

CFD Simulation of Slot holes with Leading and Trailing edge serration in Vertical Axis wind turbine blade

Submitted in partial fulfilment of the requirements for the award of

Bachelor of Engineering degree in
Aeronautical Engineering

By

RAGAVENDHAR K A (Reg. No. 38260033)

GOKULAKANNAN V (Reg. No. 38260013)



**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 RAGAVENDHAR K A (Reg. No. 38260033) and GOKULAKANNAN V (Reg. No. 38260013) who carried out the project entitled **"CFD Simulation of Slot holes with Leading and Trailing edge serration in Vertical Axis wind turbine blade"** under our supervision from September 2021 to April 2022.

Internal Guide
Mr. VENKATESH. S, M.E.

Head of the Department
Dr. P. BOOMADEVI, ME., PhD.

Submitted for Viva voce Examination held on _____

Internal Examiner

External Examiner

DECLARATION

We RAGAVENDHAR K A (Reg. No. 38260033) and GOKULAKANNAN V (Reg. No. 38260013) hereby declare that the Project Report entitled **“CFD Simulation of Slot holes with Leading and Trailing edge serration in Vertical Axis wind turbine blade”** done by us under the guidance of **Mr. VENKATESH S, 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. VENKATESH S., 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.

TABLE OF CONTENTS		
CHAPTER No.	TITLE	PAGE No.
	LIST OF FIGURES	7
	LIST OF TABLES	9
	ABSTRACT	10
1	INTRODUCTION	11
	1.1 WIND ENERGY	12
	1.2 WIND TURBINE	13
	1.3 TYPES OF WIND TURBINE	15
	1.4 APPLICATIONS OF WIND TURBINE	16
	1.5 TYPES OF VAWT	18
	1.6 SLOTTED WIND TURBINE BLADES	22
	1.7 SERRATIONS	24
2	LITERATURE REVIEW	26
3	AIM AND SCOPE	38
	3.1 AIM AND SCOPE	38
	3.2 OVERVIEW	39
4	COMPUTATIONAL FLUID DYNAMICS	40
	4.1 GENERAL	40
5	DESIGN AND CFD ANALYSIS	41
	5.1 MODELLING OF VAWT	42

	5.2	LEADING EDGE SERRATIONS	43
	5.3	TRAILING EDGE SERRATIONS	45
	5.4	LEADING AND TRAILING EDGE SERRATIONS	
6	COMPUTATIONAL DOMAIN AND NUMERICAL SOLUTIONS		48
	6.1	GENERAL	48
	6.2	BOUNDARY CREATION	48
	6.3	MESHING	49
	6.4	CFD BOUNDARY	50
7	NUMERICAL SOLUTION USING FLUENT 6.3		53
	7.1	OVERVIEW OF FLUENT	53
8	RESULTS AND DISCUSSIONS		59
	8.1	OBSERVATIONS	59
	8.2	SLOTS WITH LEADING EDGE RESULTS	62
	8.3	SLOTS WITH TRAILING EDGE RESULTS	66
	8.4	SLOTS WITH LEADING AND TRAILING EDGE RESULTS	67
	8.5	COMPARISON TABLE	67
9	WIND TUNNEL EXPERIMENT		68
10	POWER CALCULATIONS		68
11	CONCLUSIONS		68

List of figures:

Fig.1: Principle of wind turbine

Fig.2: Principles of wind turbine Aerodynamic lift

Fig.3: Horizontal axis wind turbine

Fig.4: Vertical axis wind turbine

Fig.5: Difference between Horizontal and Vertical Axis wind turbine

Fig.6: Main designs of vertical axis wind turbine

Fig.7: Savonius style vertical axis wind turbine

Fig.8: Darrieus style vertical axis wind turbine

Fig.9: Types of Darrieus style vertical axis wind turbine

Fig.10: (a) Aerodynamic forces on airfoil

(b) Aerodynamic forces on airfoil with inclined slots

Fig.11: Semi circle cavity slot in airfoil

Fig.12: Vortices formation in Semi circle cavity slot in airfoil

Fig.13: (a) Serration in leading edge of airfoil

(b) Sinusoidal wave pattern in leading edge of airfoil

Fig.14: (a) Serration in trailing edge of airfoil

(b) Eddies formation in trailing edge of airfoil

Fig.15: (a) Dimensions of elliptical hole

(b) Pitch between elliptical holes

Fig. 16: Dimensions of Sinusoidal wave profile in leading edge of turbine blade

Fig .17: Sinusoidal wave cut pattern in leading edge of turbine blade

Fig.18: Dimensions of symmetrical triangle profile in trailing edge of turbine blade

Fig.19: Symmetrical triangle cut pattern in trailing edge of turbine blades

Fig.20: Elliptical slot holes with leading and trailing edge serration

Fig.21: 3D Turbine blade model in 3D boundary

Fig.22: Coarse mesh on Boundary and Fine mesh on turbine blade

Fig.23: Enlarged image of mesh around the turbine blade

Fig.24: Velocity inlet of the computational domain

Fig.25: Pressure outlet of the computational domain

Fig.26: Walls of the computational domain

Fig.27: Grid check in Fluent software

Fig.28: Print console of Lift force and Coefficient of lift values

Fig.29: Print console of Drag force and Coefficient of drag values

Fig. 30: 3D Contour of Velocity pathlines for the Normal Blade model

Fig. 31: 3D Contour of pressure pathlines for the Normal Blade model

Fig.32: 3D Contour of Turbulence intensity pathlines for the Normal Blade model

Fig.33: Print console of Lift force and Coefficient of lift values

Fig.34: Print console of Drag force and Coefficient of drag values

Fig.35: 3D Contour of Velocity pathlines for the Normal Blade model

Fig.36: 3D Contour of pressure pathlines for the Normal Blade model

Fig.37: 3D Contour of Turbulence intensity pathlines for the Normal Blade model

Fig.38: Print console of Lift force and Coefficient of lift values

Fig.39: Print console of Drag force and Coefficient of drag values

Fig. 40: 3D Contour of Velocity pathlines for the blade with trailing edge serration

Fig.41: 3D Contour of pressure pathlines for the blade with trailing edge serration

Fig.42: 3D Contour of Turbulence intensity pathlines for the Blade with trailing edge serration

Fig.43: Print console of Lift force and Coefficient of lift values

Fig.44: Print console of Drag force and Coefficient of drag values

Fig.45: 3D Contour of Velocity pathlines for the blade model with Slot holes with leading and trailing edge serration

Fig.46: 3D Contour of Pressure pathlines for the blade model with Slot holes with leading and trailing edge serration

Fig.47: 3D Contour of Turbulence intensity pathlines for the blade model with Slot holes with leading and trailing edge serration

LIST OF TABLES:

Table 1: Solver specifications

Table 2: Material Properties

Table 3: Velocity inlet conditions & Pressure Outlet conditions

Table 4: Reference Values

Table 5: Iteration Values

Table 6: Aerodynamic forces and aerodynamic efficiency comparison of various blade designs

Table 7: Wind tunnel readings of slot holes with leading and trailing edge serration blade model

Table 8: Coefficient of Lift and Drag calculation

ABSTRACT

Research interest in the vertical axis wind turbine has been growing rapidly in the last few years considering its relatively low cost, ease of maintenance, and noiselessness. Low starting torque is considered as one of the major drawbacks of vertical axis wind turbine. Design optimization to overcome this problem by modifying the turbine blade shape has received great attention by the wind energy community. Moreover, efforts have been directed towards augmenting the power coefficient of the turbine. In the present work, the effect of employing slotted turbine blade on the performance and starting characteristics has been investigated. The slot types a) elliptical slot holes with leading edge serration, b) elliptical slot holes with trailing edge serration and c) elliptical slot holes with both leading and trailing edge serration, were used for NACA 0018 airfoil which is commonly used in vertical axis wind turbines. The flow over the turbine was modelled using ANSYS-FLUENT code. The main observations of the present study show that the VAWT with slotted Airfoil has a lower optimum tip speed ratio compared with the Baseline turbine. This is associated with the ability to generate higher torque at lower rotational speeds. The analysis of the aerodynamic behavior of the slotted airfoil turbine shows that the airfoil slot delays the separation at high angles of attack, and therefore improves the torque and power coefficient at low tip speed ratio.

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.

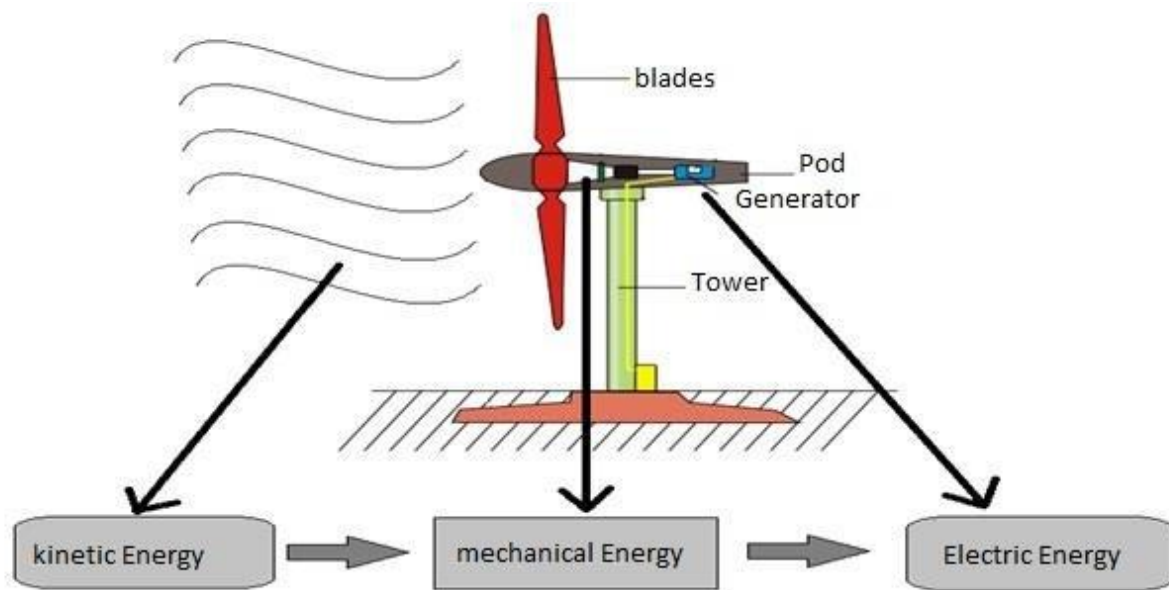


Fig.1: Principle of wind turbine

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, $P = \frac{1}{2} \rho A V^3$ (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: $C_p = 0.59$ pmax = 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,

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

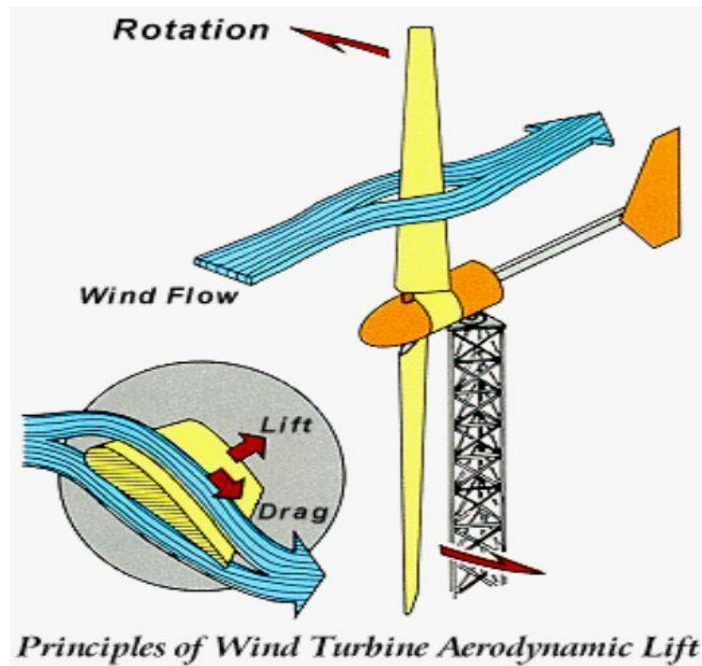


Fig.2 : Principles of wind turbine Aerodynamic lift

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.

- **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.

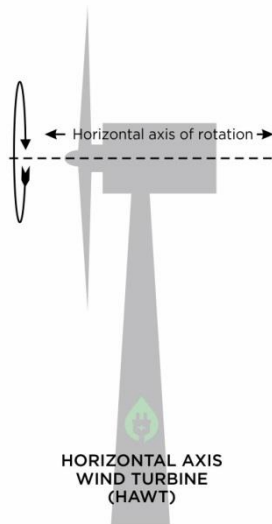


Fig.3 : Horizontal axis wind turbine

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.

- VERTICAL-AXIS TURBINES

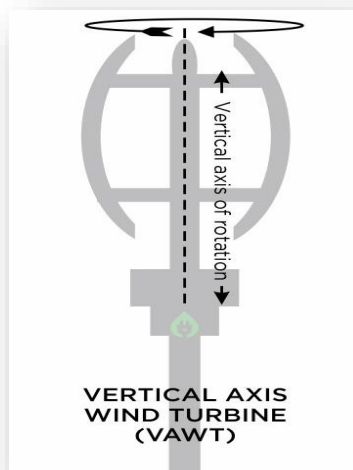


Fig.4 : Vertical axis wind turbine

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:

1.4.1 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.

1.4.2 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.

1.4.3 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.

Difference between Horizontal and Vertical Axis Wind Turbine:

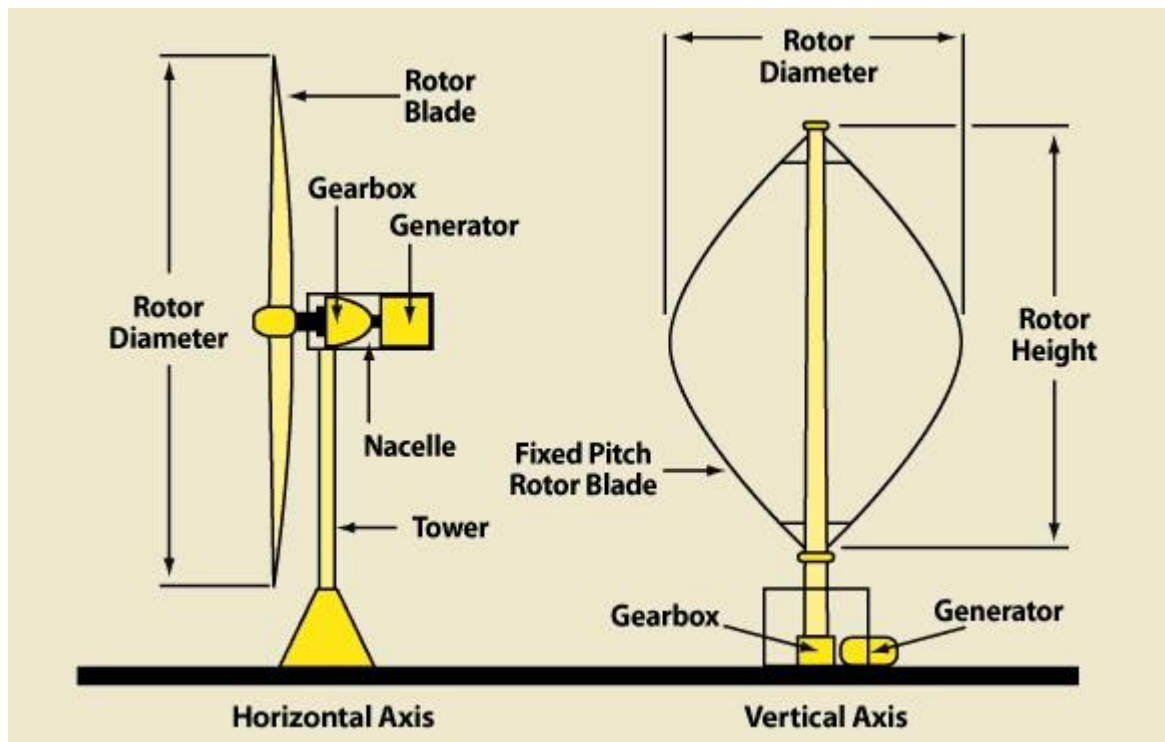


Fig.5 : Difference between Horizontal and Vertical Axis wind turbine

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.

