

CFD INVESTIGATIONS AND MODAL ANALYSIS ON TRIANGULAR SLOT AT VARIOUS POSITIONS ON NACA 63215 AIRFOIL IN HORIZONTAL AXIS WIND TURBINE BLADE

Submitted in partial fulfillment of the requirements for the award of
Bachelor of Engineering Degree in
Aeronautical Engineering

By

DEEPTHI BOLLA (Reg. No. 38260007)

SRIRAM KARTHIK (Reg. No. 38260023)



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

APRIL 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 DEEPTHI CHOWDAFRY BOLLA (Reg. No. 38260007) and SRIRAM KARTHIK.P (Reg. No. 38260701) who carried out the project entitled "**CFD INVESTIGATIONS AND MODAL ANALYSIS ON TRIANGULAR SLOT AT VARIOUS POSITIONS ON NACA 63215 AIRFOIL IN HORIZONTAL AXIS WIND TURBINE BLADE**" under our supervision from September 2021 to April 2022.

Internal Guide
Dr. MANIGANDHAN

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

Submitted for Viva voice Examination held on 5-5-2022

Internal Examiner

External Examiner

DECLARATION

We DEEPTHI CHOWDARY BOLLA (Reg.No.38260007) and SRIRAM KARTHIK.P (Reg.No.38260701) hereby declare that the Project Report entitled “**CFD INVESTIGATIONS AND MODAL ANALYSIS ON TRIANGULAR SLOT AT VARIOUS POSITIONS ON NACA 63215 AIRFOIL IN HORIZONTAL AXIS WIND TURBINE BLADE**” done by us under the guidance of **Dr. MANIGANDHAN** is submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering degree in Aeronautical Engineering.

DATE:

1.

PLACE: CHENNAI

2.

SIGNATURE OF THE CANDIDATE

ACKNOWLEDGEMENT

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

We 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 Mechanical Engineering for providing me necessary support and details at the right time during the progressive reviews.

We would like to express my sincere and deep sense of gratitude to my Project Guide **Dr. MANIGANDHAN** for his valuable guidance, suggestions and constant encouragement paved the way for the successful completion of my project work.

We 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

A wind turbine is a rotating gadget that converts the energy of the wind into electricity. Wind power is one of the most practical sources of clean, renewable energy. To convert wind power into mechanical power, the rotor blade is the most important component of a wind turbine generator. Airfoil-shaped go sections are found on the majority of commercial-grade wind mill blades. During low wind speeds, their blades seem greener than normal. CFD is a branch of fluid mechanics that uses numerical methods and algorithms to address fluid flow issues. Aerodynamic performance of horizontal-axis wind turbine blades may be improved through CFD stimulation of the triangular slot with the primary and trailing facet. In order to harness the power of the wind, wind turbines are necessary. In wind turbine wind help to rotate the blades around rotor and collects the kinetic energy. Wind flow over the blades to create lift so in over project we introduce triangular slots in the blade to increase the lift. We design blade using CATIA V5 with triangular slots on wind turbine blade of NACA 61325 airfoil, and then carried out the computational analysis using ANSYS software for various blades we design and compared them. For experimental analysis we fabricate the model using PLA and carried out wind tunnel testing at various speed. We studied the flow over the model The results shows that lift is produced more for triangular slots compare to other slots that we place. We did tapings for the fabrication and we did the wind tunnel testing for the model the experimental analysis of the experiment NACA 61325 airfoil the kinetic energy of the model wind tunnel of the testing for the wind tunnel testing triangular slots of the model.

TABLE OF CONTENTS

CHAPTER No.	TITLE	PAGE No.
	ABSTRACT	v
	LIST OF FIGURES	ix
	LIST OF TABLES	x
	LIST OF SYMBOLS AND ABBREVIATIONS	xi
1	INTRODUCTION	12
	1.1 Types of wind turbine blade	13
	1.1.1 Horizontal axis wind turbine blade	13
	1.2 Advantages of HAWT	14
	1.3 Disadvantages of HAWT	15
	1.4 Traditional HAWT Advantages	16
	1.5 Traditional HAWT disadvantages	16
2	LITERATURE SURVEY	18
3	AIM AND SCOPE	24
	3.1 Aim of the project	24
	3.2 Scope of the project	24
4	MATERIALS AND EXPERIMENTAL METHOD	25

CHAPTER No.	TITLE	PAGE No.
4.1	METHODOLOGY	25
4.2	EXPERIMENTAL METHOD	25
	4.2.1 Computational fluid dynamics	25
	4.2.2 Computational domain numerical simulation	26
	4.2.3 Modelling	26
	4.2.4 Boundary plane	28
	4.2.5 Velocity inlet boundary conditions	28
	4.2.6 Pressure outlet boundary conditions	29
	4.2.7 Wall boundary conditions	30
	4.2.8 Meshing	30
	4.2.9 Wind turbine used	32
	4.2.10 Principle of wind turbine	33
4.3	METHOD OF OPERATION	33
4.4	CALCULATION	35
4.5	EFFICIENCY	36
	4.5.1 Advantages	36
	4.5.2 Disadvantages	37
4.6	SOFTWARE REQUIRED	37
	4.6.1 Introduction to catia v5	37
	4.6.2 History of catia	37
	4.6.3 Scope of application	38
4.7	ANSYS	41
	4.7.1 Introduction of ansys	41
	4.7.2 Overview of ansys work bench	42
	4.7.3 Getting started to ansys	43
	4.7.4 Analysis and report in ansys	45
4.8	FABRICATION	58

CHAPTER No.	TITLE	PAGE No.
5	RESULTS AND DISCUSSION	59
	5.1 Experimental analysis	59
6	CONCLUSION	62
	REFERENCES	63

LIST OF FIGURES

FIGURE No	TITLE	PAGE No.
1.1	Horizontal axis wind turbine	14
4.1	Velocity inlet	29
4.2	Pressure outlet	29
4.3	Walls	30
4.4	Meshing on Boundary	31
4.5	Meshing on triangular slots	31
4.6	Importing geometry with boundary	32
4.7	Principle of H darrieus wind turbine	33
4.8	NACA 63215 airfoil creation in CATIA	40
4.9	NACA 63215 airfoil blade (HWAT)	40
4.10	Analysis input parameters	45
4.11	Analysis input parameters	46
4.12	Parameters of input ansys	47
4.13	Leading edge slot scaled residual plots	48
4.14	Leading edge Cd vs iteration	48
4.15	Leading edge Cl vs iteration	49
4.16	Pressure contour image for leading slot	49
4.17	Velocity contour for leading edge	50
4.18	Normal triangular slot in the wind blade Cd vs iteration	51
4.19	Normal triangular slot Cl vs iteration	51
4.20	Pressure contour for normal slot	52
4.21	Normal triangular slot scaled residuals	52
4.22	Velocity contour for normal slot	53
4.23	Trailing edge Triangle slot Cd vs iteration	53
4.24	Trailing edge Triangle slot Cl vs iteration	54
4.25	Pressure contour for trailing edge	54
4.26	Trailing edge Scaled residuals	55

FIGURE No	TITLE	PAGE No.
4.27	Velocity contour for trailing edge	55
4.28	Middle Triangle Slot Cd vs iteration	56
4.29	Middle Triangle Slot Cl vs iteration	56
4.30	Middle triangle slot Scaled residuals	57
4.31	Pressure contour for middle triangular slot	57
4.32	Velocity counter for middle triangular slot	58
4.33	Fabrication images	58

LIST OF TABLES

Table No	TITLE	PAGE No.
4.1	Velocity input	27
5.1	Experimental readings	59
5.2	Cp ports	60
6.1	Result table CFD	62

LIST OF SYMBOLS AND ABBREVIATIONS

Cl	coefficient of lift
cd	coefficient of drag
cm	coefficient of moment
ρ	density
A	area
V	velocity
L	lift
AR	aspect ratio
E	efficiency factor
AOA	angle of attack
RPM	rotations per minute
Hf	manometer reading
Cp	coefficient of pressure
Ps	static pressure
Pt	total pressure
E	eccentricity
Cd	drag polar
Xnp	neutral point
Vh	tail volume ratio

CHAPTER 1

INTRODUCTION

Using a wind turbine, kinetic energy from the wind may be transformed into electricity. Massive constructions known as wind farms are currently producing more than 650 gigawatts of power, with an average of 60 GW added each year to the total. As their dependence on fossil fuels diminishes, they're becoming an increasingly important source of intermittent renewable power. Study claims that wind has the lowest "relative greenhouse gas emissions, the least water usage wishes and... the most favourable social consequences" compared to photovoltaic, hydro, geothermal and coal... wind It is common for smaller windmills to be used for auxiliary power for boats and caravans, as well as to power website visitors' warning signs. huge generators can make a contribution to a home electricity supply at the same time as selling unused electricity decrease returned to the software program supplier through the electric grid. Auxiliary power for boats and caravans is provided by smaller wind generators, which are also used to power warning signals and charge batteries in the vehicles. large generators can make contributions to a home energy supply at the same time as promoting unused energy once more to the software dealer via the electric grid. Wind power is one of the most underutilised renewable energy resources that might meet the world's expanding electrical needs. Generators are using the extra wind capacity in the region to generate power from wind. The windmills are being built with a lot of money invested in them. Megawatt industrial-scale wind turbines cost somewhere about \$3 million to build. Wind turbines are expected to provide electricity for more than twenty-five years on a consistent basis. The wind turbine's braking system is a critical component in ensuring the turbine's smooth operation. The brake device is used to profit from electrical modulation, aided mechanical braking, and over tempo reduction. In addition to revealing the severe wind speed (over Beaufort large range six), wind turbine rotor rotation speeds will increase significantly as a result, causing damage to the wind turbine blades or causing the turbine to fall over. As a result, an efficient braking mechanism is required to slow down the rotor's rotating speed in the direction of the increased wind speed. Although both the mechanical and aerodynamic braking systems are components of the wind turbine, the aerodynamic braking system is the principal braking mechanism. Pitch-successful pointers, tip vanes,

spoilers, flip-tips, and ailerons are among the most important aerodynamic braking devices. The main and most powerful aerodynamic braking mechanism is the active pitch control, i.e., pitch in a function-guided manner. It allows for more precise control of the wind turbine.. In a nutshell, pitching is the adjustment of the blade angle of the wind turbine blade in order to maximise appropriate energy from wind and to protect the turbine from unexpected forces as a consequence of high-speed wind. It is possible to modify the angle of attack by pitching the whole turbine blade to regulate rotational speed and energy production. The virtual controller provides a signal to the blade pitch mechanism, which immediately pitches (turns) the rotor blades slightly out of the wind when the rotor pacing gets too extreme. Slowing down the turbine's rotational speed and the torque it transfers to the shaft is accomplished by a shift in attack mindset, which allows for normal rotational speed and torque to be stored below a fast threshold.

1.1 TYPES OF WIND TURBINE

Th ewind turbines are classified based on many types. In general, we classify based on the axis of the blades, namely Horizontal Axis Wind turbine and Vertical Axis Wind turbines.

1.1.1 Horizontal axis wind turbines (HAWT)

HAWT, also known as a horizontal axis wind turbine, is the kind of wind turbine that most people think of when they hear the term "wind turbine." A HAWT resembles a wind turbine in that it has blades that rotate horizontally, much like propellers on an aeroplane. Horizontal axis wind turbines feature the number one rotor shaft and electric generator at the top of a tower, and they like to face the wind. A smooth wind vane positioned rectangularly with the rotor (blades) of small turbines directs the turbine towards the wind, whereas larger generators often utilise a wind sensor and servo motor to expose the turbine to the wind. Most large wind turbines contain a gearbox that converts the sluggish revolution of the rotor into a quicker spin that is more suited to pressuring an electrical generator.



Fig.1.1 HAWT

turbines are often oriented upwind of towers because of the turbulence they create. Blades on wind turbines are stiffened to prevent them from being forced into the tower by high wind forces. Adding to this, the blades are positioned a considerable distance from the front of the tower and are sometimes slanted slightly upward. Despite the turbulence issue, downwind machines have been developed due to the fact that they do not need an additional mechanism to keep them in sync with the wind. In addition, the blades may bend in strong gusts, reducing the swept area and, hence, the wind resistance. The majority of HAWTs are upwind machines due to the fact that tiredness is a common side effect of flying in turbulent air and so must be avoided at all costs.

1.2 HAWT ADVANTAGES

When wind shear is present, the higher tower base provides better access to stronger winds. In certain wind shear locations, the wind speed may increase by 20% and the electrical production by 34% for every 10 metres up.

A high level of overall performance is ensured by the fact that the blades rotate completely perpendicular to the wind. However, every horizontal axis wind turbine, and the majority of proposed aircraft-based, airborne wind turbine designs, involve a variety of reciprocating motions, forcing the airfoil surfaces to back off in the direction of the wind for some of the cycle. When competing against the wind, backtracking results in decreased overall performance.

1.3 HAWT DISADVANTAGES

- Big tower creation is wanted to manual the heavy blades, gearbox, and generator.

- Their pinnacle makes them obtrusively seen at some stage in massive regions, disrupting the look of the landscape and once in a while growing community competition. A blade passing into the wind shadow of the tower may induce fatigue and structural failure in downwind variations (for this reason, the general public of HAWTs use an upwind layout, with the rotor going via the wind withinside the front of the tower).

When operating in high winds, HAWTs need an additional yaw control system to keep the blades aligned with the direction of the wind and prevent the turbine from spinning and damaging or pooring itself.

- Cyclic Stresses & Vibration - the whirling blades serve as a gyroscope as the turbine rotates to face the wind. the turbine is curved into an advanced or backward somersault as it pivots via gyroscopic precession When a wind turbine's blades are horizontal, pressure is at its lowest, and when they are vertical, pressure is at its highest. Because to the cyclic twisting, turbine blade roots, hubs, and axles might get overly worn. This kind of windmill, known as a "HAWT," has its primary rotor shaft constructed horizontally. The primary benefit of this partnership is the elimination of the need for wind turbines to be aimed directly into the wind. This is an added benefit on windy or choppy websites. a bonus. By using a Horizontal axis, the generator and other key additives may be located closer to the ground, eliminating the need for the tower to manually operate them. This arrangement also facilitates restoration. When flying against the wind, the primary drawback of a HAWT is the drag it causes. Because it is difficult to put Horizontal-axis turbines on towers, they are often attached to the ground or the roof of a building, for example. There is less wind power available for a given turbine length at lower altitudes because to the lower wind speed. Turbulent air near the ground and other things may cause a waft, which may present concerns of vibration, noise, and bearing wear, which may also increase the protection or reduce its provider's lifespan moreover similarly. However, when a wind turbine is installed on a rooftop, the building usually redirects wind over the roof, doubling the wind speed at the turbine. In order to get the most wind power with the least amount of wind turbulence, the rooftop installation turbine tower's height should be positioned at 50% of the building's peak.

1.4 TRADITIONAL HAWT ADVANTAGES

- Strong supporting towers are no longer required since the generator, gearbox and other components are all located on the ground.
- It has a lower manufacturing cost than horizontal axis wind turbines, which are more expensive to build.
- The yaw pressure and pitch mechanism are not desired since there is no need for the turbine to point in the wind's direction in order to be green. "
- It's easier to put up than other wind turbines.
- Transporting it from one place to another is a breeze.
- Inexpensive remodelling costs
- It's possible to put them in urban areas.
- Human and avian safety is ensured due to the moderate speed of the blades.
- For areas with extreme weather, such as the highlands, where mountain huts depend on their power, these generators are ideal.

1.5 TRADITIONAL HAWT DISADVANTAGES

- As one blade of the wind turbine operates at a time, the total performance of the wind turbine may be much lower than that of HAWTS.
- They require an initial push to start; this first push needed to make the blades start spinning on their own has to be initiated by a tiny motor.
- Even though they are less environmentally friendly than horizontally oriented wind turbines, they are still a lot more efficient.
- When air wafts near to the ground, it causes turbulence, which results in high levels of vibration.
- Vibration will cause bearing wear, which will lead to an increase in the cost of protection.
- A possible goal would be to generate noise pollution.
- Guy wires may be necessary for HAWTs to keep it stable (man wires are impractical and heavy in farm regions).

