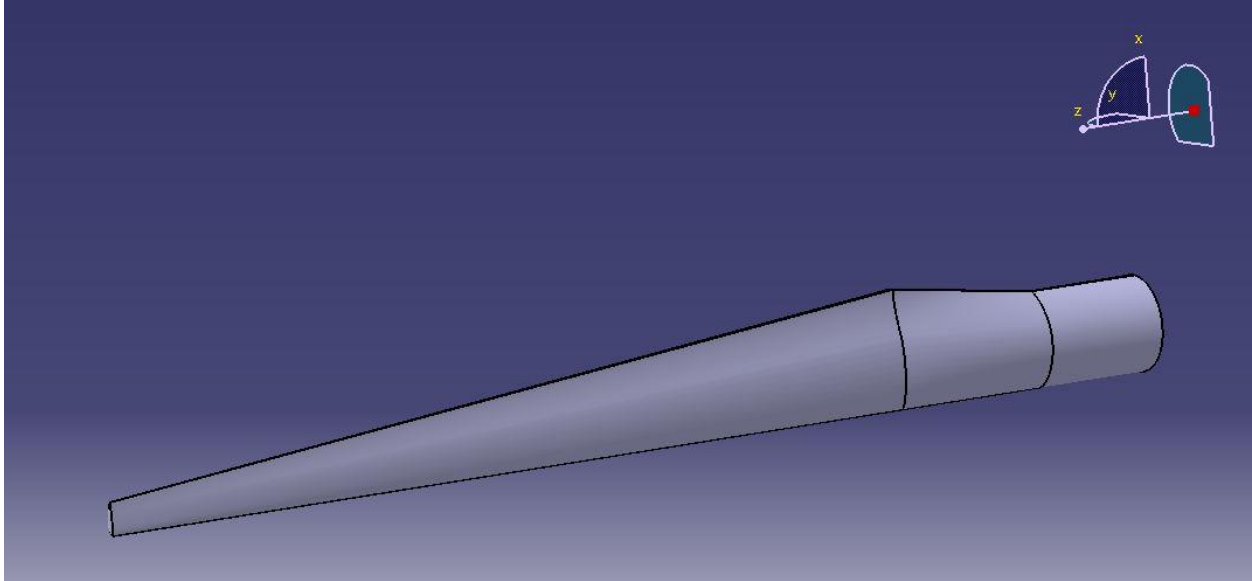


Fig.5.4 : 3D CAD model of NACA 63215 airfoil blade

- Initially NACA 63215 coordinates were collected from www.airfoiltool.com and imported into CATIA V5 software by using view macros option in Microsoft excel.
- Modifying the airfoil chord length and width to create different cross section of blade in longitudinal direction by using transform features and datum planes.
- Creating the full length blade by connecting the cross section airfoils by using Mutisection solid feature

5.6) Creating 3 Blades :

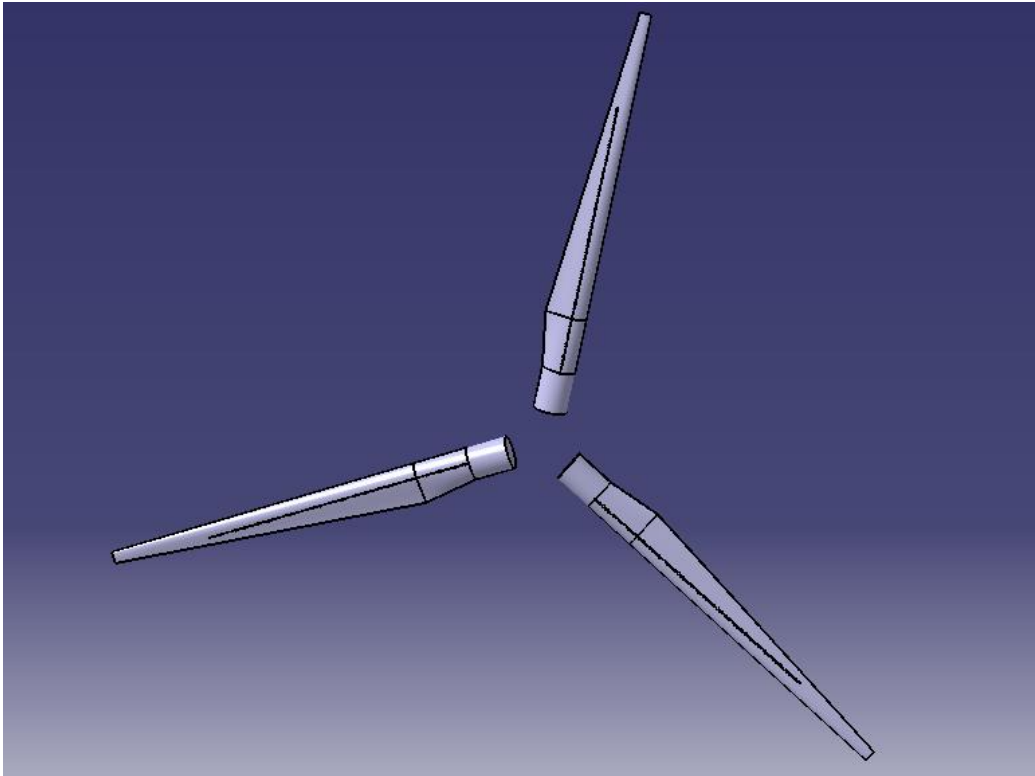


Fig.5.5 : Creating 3 blades using circular pattern

In the help of Circular pattern feature in CATIA V5 software to create 2 similar blades by selecting the finished blade. Finally, 3 blades were created.

CHAPTER - 6

COMPUTATIONAL DOMAIN FOR SIMULATION

6.1) General:

The pre-processing of importing the 3D model of blade and meshing of model has been carried out using ANSYS Fluent code in ANSYS Workbench software. ANSYS Fluent is designed to help analysts and designers build and meshing the models for use in Computational Fluid Dynamics (CFD) and other scientific applications. ANSYS Fluent receives user input by means of its Graphical User Interface (GUI).

In this study, the imported model include the boundary which are modeled by using this software. Meshing is done after the modeling processes.

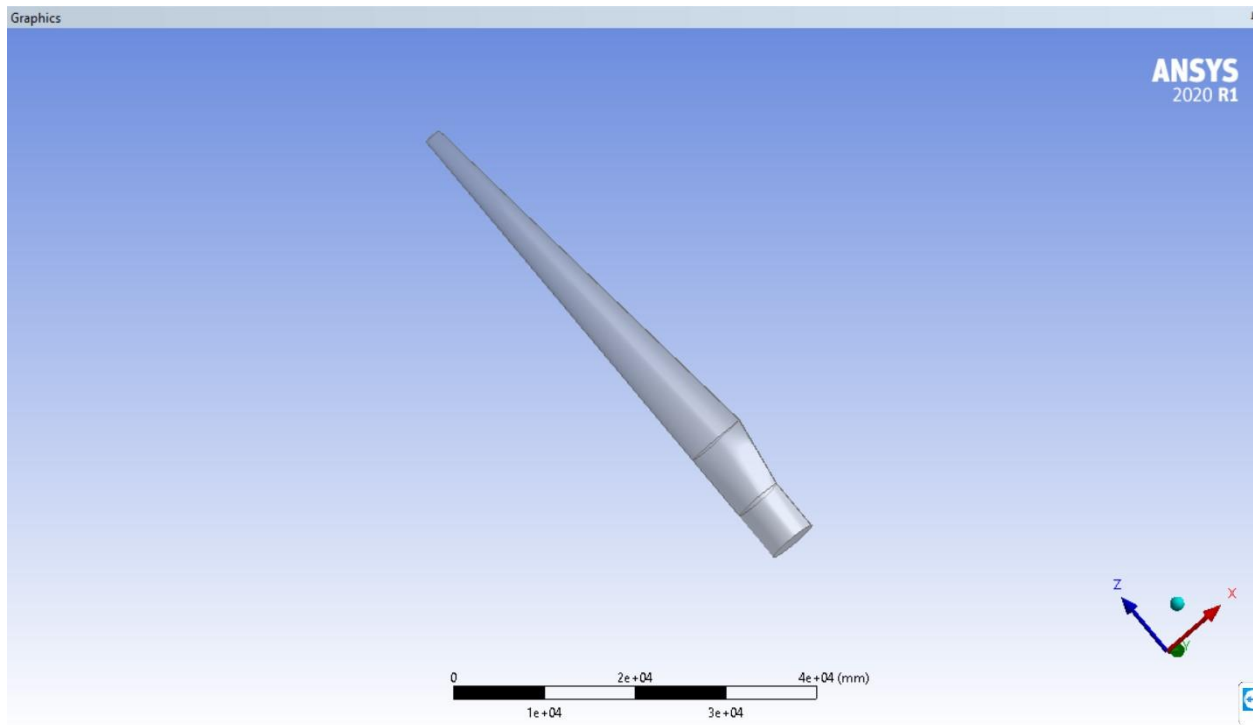


Fig.6.1 : Importing 3D STEP model into ANSYS Fluent

6.2) Boundary creation :

3D boundary with blade model is depicted in this Figure below.

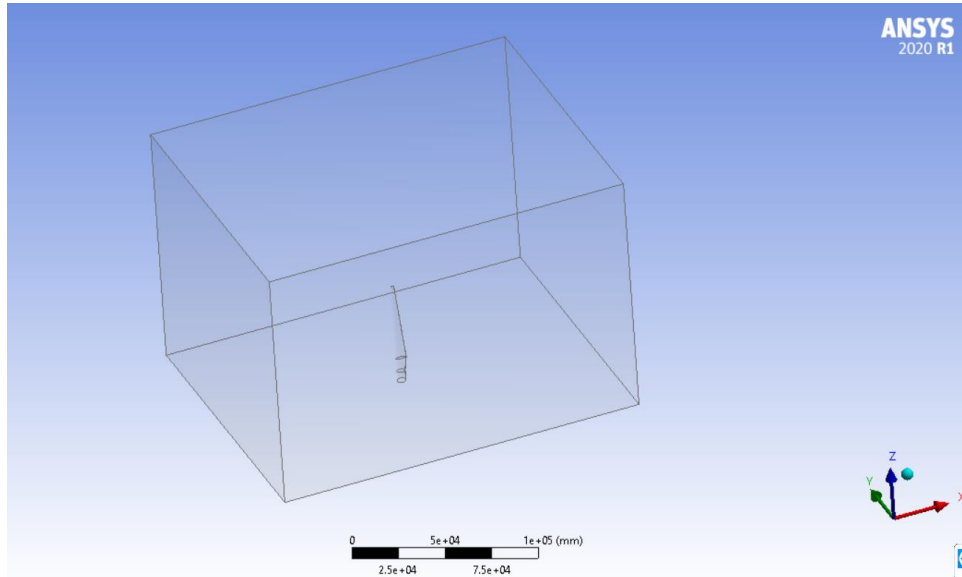


Fig.6.2 : 3D Turbine blade model in 3D boundary

The geometrical specifications of the boundary are

Length of boundary : 200000mm

Width of boundary : 160000mm

Height of boundary :80000mm

Details View	
[-] Details of Enclosure1	
Enclosure	Enclosure1
Shape	Box
Number of Planes	0
Cushion	Non-Uniform
☐ FD1, Cushion +X value (>0)	1e+05 mm
☐ FD2, Cushion +Y value (>0)	80000 mm
☐ FD3, Cushion +Z value (>0)	80000 mm
☐ FD4, Cushion -X value (>0)	1e+05 mm
☐ FD5, Cushion -Y value (>0)	80000 mm
☐ FD6, Cushion -Z value (>0)	0.0001 mm

Fig.6.3 : 3D Boundary specifications

6.3) Meshing :

Meshing is the process of dividing a continuous unit into smaller individual units.