Vertical axis wind turbine has been growing rapidly in the last few years considering its relatively low cost, ease of maintenance, and noiselessness compared with horizontal axis wind turbines

1.5 Types of Vertical Axis Wind Turbine :

There are two main designs of VAWT, called *Savonius* and *Darrieus*. These designs are quite different in the way they capture the wind energy.

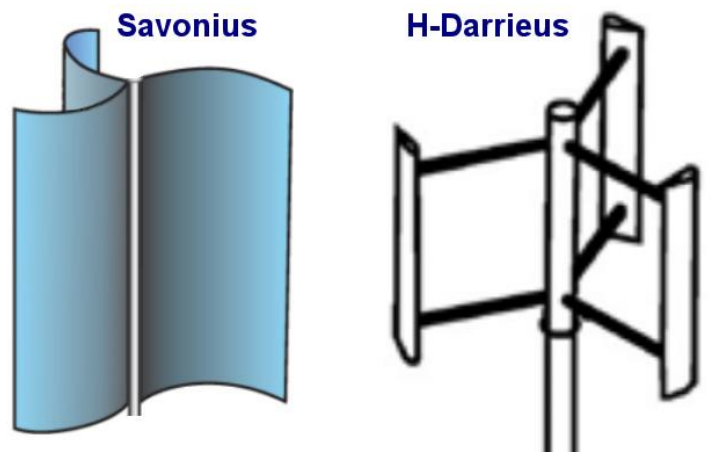


Fig.6: Main designs of vertical axis wind turbine

1.5.1 Savonius Wind Turbines: The Resistance Runner

Savonius wind turbines have blades built around the vertical shaft in a helix form, which basically looks like DNA or fusilli pasta. Wide, solid wind-receiving area of the blades is one of the most significant features of a Savonius wind turbine.

When in operation, Savonius wind turbines rely on the flow resistance mechanism to turn their rotors. In simple words, the dynamic pressure of the wind against the blades pushes the rotor into rotation. At the same time, the opposite side of the blades encounters a force of aerodynamic resistance or “drag.” This is just like what we experience when cycling or running: There’s always the air flow coming against us. Because of this, Savonius wind turbines can only turn as fast as the wind speed.

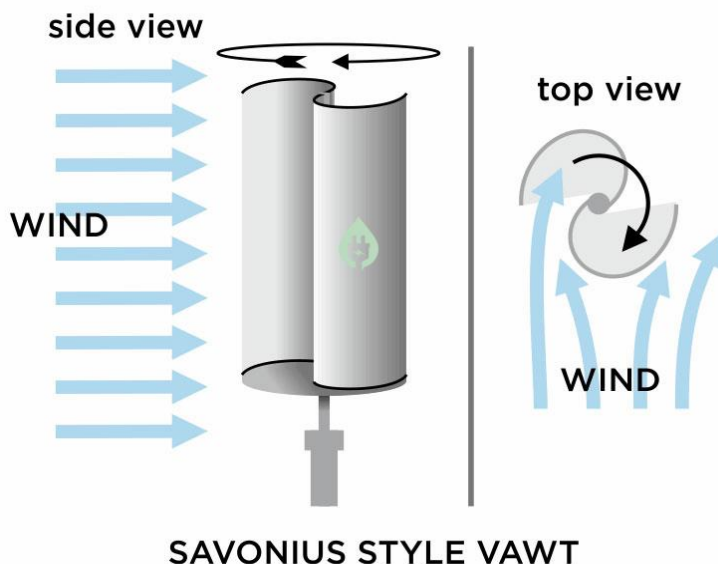


Fig.7:Savonius style vertical axis wind turbine

1.5.2 Darrieus: The Uplift Runner

Often called the “egg-beater” and named after the French inventor Georges Darrieus, classic Darrieus turbines have long, curved wings with each end attached to the top and bottom of the rotor shaft. Another model of the Darrieus turbines has three straight wings connected to the shaft parallelly, forming the “H” shape.

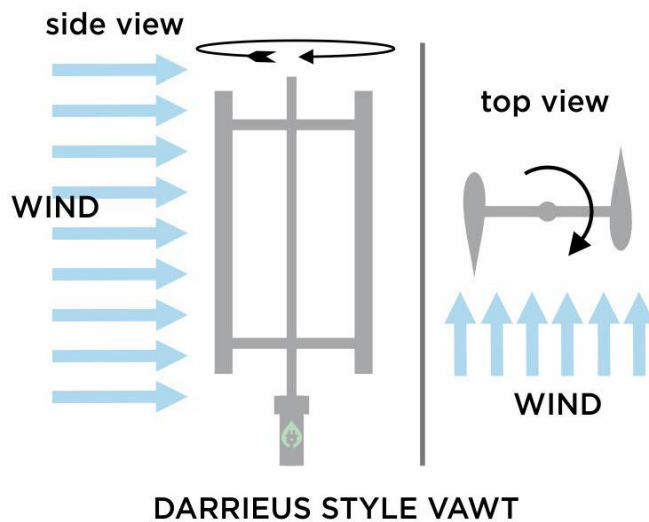


Fig 8. Darrieus style vertical axis wind turbine

In terms of operation, Darrieus utilizes the “lift” aerodynamic force to rotate. By flowing around the structure, the wind creates a suction on the front side of the turbine, driving the wings to rotate. Because of the shape of the wings, they do not experience as much drag as Savonius turbines do. Once the rotation starts, Darrieus wind turbines are able to accelerate to rotate faster than the wind speed.

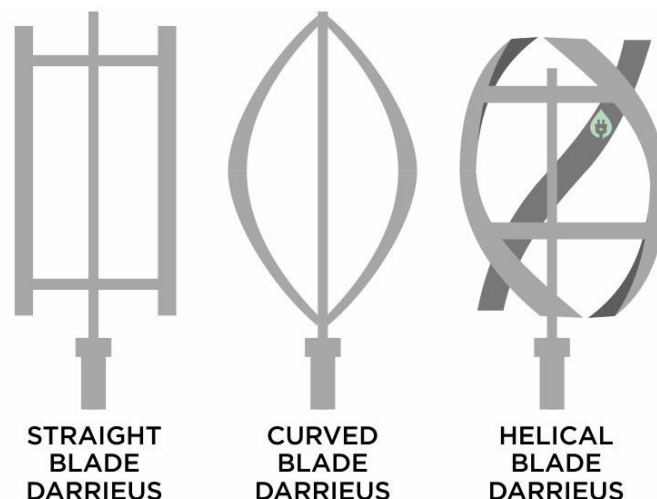


Fig 9. Types of Darrieus style vertical axis wind turbine

In Darrieus style vertical axis wind turbines, three types of designs are there. They are

1. Straight blade darrieus VAWT
2. Curved blade darrieus VAWT
3. Helical blade darrieus VAWT

1.5.3 Advantages of Vertical Axis Wind Turbine :

- Due to certain structural characteristics, vertical axis wind turbines are typically smaller than horizontal axis wind turbines. This adds to the number of advantages such as low noise emission and high turbulence endurance that make them perfect for electricity generation in populated or urban areas as well as in unstable, gusty wind.
- Since vertical axis wind turbines have less energy efficiency and power output, it is common to group multiple turbines into a wind power plant or use them as one of many electricity sources of a building or campus. It is possible to connect a vertical axis wind turbine into the local power grid or with an external storage battery. Although not as common as horizontal axis wind turbines, vertical axis wind turbines are also used on offshore platforms to support various industrial activities.
- Vertical axis wind turbines are more geographically flexible and not restricted as much by governmental policies. This makes them great alternatives for conditions unsuitable for horizontal axis models.
- The main advantage that VAWTs have over HAWTs is that they do not need to yaw. They collect wind energy from all directions all the time. Because of the way the blades are designed, they always spin the same direction, no matter where the wind is coming from this makes VAWTs better at capturing wind energy in city rooftops and built up areas where turbulent wind is changing direction all the time. VAWTs take up less room than an equivalent power HAWT, and are often chosen for areas where space is limited.

1.5.4 DISADVANTAGES OF VAWT:

- Low starting torque is considered as one of the major drawbacks of Darrieus type vertical axis wind turbine
- Power output is less compared with horizontal axis wind turbines

1.6 SLOTTED WIND TURBINE BLADES:

The Darrieus type wind turbine is a lift-type VAWT, where the lift force exerted on the turbine blade is of main concern. Basically, the lift and drag forces are generated due to difference in pressure between the airfoil upper and lower surfaces. Negative pressure is produced on the suction side of the airfoil, while positive pressure is generated on the pressure side. This scenario persists for small angles of attack. At higher angles of attack, flow separation occurs due to the positive pressure gradient towards the trailing edge of the airfoil.

1.6.1 INCLINED SLOTS:

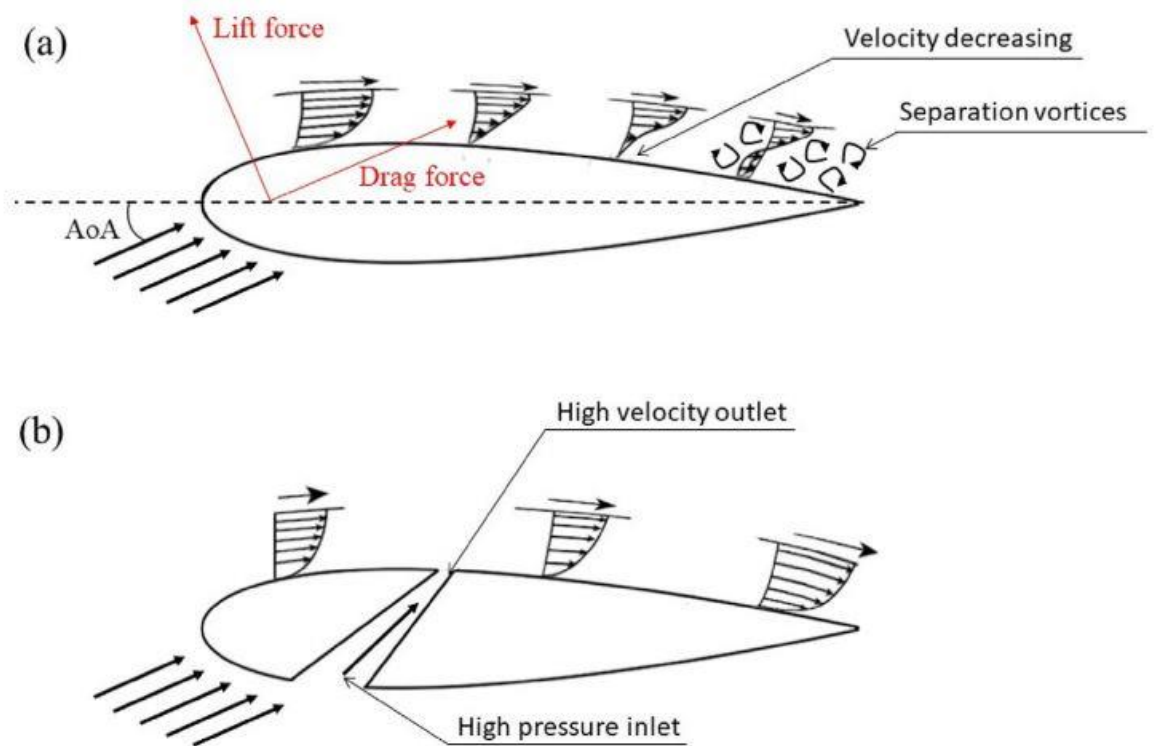


Fig 10. (a) Aerodynamic forces on airfoil

(b) Aerodynamic forces on airfoil with inclined slots

Fig. 10 shows the typical flow behavior along an airfoil section with initiating flow separation at the trailing edge for a conventional airfoil (Fig. 10-a) and a slotted airfoil (Fig. 10-b). The slot, as depicted in Fig. 10-b, extends from the pressure side (lower side) to the suction side (upper side). Due to the difference in pressure between the two sides, a jet flow is formed through the slot, and high momentum is injected into the boundary layer of the upper surface. The effect of this jet is to eliminate separation at mild angles of attack and extend the high lift range of the airfoil.

1.6.2 SEMI CIRCLE CAVITY SLOT:



Fig.11: Semi circle cavity slot in airfoil

In this type, a semi-circle profile cut is presented near the leading edge of airfoil shape. It looks like a cavity in a shape. So, it is called as semi-circle cavity slot.

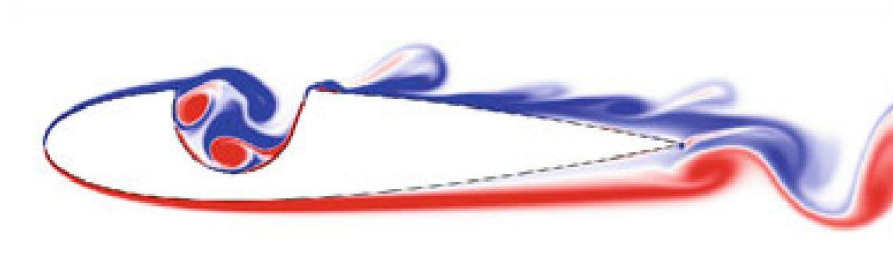


Fig.12: Vortices formation in Semi circle cavity slot in airfoil

In semi-circle cavity slots, trapping of vortices in a cavity as a drag reduction measure for thick airfoils. This type of slots are used to improve the aerodynamic efficiency of turbine blades.

1.7 SERRATIONS:

A great deal of research has been directed toward the application of different types of serrations for the control of airfoil trailing- and leading-edge noise. Prior research has shown that such passive treatments are capable of reducing the airfoil noise over a wide range of frequencies. However, the aerodynamic performance of such treated airfoils has received very little research attention.