CHAPTER-2

LITRATURE REVIEW

A computational fluid dynamics study of the horizontal axis turbine blade with tubercles at the forefront It was a team effort that included the contributions of Wei Cheng Wang; Chi Yang; Yang You Lin; and San-Yih Lin. To further understand HAWT blade performance with NACA0015 surface section, researchers turned to ANSYS FluentTM, a commercial machine fluid dynamics computer code. Pressure linked Equations formula is used to solve the incompressible Navier-Stokes equations using the Semi Implicit approach. During the turbulence flow simulations, the k- ϵ shear stress transport turbulence model stood up as the most accurate and efficient. Using experimental data, we were able to validate the modelling findings for lift and drag coefficients between zero and forty degrees of attack. In order to verify the simulation's correctness, the grid numbers and time step sizes were scrutinised. As a follow-up to test the 3D impact, a 2.5D version was created and then compared to a second version. This was followed by a comparison of the predictions of thrust from the tubercle-vanguarded knife and the straight blade. With the new rotary engine blade, the thrust of HAWT was lower than with the straight blade. Because of the vortex structure created at the leading edge of a rotary engine blade, decreasing wavelengths and rising amplitudes reduced thrust values the most. The H-Rotor type HAWT with turbine blades altered with a tubercle leading edge was distributed for CFD simulation to forecast performance. For NACA0015 surface, the findings of lift and drag coefficients ranging from zero to forty AOAs were initially legitimate based on experimental information from the literature For the second simulation, a grid size of 1445.5 and a time step size of 1.0 were both selected after careful consideration of the available options. To verify the second model, a 2.5D model was created. As a result of the vortex formation, a significant deviation was seen at rotation angles between 180 and 270 degrees. However, the outcomes predicted by these two models were quite close to one another. For the altered rotary engine blade, the maximum thrust was 20 N–240 N less than for the straight blade, with a difference of 1 percent to 10 percent. Both the maximum and lowest thrusts decreased in amplitude or wavelength with increasing amplitude or wavelength decreases. Vortex structure alteration, decreased speed magnitude of vortex, deviation of pressure gradient and vortex migration toward edge of

the blade were all factors that contributed to a drop in thrust for modified blades as seen by flow stream studies. that focus on the turbine blade's trailing edge follow this research.

Computational Fluid Dynamics (CFD) Simulation of a Horizontal Axis Wind Turbine - Dr P T Saravanakumar and his colleagues S Dinesh, M Usha, and V S Vemeera

A rotational device that collects energy from the wind might be a turbine. One of the most promising renewable energy sources is wind power. To transform wind energy into mechanical energy, the control surface of a wind turbine generator is essential. Surface-formed cross sections are used in wind turbine blades for commercial applications. At lower wind speeds, these blades prove to be quite cost-effective. A subfield of hydraulics known as computational fluid dynamics (CFD) makes use of numerical methods and algorithms to analyse fluid flow issues. In order to bridge this gap, HAWT research is focusing on creating turbines with a power output of between 300W and 500W. A low-speed, high-turbulence wind has an advantage over a low-turbulence wind in this way. CFD is used to model a variety of airflows and directions, as well as the HAWT's efficacy, in order to better understand how it works. This study examines the pros and drawbacks of different HAWT systems. Because of the absence of research and economically feasible energy solutions for distant locations that are not connected to integrated grid systems, HAWT offers significant advantages over HAWT in terms of building integration, poor self-start, and low power constant. In the range of 2kw up to 4kw, HAWT will provide an energy solution. When selecting an acceptable TSR range, the coefficient of power is also improved. Although HAWT's axis isobilateral design makes them unidirectional turbines, they can adapt effectively to variations in wind flow patterns when installed in an urban environment. There are numerous machine models that can numerically test the performance of the turbine and provide a significant advantage over the standard experimental approach for estimating HAWT performance. P.Larin, M.Paraschivoiu, and C.Aygun used CFD mostly for synergistic wind turbine integration studies. Light-emitting diodes (LEDs) were used to investigate cup-shaped turbines as well as turbines having 30° circumferential incisions on the front and rear of the blades. Six- and seven-bladed turbines performed best, according to the study into blade diversity. After cutting the blades, the facility constant was increased and the peak power coefficient moved to the next TSR by conducting a circumferential blade length analysis. There was an

improvement in the influence coefficient of 0.24 for the seven bladed rotating engine with double cut blades compared to the conventional bladed turbine put in the free stream with a power coefficient of 0.043 for conventional blades and 0.0246 for the double cut blades. Even though a turbine with a decent power coefficient was successfully installed on a 30.48m tall structure, further research into other building heights is needed.

Comparison of low-order mechanics models and RANS CFD for full scale 3D Horizontal axis rotary engines-P.L Delafin, T. Nishino, A. Kolios, L.Wang, The findings reveal that the three procedures provide very comparable results and correlate well with the experimental data for the turbine at its best (medium) tip speed magnitude relation. If the mesh was finer, the CFD findings at the low tip speed ratio may be closer to trials, according to a second CFD analysis of a pitching airfoil published in this research Since this work has shown that DMST models may fail to anticipate the optimal tip speed magnitude relation and may also overestimate or underestimate the rotary engine force and thrust forces, they should be utilised cautiously. Wind boosters for Horizontal Axis Wind Turbines at low wind speeds have been analysed and the best design has been determined. HAWT's mechanical performance may be improved in low-speed winds by using a custom wind booster. With a wind booster installed, CFD study shows that the HAWT is able to spin more quickly. Also, it has the ability to produce more mechanical power. A wind booster's ideal design may be discovered via optimization. The HAWT with the greatest wind booster is capable of producing mechanical power that is even greater than that of the original HAWT. An good agreement between the CFD model and the actual results of the ideal wind booster under real wind circumstances at low speed confirms its validity. M.M.S.R.S.Bhargav, Velamati Ratna Kishore, and Vaitla Laxman used a three-dimensional CFD model to examine the impact of unstable wind conditions on the Horizontal Axis Turbine. Darrieus HAWT performance under variable wind conditions is evaluated using a CFD model. This research evaluates the results of, U_{amp} , and f_c . $U_{amp}=30$ percent, $f_c=1$ rate, and $mean=2$ are taken into consideration when cycle averaged unstable wind CP is 0.3 and uniform flow CP is 0.34. The results of the performance investigation show that the wind CP curves don't all follow the same pattern. U_{amp} is explored by performing simulations at 10%, 20%, 30%, 40%, and 50% U_{amp} . According to research, when U_{amp} increases from 10% to 50%, the cycle averaged

continuous hyperbolic CP fluctuates between 0.284 and 0.329. The simulation is conducted at $f_c=0.5$ Hz, 0.75 Hz, one rate, 1.5 Hz, and a pair of Hz in order to examine the impact of fluctuation frequency variation. For $f_c \geq 1$ Hz, increasing f_c results in a little drop in CP. At $f_c = 1$ Hz, the greatest cycle averaged CP peak of 0.31 Hz is detected. Consequently, preparing for 1 Hz while working with HAWT is beneficial. By performing the simulations at mean=1.11, 1.3, 1.81, 2, and 2.5, we can see how the Tip Speed Magnitude Relationship behaves. Cycle-averaged CP will rise with an increase in mean and reaches a maximum price of 0.31 at mean=2. CP decreases as a consequence of an extra rise in the mean on the far side 2 To put it another way, the curve's steepness increases as the mean of the two numbers increases. That's why during wind analysis, it's best to run HAWT at mean 2 in order to obtain the best results. Higher U amp, f_c getting ready to one rate, and mean 2 increase HAWT's performance under varying wind circumstances. Only in the scenario when fluctuation amplitude and frequency have been analysed by Danao and colleagues, are their findings conflicting. The lower blade tip Z/H values of 0.033 and 0.167 are used to study vorticity contours in two different planes, respectively. Consequently, the blade tip has a significant influence in determining the CPI (CP). Research on HAWT's specific performance requires 3D simulations, as seen by here. Wind turbine foundation wave loads drop as wave slope ka increases in a computational fluid dynamics (CFD) model developed by Yu-Hsien Lin, Jing-Fu Chen, and Po-Ying Lu to examine the effects of nonlinear waves on different turbine foundations. Lower wave steepness has the most impact on the maximum wave load on gravity-based support systems, followed by support stands and monopiles. In this way, the incident wave may be physically removed by installing sloping bottom on the outlet border. The two-point technique has consistently shown reflection rates < 0.02 for the next scenarios of wave-structure interaction via a series of numerical tests using a wide range of wave heights H_0 and times T_0 . When it comes to upper wave steepness, the kind of foundation has a greater impact on R_u , Max/a than any other support structure does. Contrary to popular belief, the foundation type has little effect on the minimum R_u , Max/a of any given support system. R_u , Max/a for every support structure generally rises with the increasing of wave slope ka . In contrast, when ka rises, it becomes harder to tell apart the minimal R_u , Max/a of each support structure. The semi-chemical formula yields

a stepwise rise parameter m for estimating run-up heights while taking into account wave mechanics and different kinds of foundation. Run-up parameter m falls when the normalised maximum runup parameters R_u , Maxim/a and wave steepness ka increase. The wave load should be taken into account while estimating run-ups, especially in the case of smaller waves. m values for smaller waves are greater than those for larger waves, which indicates this.

Study by Sean Quallen and Tao Xing on the effects of a variable speed generator torque controller on a floating offshore wind turbine system The findings are compared to NREL's publicly available OC3 results, which use their fast programme to speed up the analysis process. A progressive technique is used to verify the strategy in simulations. Since there are so many mechanical resolution differences between this CFD problem solver (CFD-SIM) and the one developed by National Renewable Energy Labs (NREL), the first step in developing a baseline expectation for rotor torsion is to run an OC3LC2.1a simulation (Case 1). NREL's projections for maximum, minimum, and mean rotor torque were all within six percent of the actual findings obtained by OC3 participants. Case two (OC3 LC 2.1b conditions) is used for the second simulation (Case two). In order to investigate the effects of torsion control on rotor torque, the platform's mechanical phenomenon rotor model and VS management are now active. Mean, maximum, and lowest rotor torque differences of 4.4% between Case 2 and NREL's results are consistent, which supports the functionality of the VS controller. Straight bladed horizontal axis turbine (Part I: Predicting mechanical loads and performance)- Qing'an Li, Takao Maeda, Yasunari Kamada, Junsuke Murata, and Toshiaki Kawabata- Kento Shimizu When the tip speed-amplitude relation is 2.29, the height suction increases up to CP14 27.14 at an angle of letter of the alphabet 14 90_, in line with the pressure distribution engaged in the centre of the blade span. It's also possible that some of the flow has been separated from the trailing edge, but it's still possible that the blade is still sucking air up from the upper area. This means that the maximum cost of negative pressure is reduced as spanwise positions increase. Furthermore, it should be noticed that the activity values at the spanwise point of $z/(H/2)$ 14 0.55 exhibit a completely distinct trend due to the support structure that may create related vortex. Torsion constants of CQS 14 0.342, 0.178, 0.178 and 0.181, respectively, may be obtained only if the spanwise locations of $z/(H/2)$ 14 0, 0.55, 0.70

and 0.80 are targeted in the measurements. Furthermore, the torque coefficient is nearly 50% larger at its maximum in the centre of the blade span of $z/(H/2) = 0$, compared to the opposite cross sections. At the blade's centre height of $z/(H/2) = 0$, the facility coefficient indicates the greatest value, and declines gradually as the blade tip approaches. $z/(H/2) = 0.55$ is also the smallest facility constant in the spanwise position, with a value of 0.55. The straight-bladed HAWT's power coefficient is greatly enhanced by eliminating the three-dimensional influences on the flow around the blade and hence the support structure. Construction experiments and computational fluid dynamics (CFD) will be used to examine the impact of turbine support structures on power coefficients in our future research projects.

rotor design and analysis-MD Saddam Hussien, Dr. Krambabu M. Ramji, E. Srinivas Two HAWTs have been tested side by side to see how they compare to each other in terms of performance. Initial 3-D CFD calculations have been performed in order to gather background information. Static three-dimensional assessments have been done on three distinct configurations. During the dynamic studies, only two of these configurations were used: straight 900 and twisted 900. There was an 8.75 percent increase in power production from the turbine blades during the whole cycle of stable simulations.

CHAPTER-3

AIM AND SCOPE OF THE PRESENT INVESTIGATION

3.1 Aim of the project

The main aim of the project is to improve the aerodynamic performance by inserting the triangular slots on blade and to measure the lift and drag producing by the wind turbine blade and doing CFD and modal analysis for the NACA 63125 airfoil horizontal axis wind turbine blade.

3.2 Scope of the project

After getting the results of analysis we can use this model in daily life for converting mechanical energy into electrical energy. The triangular slots may create a major impact on the wind turbine blade design so we used the advanced airfoil for this project. The scope of the project is can increase the slots on blade for more drag and lift. Can do modifications to the design as well as our requirement and more over for this project we are going to patent the work we did and we are suppose to make a journal as well. The scope of the project is depends upon the modifications of ours to make in future it is very useful for our higher studies and future work. The scope of the work is a very vast as other projects in the market so the design is stronger as well and moreover the work of the project is very useful for future generation to use as reference and the some of the projects the scope of the project is very huge in the outdoor can use this project as reference. As wind turbines blades have similar properties of aircraft wing's they introduce slots so that the lift produced will increase which helps for wind turbine to increase speed so that the electricity generated will be more. Mechanical energy may be generated from kinetic energy via the use of rotors. To transform kinetic energy into mechanical energy, a wind turbine uses a permanent magnet synchronous generator (PMSG). The power generated by these generators is direct current (DC). The yaw system of the wind turbine is caused by the rotor of the wind turbine being oriented toward the wind. So basically in wind turbine the turbine helps to convert energy into rotations with the help of rotor and generations helps to convert this rotations into Electricity.

CHAPTER-4

MATERIALS AND EXPERIMENTAL METHOD

4.1 METHODOLOGY

- Literature survey
- Creating 3D Models of Triangular Slot at Leading, Middle and Trailing edges in the Horizontal axis wind turbine blades
- Convert the 3D Model into STEP Files, import the STEP files into ANSYS workbench software
- Creation Computational Domain and Meshing
- CFD analysis on Triangular Slot at Leading, Middle and Trailing edges in the Horizontal axis wind turbine blade models
- Creation of Structural meshing for Modal analysis
- Modal analysis on Triangular Slot at Leading, Middle and Trailing edges in the Horizontal axis wind turbine blade models to find natural frequency
- Comparision of CFD and Modal analysis results
- Power calculation

4.2 EXPERIMENTAL METHOD

4.2.1 Computational fluid dynamics

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. It is possible to numerically simulate turbulent flow over bluff structures using a variety of turbulence models, including Large Eddy Simulation and Reynolds Averaged Navier-Stokes Simulation (RANS). There is a time averaged mean component, a periodic component and a random fluctuation component in time-varying flow quantities like

velocity or pressure at high Reynolds numbers. Vortex shedding in the wake of a bluff body is the most common cause of the periodic component.

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. The turbulence component can only be modelled using Unsteady RANS (URANS), which is based on ensemble averaging and so resolves only the periodic component. Even though LES forecasts are more accurate than RANS predictions, RANS turbulence models, particularly two-equation turbulence models like k-, have been highly popular since they demand less computer memory and CPU time than LES.

4.2.2 Computational domain and numerical simulation

The pre-processing of model improvement and meshing of model has been finished the usage of GAMBIT software. GAMBIT is designed to assist analysts and architects build and meshing the fashions to be used in Computational Fluid Dynamics (CFD) and other scientific programs. GAMBIT receives person enter with the aid of its Graphical consumer Interface (GUI). on this look at, the diverse elements of the version consist of the shroud, diffuser frame, and the brim which are modeled through using this software program. The profiles are generated with the assist of coordinates available which have been generated. Meshing is accomplished after the modeling tactics.