(a) Face Mesh :

For the carrying out mesh, faces were created by splitting the entire domain into individual parts. Then, the faces were later joined as the meshing for each face matches. Mesh far away from the model was made coarse and near the model were done finer. This will help in reducing the running time of the flow analysis.

Meshing for the domain was done with the following features.

Type : Linear

Element : Quadrilateral

Element size : 0.5m

Meshing near the model was done with the following features :

Type : Linear

Element : Quadrilateral

Element size : 0.025m

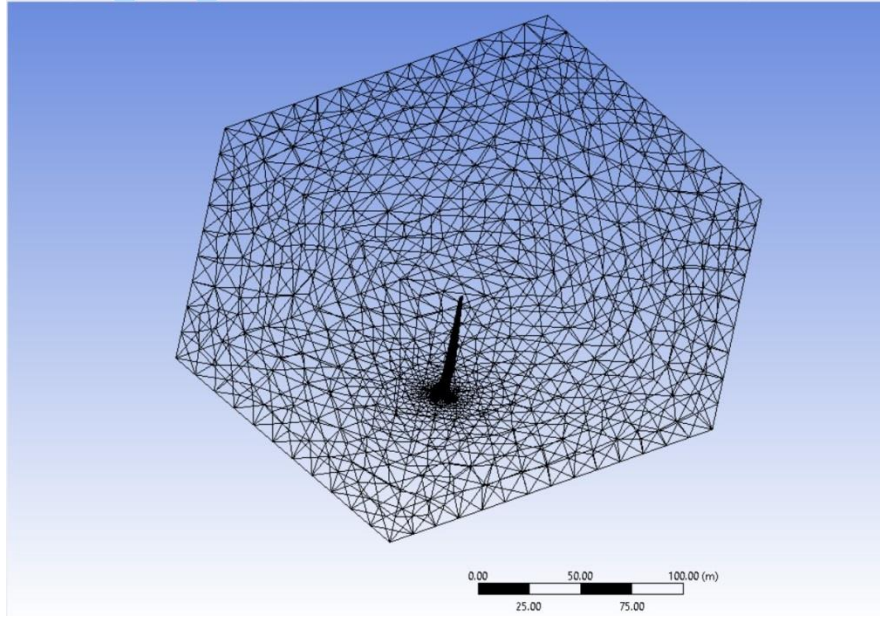


Fig.6.4 : Coarse mesh in Boundary and Fine mesh in turbine blade

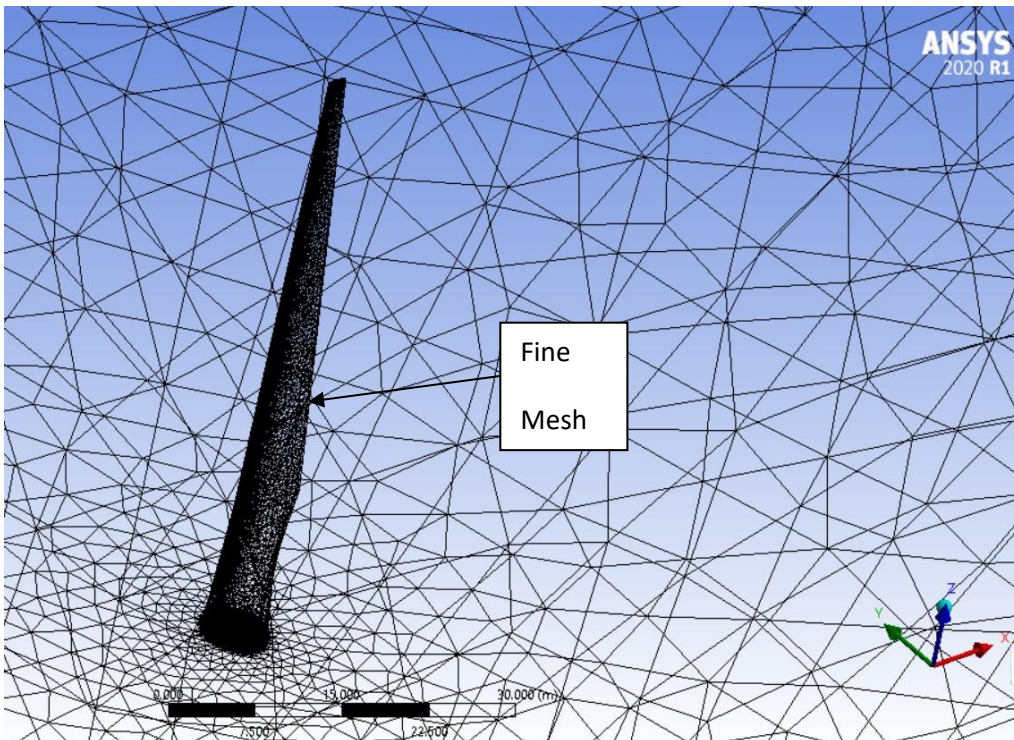


Fig.6.5 : Enlarged image of fine mesh on turbine blade

6.4) CFD Boundary :

The boundary plane set keeping in mind the axis-symmetric feature. The five boundary conditions set in this modeling are as follows :

- Velocity Inlet Boundary Conditions
- Pressure Outlet Boundary Conditions
- Wall Boundary Conditions

(a) Velocity Inlet Boundary Conditions:

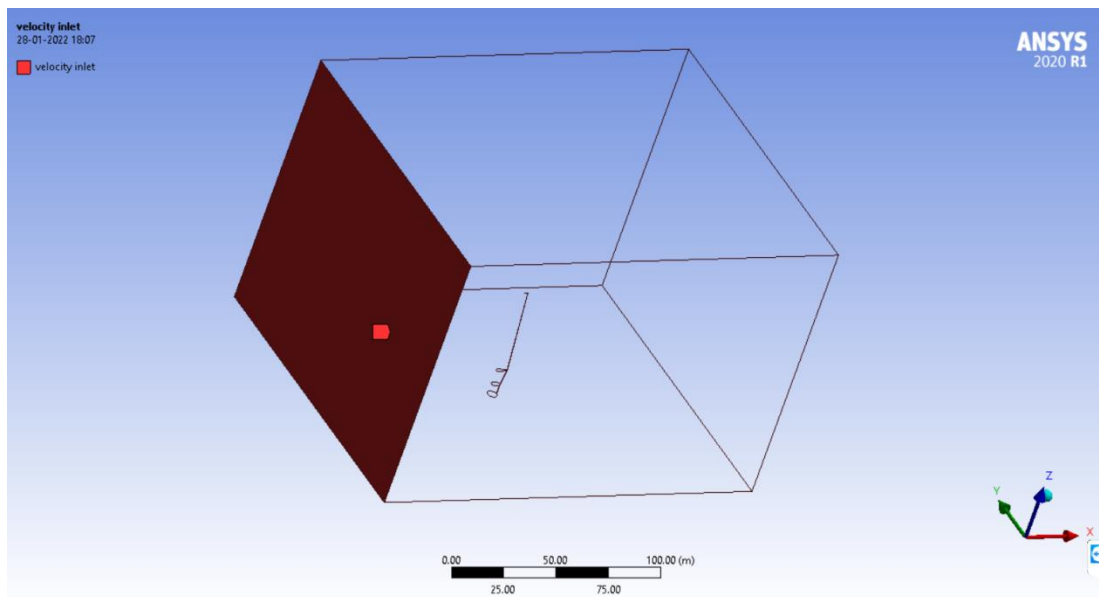


Fig.6.6 : Velocity inlet of the computational domain

The domain inlet wall is set as velocity inlet boundary. Velocity inlet boundary conditions are used to define the flow velocity, along with the relevant scalar properties of the flow, flow inlets. The total (or Stagnation) properties of the flow are not fixed, so they will rise to whatever value is necessary to provide the prescribed velocity distribution.

(b) Pressure Outlet Boundary Conditions :

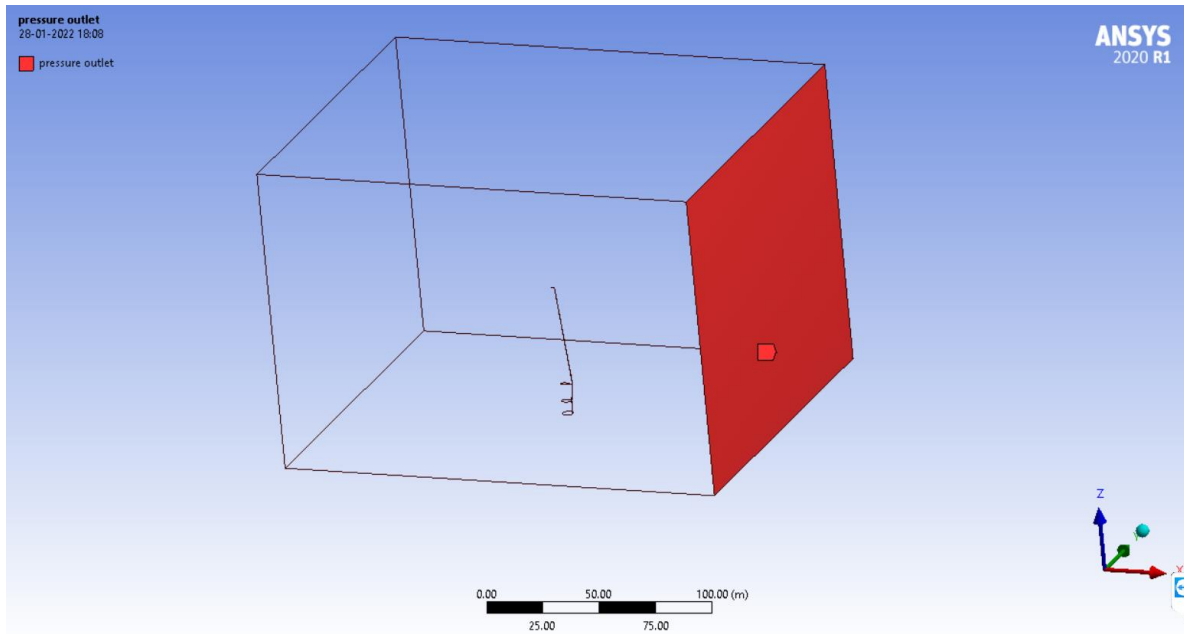


Fig.6.7 : Pressure outlet of the computational domain

The domain outlet wall is set as a Boundary Pressure outlet boundary. Pressure outlet boundary conditions require the specification of a static(gauge) pressure at the outlet boundary. The value of static pressure specified is used only while the flow is subsonic.