1.7.1 Leading edge serration:

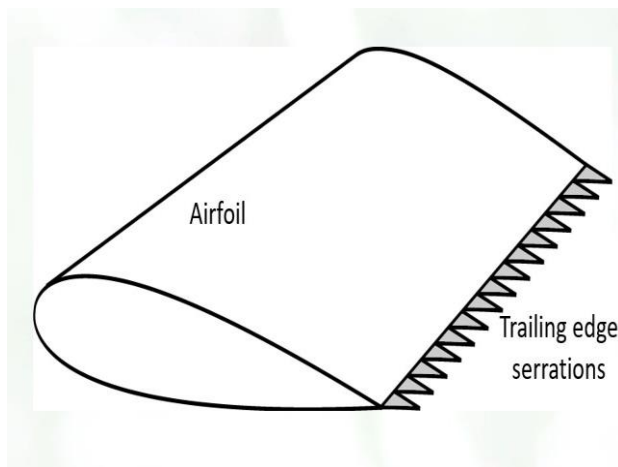


Fig.13: (a) Serration in leading edge of airfoil

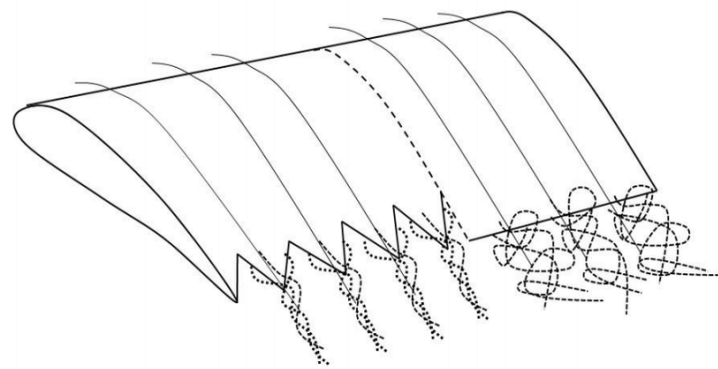
(b) Sinusoidal wave pattern in leading edge of airfoil

In leading edge serration, sinusoidal wave profile cuts were presented in leading edge of airfoil in turbine blades to control the dynamic flow separation at low tip speed ratios.

1.7.2 Trailing edge serration:



(a)



(b)

Fig.14: (a) Serration in trailing edge of airfoil

(b) Eddies formation in trailing edge of airfoil

In this serration, symmetrical triangle shape cutprofiles were placed in trailing edge of airfoil. Trailing edge serrations reduce the noise of the so called “turbulent boundary layer trailing edge noise”. This is the dominant noise source of a wind turbine.

Small eddies in the thin boundary layer can be seen as pressure fluctuations. Unfortunately, these pressure fluctuations scatter most effectively at a trailing edge which is perpendicular to the flow. In other words they will be radiated as strong sound waves.

A serrated trailing edge eliminates the perpendicularity, making the sound radiation less effective and thus less noisy.

CHAPTER - 2

2.1 LITERATURE REVIEW

1) A Trailing-Edge Noise Model for Serrated Edges

Authors : BenshuaiLyu, Mahdi Azarpeyvand, Samuel Sinayoko

This paper is concerned with the development of a theoretical model for the prediction of the sound radiated by serrated trailing-edges. The broadband noise, induced by the interaction of boundary layer with the airfoil trailing-edge, known as the turbulent boundary layer trailing-edge noise, plays a significant role in the overall airframe noise. When a turbulent boundary layer convects past the trailing-edge, the unsteady pressure with a wavenumber in the hydrodynamic range is scattered into sound with the wavenumbers in the acoustic range, leading to noise radiated from trailing-edge. Both experiments and theories reveal that the radiated sound power varies with the characteristic flow velocity to the power of five, which is thus more efficient than that when the airfoil is absent at low Mach numbers. Howe proposed a theoretical model to predict the sound generated by the serrated trailing-edge of both the sinusoidal and sawtooth profiles. Investigated analytically the trailing-edge noise reduction using novel serrations, namely, sawtooth, sinusoidal, slitted, slitted-sawtooth and sawtooth-sinusoidal. The noise reduction performance is believed to depend on two main non dimensional parameters, $k_1 h$ and $k_1 l y_0(\omega) \sigma$. the condition that $k_1 h \gg 1$, that $|a_n|$ decreases as $k_1 h$ increases. Therefore, for a fixed serration geometry high frequencies are preferable in terms of noise reduction. This also means that the root-to-tip length determines the effective frequency range where sound reduction can be achieved

2) Airfoil self-Noise reduction by non-flat plate type T.E serration

Authors: Tze Pei Chong, Phillip Joseph, Mathieu Gruber

This paper aims to reduce the airfoil trailing edge self-noise by employing non-flat plate type trailing edge serrations. This configuration offers better structural strength and integrity, as well as a more straightforward manufacturing process compared to the conventional flat plate type serrations. To increase the effectiveness of the proposed serration geometry a hybrid configuration composed of a non-flat plate type trailing edge serration with woven-wire mesh screen is employed for the reduction of the narrowband vortex shedding noise experiment setup is wind tunnel which can achieve turbulence intensity as low as 0.1% and Mach number as high as 0.3, while maintaining a low background noise. The airfoil under investigation here was held by side plates and attached to the nozzle lips. In this study, airfoil self-noise pertaining to 15 to the jet axis was examined, which is larger than the “effective” angle due to downwash deflection of the incident flow by the airfoil and affects the pressure gradient distribution on the airfoil surfaces. Turbulent boundary layers were artificially generated by placing rough sandpaper near the leading edge on both of the suction and pressure surfaces. The wall pressure spectra differ considerably at the pressure surface. Without the use of the boundary layer trip, the wall pressure spectra are of comparatively lower level and less smooth than when tripping is applied. The boundary layer becomes turbulent when the trip is introduced as manifested by the considerably higher wall pressure spectral level with the characteristic high frequency decay rate of approximately f^{-5} . The relationship between the serration parameters such as k , u and $2h$ and the radiated broadband noise has been studied theoretically by Howe [6] whereby greatest noise reductions are predicted in the high frequency range, $xh/U \gg 1$ and for $u < 45$, for which noise reductions of $10 \log_{10}[1 + (4h/k)^2]$ dB are predicted. This implies that an 18 dB broadband noise reduction can theoretically be achieved by one of the serration configurations investigated in the current study for which $h/k \gg 2$. Gruber et al. has also found that the serration length ($2h$) must be greater than the trailing edge boundary layer thickness (i.e. $h/d > 0.5$) for significant noise reductions to occur. Of all the 36 different sawtooth serrated edges that were tested by Gruber et al., maximum noise reductions were limited to 7 dB. Gruber et al. also observed noise increases of up to 3 dB at high frequencies, which they attributed to the presence of small jets through the troughs of the serration. Therefore it is found that the non-flat plate serration not only reduces the broadband self-noise significantly, but also eliminates the high-frequency noise.

3) Experimental investigation of airfoil self-noise and turbulent wake reduction by the use of trailing edge serrations

Authors: Mathieu Gruber, Tze Pei Chong, Phillip Joseph

This paper shows the comparison of the measurements of the trailing edge self-noise reduction obtained using sawtooth serrations on a NACA651210 airfoil to the theoretical noise reduction predictions developed by Howe. This also focuses on the effect of the sawtooth serrated edges on steady and unsteady aerodynamics around the airfoil. The important aspect is looked carefully about the turbulence in the wake, which is of concern for airfoil turbulence interaction noise. An experimental study was conducted in the open jet wind tunnel which reveals noise reductions of up to 5 dB over a wide frequency range by the introduction of these trailing edge designs. This paper also presents the noise measurements for a range of jet speeds and sawtooth geometries. The airfoil is at 5 degree angle of attack and the boundary layer is made turbulent by tripping. In addition, measurements of the static pressure coefficient distribution along the chord of the airfoil are also reported. This is to allow the effects on lift to be observed. The drag coefficient, obtained from wake velocity measurements downstream of the airfoil, shows no significant change due to the introduction of serrations. Noise measurements are compared to the theory derived by Howe for serrated trailing edges. Large differences between the predicted and measured noise reductions are obtained. Experimental data show that irrelevant sources of noise in the high frequency range, possibly due to the generation of vorticity along the wetted edges of the trailing edge, reduces the potential noise reduction predicted by the theoretical model. Detailed hot wire measurements of the mean and turbulent flow in the boundary layer over a single sawtooth shows that serrated edges have a small effect on the development of the boundary layer in the free stream direction. Measurements of the free stream velocity in the airfoil wake at various locations downstream of the trailing edge reveal a small reduction of the turbulence in the wake for serrated edges. It also shows more the turbulent length scale in the far wake increases for the sharper sawtooth, while decreases for the large base sawtooth. This work is relevant to reducing the noise from aircraft engines, aircraft wings and wind turbines.

4) Trailing edge noise reduction using novel surface treatment

Authors: Abbas Afshari, Mahdi Azarpeyvand, Ali A. Dehghan, Mate Szoke

This paper deals with control technique of trailing edge noise radiation due to large coherent turbulent structure. Novel surface treatment is used to indicate the capability of proposed trailing edge technique of noise reduction. It is designed in such a manner that it can demonstrate the flow and noise measurement values carried out at different surface. The experiments were carried out in an open wind tunnel of Yazd University. In the present wind tunnel, the centrifugal forward blades types fan creates low broadband noise. Microphones dimension dimensions are 2.5mm in diameter, 2.5mm in height and with a circular sensing area of 0.8mm. Microphones are embedded in the flat plate under a pinhole of diameter 0.4mm. All pinholes are created by drilling the flat plate using the accurate drill machine. Due to the its thickness the pinhole can be created vertically and positioned microphones under the pinholes at the T.E. near the T.E the, microphones have been installed inside the flat plate parallel to the surface. Measurements were carried out at zero angle of attack for three different free stream velocities, $U_{\infty} = 10, 15, \text{ and } 20 \text{ m/s}$, corresponding to chord Reynolds numbers of $Rec = 3.87 \times 10^5, 5.8 \times 10^5, \text{ and } 7.73 \times 10^5$ respectively. As the turbulent flow field around the trailing edge is the source of trailing edge noise, velocity measurements near the trailing edge of the flat plate with and without surface treatments (baseline) are studied to gain insight into the mechanism by which fences affect flow structure. Results revealed that the surface treatment can significantly reduce the surface pressure fluctuations near the trailing edge, lessen the spanwise coherence and spanwise length-scale, and reduce the convection velocity of the turbulent structures. It has also been shown that the cross-correlation between the turbulent structures within the outer region of the boundary layer and the unsteady surface pressure can be significantly reduced.

5) Poro-Serrated Trailing-Edge Devices for Airfoil Self-Noise Reduction

Authors: Alexandros Vathylakis, Phillip Joseph, Tze Pei Chong

This paper represents the continuation of the works previously published in Chong et al. And used several non-flat plate serrated trailing edges for the reduction of airfoil self-noise. The overall noise performance of the non-flat plate trailing edge serration type can be improved by the concept of poro serrated used in the current work. The bluntness-induced vortex shedding tonal noise can be suppressed by filling the gaps between the adjacent members of the saw tooth by the use of porous metal, synthetic foams, or thin brush bundles. Most important, up to 7 dB turbulent boundary layer–trailing edge broadband noise reduction can simultaneously be achieved without compromising the aerodynamic performances in lift and drag. The poro-serrated trailing edges do not cause any noise increase throughout the frequency range investigated here. The reduction of the turbulent broadband noise is primarily caused by the serration effect, but under a condition that the saw tooth surface must be solid and nonporous. The primary role of the porous metal foams in a poro-serrated trailing edge is to suppress the vortex shedding tonal noise. However, an optimum selection of the porous material is also found to be able to further reduce the broadband noise level. The new serrated trailing-edge concept developed here has the potential to improve the industrial worthiness of the serration technology in achieving low noise radiation in fan and turbine blade.

6) Optimization of the poro-serrated trailing edges for airfoil broadband noise reduction

Authors: Tze Pei Chong, Elisa Dubois

This paper deals with number of poro serrated trailing edge devices in airfoil NACA0012. Poro serrated material has a good noise absorbing capability. So here poro serrated material is used between saw tooth trailing edge. The main objective of this work is to determine whether multiple mechanism on the broadband noise reduction can co-exist on the poro serrated trailing edge. Good broadband noise reduction be achieved at high frequency but vortex shedding tone at low frequency could not be completely suppressed at high velocity when saw tooth gaps are filled with porous material of low flow resistivity. When the saw tooth gaps are filled with porous material of high flow resistivity, no vortex shedding is present but the serration effect on the broadband noise reduction becomes less effective. At optimal choice of flow resistivity for a poro serrated configuration has been identified where it can surpass the conventional serrated trailing edge of same geometry by achieving a further 1.5 dB reduction in the broadband noise while completely suppressing the vortex shedding tone. A weakened turbulent boundary layer noise scattering at the poro serrated trailing edge is reflected by the lower turbulence intensity at the near wake centerline across the whole span wise wavelength of the saw tooth.

7) Numerical investigation of leading edge noise reduction on a rod-airfoil configuration using porous materials and serrations

Authors: Christopher Teruna, Francesco Avallone, Damiano Casalino, Daniele Ragni

A lattice-Boltzmann method has been employed to study the aeroacoustics and aerodynamics of airfoils equipped with leading edge treatments, namely the porous leading edge and leading edge serrations. The present study aims to identify the differences in noise reduction mechanisms between the two treatments. Within the context of turbomachinery applications, the airfoils undergo aerodynamic excitation due to the impingement of turbulent wake shed by an upstream rod. Two airfoil profiles are considered: NACA 0012 and NACA 5406; the latter mimics geometrical features and aerodynamic loading distribution of the outlet-guide vane in a turbofan test rig. Simulations are carried out at a freestream Mach number of 0.22, corresponding to Reynolds number based on the rod diameter of 48 000. The serrations are designed to follow a sinusoidal planform shape, whereas the porous leading edge is based on a Ni-Cr-Al metal-foam with homogeneous and isotropic properties. It is found that the porous leading edge attenuates noise by dampening surface pressure fluctuations due to the reduced blockage effect compared to the solid one. Differently, the leading edge serrations promote destructive interference of noise sources along the span. When applied against turbulent inflow with tonal characteristic, such as that induced by the impingement of Kàrmàn vortex street in the rod wake, the latter is more effective. On the other hand, both treatments are found to produce similar broadband noise reduction. When comparing aerodynamic performances, it is found that under a lifting condition, cross-flow is present through the porous material which results in lift reduction and drag increase. A serrated porous leading edge is then proposed to combine the benefits of the two leading edge treatments. This results in optimal noise reduction performances and lower aerodynamic penalty with respect to the fully porous leading edge

8) Design and Fabrication of Serrated Wing

Authors: Jitendra Singh, Devender Sharma

This paper is an investigation on different theories evolved for noise reduction due to serrated trailing edge of an airfoil. The major objective is to explain the most relevant theory for involved in airfoil trailing edge noise reduction. This paper present the result of comparison and affiliation between Howe's theory, Oerleman's theory , Gruber noise reduction theory at different Reynolds number. Limitation discovered comparing these theories for transforming flat plate aerofoil to serrated trailing edge aerofoil can be overcome by novel surface treatment and also with different non flat design comprised of poroserrated material

9) Numerical Investigation of Darrieus Wind Turbine with Slotted Airfoil Blades

Authors: Omar Sherif Mohamed, Ahmed Khaled Etman, Ahmed Ibrahim

The present investigation has highlighted the advantages of using slotted airfoil blade in Darrieus type VAWT. The performance of slotted airfoil turbine was examined using CFD tools. Initially, the slot concept was applied to static NACA0018 airfoil at different angles of attack to verify its advantages on flow separation at high angles of attack. Thus, the slot geometrical parameters were optimized based on static behavior results. Thereafter, the selected slotted airfoil configuration was applied to the Darrieus rotor. The effect of the slot was investigated from different perspectives. The major findings of the present work can be summarized as follows:

- The slotted airfoil effectively delays the boundary layer separation, where at high angles of attack the slot allows part of high pressure flow to pass into the suction side of the airfoil causing an acceleration in the flow due to the nozzle shape of the slot. Consequently, this minimizes the drag force and increases the lift force at high angles of attack. Hence, the slotted airfoil achieves similar glide ratio to baseline airfoil; however, the peak glide ratio covers wider range of angles of attack while using the slotted airfoil.
- The application of slotted airfoil blade to Darrieus type turbine significantly augments both the torque coefficient and the power coefficient at low TSR, while at high TSR the power coefficient of SA turbine notably decreases.
- The SA turbine has a maximum torque coefficient of 0.15 at $TSR = 2$, which is approximately 3 times that of the BL turbine at the same TSR.
- At $TSR = 2$ the SA turbine calculated power coefficient is 0.3 which is approximately 3 times that of the BL turbine at the same TSR. However, at $TSR = 3$ the SA turbine computed power coefficient is 0.19 which is 54% of that of the BL turbine.
- It can be concluded that the self-starting capability of SA turbine is far better than that of BL turbine. While the rotor is at rest with no rotation, the SA turbine shows higher torque coefficient at all azimuth positions. Finally, it is important to mention that future works, which aim to focus on wind tunnel experimental study of slotted airfoil Darrieus type turbine, will need to be undertaken

10) Flow around a NACA0018 airfoil with a cavity and its dynamical response to acoustic forcing

Authors: W. F. J. Olsman, J. F. H. Willems, A. Hirschberg, T. Colonius, R. R. Tieling

Trapping of vortices in a cavity has been explored in recent years as a drag reduction measure for thick airfoils. If, however, trapping fails, then oscillation of the cavity flow may couple with elastic vibration modes of the airfoil. To examine this scenario, the effect of small amplitude vertical motion on the oscillation of the shear layer above the cavity is studied by acoustic forcing simulating a vertical translation of a modified NACA0018 profile. At low Reynolds numbers based on the chord ($O(10^4)$), natural instability modes of this shear layer are observed for Strouhal numbers based on the cavity width of order unity. Acoustic forcing sufficiently close to the natural instability frequency induces a strong non-linear response due to lock-in of the shear layer. At higher Reynolds numbers (above 10^5) for Strouhal number 0.6 or lower, no natural instabilities of the shear layer and only a linear response to forcing were observed. The dynamical pressure difference across the airfoil is then dominated by added mass effects, as was confirmed by numerical simulations

CHAPTER - 3

3.1 Objective and scope:

Aim:

The objective of the present study is to evaluate the influence of blade shape modification 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 set up of the wind turbine by establishing 3D CFD models of wind turbine blade with the varying as follows

Blade length as 0.660m and Rotor diameter as 0.526m which is initially fixed in the Reference model for the following three cases;

Slot holes with Leading edge serration : By applying the slot holes and serration in the leading edge of the blade and maintaining all the other dimensions of the model constant.