4.2.3 Modelling

The coordinates are provided for the improvement of the 2d version of the diffuser. For design purposes, the diffuser can be visible as a aggregate of three separate sections particularly, the shroud, diffuser body, and the brim. The diffuser is modeled with out wind turbine in this take a look at. The geometrical specs of the diffuser are diagrammatically depicted on this parent under. Coordinates are generated according to the geometry of the diffuser and edges are created to finish the profile of the model. The geometrical specifications of the diffuser model selected for the reference are given in desk

velocity Input = 10m/s

Table No.4.1 (start loft and curve results)

start curve	start loft	result
0	100	0.189
0	95	1.21
0	90	2.172
0	80	3.935
0	70	5.496
0	60	6.845
0	50	7.941
0	40	8.705
0	30	9.003
0	25	8.912
0	20	8.606
0	15	8.018
0	10	7.024
0	7.5	6.3
0	5	5.332
0	2.5	3.922
0	1.25	2.841
0	0	0
0	1.25	-2.841
0	2.5	-3.922
0	5	-5.332
0	7.5	-6.3
0	10	-7.024
0	15	-8.018
0	20	-8.606
0	25	-8.912

0	30	-9.003
0	40	-8.705
0	50	-7.941
0	60	-6.845
0	70	-5.496
0	80	-3.935
0	90	-2.172
0	95	-1.21
0	100	-0.189

4.2.4 Boundary Plane

The boundary plane was constructed with axis symmetry in mind. The following are the five boundary conditions used in this simulation:

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

4.2.5 Velocity Inlet Boundary Conditions:

The speed inlet boundary is set to the inlet wall of the space. drift speed is defined by velocity inlet boundary conditions and the scalar parameters of waft and flow inlets. The go with the flow's overall (or Stagnation) residences are not set, so they may rise to whatever charge is necessary to meet the velocity distribution. its support for more than one range of designs, its miles are taken into consideration as a cax program sometimes named as 3-dimensional product life cycle management software. It issues collaborative engineering designs through an integrated cloud carrier and feature assistance is to be used for multiple disciplines. It offers the special capability other than versioning any product but to achieve something in real life operations within the age of experience For protection and performance motives the elevator/ rudder is ready with a movable floor known as trim tab like elevator trim and rudder trim.

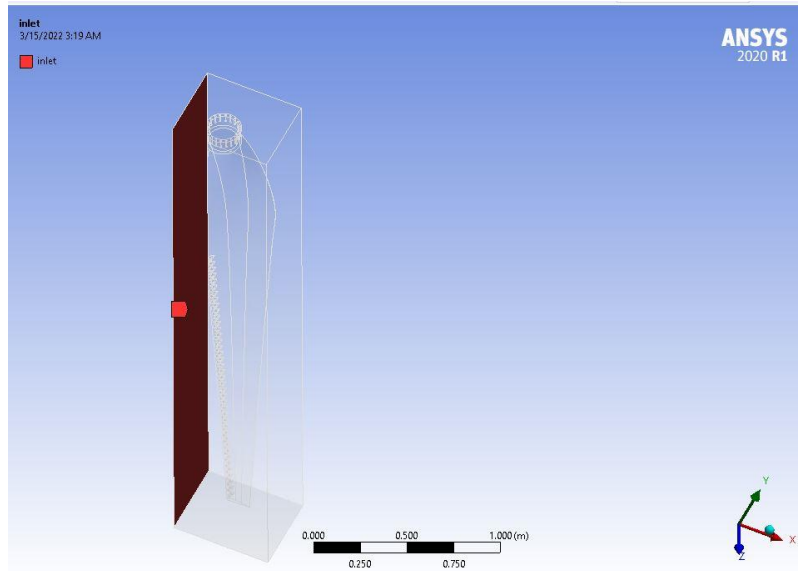


Fig.4.1 Velocity Inlet

4.2.6 Pressure Outlet Boundary Conditions

The area outlet wall serves as a boundary stress outflow border. At the opening border of a pressure outlet, a static (Guage) stress must be specified. Subsonic flow is the optimal moment to apply the static strain rate unique.

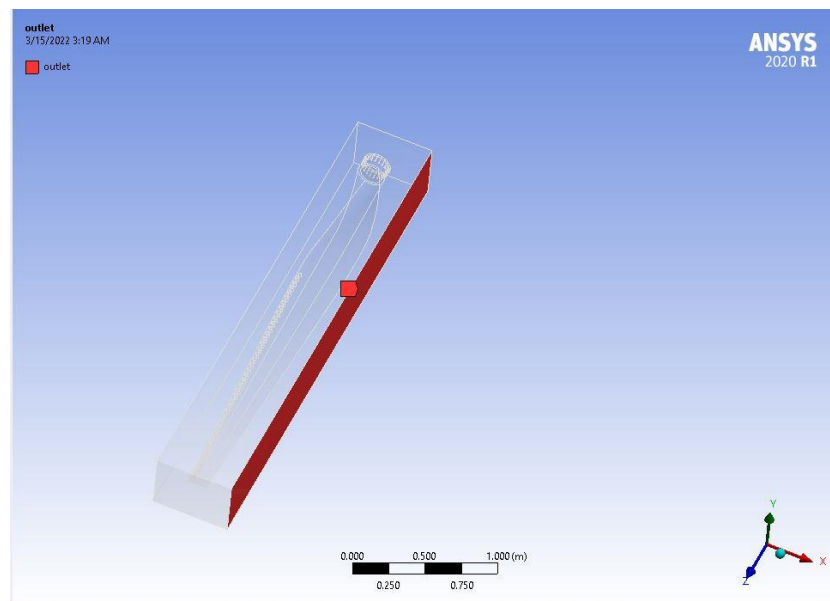


Fig.4.2 Pressure Outlet

In fig 4.2 we used to check the pressure outlet of the design.

4.2.7 Wall Boundary Conditions:

This is a good approximation of the border of the wall. Using wall boundary conditions, fluid and powerful zones may be contained. However, a tangential velocity item in terms of translation or rotational movement of the wall boundary may be specified to make the walls slide, or shear can be used to make the walls slip. Using information from the waft field, we can derive the shear pressure and heat transfer between fluid and wall.

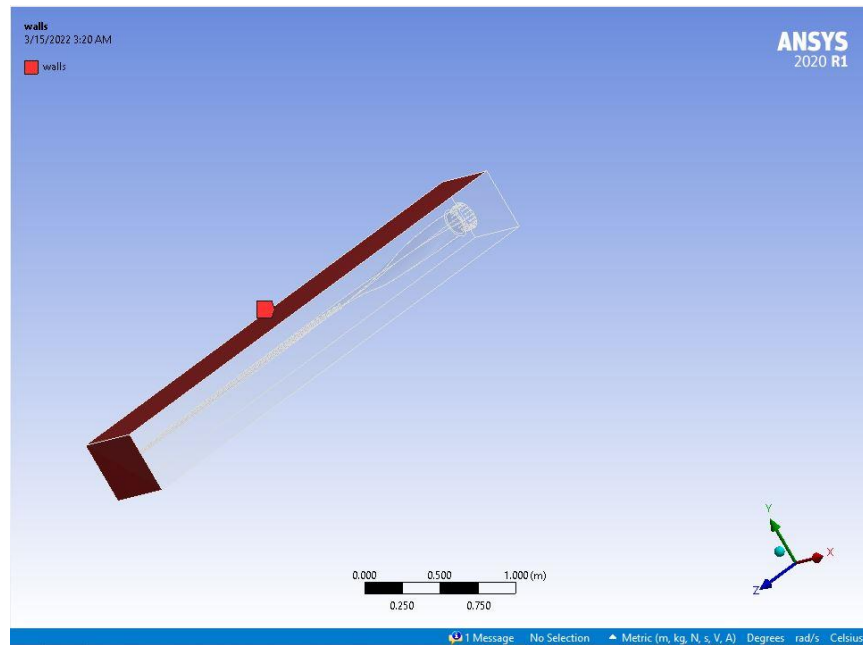


Fig.4.3 Walls (other 4 sides)

4.2.8 Meshing:

Meshing is the procedure of dividing a continuous unit into smaller character gadgets. computational domain

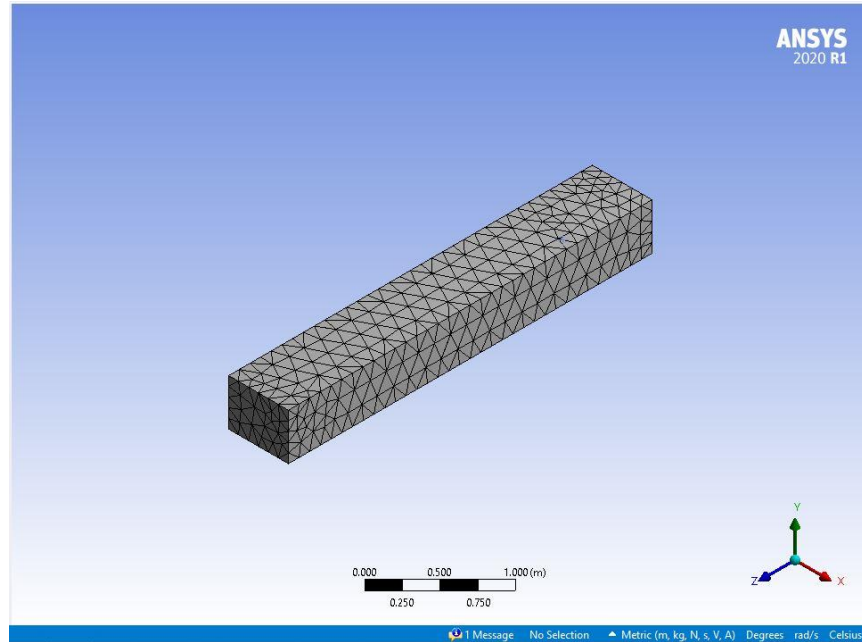


Fig.4.4 Meshing on boundary

In fig 4.4 we did meshing on boundaries of our design

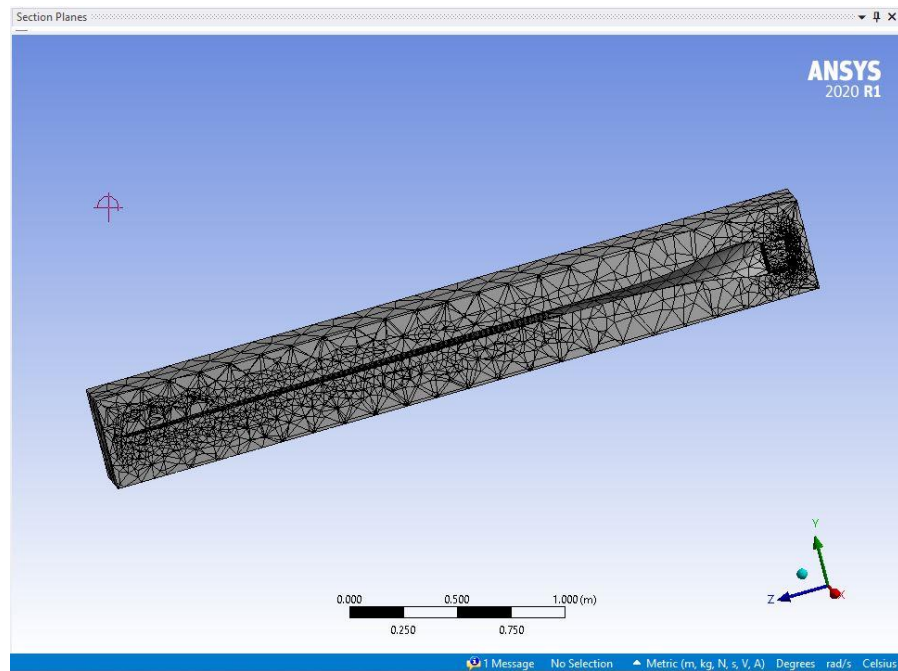


Fig.4.5 Meshing on Triangular Slot

In fig 4.5 we did meshing for triangular slots of the design

4.2.9 Wind turbine used

It is one of the most promising wind energy converters for situations where the wind direction changes rapidly, such as in the built environment. Make sure they can start on their own and stay up and enhance their performance while utilising this sort of normally modest equipment. Due to its rotation around a Horizontal axis, the aerodynamics of the turbine is more sophisticated than a horizontal axis wind turbine, and our knowledge and understanding of these mills is still a long way from complete. Recent investigations on the turbine's design factors such as solidity, blade shape and pitch are reviewed in this study. Furthermore, the essential task of designing a turbine that can start on its own is given careful consideration. In addition, this study describes the major research methodologies for assessing the turbine, which will pick up achievements and promise for future observation.

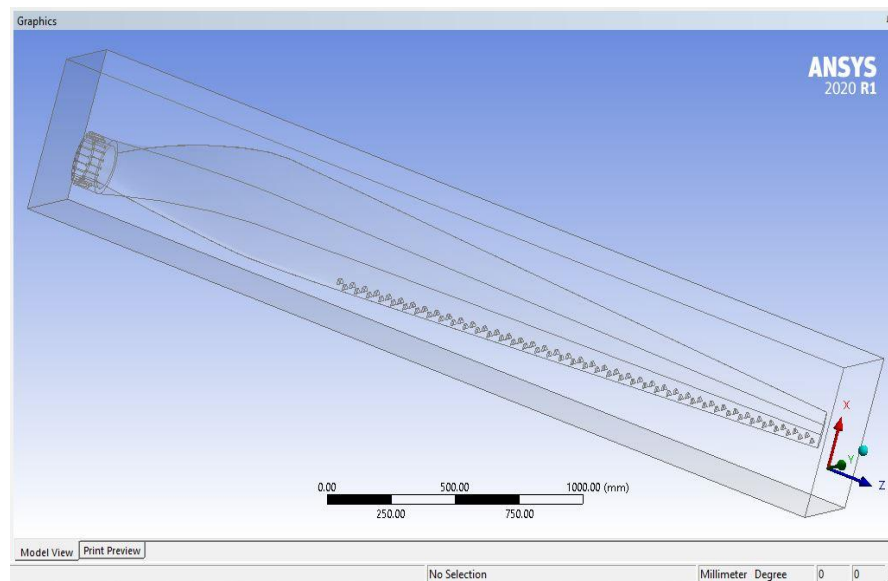


Fig.4.6 Importing geometry with boundary

This importing geometry with boundary of our design.

4.2.10 Principle of h Darrieus Wind Turbine

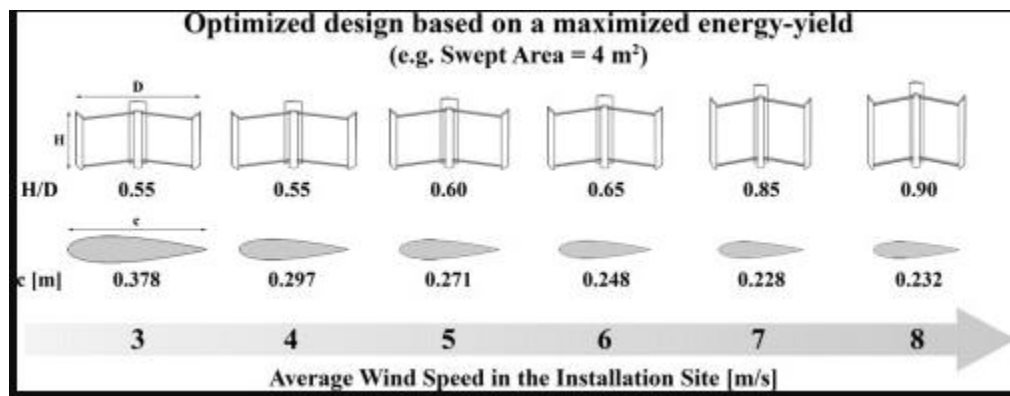


Fig.4.7 Principles of wind turbine

those generators aren't self-starting however it requires a motor that's small powered to start the revolution. as soon as it has sufficient speed then the wind flows across the airfoil's starts to supply torque & the rotor can be driven in the place of the wind. within the Darrieus turbine, mini Savonius rotors are located at the shaft to start a revolution. these lessen the Darrieus turbine once it gets going however they make the complete tool very easier in addition to easier to preserve.