(c) Wall Boundary Conditions :

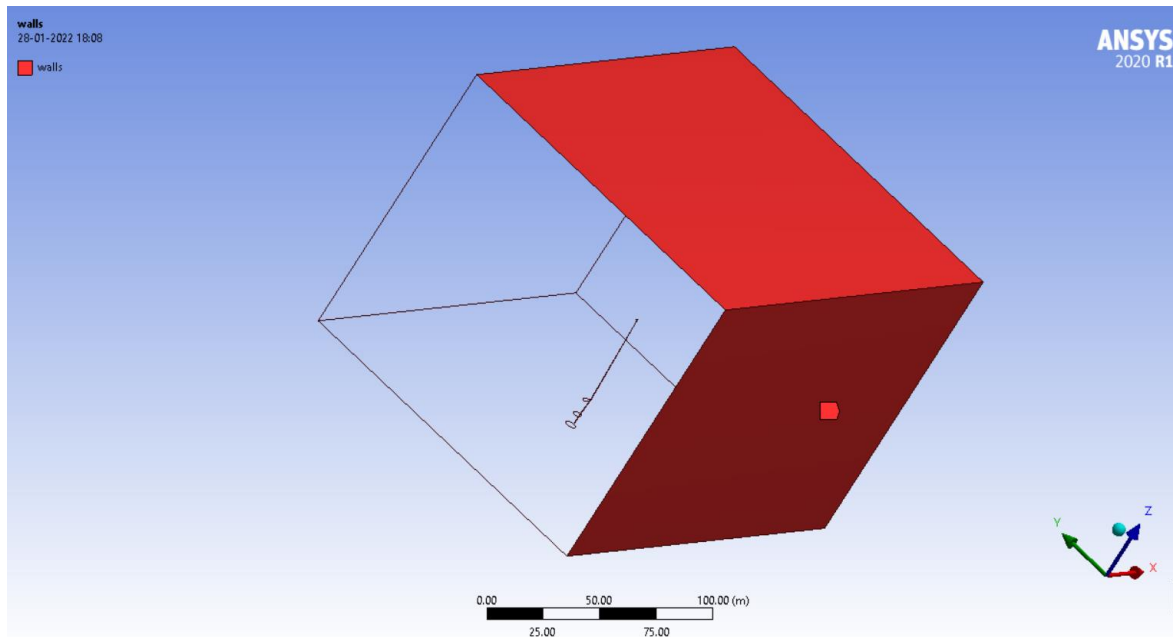


Fig.6.8 : Walls of the computational domain

The model wall is set as wall boundary. Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary conditions is enforced at walls by default, but it can be specified that a tangential velocity component in terms of translational or rotational motion of the wall boundary, or model a slip wall by specifying shear. The shear stress and the heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

CHAPTER -7

3D NUMERICAL SIMULATION IN ANSYS FLUENT :

7.1) OVERVIEW OF FLUENT :

Numerical simulation has been carried using fluent software. This software is used for modeling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, including the ability to solve the flow problems using unstructured meshes that can be generated about complex geometries with the relative ease. It is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible.

All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The reference procedural steps for solving a problem in FLUENT include :

- 1) Define the modeling goals
- 2) Create the model geometry and grid
- 3) Set up the solver and physical models
- 4) Compute and monitor the solution
- 5) Examine and save the results
- 6) Consider revisions to the numerical or physical model parameters, if necessary

7.2) ANALYSIS :

The analysis has been carried in fluent by importing the meshed file saved in gambit.

The steps that have followed are given below which include all the conditions and the boundary values for the problem statement.

Grid Check

```
Domain Extents:
  x-coordinate: min (m) = 0.000000e+000, max (m) = 2.529700e+001
  y-coordinate: min (m) = 0.000000e+000, max (m) = 9.761000e+000
Volume statistics:
  minimum volume (m3): 2.198033e-004
  maximum volume (m3): 4.570068e-001
  total volume (m3): 2.468763e+002
Face area statistics:
  minimum face area (m2): 1.891260e-002
  maximum face area (m2): 6.842351e-001
Checking number of nodes per cell.
Checking number of faces per cell.
Checking thread pointers.
Checking number of cells per face.
Checking face cells.
Checking bridge faces.
Checking right-handed cells.
Checking face handedness.
Checking face node order.
Checking element type consistency.
Checking boundary types:
Checking face pairs.
Checking periodic boundaries.
Checking node count.
Checking nosolve cell count.
Checking nosolve face count.
Checking face children.
Checking cell children.
Checking storage.
Done.
```

Fig.7.1 : Grid check in Fluent software

Initially grids of computational domain (or) CFD boundary is checked when model is imported into ANSYS Fluent software.

7.3) Solver and Material Selection and Operating Condition Defining :

(a) Solver :

The solver is defined first .Energy equation is taken into consideration.The viscous medium is also taken.First the analysis is carried using turbulent flow with k- ϵ model,specifically realizable.

The following conditions are taken in solver :

Solver	Pressure Based
Formulation	Implicit
Time	Steady
Space	3D
Velocity Formulation	Absolute
Porous Formulation	Physical Velocity
Gradient options	Green-Gauss Cell
Viscous Model	k- ϵ

Table 1 : Solver specifications

(b) Selection of Material and Operating Conditions

Material selected is air. The Properties of the fluid are as follows.

The following values are entered in Viscous and Material Selection.

Material Type	Fluid
Fluid	Air
Density	1.225 kg/m ³
Viscosity	1.7894 x 10 ⁻⁵ kg/ms
Pressure	101325 Pa

Table 2 : Material Properties

7.4) Boundary conditions :

The boundary conditions selected are as follows :

- Velocity Inlet Boundary Conditions for the domain inlet.
- Pressure Outlet Boundary Conditions for the domain outlet.
- Wall Boundary Conditions for the model wall.
- Symmetry Boundary Conditions for the upper edge of the domain.
- Axis Boundary Conditions for the bottom edge of the domain.

The velocity inlet conditions set are as tabulated below.

Inlet	Velocity Inlet ,Velocity =10 m/s
Specific method	Intensity & Viscosity ratio
Turbulent Intensity	0.1
Viscosity ratio	10
Outlet	Pressure Outlet
Specific method	Intensity & Viscosity ratio
Turbulent Intensity	0.1
Viscosity ratio	10

Table 3 : Velocity inlet conditions & Pressure Outlet conditions

7.5) Residual Monitoring :

The computation is done from the inlet.

Residual Monitoring is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity - 0.000001

X-Velocity - 0.000001

Y-Velocity - 0.000001

k- ϵ - 0.000001

Later the reference values are entered and the number of iterations is then set up and iterations starts. The iteration continues till the convergence is reached.

Report Reference Values :

In reporting paramaters, the reference values are given in the following

7.6) Report Reference Values

In reporting parameters, the reference values are given as following

Density kg/m ³	1.225
Enthalpy, J/kg	0
Length, m	1
Pressure, Pa	0
Temperature, K	288.16
Velocity, m/s	10
Viscosity, kg/ms	0.17894×10^{-4}
Ratio of specific heat	1.4
Reference Zone	Fluid

Table 4 : Reference Values

In the reference values, the area of the model differs in each case of dimension changes.

7.7) Iteration :

The iteration process is started after setting the reference values. The following values are given as the requisites in the iteration.

Time Step size	0.001
No. of Time Steps	1000
Reporting Interval	10
Maximum Iteration per time step	20

Table 5 : Iteration Values

Based on all the above values are tabulated, the analysis of the flow was done in fluent. Results can be seen as plot in the fluent software.

CHAPTER 8

RESULTS AND DISCUSSION

8.1) Result sets

There are three sets of results for the analysis of the problem. The results were obtained for the analysis of the variation of free stream velocity and for the determination of an efficient NACA airfoil set up of the wind turbine using 3D CFD models of wind turbine blades with varying the following shapes as follows

The parameter variations done are as follows.

NACA 4415 : This is a reference model which has a airfoil of NACA 4415 and blade length is 60m

NACA 0015 : By applying NACA 0015 airfoil to the blade and maintaining all the other dimensions of the model constant.

NACA 2412 : By applying NACA 2412 airfoil to the blade and maintaining all the other dimensions of the model constant

NACA 63215 : By applying NACA 63215 airfoil to the blade and maintaining all the other dimensions of the model constant

8.2) Reference (NACA 4415 Airfoil blade) results :

The observation obtained for the Reference model, that is for the Normal blade model which has a airfoil of NACA 4415 and blade height of 60m is described below.

a)Aerodynamic forces observation :

```

Forces - Direction Vector (1 0 0)
Zone      Forces (n)
Pressure  Viscous      Total
wall-slot_holes_with_1.s_and_t.s -3.3219586  0.0068045112  -3.315154
-----
Net      -3.3219586  0.0068045112  -3.315154
Coefficients
Pressure  Viscous      Total
-0.054236058  0.00011109406  -0.054124964
-----
-0.054236058  0.00011109406  -0.054124964

```

Fig.8.1 : Print console of Lift force and Coefficient of lift values of NACA 4415 airfoil blade

```

Forces - Direction Vector (0 1 0)
Zone      Forces (n)
Pressure  Viscous      Total
wall-slot_holes_with_1.s_and_t.s 0.60883819  0.099003483  0.70784167
-----
Net      0.60883819  0.099003483  0.70784167
Coefficients
Pressure  Viscous      Total
0.0099402154  0.0016163834  0.011556599
-----
0.0099402154  0.0016163834  0.011556599

```

Fig.8.2 : Print console of Drag force and Coefficient of drag values of NACA 4415 airfoil blade