Slot holes with Trailing edge serration : By applying the slot holes and serration in the leading edge of the blade and maintaining all the other dimensions of the model constant

Slot holes with Leading and Trailing edge serrations : By applying the slot holes and serration in both leading and trailing edge of the blade and maintaining all the other dimensions of the model constant

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.

3.2 Overview of this Report:

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

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

5.1 3 MODELING OF VERTICAL AXIS WIND TURBINE BLADES

WITH VARIATIONS:

The coordinates are provided for the development of the NACA 0018 airfoil wind turbine blade model from airfoil tools website. For design purposes, the blade can be seen as a combination of three separate variations namely, slot holes with leading edge serration, slot holes with trailing edge serration and the slot holes with leading and trailing edge serrations.

NACA 0018 airfoil coordinates are generated in CATIA by using view macros in Microsoft excel according to the geometry of the blade is created to complete the model.

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

Airfoil used : NACA 0018

Blade length : 0.660 m

Rotor diameter: 0.526 m

Rotor radius : 0.263 m

Requirements:

Software Used : CATIA V5

Processor : Intel Core i3

RAM : 4GB

Operating System : Windows 10

Features Used:

EXTRUDE : This feature is used to extrude the length of the blade

ROTATE : This option is used to set the wind turbine blade angle 0 deg into +5 deg.

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

SLOT : This feature is used create elliptical slot holes.

POCKET : This feature is used to create leading and trailing edge serration

SLOT HOLES:

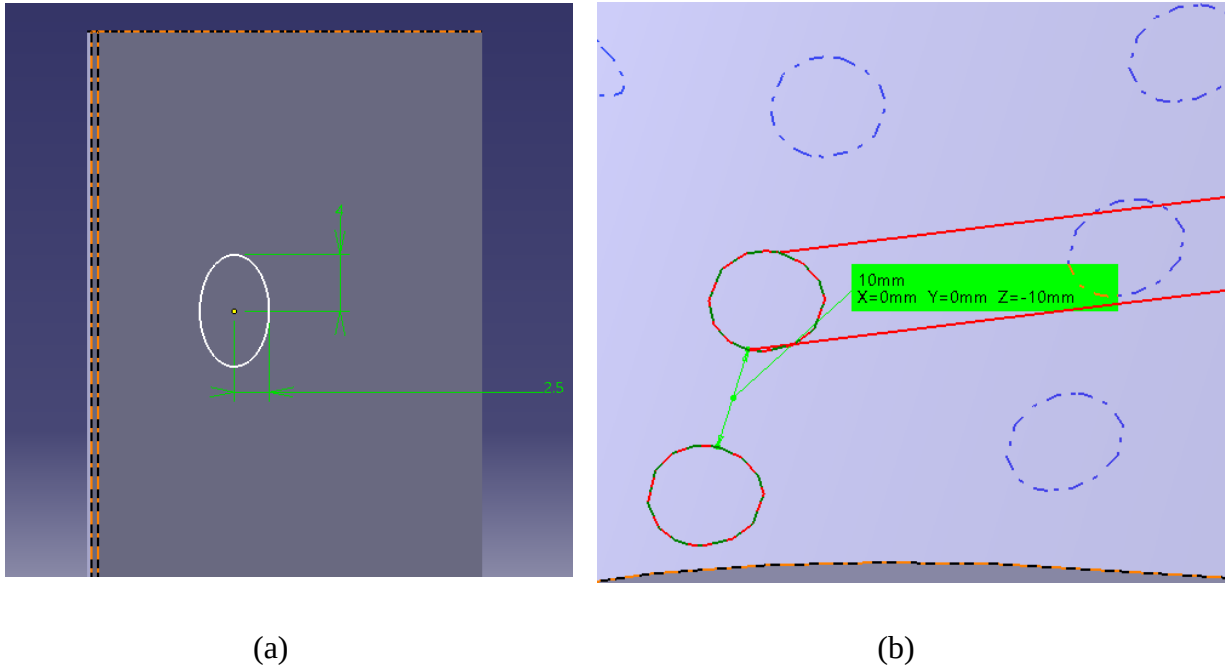


Fig.15: (a) Dimensions of elliptical hole

(b) Pitch between elliptical holes

Created inclined elliptical holes started near leading edge and ended at trailing edge of turbine blade using slot feature in CATIA software.

These slot holes are used to control the airflow and reduce the blade weight.

The slot hole geometry parameters are

Hole Type : Elliptical

Inclination Angle : - 15 deg

Length (Major axis distance) : 8mm

Width (Minor axis distance) : 5mm

5.2 LEADING EDGE SERRATION:

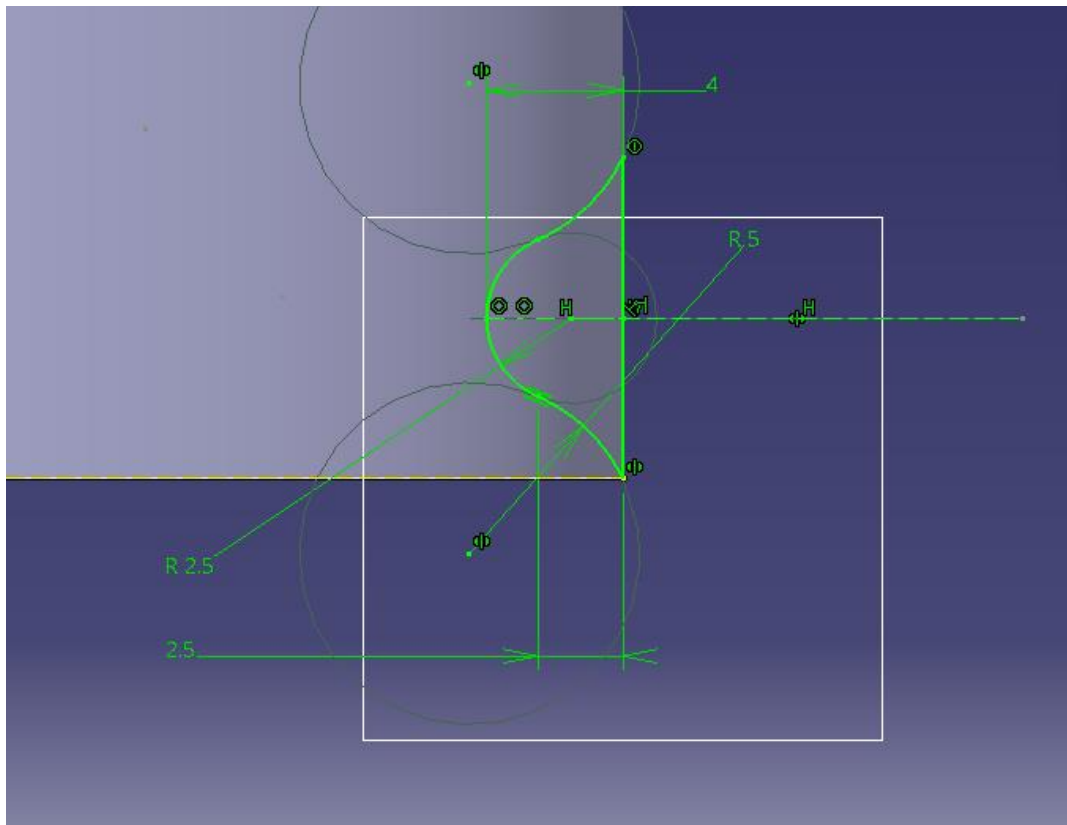


Fig. 16: Dimensions of Sinusoidal wave profile in leading edge of turbine blade

Sinusoidal wave cut profile is created in leading edge and near the elliptical slot holes. Circles with tangent constraints is used to make this profile at sketching in CATIA software

Geometry parameters of sinusoidal wave profile is

Length : 10mm

Width : 4mm

Fillet radius : 5mm

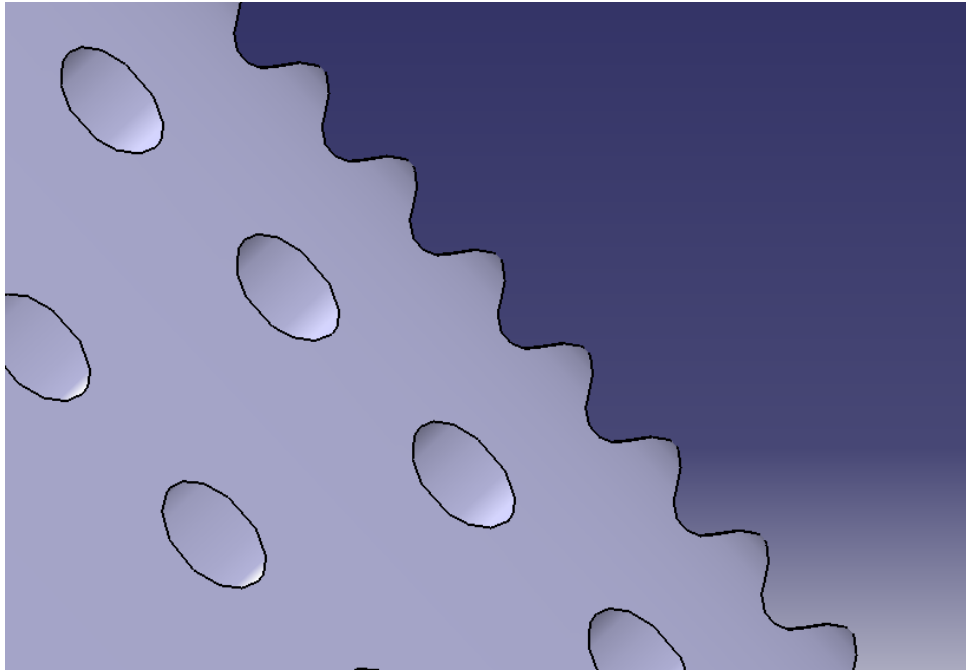


Fig .17 : Sinusoidal wave cut pattern in leading edge of turbine blade

Sinusoidal wave pattern cuts were created near leading edge using rectangular pattern feature at part modeling in CATIA software. This type of sinusoidal wave cut patternis used to control the dynamic flow separation at low tip speed ratios and reduce the weight of the blade.

5.3 TRAILING EDGE SERRATION :

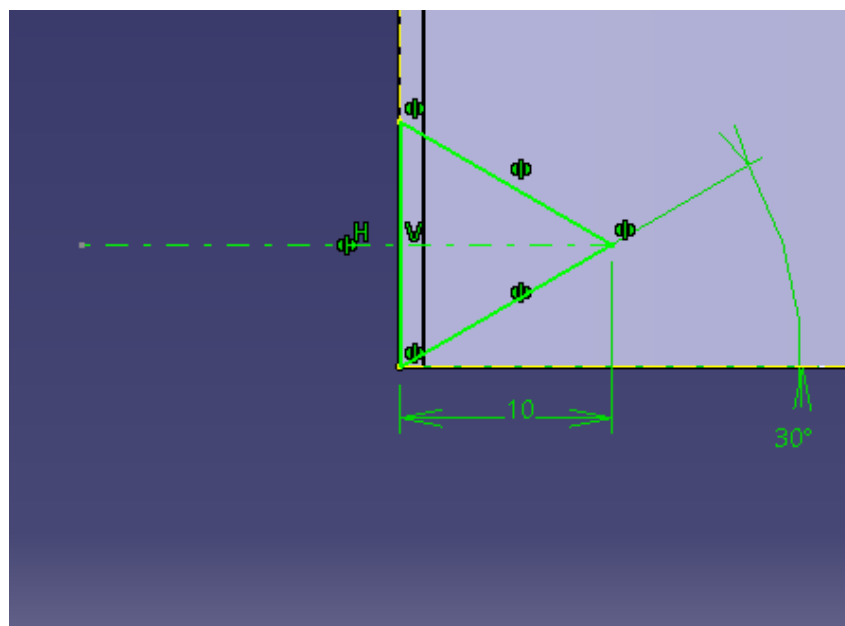


Fig.18: Dimensions of symmetrical triangle profile in trailing edge of turbine blade

Symmetrical triangle cut profile is created in trailing edge of turbine blade. Line option is used to make this profile at sketching in CATIA software

Geometry parameters of trailing edge serration is

Slot type : Triangle

Each side : 10mm

Side angle : 30 deg

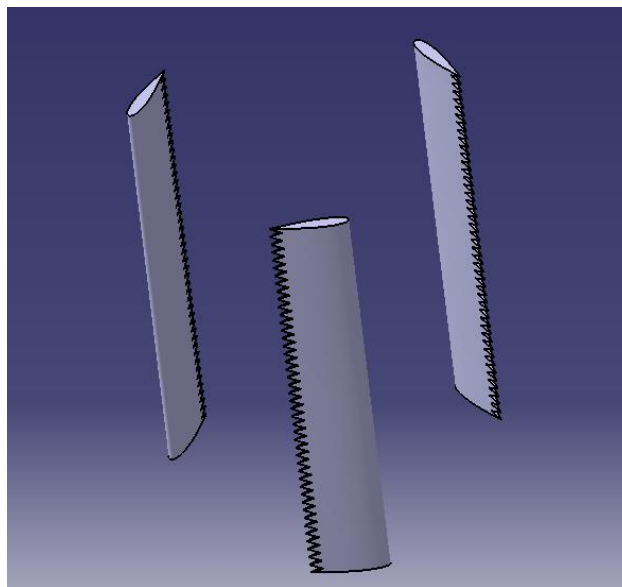


Fig. 19 : Symmetrical triangle cut pattern in trailing edge of turbine blades

Symmetrical triangle cut pattern in trailing edge of turbine blade is created by using rectangular pattern at part modeling in CATIA software. This type of cut pattern is used to eliminate the perpendicularity, making the sound radiation less effective and thus less noisy.