4.3 METHOD OF OPERATION

Asymmetrical airfoils are placed in the design's unique variants, with zero rigging perspective, meaning that the airfoils are positioned in relation to the structure on which they may be attached. In contrast to traditional wind turbines, which must be turned to face the wind, this design is equally strong regardless of the direction of the wind. Rotor spins and the airfoils move in a circular motion. This incoming airflow is vectorially delivered to the wind such that the resulting airflow generates a variable little amazing angle of attack on the blade. Internet pressure is generated in this way, aiming obliquely ahead along a predetermined "line of movement". This pressure may be transferred inwards beyond the turbine axis at a particular distance, providing the shaft a pleasant torque, and so enabling it to spin along the route it is currently travelling. The rotor's aerodynamic concepts are similar to those of autogiros and regular helicopters in autorotation. Airfoil movement over the lower back of the apparatus causes a change in the angle and direction of attack, but produced pressure is still obliquely inside the path

of rotation since the wings are same and the rigging angle is equal. When the wind is blowing, the rotor rotates at a faster pace than usual. An electrical generator may be used to collect and turn the torque and speed-related energy into useful energy.

Since the words "carry" and "drag," in aeronautics parlance, refer to forces present in and together with the incoming internet's relative airflow, these terminology may no longer be relevant. There are two opposing forces at work: tangential force, which draws the blade around, and radial force, which pushes the bearings out of alignment. no rotational pressure is generated while the rotor is motionless, even when the wind speed climbs significantly—the rotor should already be rotating to create torque. Therefore, the design is not frequently self-initiated. Darrieus rotors may self-begin in unusual circumstances, thus a brake is required to keep it in place while it is stopped. The design has a flaw in that the angle of attack changes while the turbine rotates, resulting in the maximum torque being generated by each blade at two different points in its cycle (front and back of the turbine). As a consequence, a sinusoidal (pulsing) power cycle develops, making the design process more difficult. In particular, almost all Darrieus generators feature resonant modes where, at a specified rotating velocity, the pulse occurs at a high frequency of the blades which may (ultimately) deteriorate. As a result, most Darrieus mills feature mechanical brakes or other speed-control mechanisms to keep the turbine from spinning at high rates for an extended period of time.. Every other issue emerges because the majority of the spinning mechanism's mass is on the perimeter rather than the hub, as is the case with a propeller that travels for miles. This results in very high centrifugal forces on the mechanism, which necessitates a more powerful and heavier mechanism than in any other circumstance. This may be done by forming wings into a "egg-beater" shape (a "troposkein" shape, borrowed from the Greek for "the shape of a spun rope") so that they are self-helping and no longer need as hefty supports and mountings. The Darrieus arrangement is theoretically less luxurious in this configuration than a regular one since the blades that generate torque toward the generator located at the base of the turbine are under the most strain. this. The simplest forces that must be counterbalanced Horizontally, there is the compression load caused to the blades bending outward (which attempts to "squeeze" the tower) and the wind pressure that attempts to blow the whole turbine over, half of which is transferred to the lowest and the

other half is easily countered using guy wires. Traditional layouts are compared to wind-driven tower tops. Additionally, guy wires cannot be used to counteract this weight since the propeller rotates both above and below the tower's top. As a result, the traditional design requires for a sturdy tower whose height increases substantially in direct proportion to the size of the propeller. Modern designs are able to adjust for the varied speed and varying pitch of most of the towers in the area.

Despite the fact that Darrieus design has certain benefits, there are many bigger downsides, particularly with larger machines inside the MW range, in standard assessment. The Darrieus design makes use of much more expensive fabric in the blades, but the majority of the blade is too near to the ground to provide any meaningful strength. At its lowest point, the wing tip is expected to be at least forty metres above the ground, in order to optimise the generation of power and its life span. There is currently no recognised material (not even carbon fibre) that can fulfil the requirements of cyclic loads.

4.4 CALCULATION

The output power of shrouded 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

d= Rotar Diameter (m) = 0.525 m

r=Rotar Radius(m) = 0.263m

A - Swept Area (m^2) = πr^2 (m^2) = $3.14 \times (0.263)^2 = 0.1017\text{m}^2$

v- Wind velocity (m/s)

Cp- Power Coefficient (No unit) = 0.45 (Betz Limit)

Average velocity in Throat section (location of wind turbine blades) = $v_{\text{avg}} = 5.88\text{m/s}$

$$P = 0.5 \times 1.225 \times 0.1017 \times (5.88)^3 \times 0.45$$

$$P = 5.69 \text{ W}$$

4.5 EFFICIENCY

As a result of the Darrieus Wind Turbine's operating characteristics and design, its efficiency is much lower than that achieved by the horizontal axis wind turbine. The HAVT

(horizontal axis wind turbine) can convert 40% – 50% of the KE (kinetic strength) it generates into actual electrical strength, which is the typical performance percentage. Alternatively, a Savonius Horizontal axis wind turbine performs between 10% and 17%, and a Darrieus Horizontal axis wind turbine performs between 30% and 40%. Savonius wind turbine can generate sufficient strength to maintain yearly usage of a normal two-individual own family.

4.5.1 Advantages

The rotor shaft of this turbine is Horizontal. So it's far possible to situate the weight just like a generator otherwise a centrifugal pump at earth level. When the generator isn't moving, the wire to the weight isn't twisted and no brushes are needed for large twisting angles.

- The rotor receives wind from all directions.
- without difficulty organized within the buildings
- The association of the windmill over a constructing is probably bigger compared to a horizontal-axis windmill.
- Scalability protection for workers
- Very smooth to perform, so that they don't disappointed human beings in housing neighborhoods.
- portable from one area to any other.
- it is allowable anywhere taller structures are forbidden.
- consists of low-velocity blades in order that it reduces the chance to birds & human beings.
- it really works in severe climate through choppy winds in addition to mountain situations.

4.5.2 Disadvantages

- Low performance
- it is tough to begin as compared to the Savonius wind turbine
- Self-starting mechanism

- Rotation performance is low
- Low performance
- Wind pace is much less available
- aspect wear-down

4.6 SOFTWARE REQUIRED

4.6.1 Introduction to catia v5

Computer Aided Three Dimensional Interactive Application is a CAD/CAM/CAE industrial application package developed by attack systems in France. CATIA is the foundation of the attack systems product lifecycle management software package, written in the C++ programming language.

CATIA competes with Siemens NX, Pro/E, Autodesk Inventor, and many others in the CAD/CAM/CAE sector.

4.6.2 History of Catia

After Marcel Dassault, a French aircraft manufacturer, started using CADAM CAD software to design the Mirage fighter jet in 1977, CATIA quickly spread to other sectors, including aerospace, automotive, shipbuilding, and more. Although it was originally known as CATI (for "Interactive Aided Three-Dimensional Layout"), in 1981 Dassault renamed it CATIA after creating a subsidiary and signing a non-unique distribution agreement with IBM.

Since its first purchase in 1984, Boeing has been CATIA's top client. UNIX was chosen as the platform for porting CATIA version 3 from mainframe computers in 1988. CATIA was chosen as the primary 3-D CAD tool by General Dynamics electric Boat Corp. in 1990 to design the US military's Virginia class submarine. As a result of IBM's purchase of CADAM in 1992, CADAM V4 was released the following year. For the IBM AIX operating system, it was ported to four UNIX systems, including Sun Microsystems' SunOS and Hewlett-HP-UX. Packard's

CATIA V5 was introduced in 1998 and is compatible with UNIX, Windows NT, and Windows XP since 2001, when it was entirely redesigned from the ground up.

CATIA V6 was released by Dassault in 2008. as a result of this, consumer guides for operating systems other than Microsoft Windows are no longer available.

4.6.3 Scope of Application

It is sometimes referred to as a 3-D Product Lifecycle Management software package since CATIA covers several layers of product development (CAD), including conception, design, manufacturing (CAM) and engineering (CAE).

In addition to surface and form planning, mechanical engineering, and system and systems engineering, CATIA facilitates cross-disciplinary collaboration. Form and surface design..

Surface, reverse engineering and visualisation tools are available in CATIA to let you develop, control and evaluate complex novel forms. From A surfaces that have been subdivided, styled, and refined to mechanically functional surfaces. From 3D sketching, sheet metal, composites, moulded, cast or tooling parts and assemblies up to the defining of mechanical assemblies are all supported by CATIA. Tools for comprehensive product definition, including tolerances and kinematics definitions, are available in this system. To provide documentation for manufacturing: CATIA allows the design of electronic, electric and distributed systems as well as fluid and HVAC systems. CATIA provides a way for modelling complex and intelligent goods using the structures engineering process. Essentials definition, architecture, behaviour modelling and embedded software era are all covered.

Utilities programming interfaces allow for the customization of CATIA (API). CATIA V5 and V6 may be customised utilising the primary and C++ programming languages using the CAA (thing utility structure); a COM-like interface for CATIA.

Although NURBS was incorporated in subsequent versions of CATIA V4, the piecewise polynomial floor was the primary floor method in V4.

The powerful engine in CATIA V4 does not have any manifolds. Catia V5 contains a parametric strong/floor-based bundle that uses NURBS as the primary surface depiction and has various workbenches that support KBE..

Enova, Smartian, and several CAE analysis software may all be painted with V5.

From 3D sketching, sheet metallic, composite, and moulded, solid, or tooling components, to mechanical assemblies, CATIA allows for the introduction of 3-D components. Complete product specification, including practical tolerances and kinematics definition, is provided by the equipment in this article. Computer-aided design (CAD): CATIA helps the layout of electronic as well as distributed systems including fluid and HVAC systems, as well as their documentation for manufacture. structures CATIA provides a system engineering method for modelling complex and smart goods. Modeling behaviour and digital products or embedded software are all included in this course. It also includes defining the prerequisites.

4.6.3.1 3d images from catia

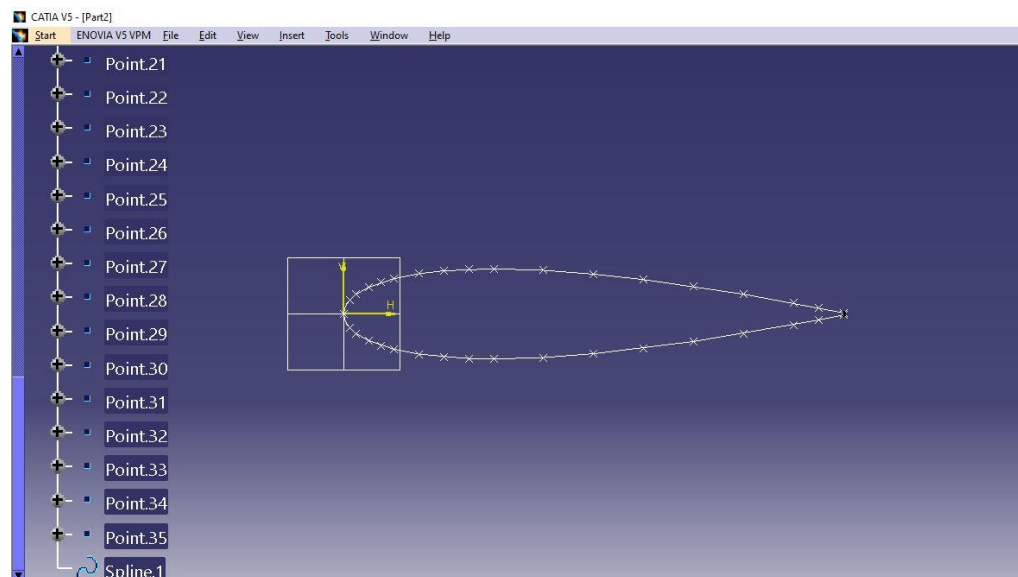


Fig.4.8 Airfoil creation

NACA 63215 airfoil creation in CATIA we have used NACA 63215 airfoil to design wind turbine. NACA series airfoils plays major role in aerodynamic industry. Slots are introduced in airfoil to increase the lift produce. We design wind turbine using this airfoil and then we introduce triangular slots in the wind turbine

4.6.3.2 NACA 63215 airfoil blade (HAWT)

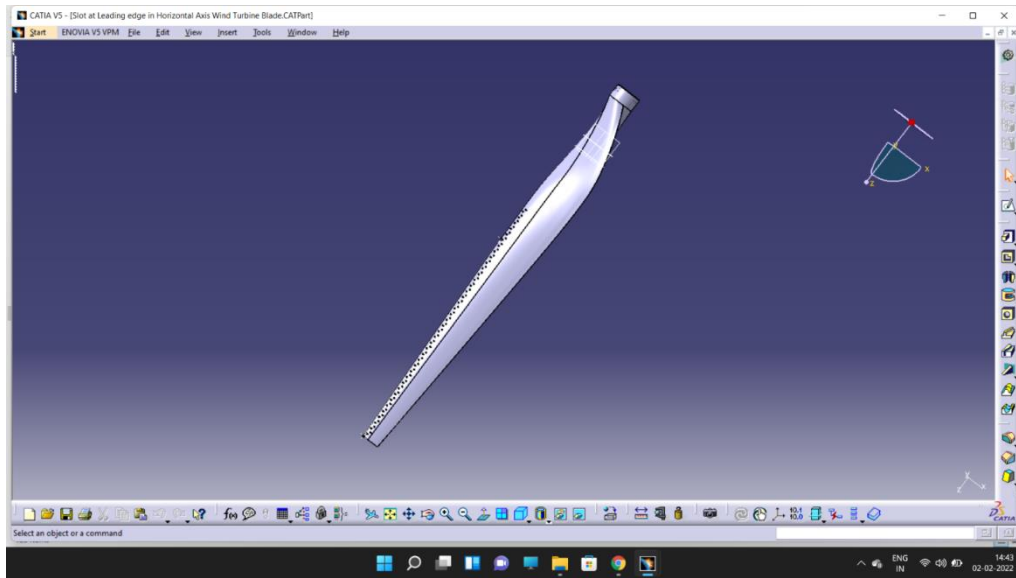


Fig.4.9 Airfoil blade

After designing in CATIA the blade is like this. We design wind turbine blade in CATIA V5 and introduce triangular slots in blade these slots are used to increase lift. As the air flows over the surface these slots will disturb the flow of the air over the surface this disturbance will help with to increase the lift. In aerodynamic industry slots are being introduced a lot nowadays.

4.7 ANSYS

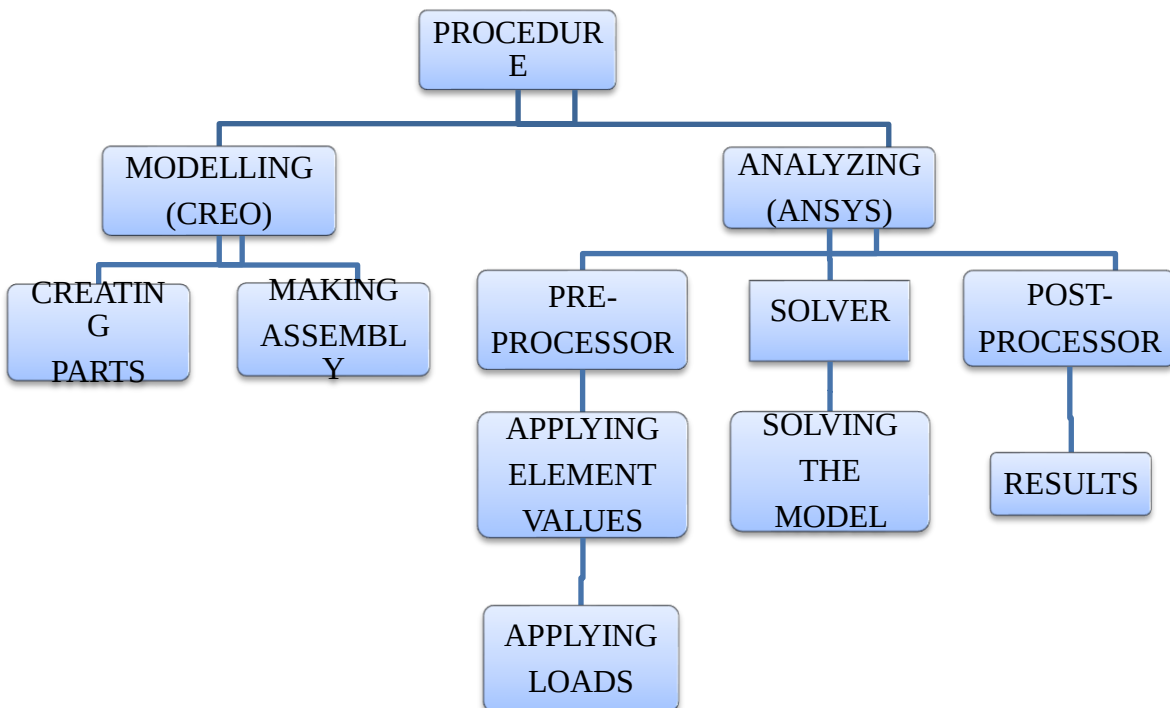
4.7.1 Introduction to ansys

For structural analysis, ANSYS Mechanical is a finite element assessment tool that incorporates linear, nonlinear, and dynamic analyses. There are finite factors to represent behaviour in this computer simulation tool, as well as support for cloth models and equation solvers for a wide range of mechanical layout difficulties. Also included in ANSYS Mechanical is acoustics, piezoelectric, thermal-structural, and thermo-electric powered assessment.