Lift Force = 3.315154 N

Drag Force = 0.70784167 N

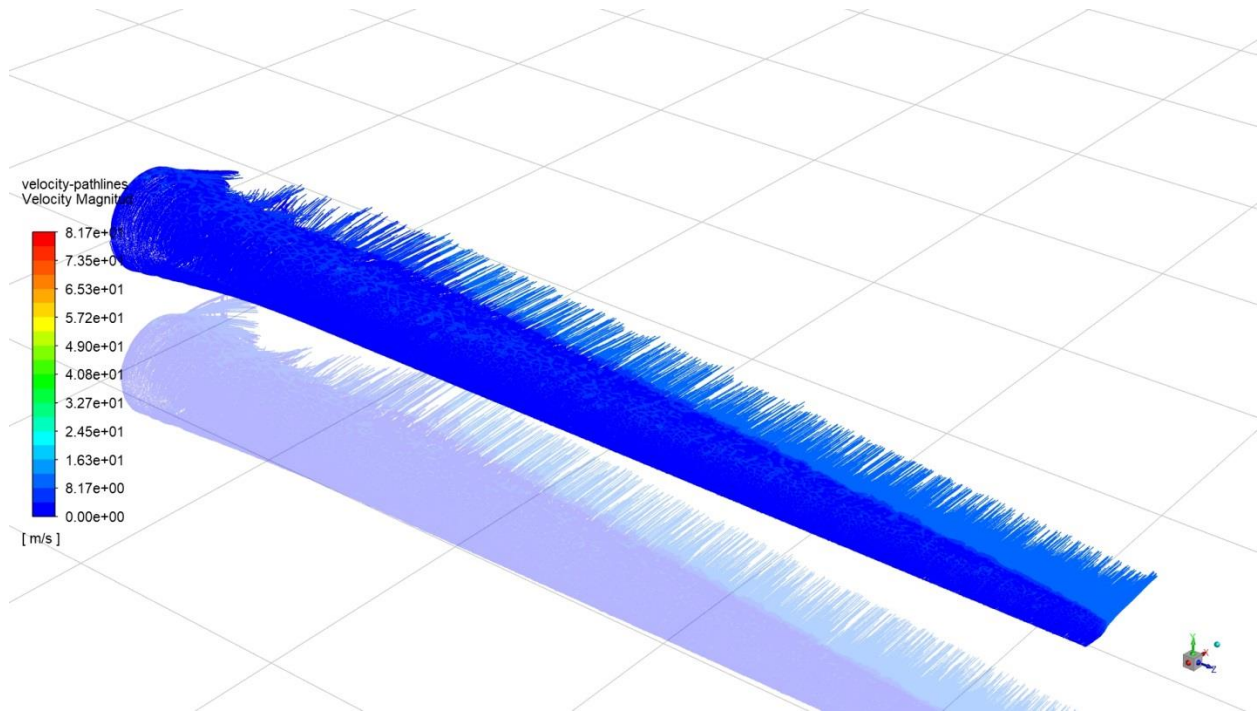
Coefficient of Lift = 0.054124964

Coefficient of Drag = 0.011556599

Aerodynamic efficiency (η) = Lift Force / Drag force = 4.683

Cl/Cd = 4.683

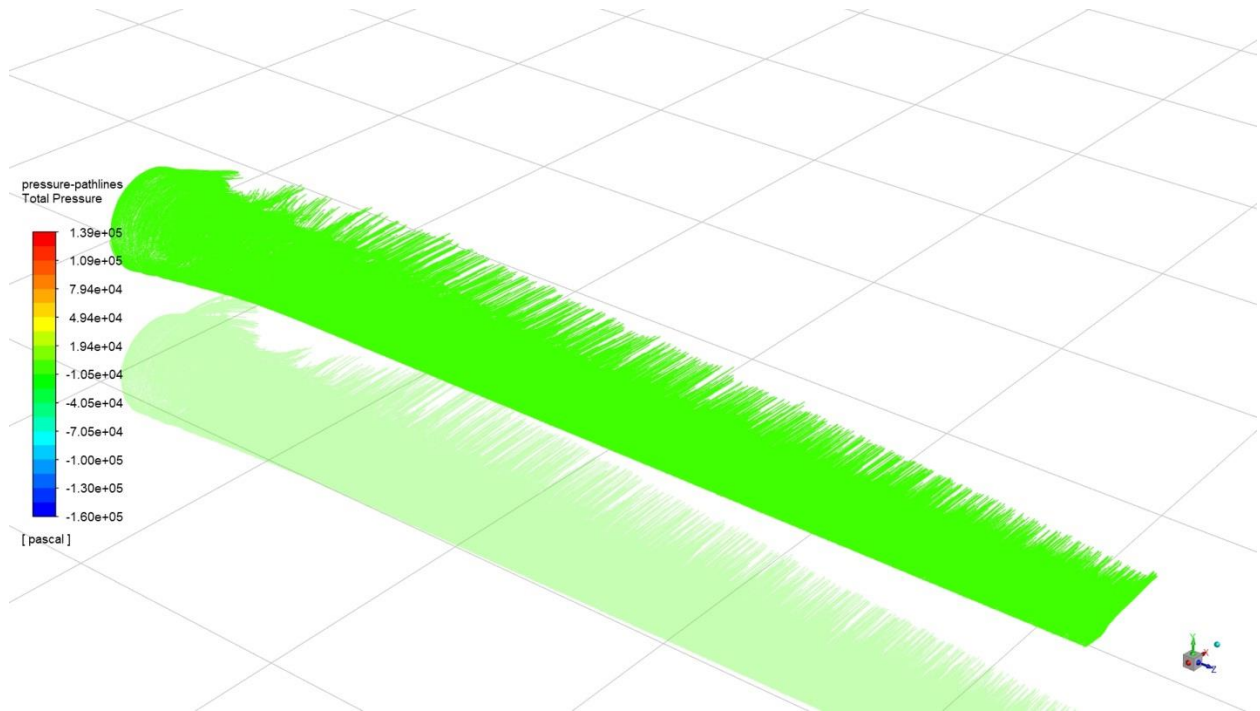
b)Velocity observation :



**Fig.8.3 : 3D Contour of Velocity pathlines of Reference blade model
(NACA 4415 airfoil blade)**

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 16.30 m/s. The velocity is maximum at the trailing edge portion. The velocity behind the leading edge limit is 0 – 8.17 m/s due to the vortices formation owing to the flow separation.

c)Pressure observation :



**Fig.8.4 : 3D Contour of pressure pathlines of Reference blade model
(NACA 4415 airfoil blade)**

The maximum pressure reached in this case is observed to be 139 Pa. The pressure is maximum at both leading and trailing edge portion. The pressure behind the leading edge limit is -16 Pa to -13 Pa due to the vortices formation owing to the flow separation.

d)Turbulence intensity observation :

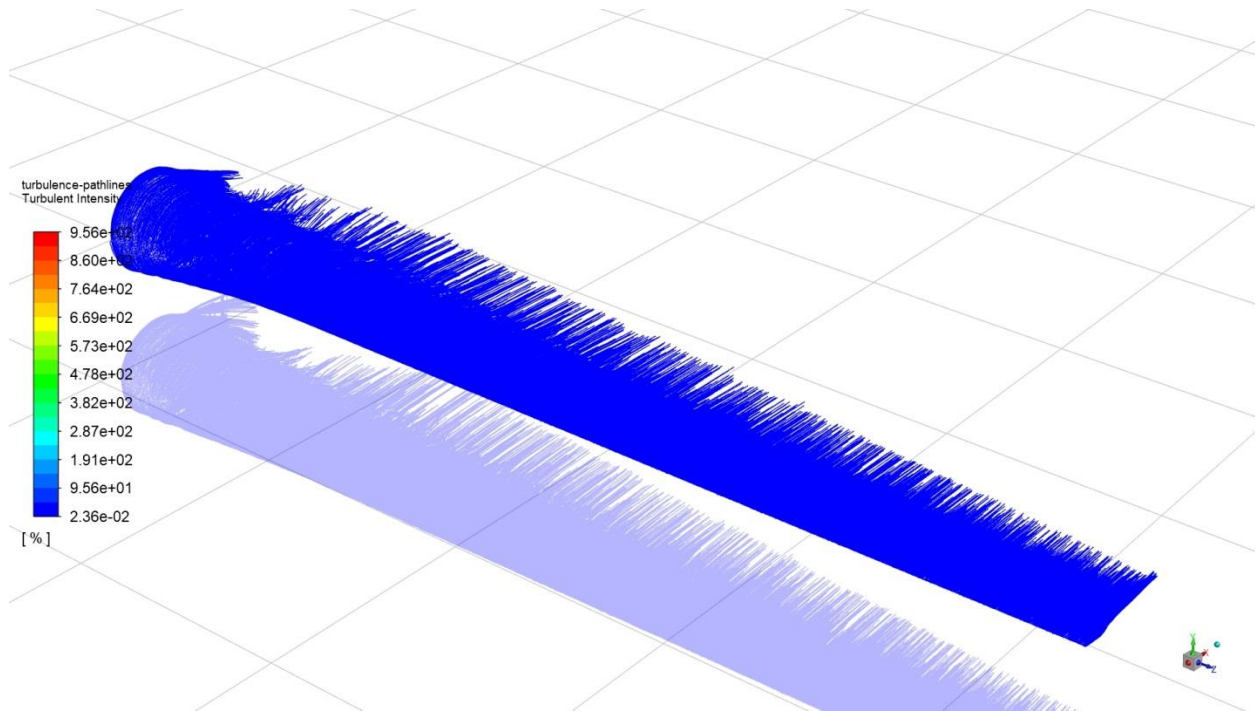


Fig.8.5 : 3D Contour of Turbulence intensity pathlines of Reference blade model (NACA 4415)

In Fig.8.5, visualization of the turbulence intensity pathlines that shows the changes in turbulence intensity level at different areas in the flow field using different colors.