5.4 SLOT HOLES WITH LEADING AND TRAILING EDGE SERRATION :

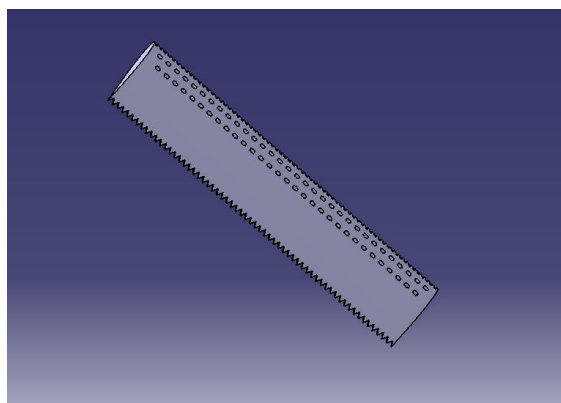


Fig. 20 : Elliptical slot holes with leading and trailing edge serration

In this type, Slot holes with both leading and trailing edge serrations are presented. Slot and rectangular pattern features are used to create elliptical slot holes and serrations. This blade design, controldynamic flow separation at low tip speed ratios and noise. Blade weight is smaller compared with other types of blade designs.

CHAPTER - 6

COMPUTATIONAL DOMAIN AND NUMERICAL 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.

6.2) Boundary creation:

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

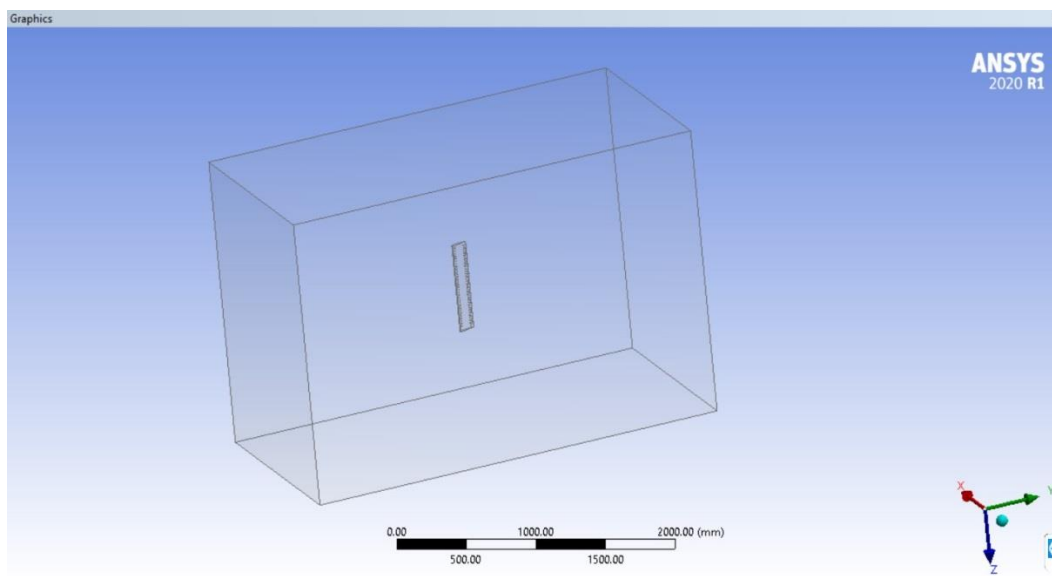


Fig.21: 3D Turbine blade model in 3D boundary

The geometrical specifications of the boundary are

Length of boundary : 8000mm

Width of boundary : 4000mm

Height of boundary :6000mm

6.3) Meshing:

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

Face Mesh:

For the carrying out mesh, faces were created by splitting the entire domain into individual parts. Then, the faces are 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

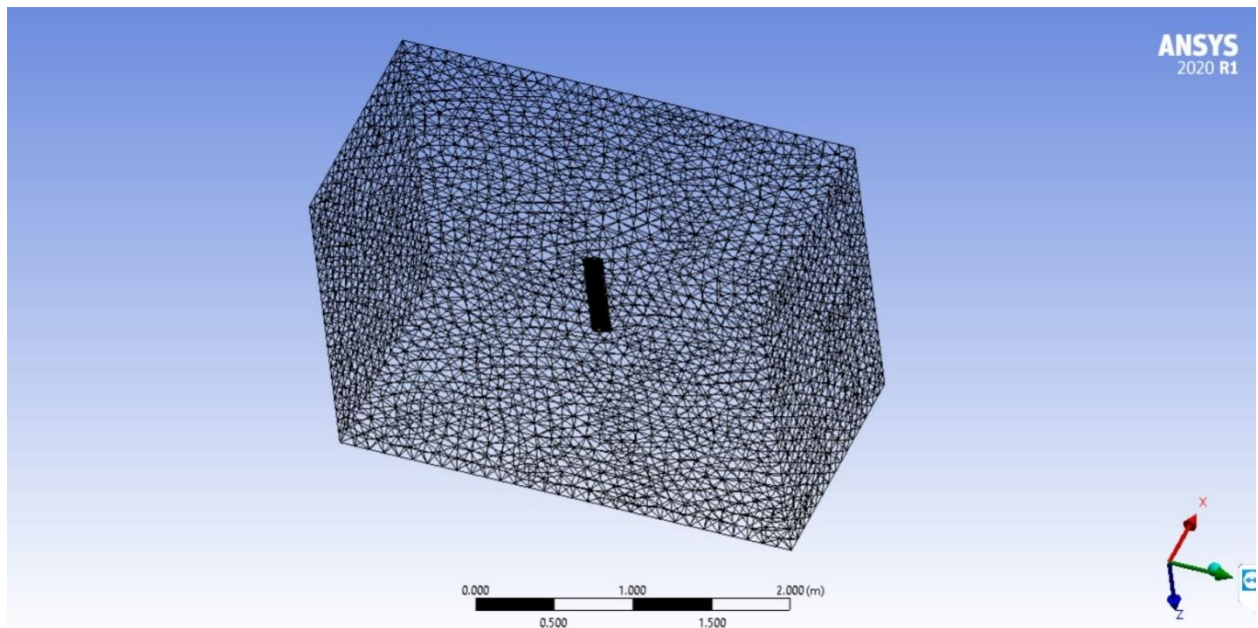


Fig.22: Coarse mesh on Boundary and Fine mesh on turbine blade

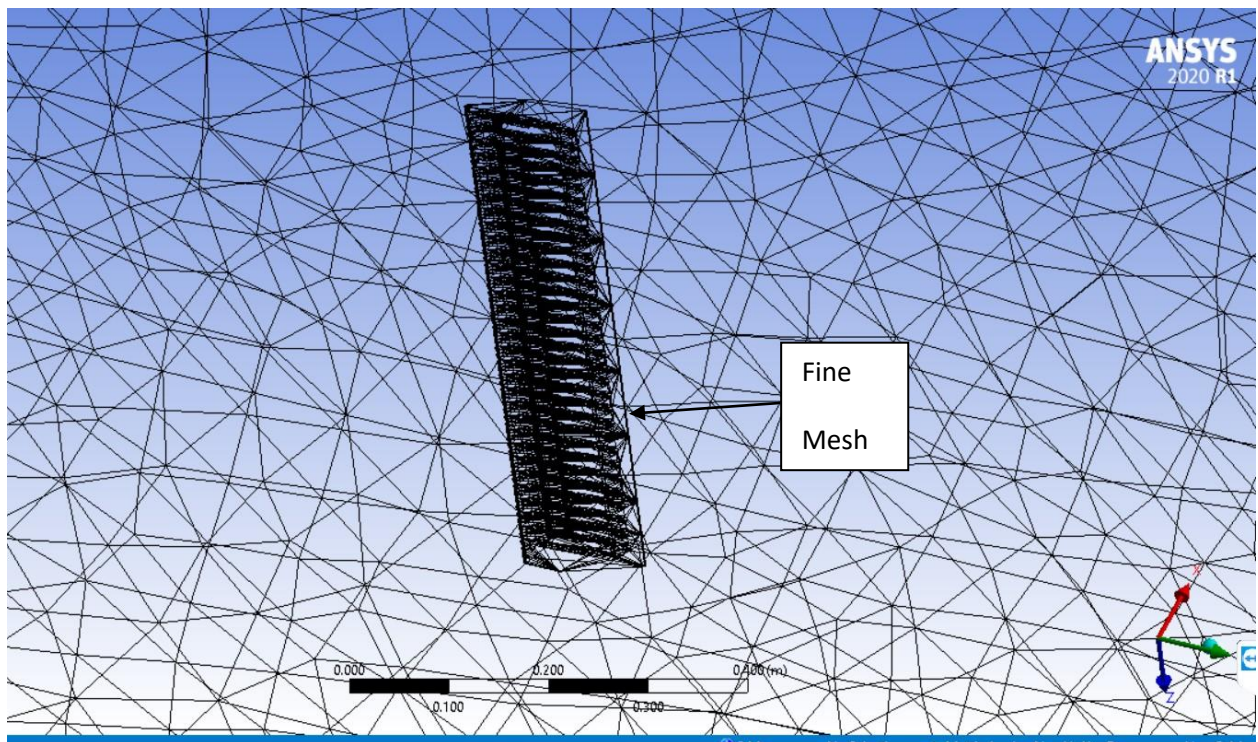


Fig.23: Enlarged image of mesh around the turbine blade

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

6.4.1) Velocity Inlet Boundary Conditions:

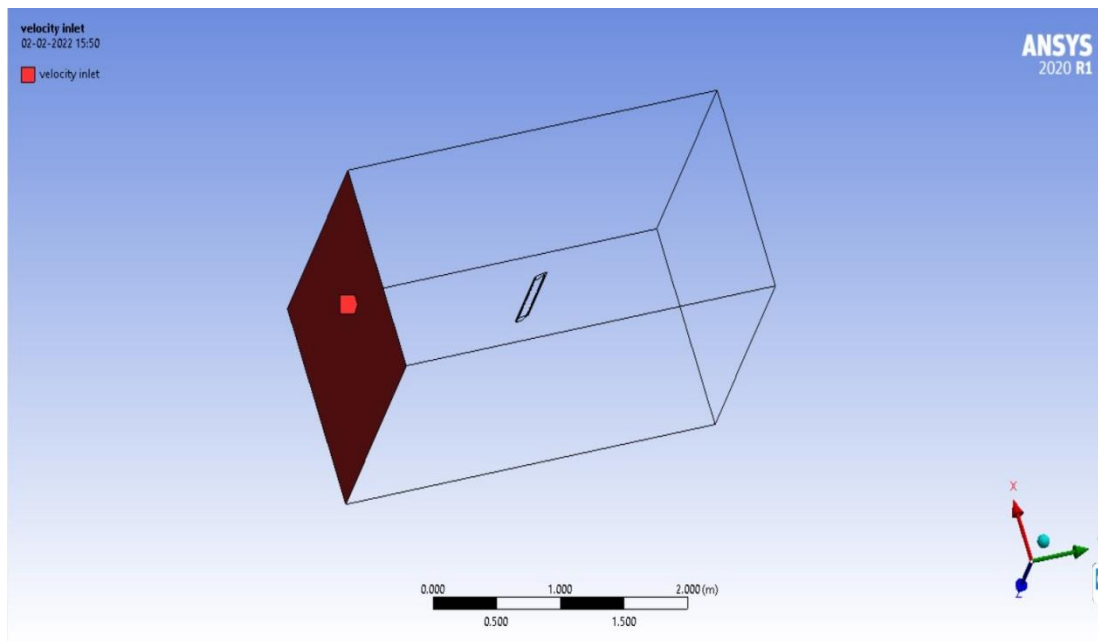


Fig. 24: Velocity inlet of the computational domain

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, at 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.

6.4.2) Pressure Outlet Boundary Conditions:

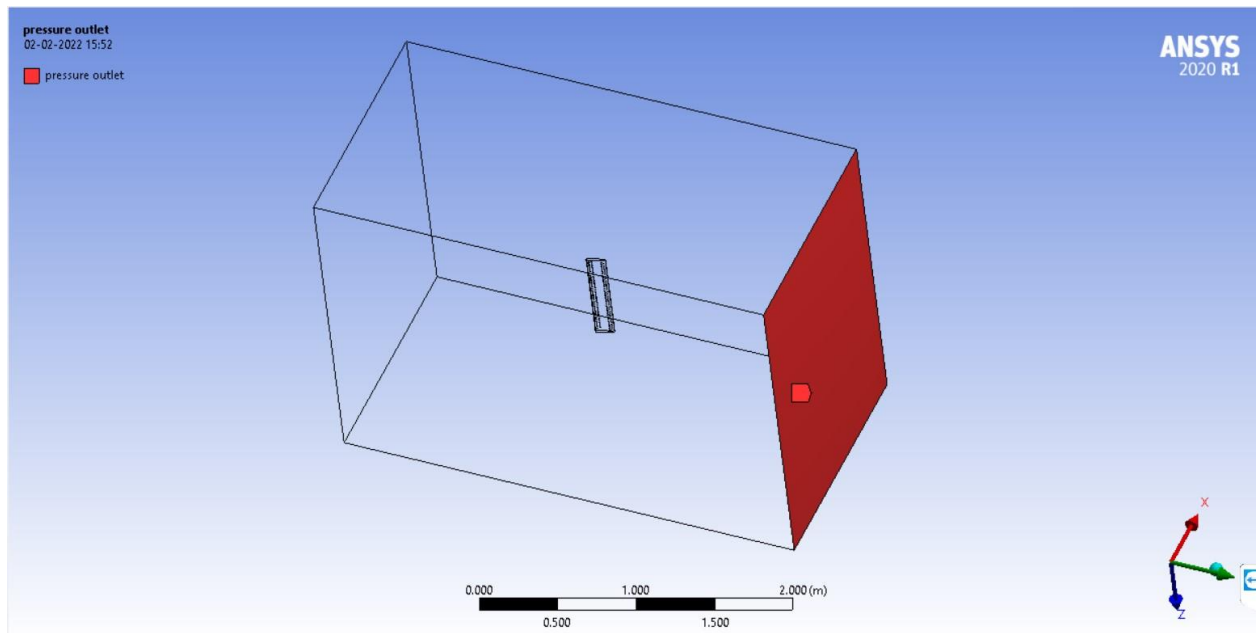


Fig.25 : Pressure outlet of the computational domain

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.