ANSYS offers a complete software program match that spans the entire range of physics, presenting a gateway to actually any subject of engineering simulation that a design method calls for. Agencies around the sector accept as true with ANSYS to deliver the excellent cost for his or her engineering simulation software program investment. The software used for evaluation is ANSYS 14.

4.7.1.1 Software overview:

In order to keep users from being disturbed by the complexities of programming, ANSYS is designed to be as user-friendly and simple as possible. Ansys software allows us to apply any kind of load to any type of part and to view the stress and stress acting at every point of the strong problem. The utmost amount of weight that a strong item can bear is also shown. The stresses and strains present in that version were discovered by an analysis of the provided strong model diagram created in Creo.



Design and analysis software overview

4.7.2 Overview of Ansys Work Bench

Center product solver power is combined with middle product control features needed to manage venture process in Ansys Workbench. Analyses in ansys workbench are created as structures, which may be merged into assignments. Schematic workflow is used to

manage the link between the systems involved in the activity. A local ANSYS workbench utility may be interacted with through schematic, as can a statistics-focused ansys workbench tool. This implies that the interface remains distinct, but utility statistics can connect with local ANSYS workbench facts. The mechanical application is contained in the data supplied in native workspaces for conducting schematic engineering information and design exploration.

types of cells

- Engineering Data
- Geometry
- Model/mesh
- Setup
- Solution
- Results

4.7.3 Getting started to ansys

STEP 1: Release ANSYS, by means of going to begin up menu and double clicking on Workbench record within the ANSYS

STEP 2: Once this system is released it have to look with a few device container. Visit the evaluation system fluid waft (cfx) and double click on

STEP 3: Subsequent double click on the geometry. This degree is for purchasing the required geometry read into the software, word that there is a blue question mark Icon beside the geometry textual content. looking at the bottom of the window you'll home windows one have name of Messages this title confirms that imported geometry has no trouble, subsequent window has a identify development

STEP 4: Once the window is energetic you may get a window to specify working gadgets for the version dimensions selected meters and press ok. box size dimension result in finer mesh is the more accurate is the Captured data

STEP 5: Visit the report and pick out import external geometry file

STEP 6: A window having a identify open might be seen to the consumer, pick Fil type para solid then go to the folder that has required document

STEP7: Looking on the layout modeler window, we can't see the imported Geometry but, press the generate icon that is represented by a yellow thunder icon

STEP8: Visit view and pick wireframe

STEP9: Once we completed that degree manner we've got completed from clothier model and has to proceed to meshing component

STEP10: The meshing part of the challenge has started, notice the beside Mesh they're yellow thunder icon

STEP11: Go to method and pick out tetrahedrons

STEP12: Visit the algorithm and pick patch impartial

STEP13: Press the replace icon after which press on the generate mesh

STEP14: Click on mesh, now its visible to the person the generated mesh

STEP15: Visit the workbench, you may see there may be inexperienced tick beside the mesh congratulation you could now continue to setup

The Tool Bar and Menu Bar

Open a new or present product

- shop a challenge underneath the same call
- Reconnect to updates that had been pending when the assignment become closed
- Refresh a project
- replace the assignment
- Imports a legacy database
- Set your workspace to compact or full mode
- go back to standard mission work area view

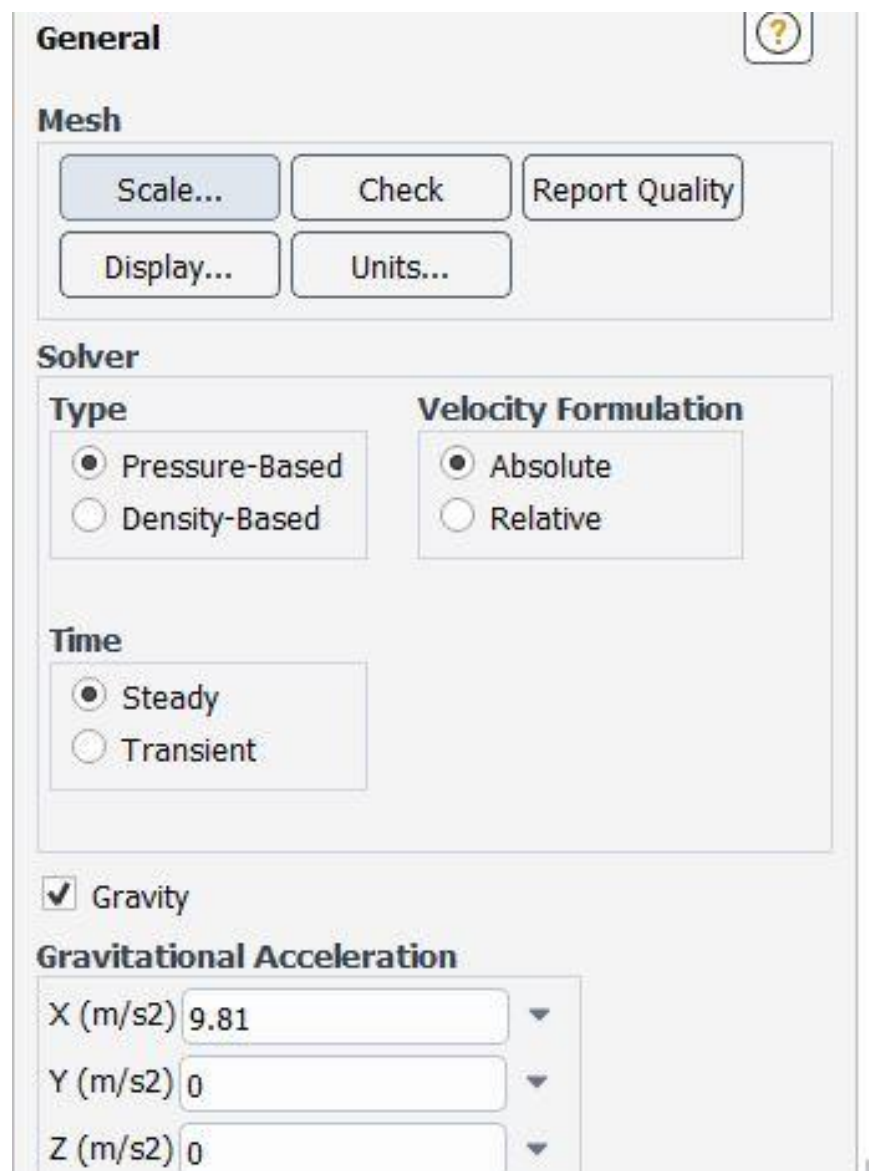
The menu bar gives get entry to

- file menu
- View menu
- gear menu
- Units menu
- assist menu the usage of the chart view

- Chart sorts
- setting chart houses
- Chart zoom, pan, and rotate
- using the triad
- Saving a chart

A variety of loads may be integrated into the stable version, and now all stresses operating at each place are represented in the diagram and colour variation is exhibited between different types of stresses.

4.7.4 ANALYSIS & REPORT IN ANSYS



The screenshot shows the 'General' tab of the ANSYS Analysis input parameters dialog box. It is organized into several sections:

- Mesh**: Contains buttons for 'Scale...', 'Check', 'Report Quality', 'Display...', and 'Units...'.
- Solver**:
 - Type**: Radio buttons for 'Pressure-Based' (selected) and 'Density-Based'.
 - Velocity Formulation**: Radio buttons for 'Absolute' (selected) and 'Relative'.
 - Time**: Radio buttons for 'Steady' (selected) and 'Transient'.
- Gravity**: A checked checkbox.
- Gravitational Acceleration**: Three input fields for X, Y, and Z components in m/s².
 - X (m/s²): 9.81
 - Y (m/s²): 0
 - Z (m/s²): 0

Fig.4.10 Analysis input parameters

For the solver we use pressure based with absolute velocity formulation the constant time under the standard gravity. The type of the solver we use is pressure based.

Viscous Model

Model

- ☐ Inviscid
- ☐ Laminar
- ☐ Spalart-Allmaras (1 eqn)
- ☒ k-epsilon (2 eqn)
- ☐ k-omega (2 eqn)
- ☐ Transition k-kl-omega (3 eqn)
- ☐ Transition SST (4 eqn)
- ☐ Reynolds Stress (7 eqn)
- ☐ Scale-Adaptive Simulation (SAS)
- ☐ Detached Eddy Simulation (DES)
- ☐ Large Eddy Simulation (LES)

k-epsilon Model

- ☒ Standard
- ☐ RNG
- ☐ Realizable

Near-Wall Treatment

- ☒ Standard Wall Functions
- ☐ Scalable Wall Functions
- ☐ Non-Equilibrium Wall Functions
- ☐ Enhanced Wall Treatment
- ☐ Menter-Lechner
- ☐ User-Defined Wall Functions

Model Constants

Cmu: 0.09

C1-Epsilon: 1.44

C2-Epsilon: 1.92

TKE Prandtl Number: 1

TDR Prandtl Number: 1.3

Energy Prandtl Number: 0.85

Wall Prandtl Number:

User-Defined Functions

Turbulent Viscosity: none

Prandtl Numbers

TKE Prandtl Number: none

TDR Prandtl Number: none

Energy Prandtl Number: none

OK Cancel Help

Fig.4.11 Analysis of input parameters

The model we use is standard k-epsilon model with standard wall function.

With turbulent viscosity the TKE Number.

Create/Edit Materials

Name:

Material Type:

Chemical Formula:

Fluent Fluid Materials:

Mixture:

Order Materials by:
☒ Name
☐ Chemical Formula

Fluent Database...
User-Defined Database...

Properties

Density (kg/m3):

Cp (Specific Heat) (j/kg-k):

Thermal Conductivity (w/m-k):

Viscosity (kg/m-s):

Velocity Inlet

Zone Name:

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Velocity Specification Method:

Reference Frame:

Velocity Magnitude (m/s):

Supersonic/Initial Gauge Pressure (pascal):

Turbulence

Specification Method:

Turbulent Intensity (%):

Turbulent Viscosity Ratio:

Fig.4.12 Parameters of input ansys

Magnitude and normal ti boundry is used as velocity specification method.

4.7.4.1 Leading edge slot

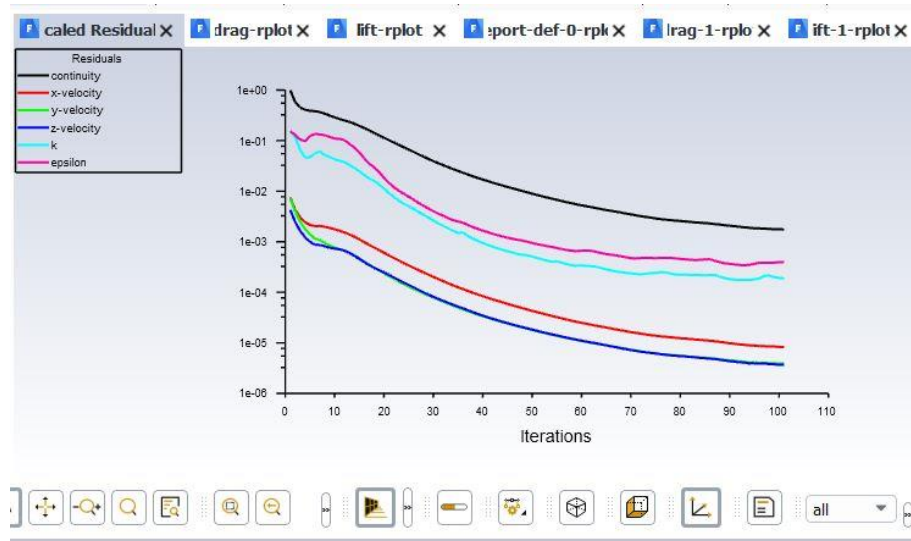


Fig.4.13 (Scaled Residual plots)

This graph shows us the scaled residual plots iterations against y boundary. The above graph shows the iterations of the leading edge slot. We place a slot at the leading edge of the airfoil.

4.7.4.2 Cd vs iteration

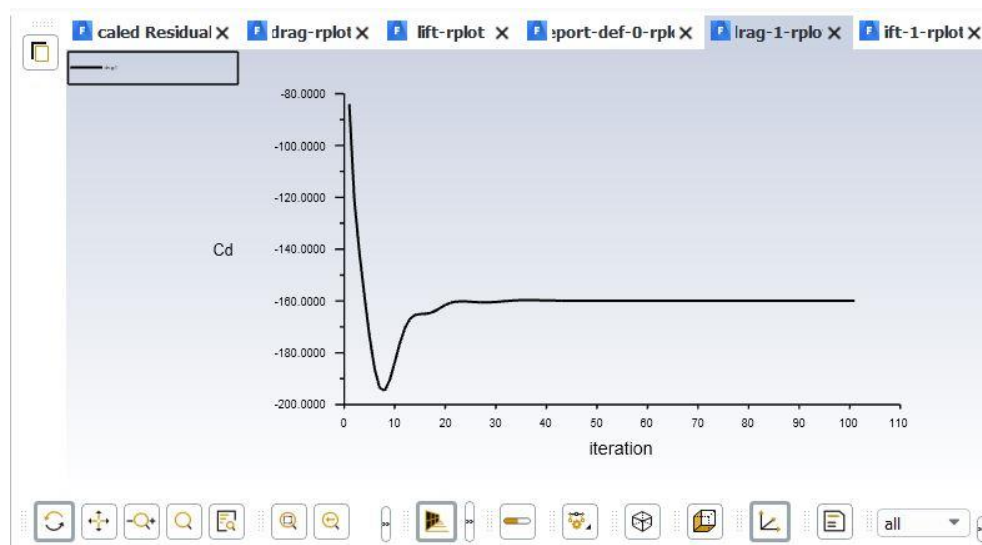


Fig.4.14 Cd vs iteration leading edge slot

Graph.No.4.2 shows C_d vs iteration. The curve shows C_d decrease at 9th iteration. And maintain a constant C_d value at 20.