8.3) NACA 0015 Airfoil blade results :

The observation obtained for the blade model which has a airfoil of NACA 0015 and blade height of 60m is described below.

a) Aerodynamic forces observation :

Forces - Direction Vector (1 0 0)						
		Forces (n)		Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-naca_0015_airfoil	1174.9546	120.85258	1295.8072	1918.2932	197.31034	2115.6036
Net	1174.9546	120.85258	1295.8072	1918.2932	197.31034	2115.6036

Fig.8.6 : Print console of Lift force and Coefficient of Lift values of NACA 0015 airfoil blade

Forces - Direction Vector (0 0 1)						
Zone	Forces (n)			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-naca_0015_airfoil	243.50201	1.8301785	245.33219	397.5543	2.9880465	400.54234
Net	243.50201	1.8301785	245.33219	397.5543	2.9880465	400.54234

**Fig.8.7 : Print console of Drag force and Coefficient of drag values of
NACA 0015 airfoil blade**

Lift Force = 1295.8072 N

Drag Force = 245.332 N

Coefficient of Lift = 2115.6036

Coefficient of Drag = 400.54234

Aerodynamic efficiency (η) = Lift Force / Drag force

= 5.282

Cl/Cd = 5.282

b)Velocity observation :

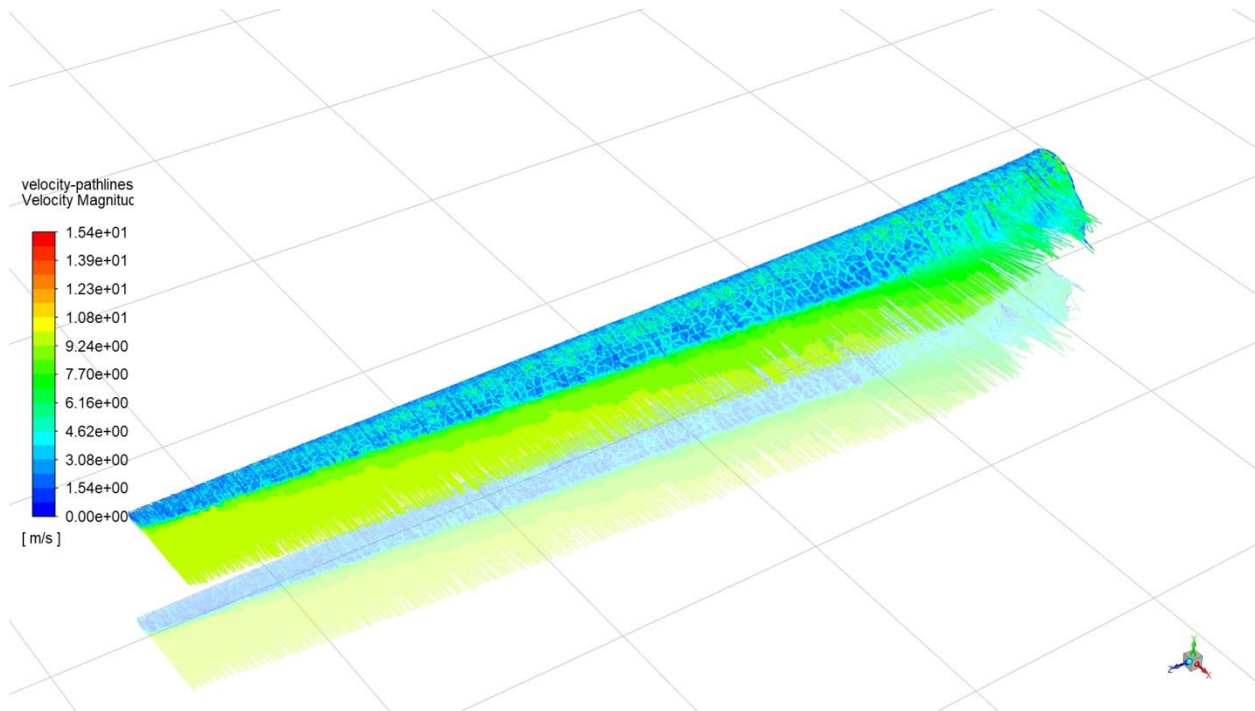


Fig.8.8 : 3D Contour of Velocity pathlines of NACA 0015 airfoil blade

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 15.40 m/s. The velocity is maximum at the flow after the trailing edge portion. The velocity behind the leading edge limit is 0 – 9.24 m/s due to the vortices formation owing to the flow separation.

c)Pressure observation :

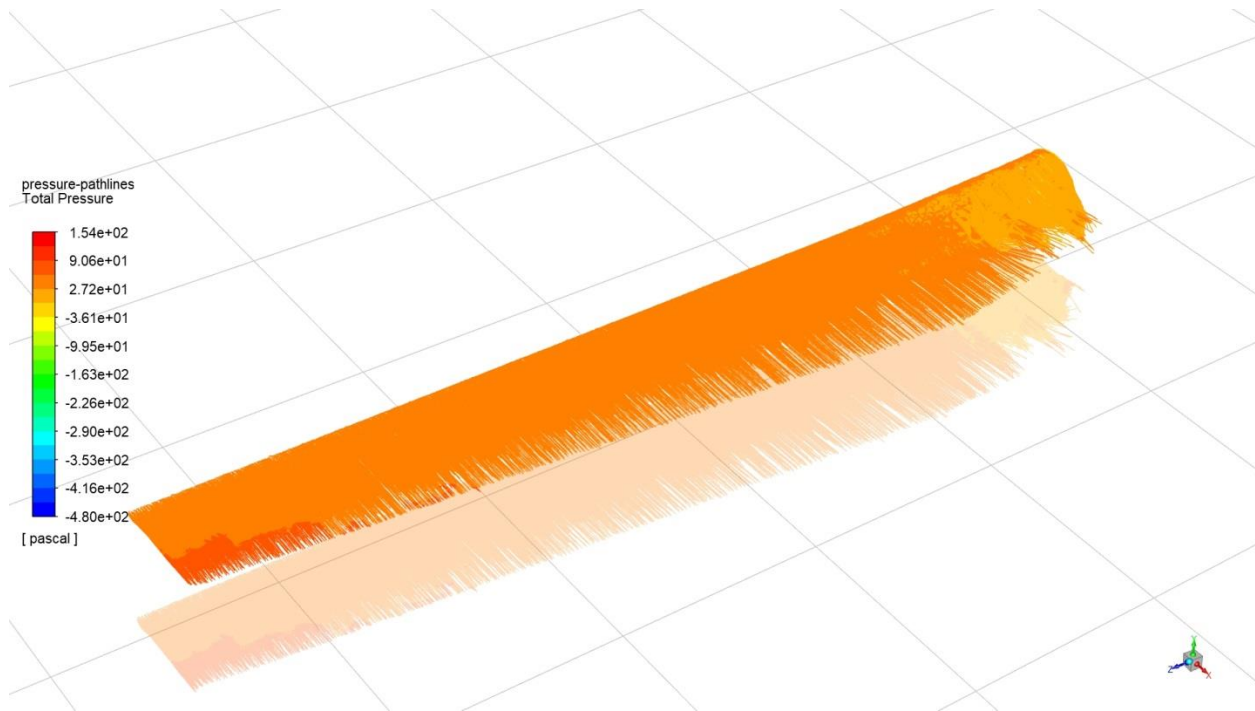


Fig. 8.9 : 3D Contour of pressure pathlines of NACA 0015 airfoil blade

The maximum pressure reached in this case is observed to be 154 Pa. The pressure is maximum at both leading and trailing edge portion. The pressure behind the leading edge limit is -36.10 Pa to 27.20 Pa due to the vortices formation owing to the flow separation.

d) Turbulence intensity observation :

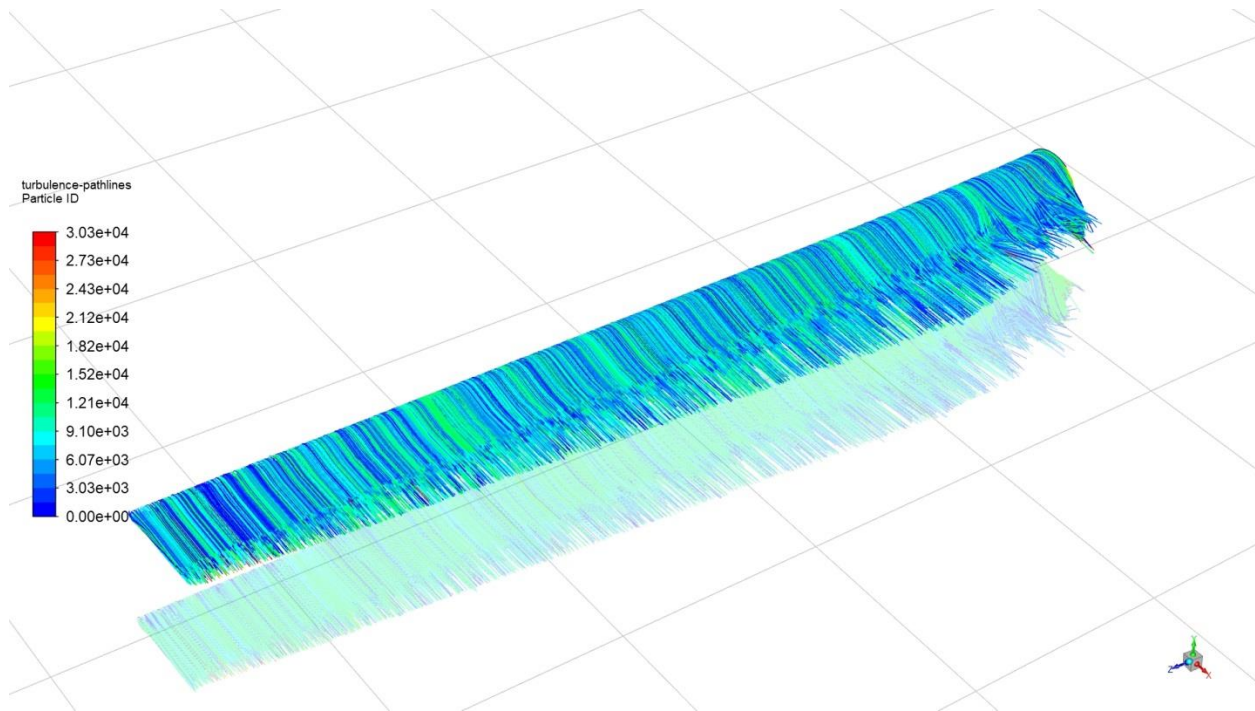


Fig.8.10 : 3D Contour of Turbulence intensity pathlines of NACA 0015 airfoil blade

In Fig.8.10, visualization of the turbulence intensity pathlines that shows the changes in turbulence level at different areas in the flow field using different colors.