6.4.3) Wall Boundary Conditions:

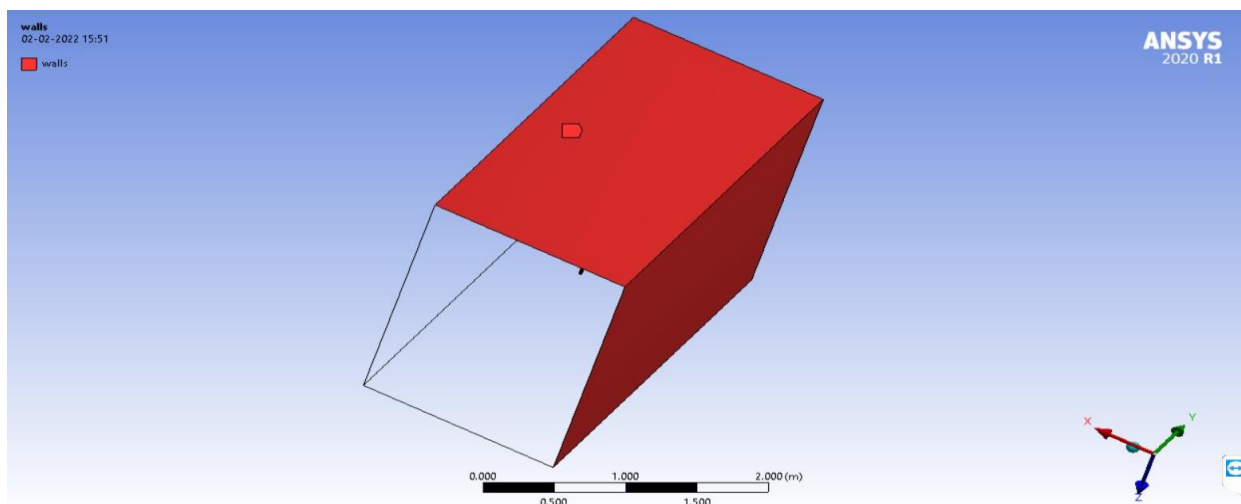


Fig.26: Walls of the computational domain

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

7.1 NUMERICAL SIMULATION USING FLUENT 6.3:

7.1.1 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

7.1.2 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.27: Grid check in Fluent software

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

7.1.3 Solver and Material Selection and Operating Condition

Defining:

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

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

7.1.4 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

7.1.5 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

7.1.6 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 x 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.

7.1.7 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:

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 dimension set up of the shrouded wind turbine using 3D CFD models of wind turbine blades with varying the following shapes as follows

The parameter variations done are as follows.

Slot holes with Leading edge serration:By varying the shape of the blade in the leading edge portion and maintaining all the other dimensions of the model constant.

Slot holes with Trailing edge serration:By varying the shape of the blade in the trailing edge portion and maintaining all the other dimensions of the model constant.

Slot holes with Leading and Trailing edge serration:By varying the shape of the blade in both leading and trailing edge portions and maintaining all the other dimensions of the model constant.

8.1 Observations for Normal blade (Reference Model):

The observation obtained for the Reference model, that is for the Normal 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 (0 1 0)							
Forces (n)				Coefficients			
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total	
wall-slot_holes_with_l.s_and_t.s	0.60883819	0.099003483	0.70784167	0.0099402154	0.0016163834	0.011556599	
Net	0.60883819	0.099003483	0.70784167	0.0099402154	0.0016163834	0.011556599	

Fig.28 :Print console of Lift force and Coefficient of lift values

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

Fig.29:Print console of Drag force and Coefficient of drag values

Lift Force = 7.078417 N

Drag Force = 3.315154 N

Coefficient of Lift = 0.11556599

Coefficient of Drag = 0.054124964

Aerodynamic efficiency (η)= Lift Force / Drag force = 2.135

Cl/Cd = 2.135

b)Velocity observation:

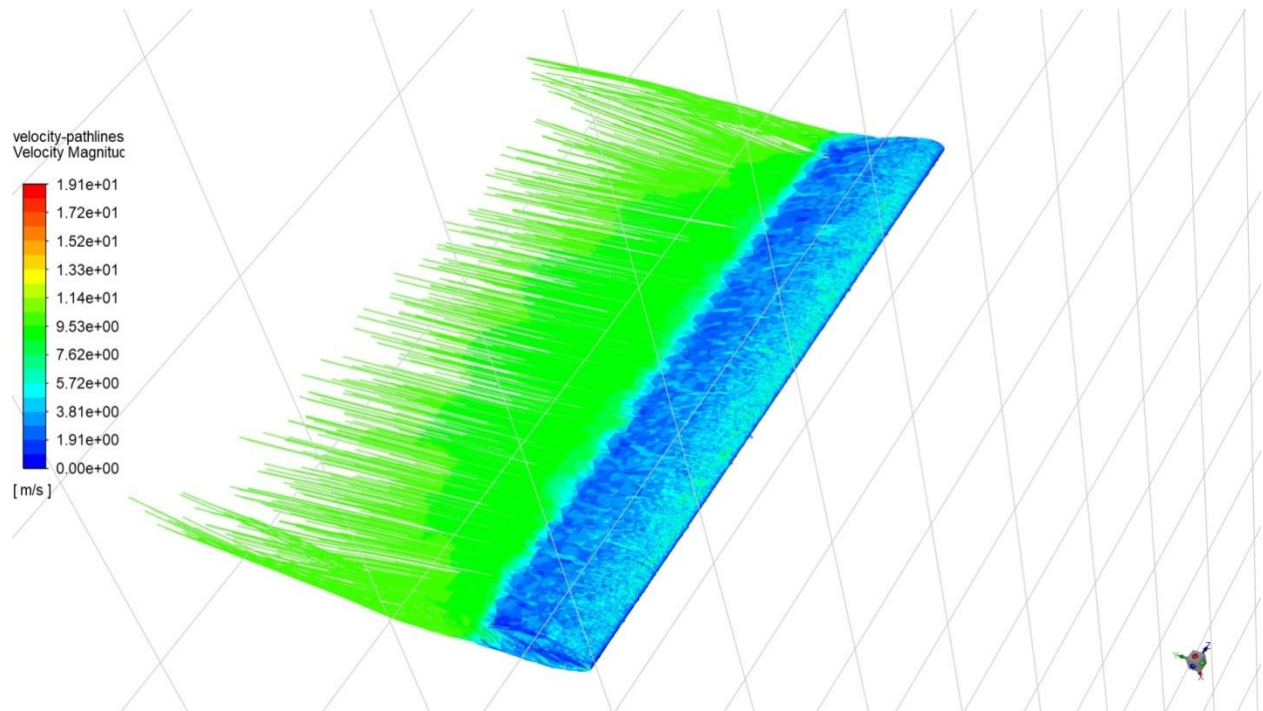


Fig. 30: 3D Contour of Velocity pathlines for the Normal Blade model

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 19.10 m/s. The velocity is maximum at the trailing edge portion. The velocity behind the leading edge limit is 0 – 5.72 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

c)Pressure observation:

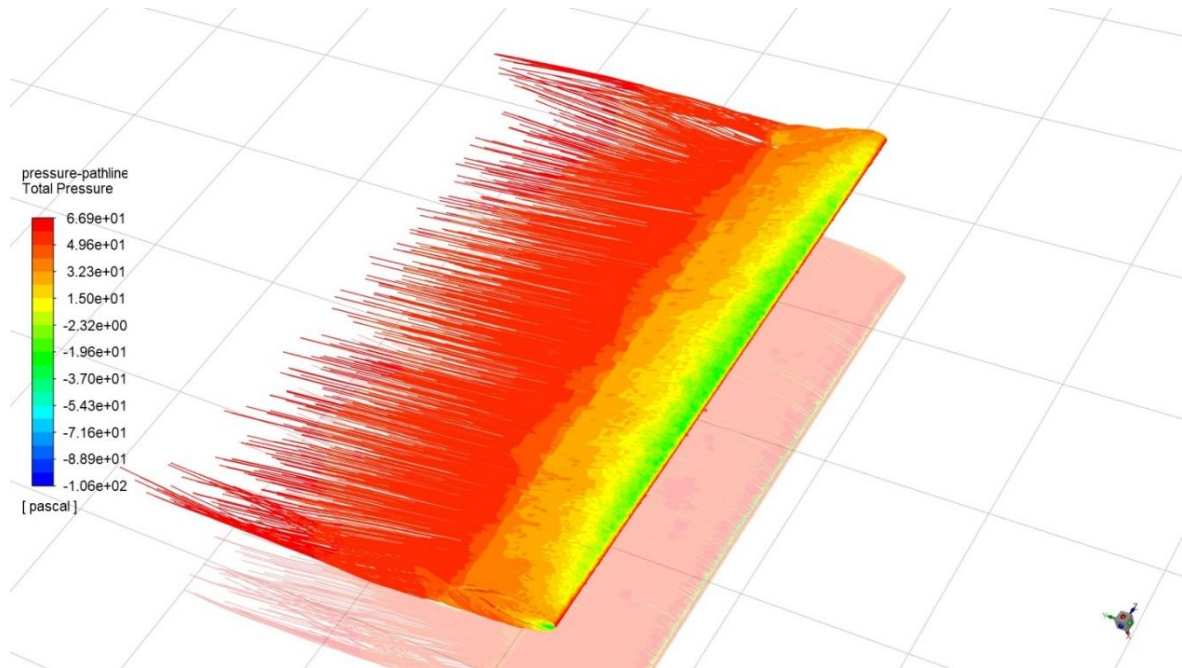


Fig. 31: 3D Contour of pressurepathlines for the Normal Blade model

The maximum pressure reached in this case is observed to be 66.90 Pa. The pressure is maximum at both leading and trailing edge portion. The pressure behind the leading edge limit is 37.00 – 66.90 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

d) Turbulence intensity observation:

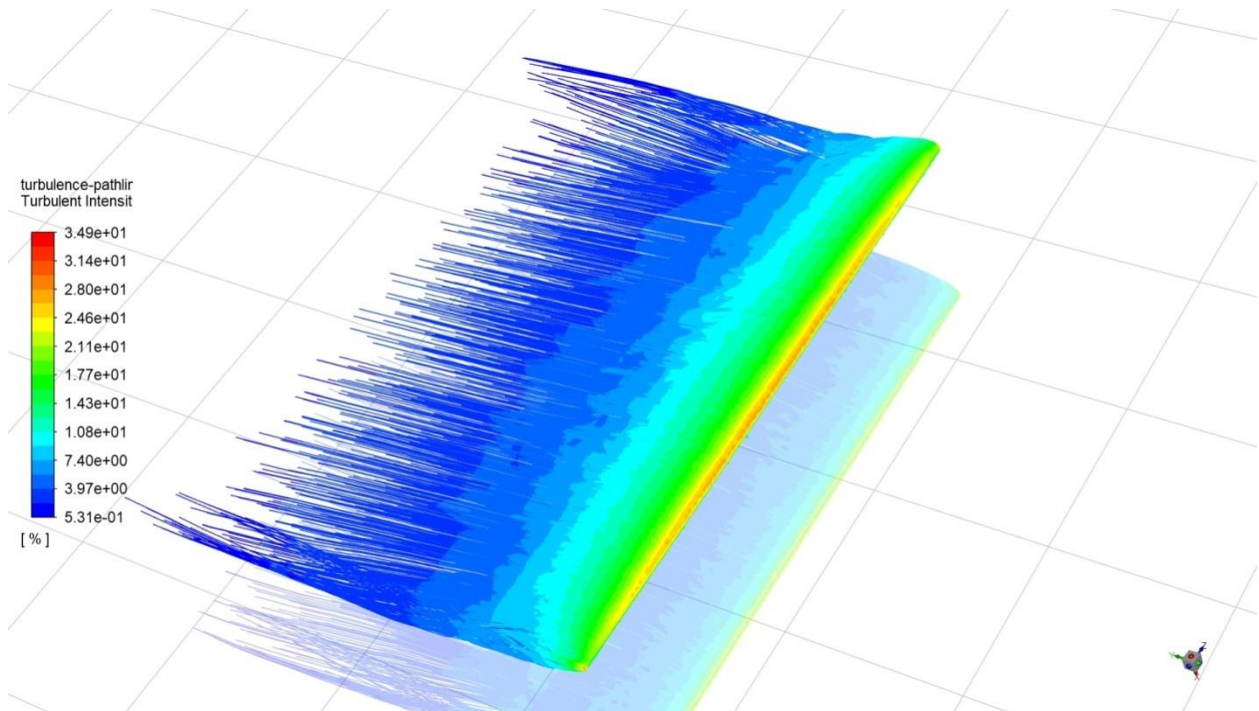


Fig.32:3D Contour of Turbulence intensity pathlines for the Normal Blade model

In Fig.32, visualization of the turbulence intensity pathlines that shows the changes in turbulence intensity level at different areas in the flow field using different colors.

8.2) SLOT HOLES WITH LEADING EDGE SERRATION RESULTS :

The observation obtained for the blade model with slot holes and serration at leading 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:

lift-force		cl	
Lift	(n)	Cl	()
wall-slot_holes_with_1.s	3.6784526	wall-slot_holes_with_1.s	0.060056369

Fig.33: Print console of Lift force and Coefficient of lift values

drag-force		cd	
Drag	(n)	Cd	()
wall-slot_holes_with_1.s	0.98696965	wall-slot_holes_with_1.s	0.01611379

Fig.34: Print console of Drag force and Coefficient of drag values

Lift Force = 3.6784526 N

Drag Force = 0.9869696 N

Coefficient of Lift = 0.060056369

Coefficient of Drag = 0.01611379

Aerodynamic efficiency (η) = Lift Force / Drag force = 3.7270

Cl/Cd = 3.7270

b)Velocity observation:

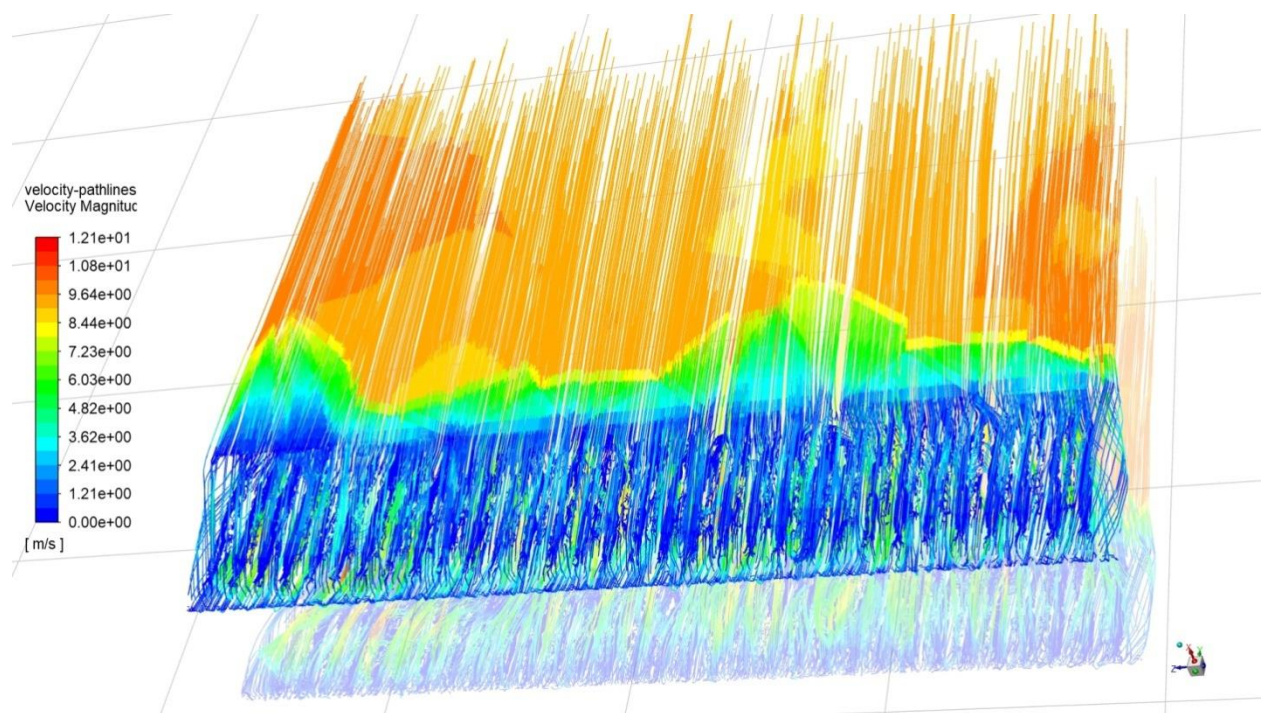


Fig.35: 3D Contour of Velocity pathlines for the Normal Blade model

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 12.10 m/s. The velocity is maximum at the flow after the trailing edge portion. The velocity behind the leading edge limit is 0 – 7.23 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