4.7.4.3 C_l vs iteration

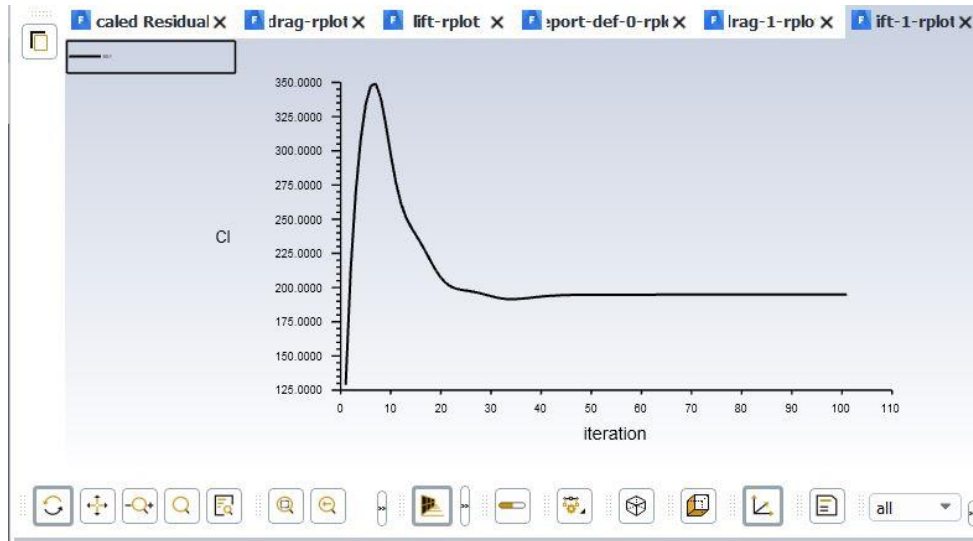


Fig.4.15 C_l vs iteration leading edge

Graph.no.4.3 shows us C_l vs iteration for leading edge slot. The curve increase at 9 for leading edge slot and then maintain constant line

4.7.4.4 pressure contour

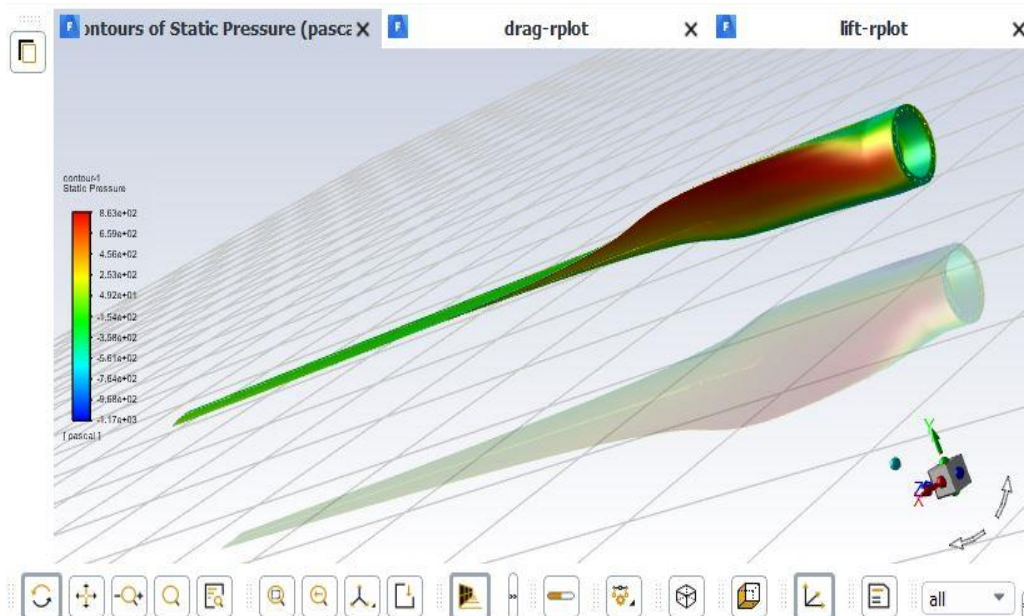


Fig.no.4.16 Pressure contour image for leading slot

Pressure contour for the blade at leading edge looks like this. The above picture shows the pressure acting max at the area covered in red. The pressure acting will be low at the area covered in blue.

4.7.4.5 Velocity contour

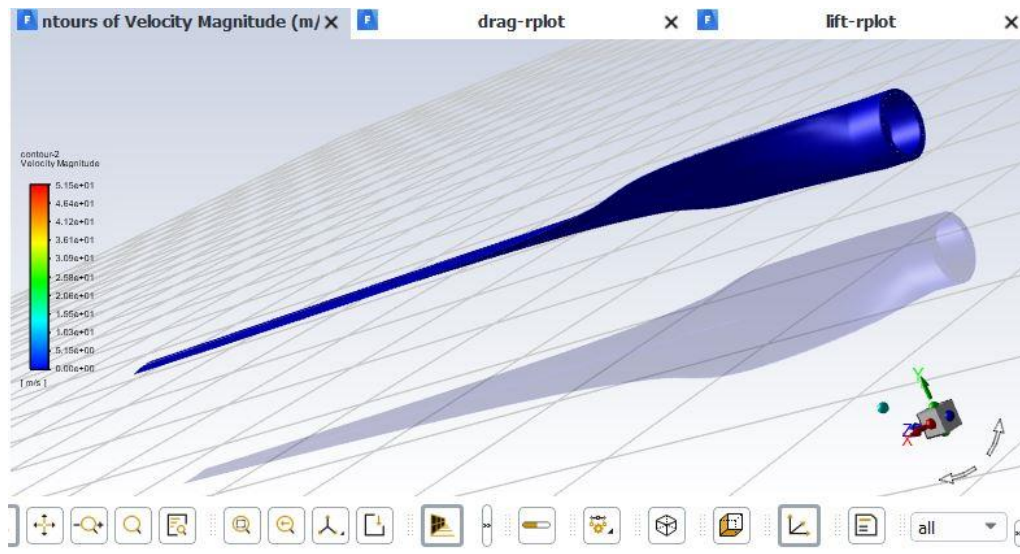


Fig.no.4.17 Velocity contour image for leading edge

Velocity contour for the leading edge slot looks like this. The above picture shows the velocity at the leading edge.

4.7.4.6 Normal triangular slot in the wind blade

4.7.4.7 Cd vs iteration

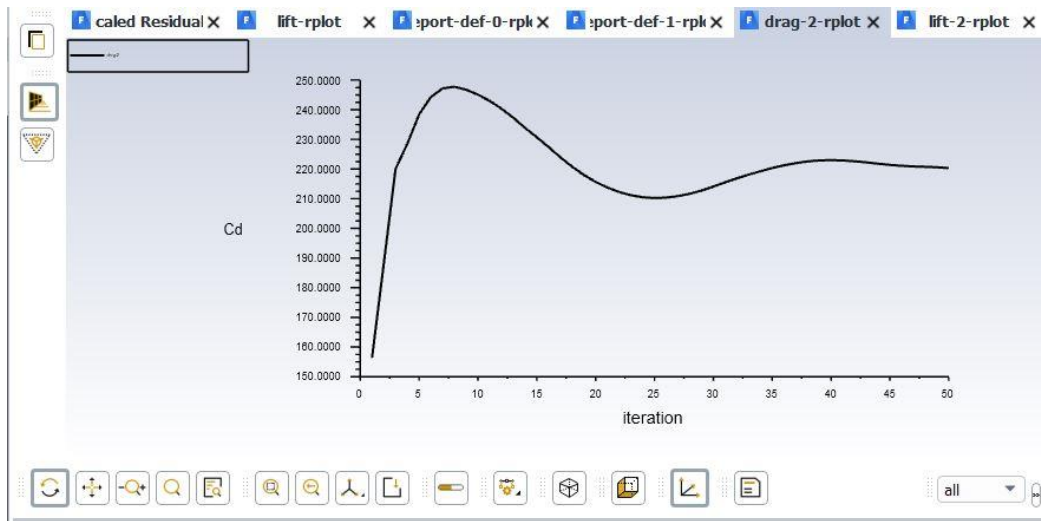


Fig.4.18 (Normal triangular slot in the wind blade Cd vs iteration)

This graph shows us Cd vs iteration for normal triangular slot. The curve shows maximum at 6 and then decrease till 25 the increase slightly.

4.7.4.8 CI vs Iteration

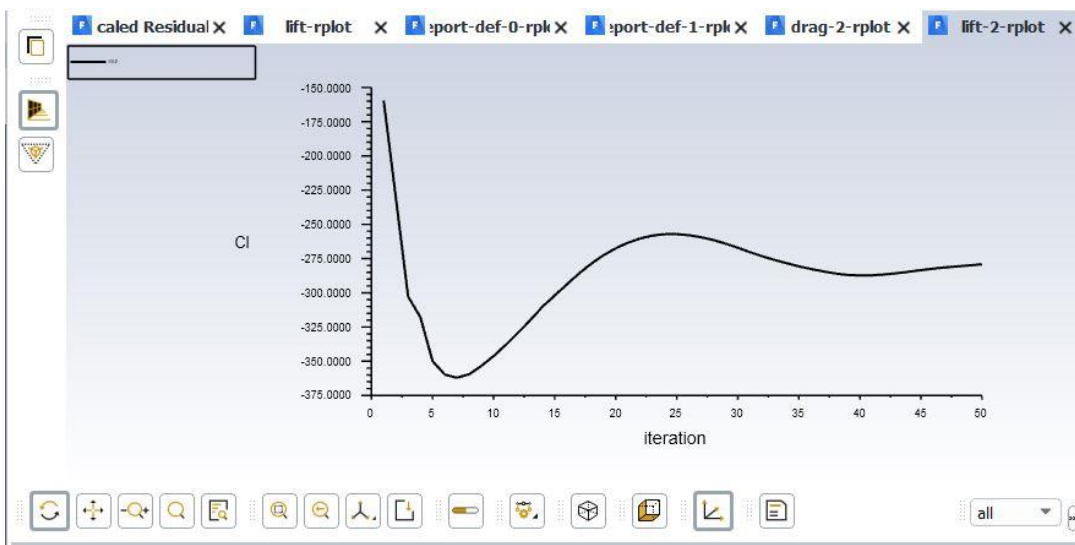


Fig.4.19 Cl vs iteration

This graph shows us Cl vs iteration for normal triangular slot. The curve minimize till 6 then increase till 25 and then gradually decrease.

4.7.4.9 Pressure contour

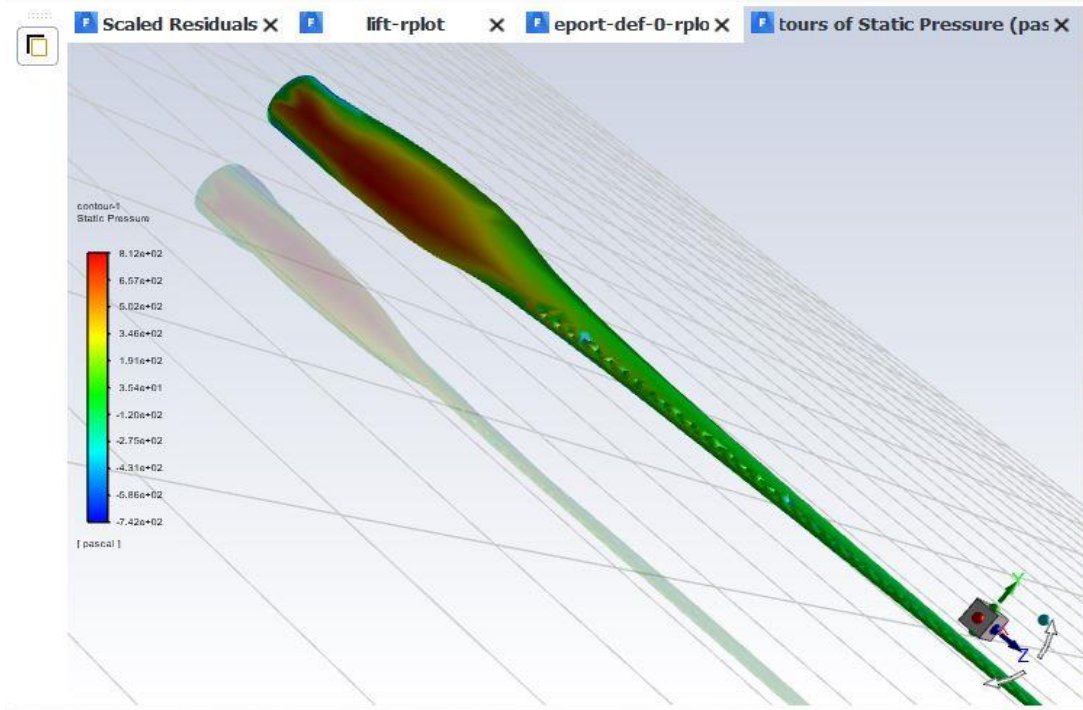


Fig.no.4.20 Pressure contour for normal triangular slot

This fig is for pressure contour for a normal triangular slot.

4.7.4.10 Scaled residuals

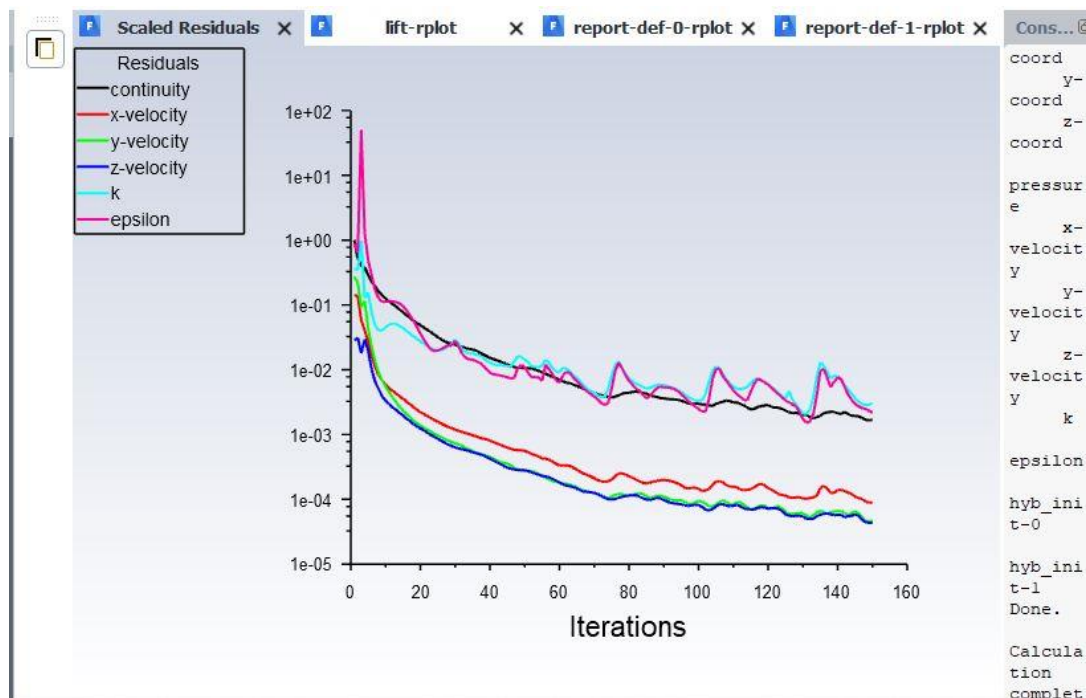


Fig.4.21 scaled residuals

This graph shows us scaled residuals for normal triangular slot.