8.4) NACA 2412 Airfoil blade results :

The observation obtained for the blade model with slot holes and serration at trailing edge , that is for the blade model which has a airfoil of NACA 0018 and blade height of 0.660 m is described below.

a)Aerodynamic forces observation :

Forces - Direction Vector (1 0 0)						
Zone	Forces (n)			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-naca_2412_airfoil	1505.0159	149.15564	1654.1716	24.571689	2.4351942	27.006883
Net	1505.0159	149.15564	1654.1716	24.571689	2.4351942	27.006883

**Fig.8.11 : Print console of Lift force and Coefficient of lift values of
NACA 2412 airfoil blade**

Forces - Direction Vector (0 0 1)						
Forces (n)			Coefficients			
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-naca_2412_airfoil	301.76414	2.8373768	304.60152	4.9267614	0.046324518	4.973086
Net	301.76414	2.8373768	304.60152	4.9267614	0.046324518	4.973086

**Fig.8.12 : Print console of Drag force and Coefficient of drag values of
NACA 2412 airfoil blade**

Lift Force = 1654.1716 N

Drag Force = 304.60152 N

Coefficient of Lift = 27.006883

Coefficient of Drag = 4.973086

Aerodynamic efficiency (η) = Lift Force / Drag force = 5.431

Cl/Cd = 5.431

b)Velocity observation :

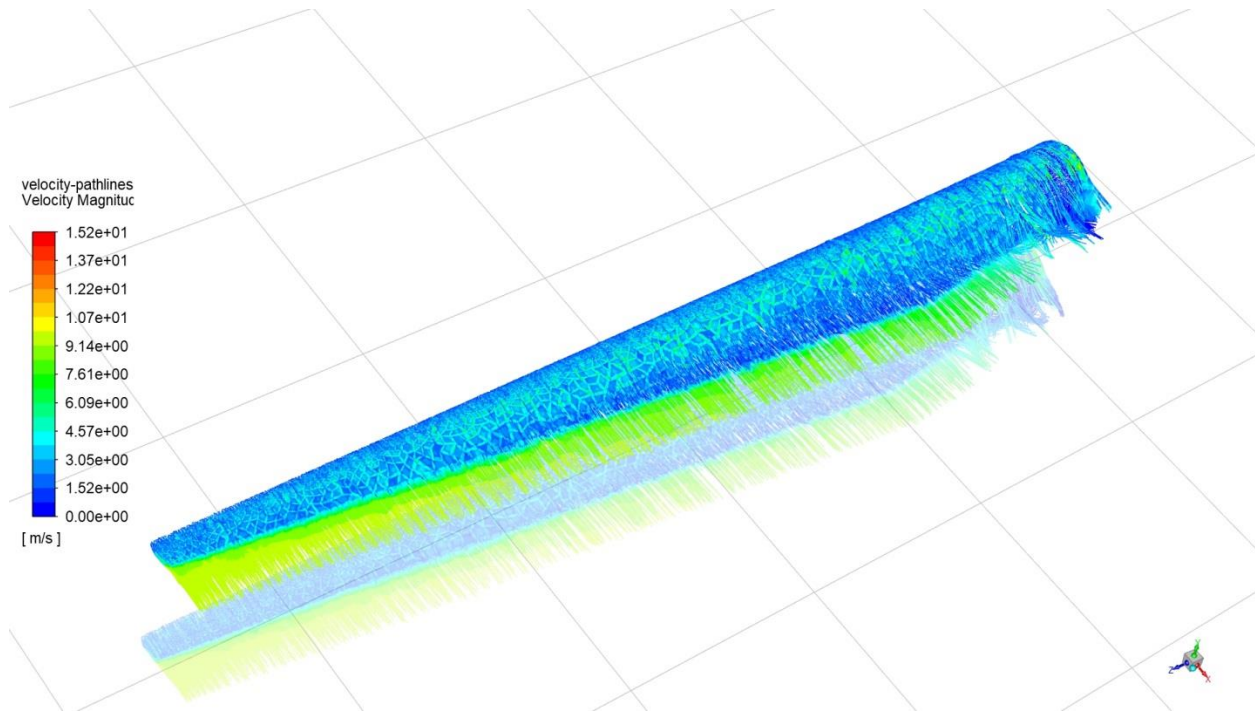


Fig.8.13 : 3D Contour of Velocity pathlines of NACA 2412 airfoil blade

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 12.80 m/s. The velocity is maximum at the flow after the trailing edge portion. The velocity behind the trailing edge limit is 0 – 4.57 m/s due to the vortices formation owing to the flow separation.

c)Pressure observation :

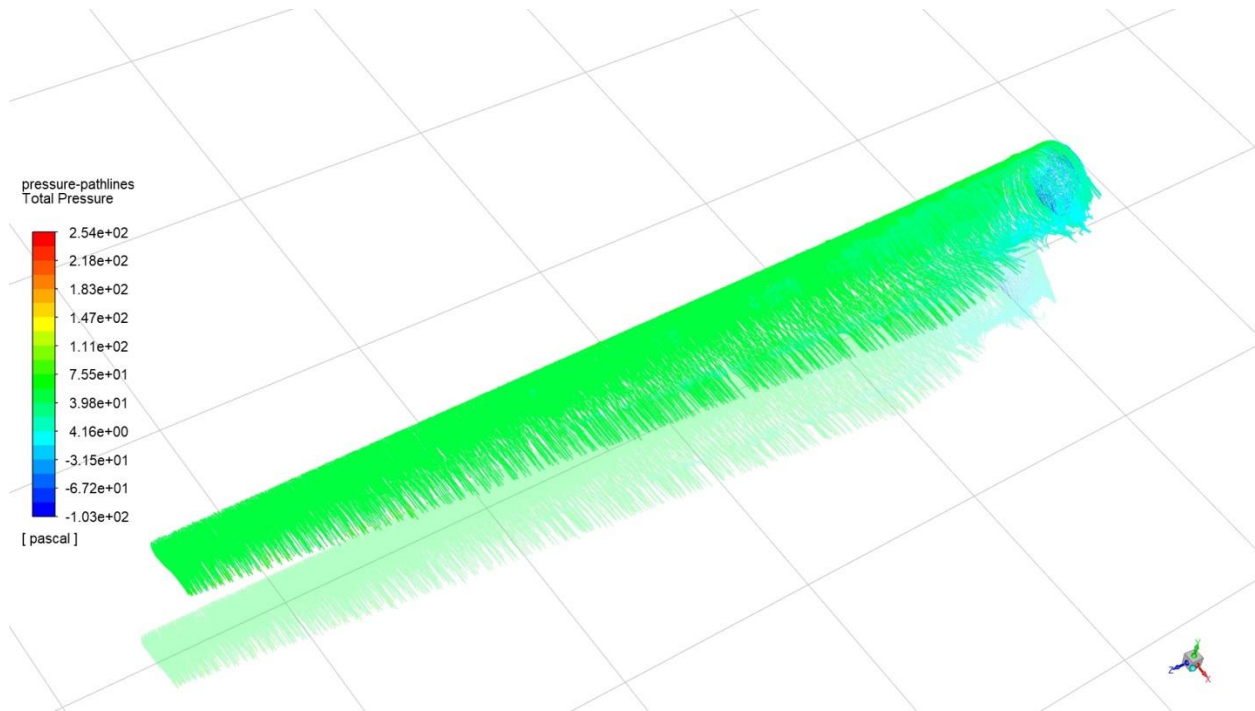


Fig.8.14 : 3D Contour of pressure pathlines of NACA 2412 airfoil blade

The maximum pressure reached in this case is observed to be 254.00 Pa. The pressure is average at both leading and trailing edge portion. The pressure behind the trailing edge limit is 4.16 Pa – 39.80 Pa due to the vortices formation owing to the flow separation.

d)Turbulence intensity observation :

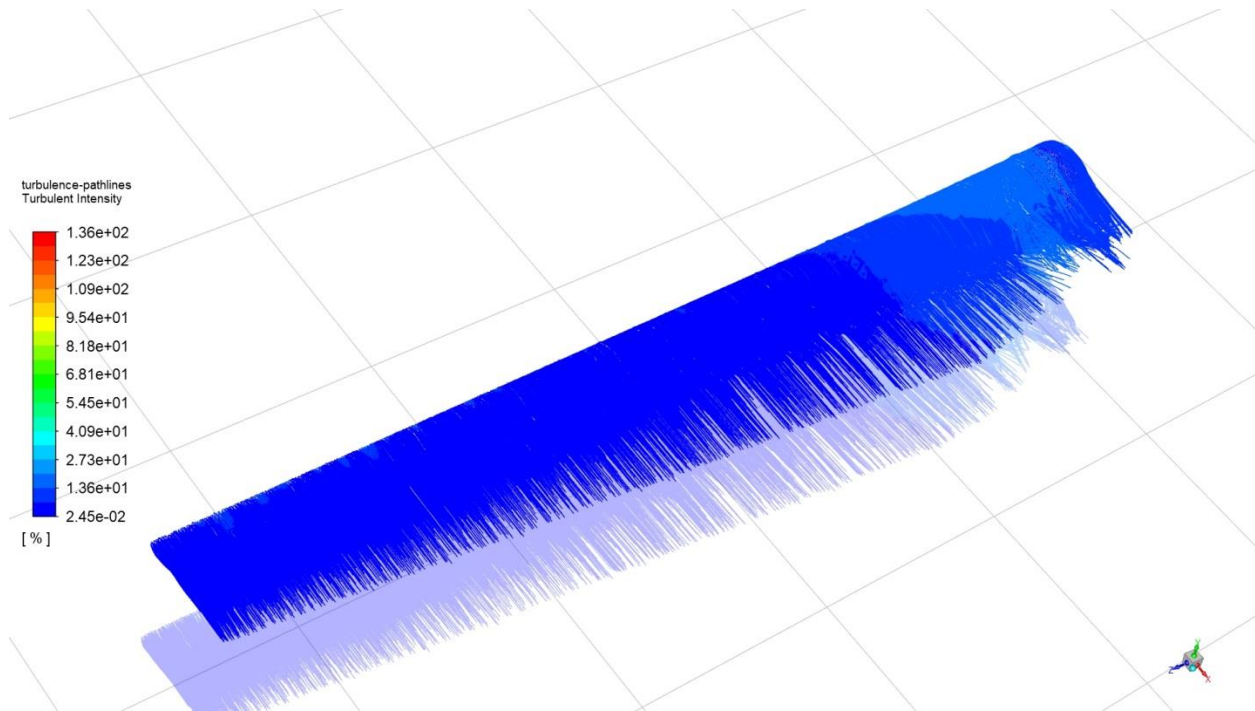


Fig.8.15 : 3D Contour of Turbulence intensity pathlines of NACA 2412 airfoil blade

In Fig.8.15, visualization of the turbulence intensity pathlines that shows the changes in turbulence level at different areas in the flow field using different colors.