C)Pressure observation:

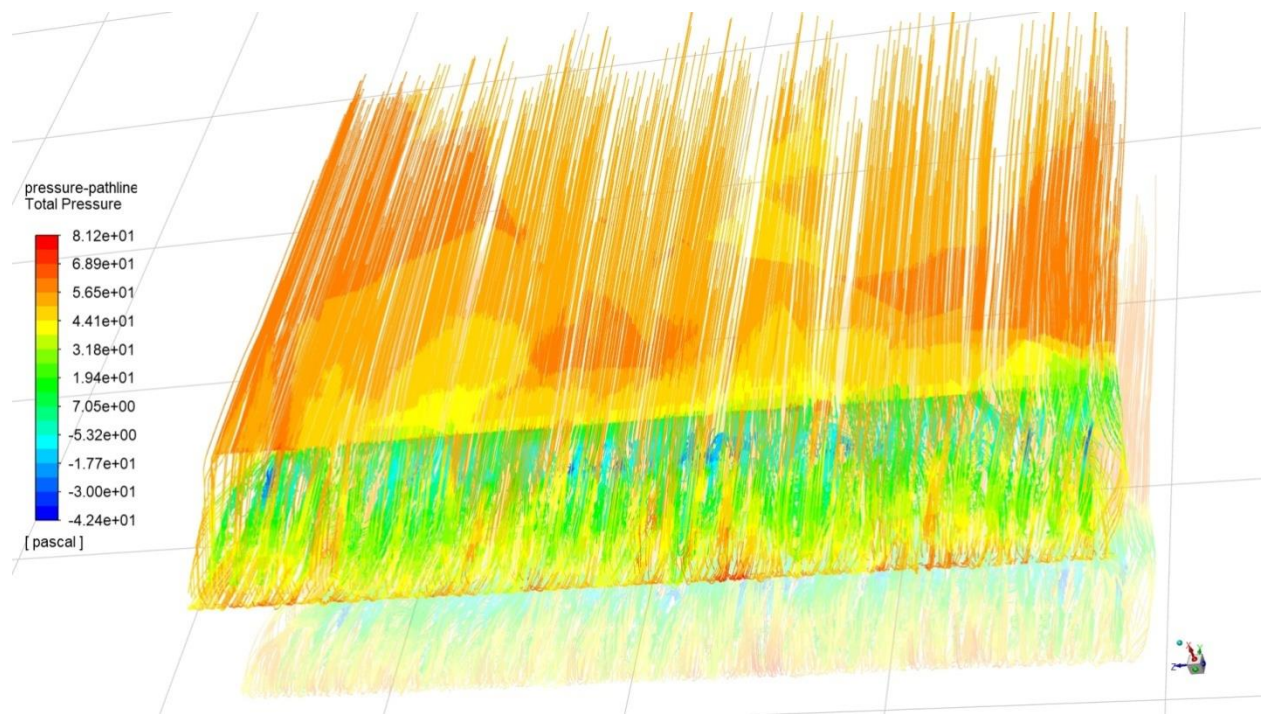


Fig. 36: 3D Contour of pressurepathlines for the Normal Blade model

The maximum pressure reached in this case is observed to be 81.20 Pa. The pressure is maximum at both leading and trailing edge portion. The pressure behind the leading edge limit is -42.40 – 68.90 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

d) Turbulence intensity observation:

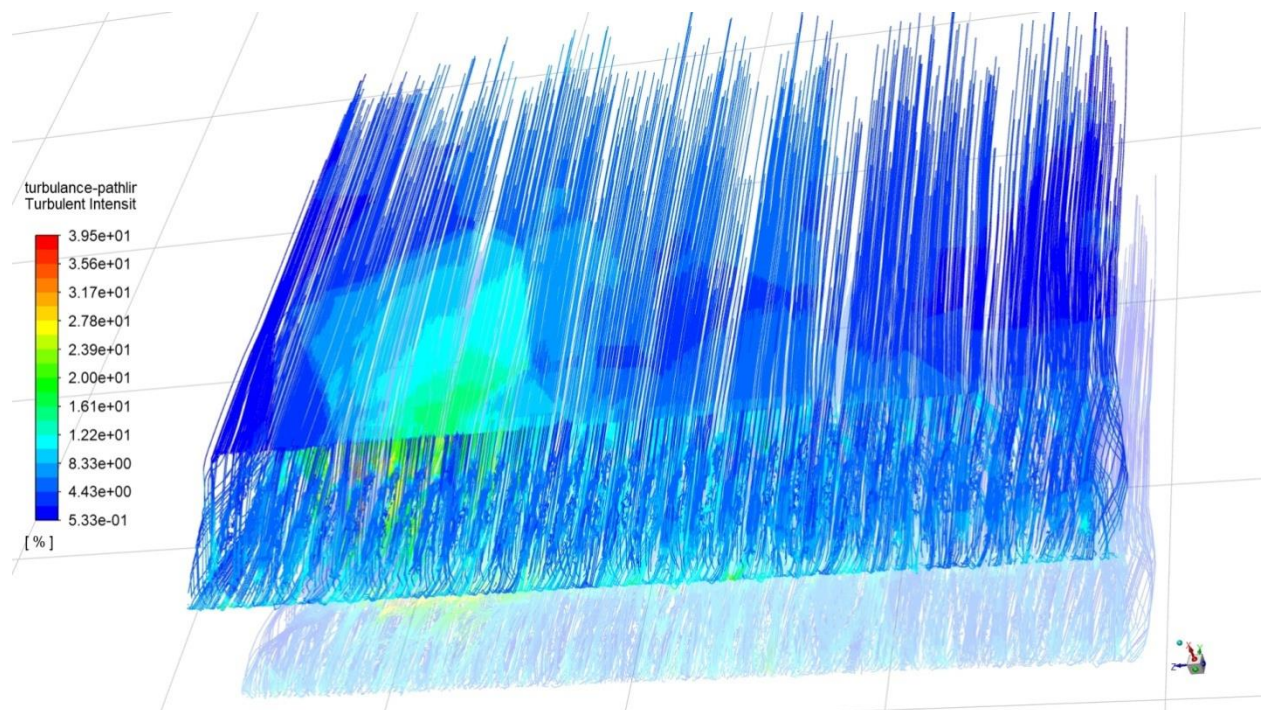


Fig.37:3D Contour of Turbulence intensity pathlines for the Normal Blade model

In Fig.37, visualization of the turbulence intensity pathlines that shows the changes in turbulence level at different areas in the flow field using different colors.

8.3) SLOT HOLES WITH TRAILING EDGE SERRATION 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 :

lift-force		cl	
Lift	(n)	Cl	()
wall-slot_holes_with_1.s	3.6523124	wall-slot_holes_with_1.s	0.059629591

Fig.38 :Print console of Lift force and Coefficient of lift values

drag-force		cd	
Drag	(n)	Cd	()
wall-slot_holes_with_1.s	1.0108856	wall-slot_holes_with_1.s	0.016504255

Fig.39: Print console of Drag force and Coefficient of drag values

Lift Force = 3.6523124 N

Drag Force = 1.0108856 N

Coefficient of Lift = 0.059629591

Coefficient of Drag = 0.016504255

Aerodynamic efficiency (η)= Lift Force / Drag force = 3.6130

Cl/Cd = 3.6130

b) Velocity observation :

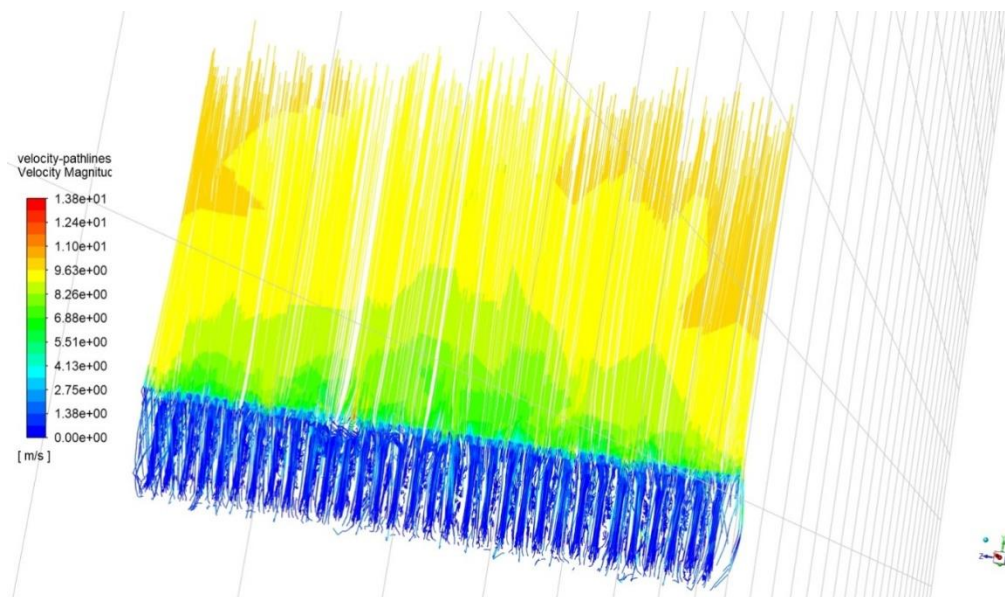


Fig. 40 :3D Contour of Velocity pathlines for the blade with trailing edge serration

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 13.80 m/s. The velocity is maximum at the flow after the trailing edge portion. The velocity behind the trailing edge limit is 0 – 8.26 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

c) Pressure observation :

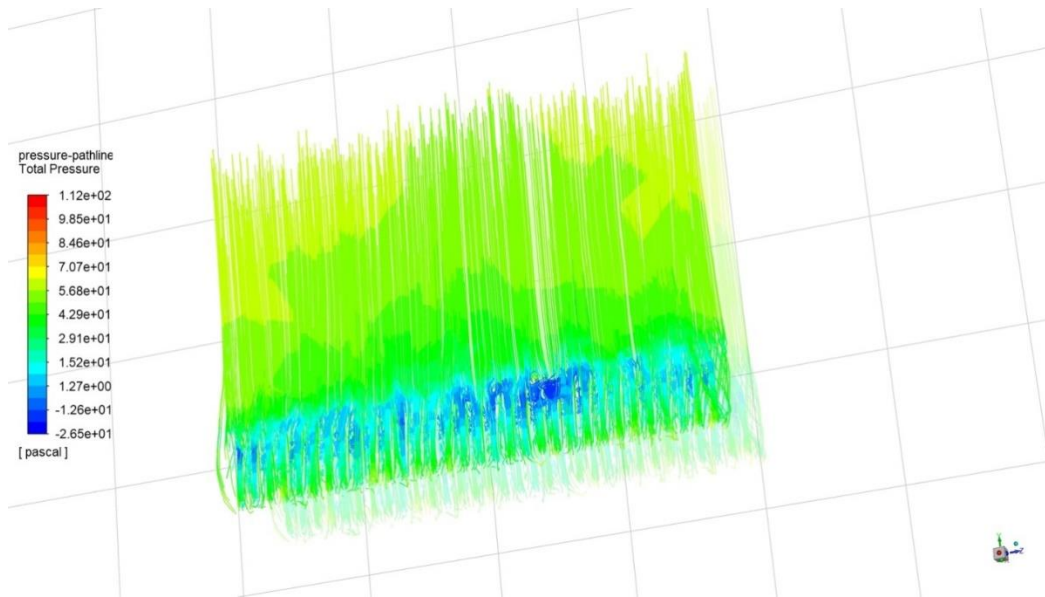


Fig.41:3D Contour of pressure pathlines for the blade with trailing edge serration

The maximum pressure reached in this case is observed to be 112.00 Pa. The pressure is average at both leading and trailing edge portion. The pressure behind the trailing edge limit is -26.50 – 56.80 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

d) Turbulence intensity observation:

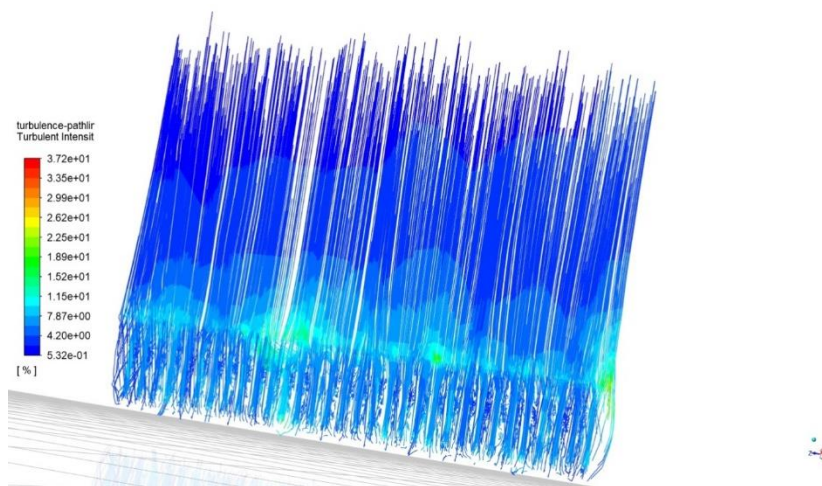


Fig.42:3D Contour of Turbulence intensity pathlines for the Blade with trailing edge serration

In Fig.42, 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) SLOT HOLES WITH LEADING AND TRAILING EDGE SERRATION RESULTS:

The observation obtained for the blade model with slot holes and serration at both leading and trailing edges , 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 :

lift		cl	
-----		-----	
Lift	(n)	Cl	()
-----	-----	-----	-----
wall-normal_blade	5.126474	wall-normal_blade	0.083697534

Fig.43:Print console of Lift force and Coefficient of lift values

drag-force		cd	
-----		-----	
Drag	(n)	Cd	()
-----	-----	-----	-----
wall-normal_blade	0.66589615	wall-normal_blade	0.010871774

Fig.44:Print console of Drag force and Coefficient of drag values

Lift Force = 5.126474 N

Drag Force = 0.665896 N

Coefficient of Lift (Cl) = 0.083697534

Coefficient of Drag (Cd) = 0.010871774

Aerodynamic efficiency (η)= Lift Force / Drag force = 7.6986

Cl/Cd = 7.6986

b)Velocity observation:

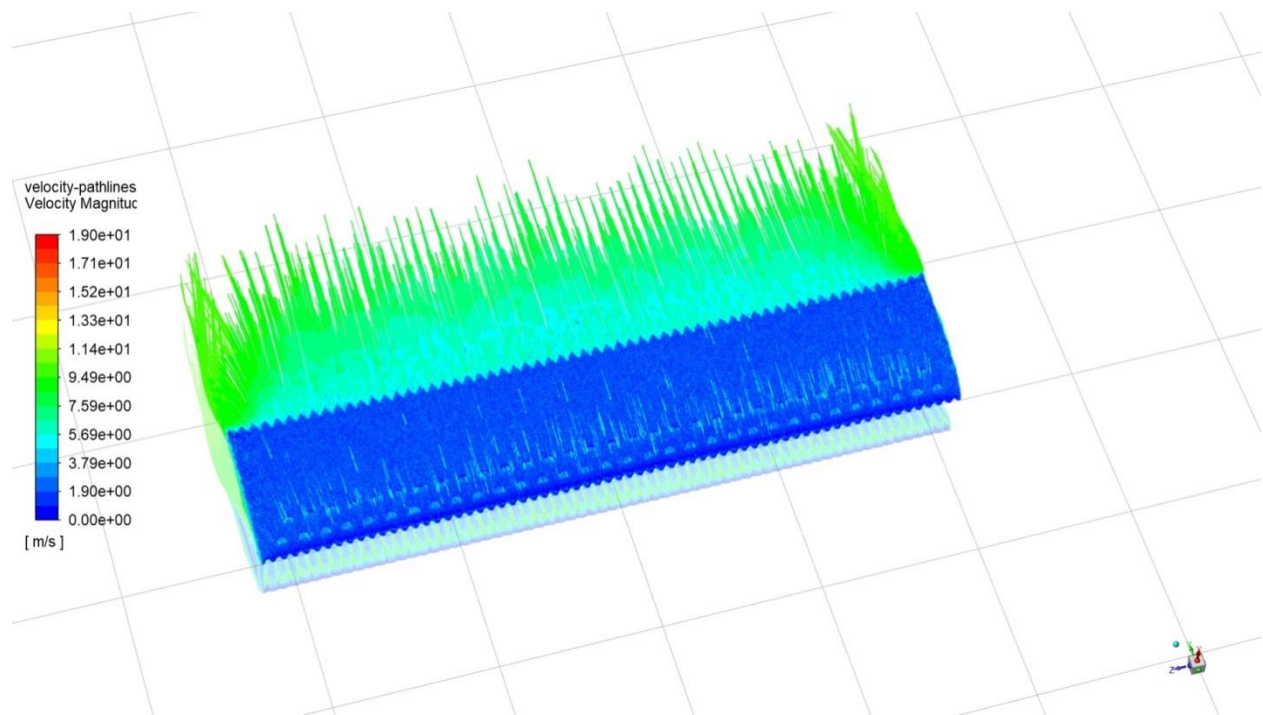


Fig.45:3D Contour of Velocity pathlines for the blade model with Slot holes with leading and trailing edge serration

The inlet velocity given is 10 m/s. The maximum velocity reached in this case is observed to be 11.40 m/s. The velocity is maximum at the flow after the trailing edge portion. The velocity behind the leading edge limit is 0 – 5.69 m/s and the velocity behind the trailing edge limit is 0 – 11.40 m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

c)Pressure observation:

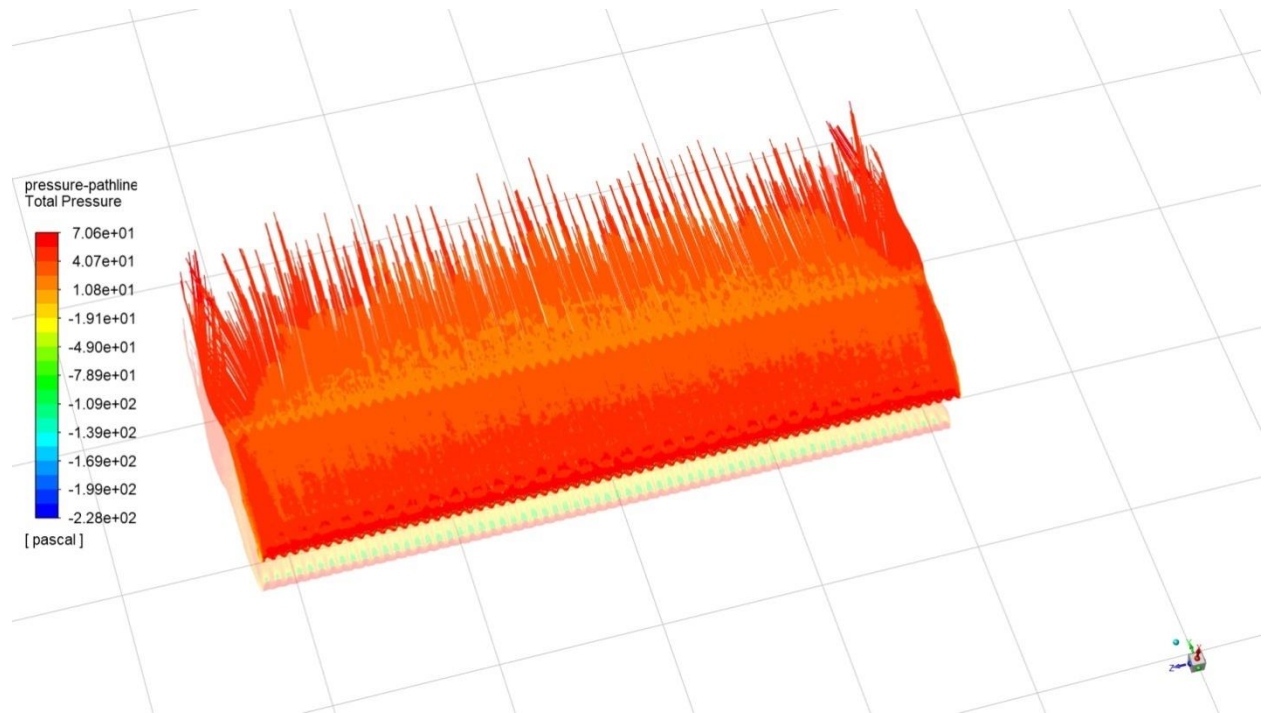


Fig.46:3D Contour of Pressurepathlines for the blade model with Slot holes with leading and trailing edge serraion

The maximum pressure reached in this case is observed to be 70.60 Pa. The pressure is maximum at both leading and trailing edge portion. The pressure behind the leading edge limit is $-109.00 - 70.60$ Pa. The pressure behind the trailing edge limit is $10.80 - 70.60$ m/s due to the vortices formation owing to the flow separation. Figure give a picture of this 3d contour.

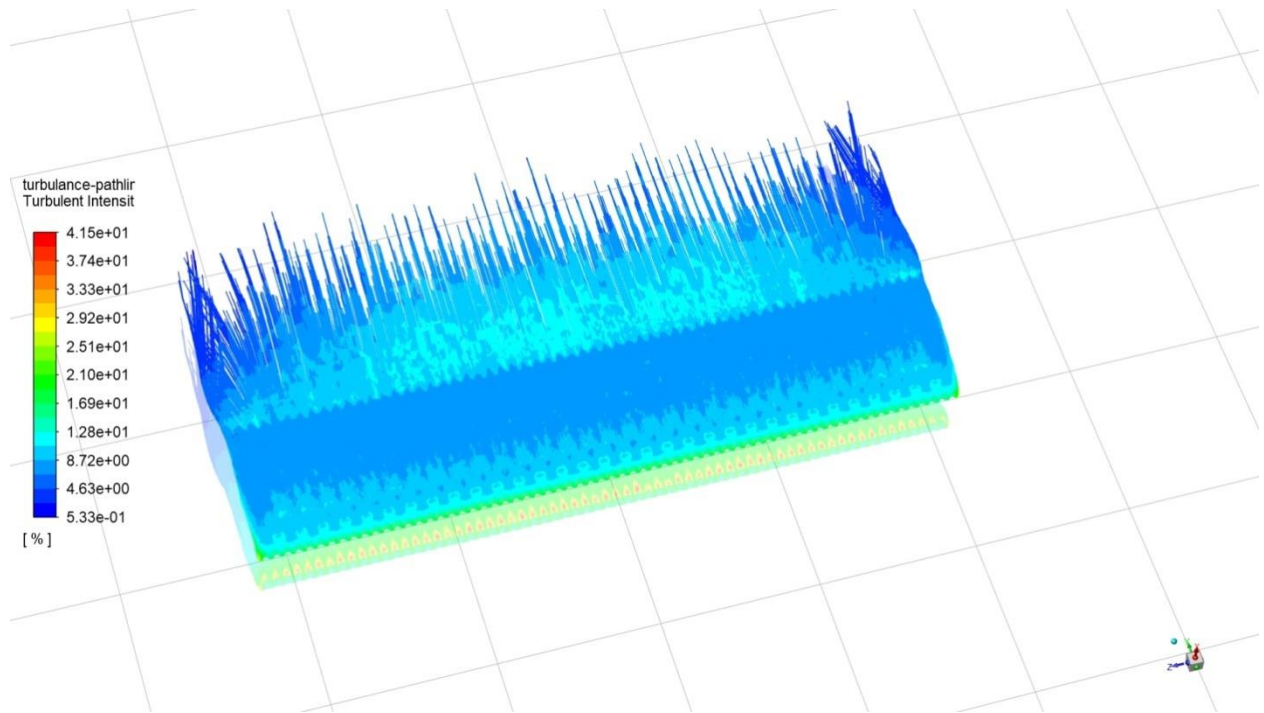


Fig.47:3D Contour of Turbulence intensity pathlines for the blade model with Slot holes with leading and trailing edge serraion

In Fig.47, 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 Comparison table for aerodynamic forces and aerodynamic efficiency values of various blade designs :

Blade Type	Lift (N)	Drag (N)	Cl	Cd	Aerodynamic efficiency(η)	Cl/Cd
Normal Blade	7.0784213	3.315154	0.11556599	0.054124964	2.1354	2.1354
Slot holes with leading edge serration	3.6784526	0.9869696	0.060056369	0.01611379	3.7270	3.7270
Slot holes with trailing edge serration	3.6523124	1.0108856	0.059629591	0.016504255	3.6130	3.6130
Slot holes with leading and trailing edge serrations	5.126474	0.665896	0.083697534	0.010871774	7.6986	7.6986

Table 6: Aerodynamic forces and aerodynamic efficiency comparison of various blade designs

In above table, aerodynamic efficiency of normal blade is 2.1354 ,slot holes with leading edge serration is 3.7270, slot holes with trailing edge serration is 3.6130 and slot holes with both leading and trailing edge serration is 7.6986.

Aerodynamic efficiency value of slot holes with leading and trailing edge serration is two times greater than slot holes with leading edge serration and slot holes with trailing edge serration and three times greater than normal blade. So, the aerodynamic efficiency of slot holes with leading and trailing edge serration 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 slot holes with leading and trailing edge serration blade model

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 :

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

Drag force = $C_D * \frac{1}{2} \rho A V^2$ (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 = 5.038 N

Drag force = 0.312 N

Aerodynamic efficiency = η = Lift force / Drag force = $5.038/0.812 = 6.172$

Aerodynamic efficiency (η) = 6.172

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 X (0.263)² = 0.217 m^2

v- Wind velocity (m/s)

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

Average velocity (location of wind turbine blades) = v_{avg} = 12.17 m/s

$$P = 0.5 \times 1.225 \times 0.217 \times (12.17)^3 \times 0.40$$

POWER OUTPUT (P) = 95.83 W

CHAPTER 11

CONCLUSION

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, Normal blade is 2.1354, Slot holes with leading edge serration is 3.7270, Slot holes with trailing edge serration is 3.613 and slot holes with leading and trailing serration is 7.6986.

Comparing the three blade models and normal blade model, slot holes with leading and trailing serration blade reached maximum aerodynamic efficiency of 7.6986. In windtunnel experiment, aerodynamic efficiency of that blade model is 6.172

Comparison of the velocity of flow along the length of blade has been made in these cases, In the case of Normal blade, average velocity 9.81 m/s is observed, In Slot holes with leading edge serration case, average velocity 10.44 m/s is observed. In the case of Slot holes with trailing edge 11.21 m/s is observed and average velocity 12.17 m/s is observed in the Slot holes with leading and trailing edge case.

Comparing the three models and reference model, Slot holes with leading and trailing edge serration 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 95.83W

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] M. Herr and W. Dobrzynski, “Experimental investigations in low-noise trailing-edge design,” *AIAA J.*, vol. 43, no. 6, pp. 1167–1175, May 2005.
- [2] M. S. Howe, “Noise produced by a sawtooth trailing edge,” *J. Acoust. Soc. Am.*, vol. 90, no. 1, pp. 482–487, Jul. 1991.
- [3] M. S. Howe, “Aerodynamic noise of a serrated trailing edge,” *J. Fluids Struct.*, vol. 5, no. 1, pp. 33–45, Jan. 1991.
- [4] M. Gruber, P. F. Joseph, and T. P. Chong, “Experimental investigation of airfoil self noise and turbulent wake reduction by the use of trailing edge serrations,” in 16th AIAA/CEAS Aeroacoustics Conference (31st AIAA Aeroacoustics Conference), 2010.
- [5] M. Gruber, P. F. Joseph, and T. P. Chong, “On the mechanisms of serrated airfoil trailing edge noise reduction,” in 17th AIAA/CEAS Aeroacoustics Conference 2011 (32nd AIAA Aeroacoustics Conference), 2011.
- [6] G. M. Lilley, “A study of the silent flight of the owl,” in 4th AIAA/CEAS Aeroacoustics Conference, 1998.
- [7] A. A. Oberai, F. Roknaldin, and T. J. R. Hughes, “Computation of trailing-edge noise due to turbulent flow over an airfoil,” *AIAA J.*, vol. 40, no. 11, pp. 2206–2216, May 2002.
- [8] D. J. Moreau, L. A. Brooks, and C. J. Doolan, “On the noise reduction mechanism of a flat plate serrated trailing edge at low-to-moderate Reynolds number,” in 18th AIAA/CEAS Aeroacoustics Conference (33rd AIAA Aeroacoustics Conference), 2012.
- [9] F. V. Hucheson and T. F. Brooks, “Effects of Angle of Attack and Velocity on Trailing Edge Noise Determined Using Microphone Array Measurements,” *Int. J. Aeroacoustics*, vol. 5, no. 1, pp. 39–66, Jan. 2006.
- [10] S. Moreau and M. Roger, “Effect of airfoil aerodynamic loading on trailing-edge noise sources,” *AIAA J.*, vol. 43, no. 1, pp. 41–52, May 2005.
- [11] T. P. Chong, A. Vathylakis, P. Joseph, and M. Gruber, “On the noise and wake flow of an airfoil with broken and serrated trailing edges,” in 17th AIAA/CEAS Aeroacoustics Conference 2011 (32nd AIAA Aeroacoustics Conference), 2011.

- [12] F. V. Huthcheson and T. F. Brooks, “Measurement of Trailing EDGE Noise Using Directional Array and Coherent Output Power Methods,” *Int. J. Aeroacoustics*, vol. 1, no. 4, pp. 329–353, Oct. 2002.
- [13] A. Vathylakis, T. P. Chong, and P. F. Joseph, “Poro-serrated trailing-edge devices for airfoil self-noise reduction,” *AIAA J.*, vol. 52, no. 11, pp. 3379– 3394, Jul. 2015.
- [14] M. Roger and S. Moreau, “Extensions and limitations of analytical airfoil broadband noise models,” *Int. J. Aeroacoustics*, vol. 9, no. 3, pp. 273–305, May 2010.
- [15] A. Afshari, M. Azarpeyvand, A. A. Dehghan, and M. Szőke, “Trailing edge noise reduction using novel surface treatments,” in *22nd AIAA/CEAS Aeroacoustics Conference*, 2016, 2016.
- [16] G. Guidati, R. Bareiß, S. Wagner, T. Dassen, and R. Parchen, “Simulation and measurement of inflow-turbulence noise on airfoils,” in *3rd AIAA/CEAS Aeroacoustics Conference*, 1997