4.7.4.11 Velocity-contour

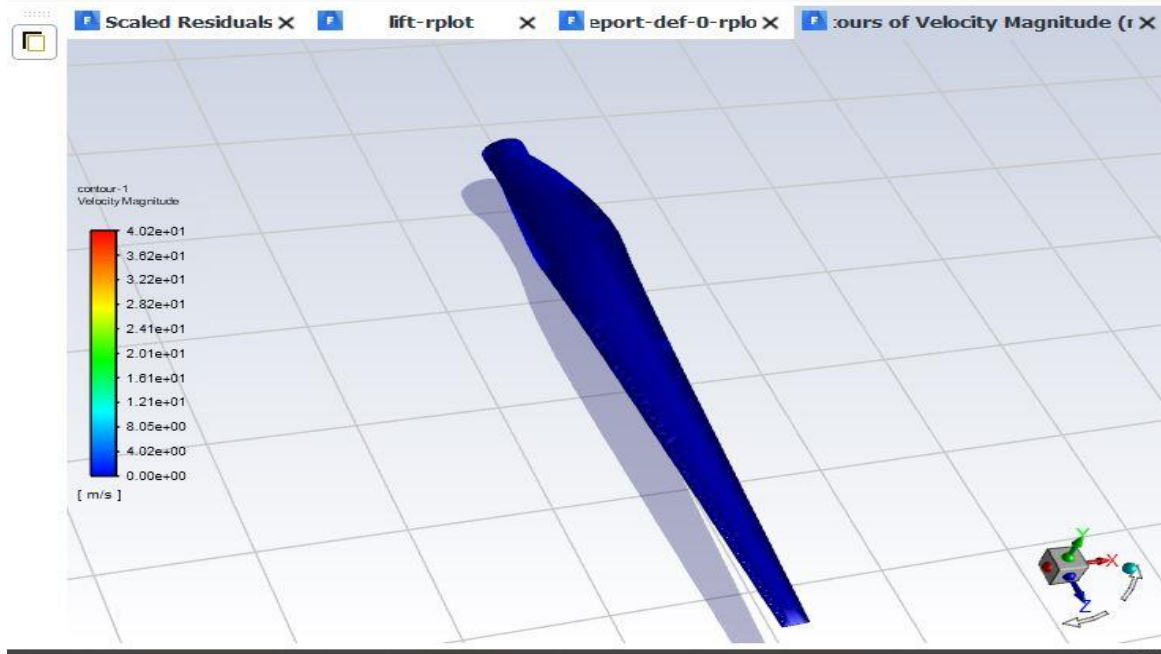


Fig.no.4.22 Velocity contour

This fig shows us the velocity contour for normal triangular slot

4.7.4.12 Trailing edge Triangle slot

4.7.4.13 Cd vs iteration

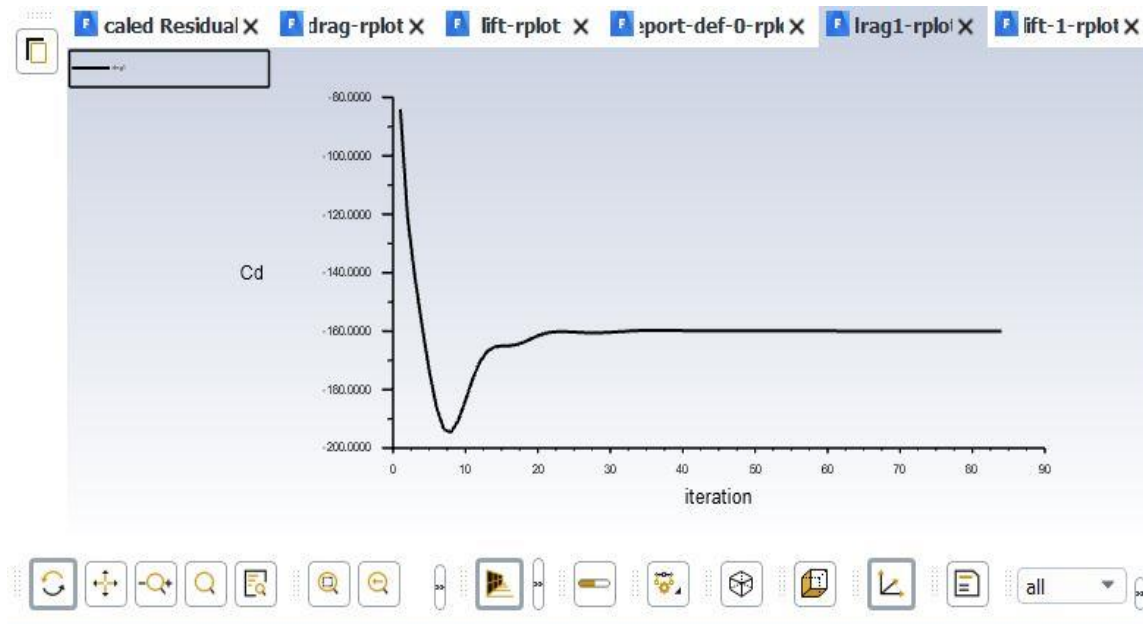


Fig.4.23 Cd vs iteration trailing edge

This graph shows us Cd vs iteration for trailing edge slot

4.7.4.14 Cl vs Iteration

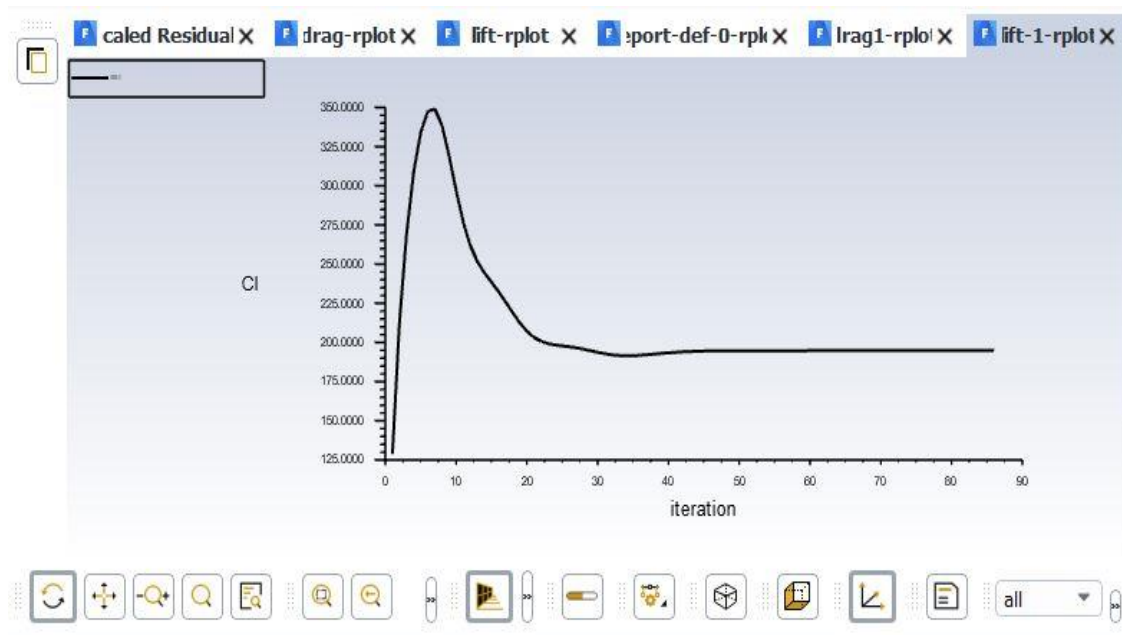


Fig.4.24 C_l vs iteration

This graph shows us C_l vs iteration for trailing edge slot

4.7.4.15 Pressure contour

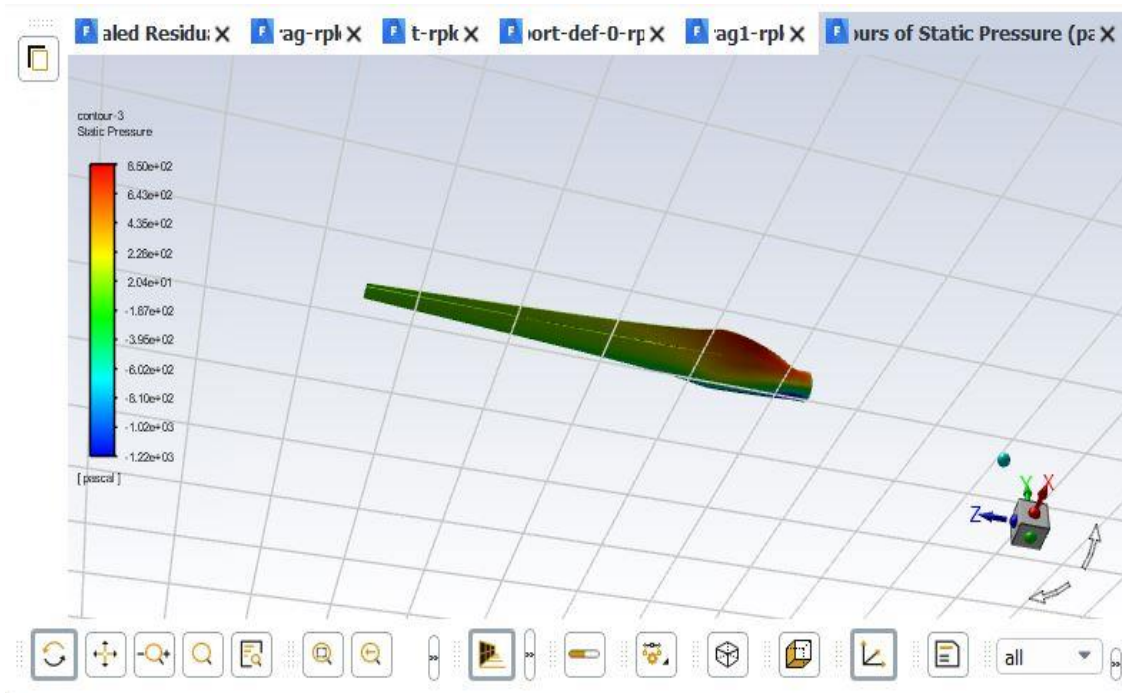


Fig.no.4.25 pressure contour for trailing edge slot

This image is for pressure contour for trailing edge slot

4.7.4.16 Scaled residuals

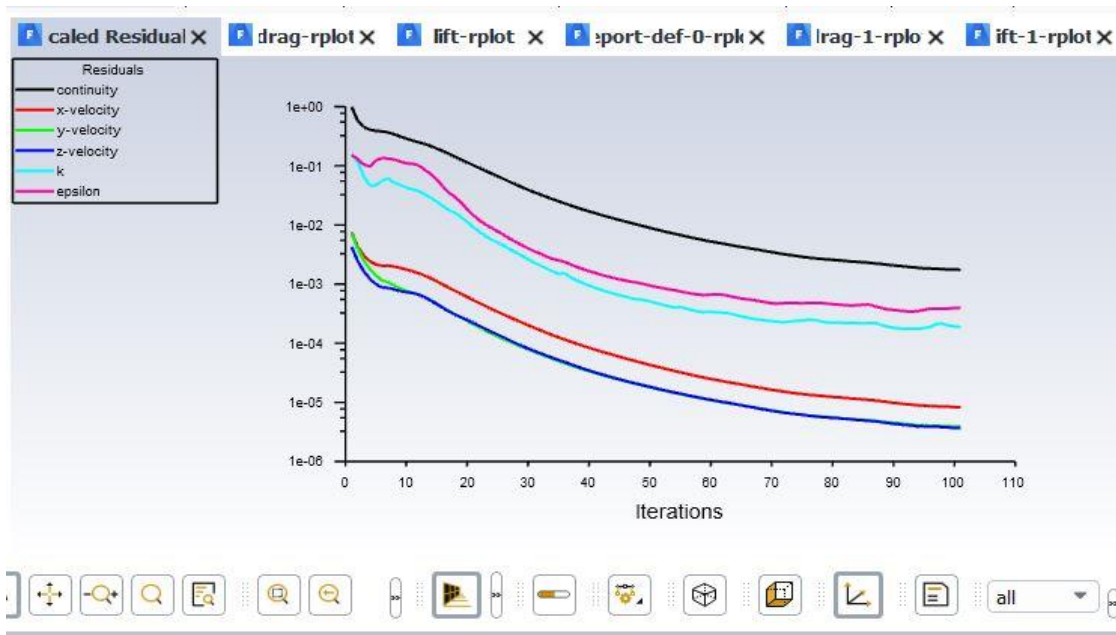


Fig.4.26 Scaled residuals

This graph shows us scaled residuals for trailing edge slot

4.7.4.17 Velocity-contour

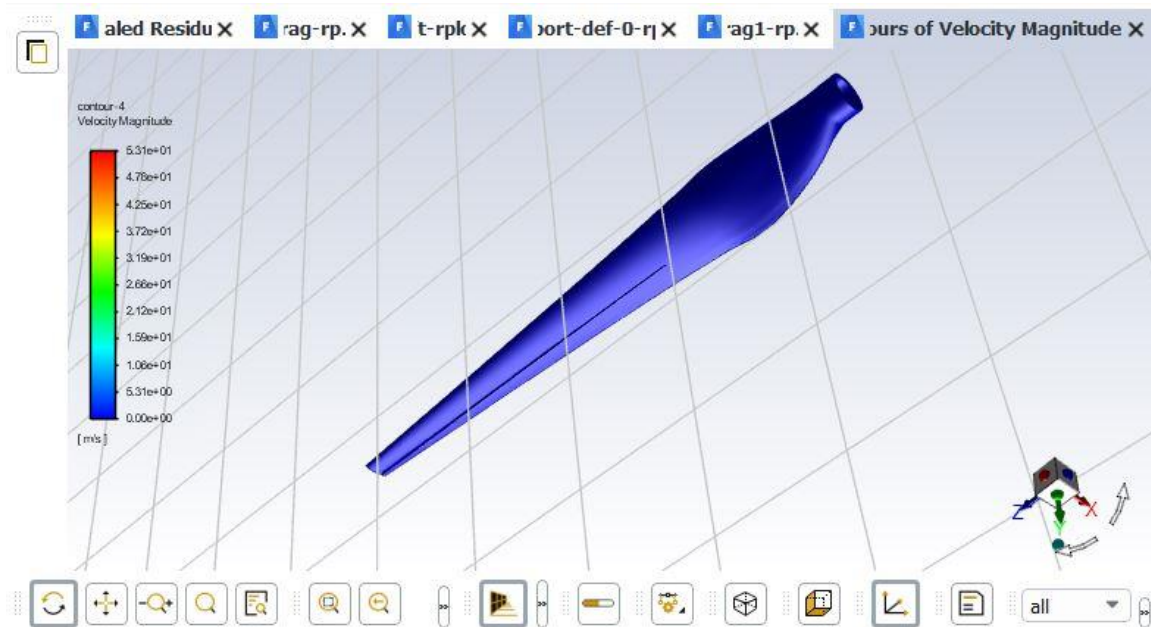


fig.4.27 Velocity contour for trailing edge slot

This fig shows us the velocity in trailing edge slot

4.7.4.18 Middle Triangle Slot

4.7.4.19 Cd vs iteration

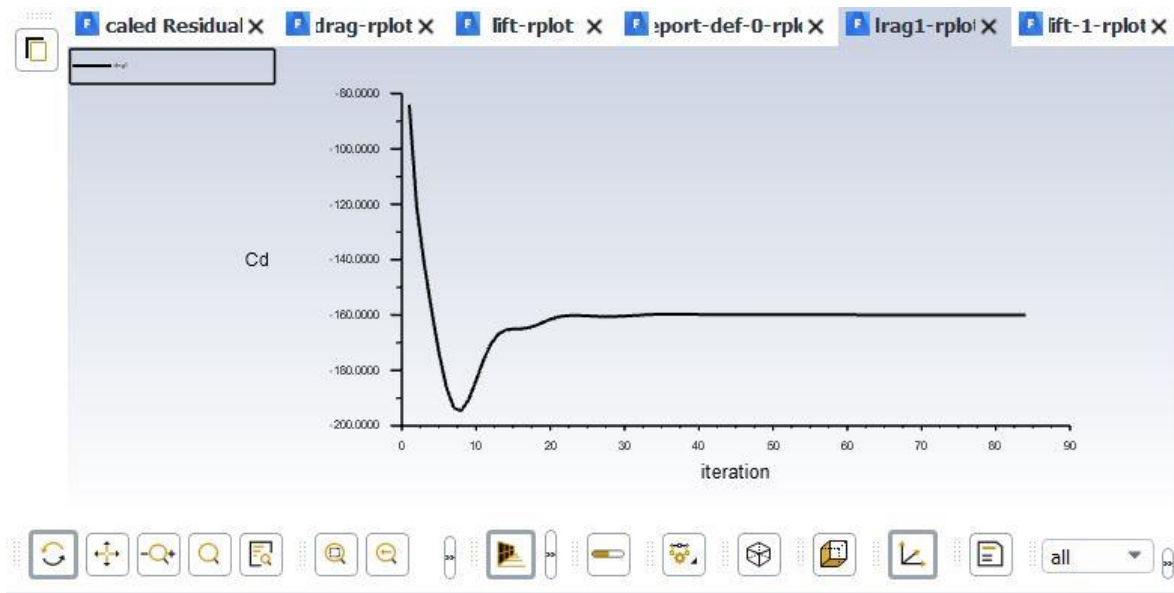


Fig.4.28 C_d vs iteration middle triangular slot

This graph shows us C_d vs iteration for middle triangular slot

4.7.4.20 C_l vs iteration

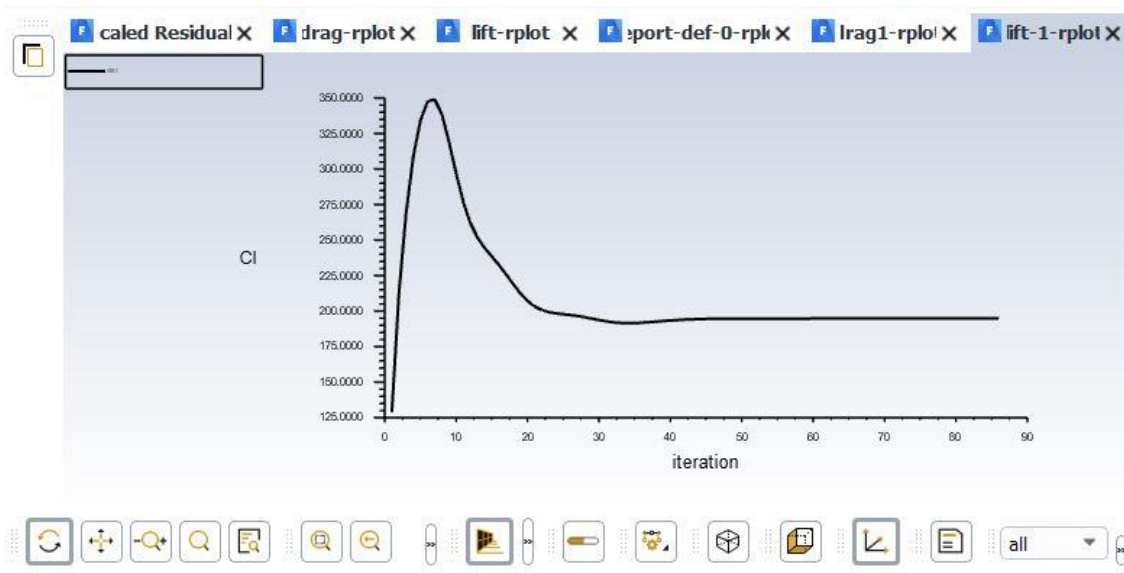


Fig.4.29 C_l vs iteration

This graph shows us C_l vs iteration for middle triangular slot

4.7.4.21 Scaled residuals

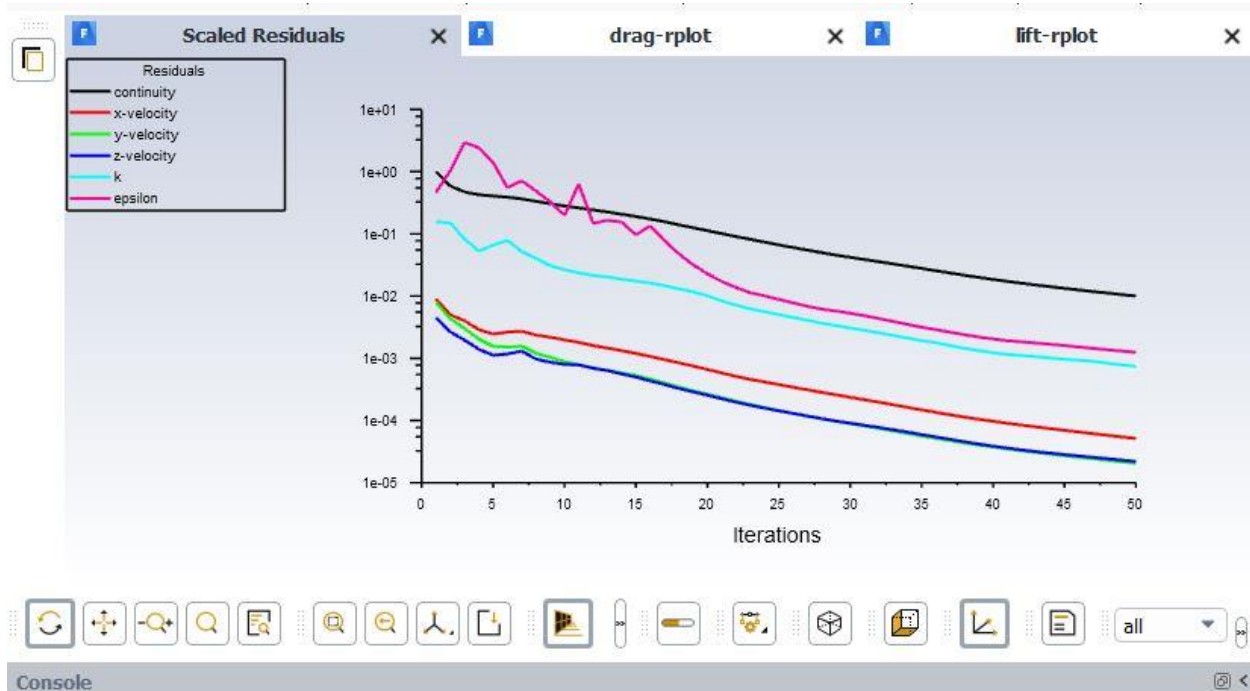


Fig.4.30 Scaled residuals

This graph shows us scaled residuals for middle triangular slot

4.7.4.22 Pressure contour

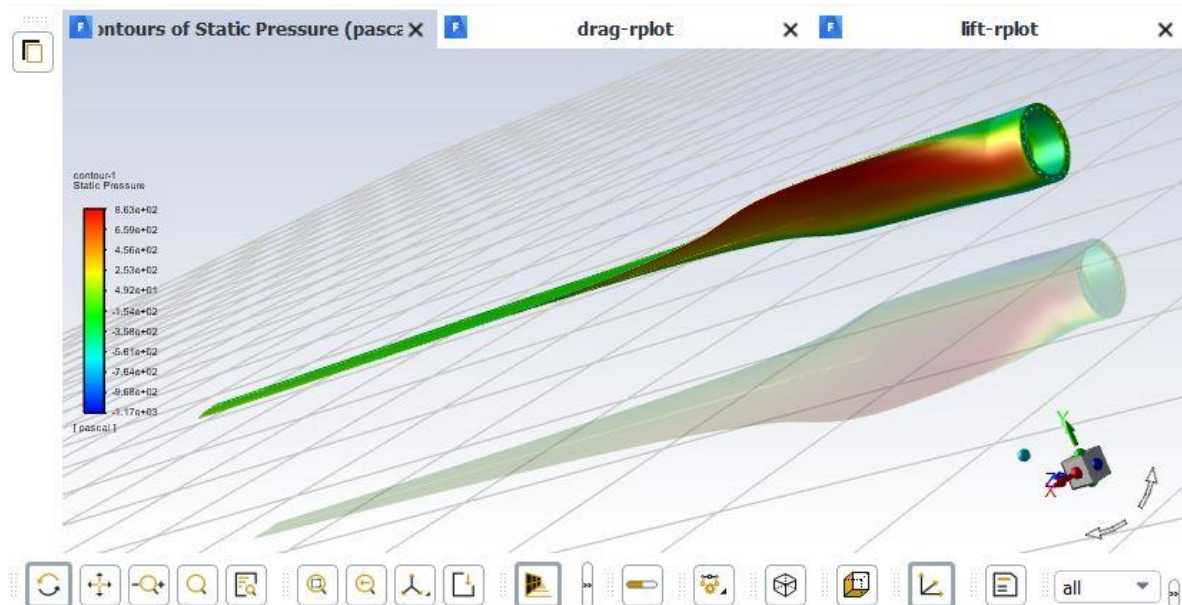


fig.no.4.31 Pressure contour

This fig shows us pressure contour for middle triangular slot

4.7.4.23 Velocity contour

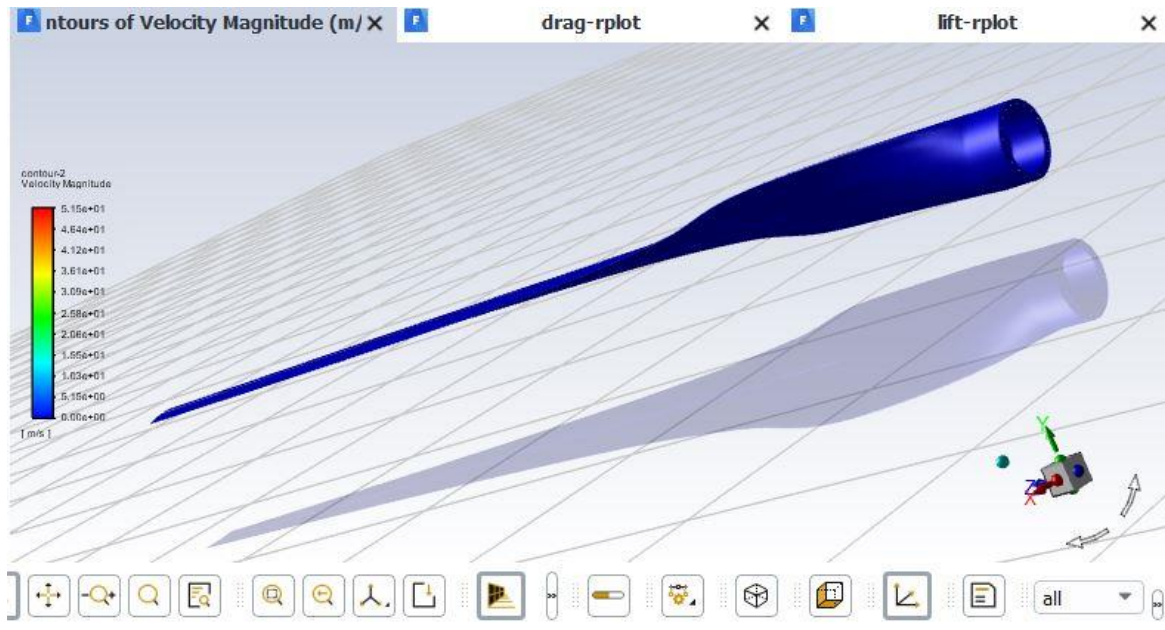


fig.no.4.32 Velocity contour

This fig shows us Velocity contour for middle triangular slot

4.8 Fabrication



Fig.no.4.33 3D Printing Model

For fabrication we did 3D printing using PLA material

CHAPTER-5

RESULT AND DISCUSSION

Finally, when we compare the graphs of already existing model and the Introduced model we have found a big difference. It shows Cd and Cl vs iteration grater than existing model with triangular slots on wind turbine blade of NACA 61325 airfoil. Which best lift producing device. The induced Cd vs iteration and Cl vs iteration at leading edge, triangular slots and trailing edge.

5.1 Experimental analysis

After completion of computation analysis, we carried out experimental analysis. For this we fabricated the design using PLA with the help of 3D printer. We insert 2mm hollow tubes into the 12 tapping with are made around the model in clock wise direction. 6 on the top surface and 6 on the bottom surface. The model with the tube is fitted to the wind tunnel and flow over the surface is studied at different RPM.

Condition of the wind tunnel

- Speed- 100,200,300 RPM
- Manometer liquid - Ethanol

These coefficient of pressure values are calculated using $CP = (PS - PS)/(Pt - Ps)$
Experimental results are as follow

Table.5.1 experimental readings

Ports/ RPM	1	2	3	4	5	6	7	8	9	10	11	12
100	0.9	0.9	0.9	0.9	0.8	0.9	0.9	0.9	0.9	0.9	0.9	0.9
200	1.2	1.0	1.0	1.2	0.9	1.0	0.8	1.2	0.9	1.0	1.2	1.21
300	1.4	1.2	1.0	1.3	1.2	1.4	1.4	1.2	1.3	1.0	1.4	1.4

This table shows us the readings we are taken when the experiment is going on.

Table.5.2 Cp ports

Cp/ Ports	1	2	3	4	5	6	7	8	9	10	11	12
	2	2	2	2	0.3	2	2	2	2	2	2	2
	1	0	0	-1	2	3	1	1	2	0	1	1
	- 0.33	0.33	-1	0	0.3	- 0.33	- 0.33	0.33	0	-1	- 0.33	-0.33

This table shows us the Cp readings for the given readings

1 Calculate x/c and y/c for each port location

Further the x/c which is distance from one tapping to the other in horizontal direction is measured from the fabricated model and y/c is the distance from each tapping to the chord line in vertical direction. Cl, Cd and will be calculated using the x/c and y/c values by choosing only the max negative value and min negative value at every angle of attack