8.5) NACA 63215 Airfoil blade results :

The observation obtained for the blade model which has a airfoil of NACA 63215, it is a 5-digit NACA airfoil and blade height of 60m is described below.

a) Aerodynamic forces observation :

lift-force		

	Lift	(n)

wall-naca_63215_airfoil		1749583.4

(a)

cl		

	Cl	()

wall-naca_63215_airfoil		28564.628

(b)

Fig.8.16 : (a) Print console of Lift force and (b) Coefficient of lift values of NACA 63215 airfoil blade

drag-force		

	Drag	(n)

wall-naca_63215_airfoil		967.08525

(a)

cd		

	Cd	()

wall-naca_63215_airfoil		15.789147

(b)

Fig.8.17 : (a) Print console of Drag force and (b) Coefficient of drag values of NACA 63215 airfoil blade

Lift Force = 17495.834 N

Drag Force = 967.08525 N

Coefficient of Lift (Cl) = 285.64628

Coefficient of Drag (Cd) = 15.789147

Aerodynamic efficiency (η) = Lift Force / Drag force = 18.09

Cl/Cd = 18.09

b)Velocity observation :

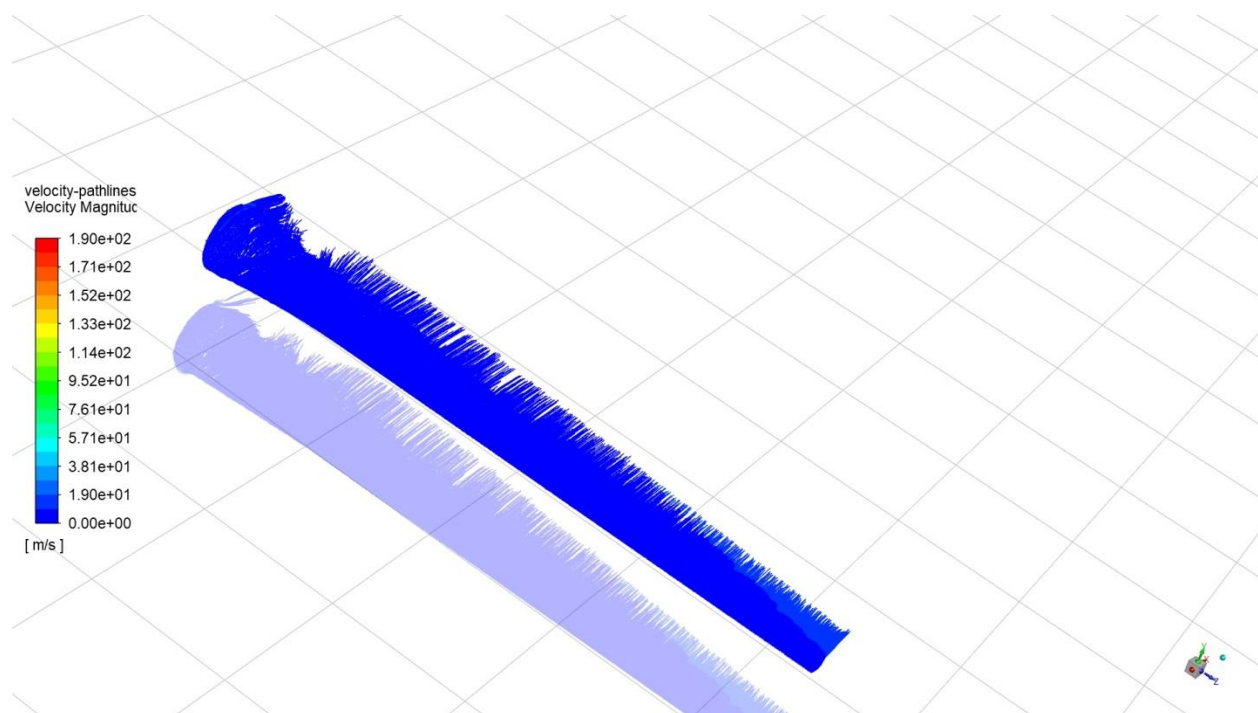


Fig.8.18 : 3D Contour of Velocity pathlines of NACA 63215 airfoil blade

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 19.00 m/s. The velocity is maximum at the flow after the trailing edge portion. The velocity behind the leading edge limit is 0 – 1.90 m/s and the velocity behind the trailing edge limit is 1.90 – 3.81 m/s due to the vortices formation owing to the flow separation.

c)Pressure observation :

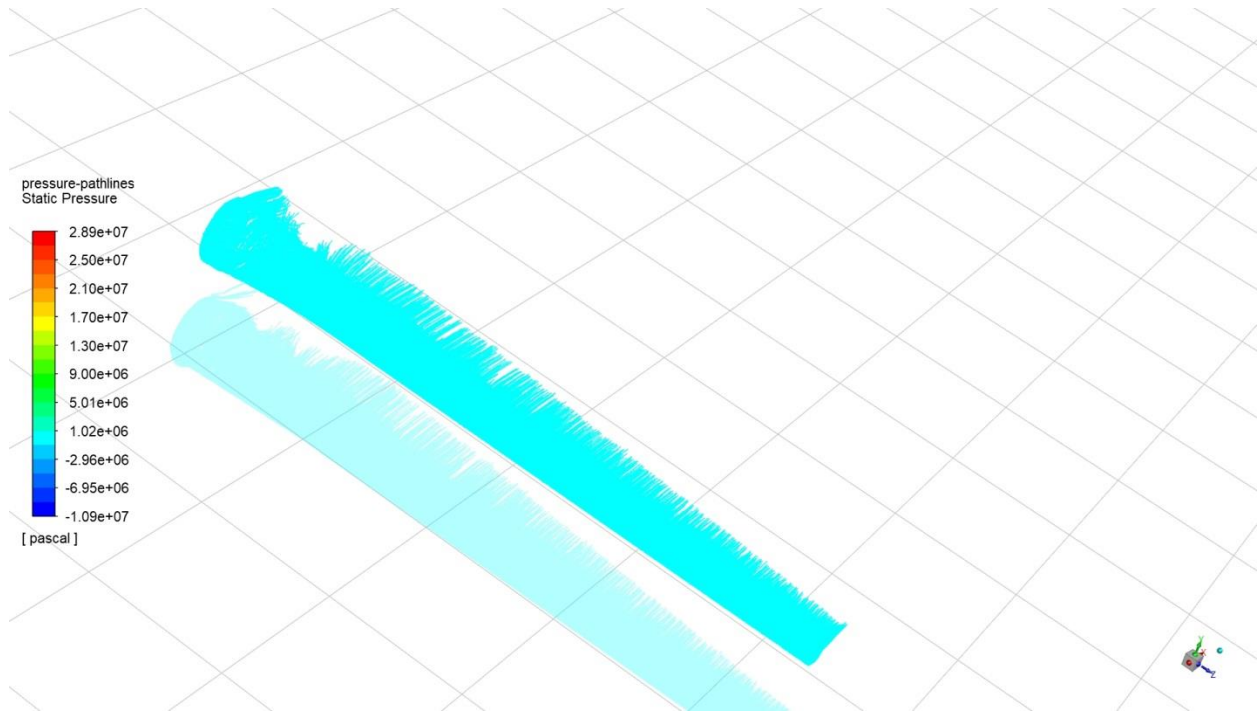


Fig.8.19: 3D Contour of Pressure pathlines of NACA 63215 airfoil blade

The maximum pressure reached in this case is observed to be 289 Pa. The pressure is minimum at both leading and trailing edge portion. The pressure behind the leading edge and trailing edge limit is -29.60 to 10.20 Pa due to the vortices formation owing to the flow separation.

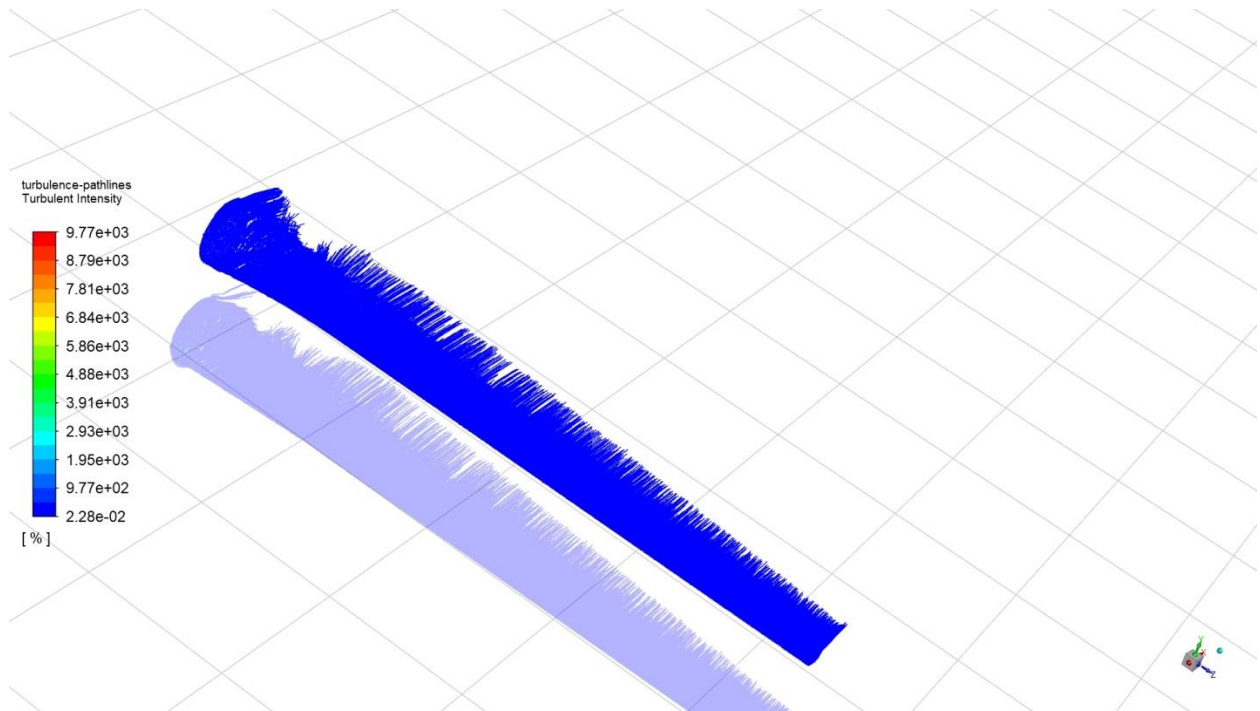


Fig.8.20 : 3D Contour of Turbulence intensity pathlines of NACA 63215 airfoil blade

In Fig.8.20, visualization of the turbulence intensity pathlines that shows the changes in turbulence level at different areas in the flow field using different colors.

8.6) Comparison table for aerodynamic forces and aerodynamic efficiency values of various blade designs :

Blade Type	Lift (N)	Drag (N)	C_L	C_D	Aerodynamic efficiency(η)	C_L/C_D
Reference Blade NACA 4415 airfoil	3.315154	0.707842	0.05412	0.01155	4.683	4.683
NACA 0015 airfoil	1295.807	245.332	2115.60	400.54234	5.282	5.282
NACA 2412 airfoil	1654.172	304.6015	27.0068	4.973086	5.431	5.431
NACA63215 airfoil	17495.83	967.0852	285.646	15.789147	18.09	18.09

Table 6 : Aerodynamic forces and efficiency comparison of various blades

In above table, aerodynamic efficiency of Reference blade which has a airfoil of NACA 4415 is 4.683, NACA 0015 airfoil blade is 5.282, NACA 2412 airfoil blade is 5.431 and the aerodynamic efficiency of NACA 63215 airfoil blade is 18.09.

Aerodynamic efficiency value of NACA 63215 is 3.5 times (approx.) greater NACA 4415, NACA 0015 and NACA 2412 airfoil blades. So, the aerodynamic efficiency of NACA 5 digit series NACA 63215 airfoil blade is more effective compared with other blades.

CHAPTER 9

WIND TUNNEL EXPERIMENT

In wind tunnel experiment, 12 pressure tapes were created in outer surface of selected turbine blade model and insert into the test section of wind tunnel. 100 to 300 RPM motor speed applied to wind tunnel and readings were calculated.

9.1) Coefficient of pressure (C_p) calculation :

Coefficient of pressure, $C_p = P_{st} - P_s / P_0 - P_s$ (No unit)

Motor Speed	C_{p1}	C_{p2}	C_{p3}	C_{p4}	C_{p5}	C_{p6}	C_{p7}	C_{p8}	C_{p9}	C_{p10}	C_{p11}	C_{p12}
100 RPM	2	2	2	2	0.3	2	2	2	2	2	2	2
200 RPM	1	0	0	-1	2	0	3	1	2	0	1	1
300 RPM	-0.33	0.33	-1	0	0.3	-0.33	-0.33	0.33	0	-1	-0.33	-0.33

TABLE 7 : Wind tunnel readings of NACA 63215 airfoil blade

9.2) Coefficient of Lift (C_L) and Drag (C_D) calculation :

Coefficient of Lift, $C_L = (C_{pu} - C_{pl}) (x/c_{max} - x/c_{min})$ (No unit)

Coefficient of Drag, $C_D = (C_{pu} - C_{pl}) (y/c_{max} - y/c_{min})$ (No unit)

Motor speed	C_{pu}	C_{pl}	C_L	C_D
100 RPM	2	0.3	1.207	0.085
200 RPM	3	0	2.13	0.15
300 RPM	-1	-0.33	-0.49	-0.035

TABLE 8 : Coefficient of Lift and Drag calculation

9.3) Aerodynamic forces - Lift (L) and Drag (D) calculation :

$$\text{Lift force} = C_L * \frac{1}{2} \rho A V^2 \text{ (N)}$$

$$\text{Drag force} = C_D * \frac{1}{2} \rho A V^2 \text{ (N)}$$

C_L = Coefficient of Lift (No unit)

C_D = Coefficient of Drag (No unit)

ρ = Density of air (kg/m^3) = 1.225 kg/m^3

A = Blade surface area (m^2)

V = wind velocity (m/s) = 10 m/s

Lift force = 16892.23 N

Drag force = 875.312 N

Aerodynamic efficiency = η = Lift force / Drag force

$$= 16892.23/1032.312 = 16.363$$

Aerodynamic efficiency (η) = 16.363

CHAPTER 10

POWER CALCULATION

The output power of vertical axis wind turbine can be calculated by using this formula

$$P = \frac{1}{2} \rho A v^3 C_P \text{ (W)}$$

$$\text{i.e., } P \propto v^3$$

P - Output power (W)

ρ - Density of air (Kg/m^3) = 1.225 Kg/m^3

r = Diameter of rotor (m) = 0.526 m

r = Radius of rotor (m) = 0.263 m

A - Swept Area (m^2) = πr^2 (m^2) = $3.14 \times (60)^2 = 11304 \text{ m}^2$

v- Wind velocity (m/s)

Cp- Power Coefficient (No unit) = 0.40 (Betz Limit)

Average velocity (location of wind turbine blades) = $v_{\text{avg}} = 11.34 \text{ m/s}$

$$P = 0.5 \times 1.225 \times 11304 \times (11.34)^3 \times 0.40 = 4038660.9655$$

POWER OUTPUT (P) = 4.039 MW

CHAPTER - 11

CONCLUSION

11.1) Summary and conclusions

Modification in blade shapes for small vertical axis wind turbines are being used to accelerate the wind velocity in the small wind regimes. Many experimental and numerical studies has been carried out on the blade shape modification like slot holes and serration to augment velocity/wind power and aerodynamic efficiency are available in the literature. At present, a numerical study has been carried using computational fluid dynamics on a brimmed windturbine blade with the variations in geometric details of a 3D blade model (without modeling the wind turbine) is used in this study. The pre-processing and numerical simulation is carried out in ANSYS software 'Fluent'. 3D modeling has been carried out using CATIA V5 software. Variations in both leading and trailing edge of blade are used in this study.

Based on the numerical simulation of flows with uniform and smooth approach velocity of 10 m/s around the wind turbine blade with the slot holes, leading and trailing edge serration cases, the following conclusions have been made.

Comparison of aerodynamic efficiency of these cases, Reference blade of NACA 4415 airfoil blade is 4.683, NACA 0015 airfoil blade is 5.282, NACA 0015 airfoil blade is 5.431 and NACA 63215 airfoil blade is 18.09.

Comparing the three blade models and reference blade model, NACA 63215 airfoil blade reached maximum aerodynamic efficiency of 18.09. In windtunnel experiment, aerodynamic efficiency of NACA 63215 airfoil blade model is 16.363

Comparison of the velocity of flow along the length of blade has been made in these cases, In the case of NACA 4415 airfoil blade (Reference blade), average velocity 10.81 m/s is observed, In NACA 0015 airfoil case, average velocity 10.44 m/s is observed. In

the case of NACA 2412 airfoil, 11.03 m/s is observed and average velocity 11.34 m/s is observed in the NACA 63215 airfoil case.

Comparing the three models and reference model, NACA 5 digit series NACA 53215 blade model has better results in average velocity and aerodynamic efficiency. So, average velocity of this model is taken into calculate power output of wind turbine. The power output is 4.039 MW

It is to be noted that

This flow simulation is done for a 3D dimensional boundary case of the model with solution.

The validation of the results obtained based on these simulation to be done using windtunnel experiments. Hence, the results can be used quantitatively.

REFERENCES

1. Jianlong Ma, Yafan Duan, Ming Zhao, Wenchun Lv, Jianwen Wang, Qilao Meng Ke, and Yongfeng Ren, "Effect of Airfoil Concavity on Wind Turbine Blade Performances", 2019
2. N Manikandan, Stalin Balasubramaniam, " Design of NACA 63215 Airfoil for a Wind Turbine", 2017
3. Emmanuel Yeboah Osei, Richard Opoku, Albert K. Sunnu, and Muiyiwa S. Adaramola , " Development of High Performance Airfoils for Application in Small Wind Turbine Power Generation", 2020
4. Hong Zongheng , Yu Tao , Chen Guanyu ,Li Xiangrui, Hong Yu , " Simulation Analysis on the Blade Airfoil of Small Wind Turbine", 2019
5. M. Birajdar, S. Kale, and S. N. Sapali, *Effects of Design Parameters on Aerodynamic Performance of New Profile Small Wind Turbine Blades*, IMECE2015, Houston, TX, USA, 2016.
6. R. K. Singh and M. R. Ahmed, "Blade design and performance testing of a small wind turbine rotor for low wind speed applications," *Renewable Energy*, vol. 50, pp. 812–819, 2013.
7. H. Shah, S. Mathew, and C. M. Lim, "A novel low Reynolds number airfoil design for small horizontal axis wind turbines," *Wind Engineering*, vol. 38, no. 4, pp. 377–391, 2014.
8. Y. Liu, C. Yang, and X. Song, "An airfoil parameterization method for the representation and optimization of wind turbine special airfoil," *Journal of Thermal Science*, vol. 24, no. 2, pp. 99–108, 2015.
9. J. L. Dowler and S. Schmitz, "A solution-based stall delay model for horizontal-axis wind turbines," *Wind Energy*, vol. 18, no. 10, pp. 1793–1813, 2015.

10. J. Chen, Q. Wang, S. Zhang, P. Eecen, and F. Grasso, "A new direct design method of wind turbine airfoils and wind tunnel experiment," *Applied Mathematical Modelling*, vol. 40, no. 3, pp. 2002–2014, 2016.
11. R. Azim, M. M. Hasan, and M. Ali, "Numerical investigation on the delay of boundary layer separation by suction for NACA 4412," *Procedia Engineering*, vol. 105, pp. 329–334, 2015.
12. G. M. McNerney, C. P. Van Dam, and D. T. Yen-Nakafuji, "Blade-wake interaction noise for turbines with downwind rotors," *Journal of Solar Energy Engineering*, vol. 125, no. 4, pp. 497–505, 2003.
13. S. S. Rodrigues and A. C. Marta, "On addressing wind turbine noise with after-market shape blade add-ons," *Renewable Energy*, vol. 140, pp. 602–614, 2019.
14. A. C. Benim, M. Diederich, F. Gül, P. Oclon, and J. Taler, "Computational and experimental investigation of the aerodynamics and aeroacoustics of a small wind turbine with quasi-3D optimization," *Energy Conversion and Management*, vol. 177, pp. 143–149, 2018.
15. S. Sanaye and A. Hassanzadeh, "Multi-objective optimization of airfoil shape for efficiency improvement and noise reduction in small wind turbines," *Journal of Renewable and Sustainable Energy*, vol. 6, no. 5, Article ID 053105, 2014.
16. S. S. Rodrigues and A. C. Marta, "On addressing noise constraints in the design of wind turbine blades," *Structural and Multidisciplinary Optimization*, vol. 50, no. 3, pp. 489–503, 2014.
17. P. Pourazarm, L. Caracoglia, M. Lackner, and Y. Modarres-Sadeghi, "Stochastic analysis of flow-induced dynamic instabilities of wind turbine blades," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 137, pp. 37–45, 2015.
18. P. Pourazarm, Y. Modarres-Sadeghi, and M. Lackner, "A parametric study of coupled-mode flutter for MW-size wind turbine blades," *Wind Energy*, vol. 19, no. 3, pp. 497–514, 2016.

19. M. M. A. Hafeez and A. A. El-Badawy, "Flutter limit investigation for a horizontal axis wind turbine blade," *Journal of Vibration and Acoustics*, vol. 140, no. 4, Article ID 041014, 2018.
20. J. Howison, J. Thomas, and K. Ekici, "Aeroelastic analysis of a wind turbine blade using the harmonic balance method," *Wind Energy*, vol. 21, no. 4, pp. 226–241, 2018.
21. B. Chen, Z. Zhang, X. Hua, S. R. K. Nielsen, and B. Basu, "Enhancement of flutter stability in wind turbines with a new type of passive damper of torsional rotation of blades," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 173, pp. 171–179, 2018.
22. E. Verstraelen, G. Habib, G. Kerschen, and G. Dimitriadis, "Experimental passive flutter suppression using a linear tuned vibration absorber," *AIAA Journal*, vol. 55, no. 5, pp. 1707–1722, 2017.