Cl = (max value upper camber- min value lower camber) (x/Cmax-x/Cmin)

100 rpm

Cl = (1.7) (18/21 – 3/21) = 1.207

200 rpm

Cl = (3) (18/21- 3/21) = 2.13

300 rpm

Cl = (-0.7) (18/21- 3/21) = -0.49

Cd = (max value upper camber- min value lower camber) (y/Cmax-y/Cmin)

100 rpm

Cd = (1.7) (1.5/21- 0.5/21) = 0.085

200 rpm

Cd = (3) (1.5/21-0.5/21) = 0.15

300 rpm

Cd = (-0.7) (1.5/21- 0.5/21) = -0.035

Table.5.3 results of Cl, Cd at 100rpm, 200rpm and 300rpm

100 RPM		200RPM		300RPM	
Cl	Cd	Cl	Cd	Cl	Cd
1.207	0.085	2.13	0.15	-0.49	-0.035

After completion of Computational analysis.

CHAPTER-6

CONCLUSION

Wind turbine blade may be very beneficial in our every day life, many engineers are running on that like designing a brand new fashions like we did. we have made the design in CATIA software by means of putting triangular slots on the turbine blade and we did evaluation in kinds they're CFD and model evaluation using ANSYS software program and the effects are stated above and under in table as well. how the raise is producing by means of the turbine blade and drag and co-green of lift and drag of the version by means of this calculations we can say the lift is improved and the drag is progressed in line with the layout we have made the analysis for ordinary blade and the slotted blade the outcomes are given above in the analysis segment so the consequences are given underneath within the desk as you could see. diverse assessments like lift, drag, co-green of carry and co-efficient of drag and ratio of co-efficient of raise to co-green of drag and their outcomes are given underneath.

Table.No.6.1 CFD results

MODEL	L	D	Cl	Cd	L/D	Cl/Cd
Normal Blade	2672.39	- 4783.68	43.63	-78.10	-0.558	-0.558
Trailing edge	1.085	0.266	0.017	0.004	4.078	4.25
Leading edge	1.299	0.311	0.021	0.005	4.176	4.2
Middle slot	0.709	0.294	0.011	0.004	2.411	2.75

Table 6.1 shows the results of normal blade, trailing edge and leading edge results

REFERENCE

- [1]. P.Larin , M.Paraschivoiu,C.Aygun - Journal of Wind Engineering and Industrial Aerodynamics, J. Wind Eng. Ind. Aerodyn. 156 (2016)
- [2]. San-Yih Lin, Yang-You Lin, Chi-Jeng Bai, Wei-Cheng Wang - Performance analysis of Horizontal-axis-wind
- [3]. turbine blade with modified trailing edge through computational fluid dynamics, Renewable Energy (2016).
- [4]. P.-L. Delafin, T. Nishino, A. Kolios, L. Wang - Comparison of low-order aerodynamic models and RANS CFD for full scale 3D Horizontal axis wind turbines, Renewable Energy (2017).
- [5]. Natapol Korprasertsak, Thananchai Leephakpreeda - Analysis and optimal design of wind boosters for Horizontal Axis Wind Turbines at low wind speed, Journal of Wind Engineering and Industrial Aerodynamics , J. Wind Eng. Ind. Aerodyn. (2016)
- [6]. M.M.S.R.S. Bhargava, Velamati Ratna Kishorea, Vaitla Laxman - Influence of fluctuating wind conditions on Horizontal axis wind turbine using a three dimensional CFD model , J. Wind Eng. Ind. Aerodyn. (2016).
- [7]. Yu-Hsien Lin, Jing-Fu Chena, Po-Ying Lu - A CFD model for simulating wave run-ups and wave loads in case of different wind turbine foundations influenced by nonlinear waves, Ocean Engineering (2017)
- [8]. Sean Quallen, Tao Xing- CFD simulation of a floating offshore wind turbine system using avariable-speed generator-torque controller, Renewable Energy 97 (2016)
- [9]. Qing'an Li,Takao Maeda ,Yasunari Kamada, Junsuke Murata,Toshiaki Kawabata,Kento Shimizu,Tatsuhiko Ogasawara,Alisa Nakai,Takuji Kasuya - Wind tunnel and numerical study of a straight-bladed Horizontal axis wind turbine in three-dimensional analysis (Part I: For predicting aerodynamic loads and performance) , Energy(2016).
- [10]. MD. Saddam Hussien, Dr. K. Rambabu,M. Ramji, E. Srinivas - Design and Analysis of Horizontal Axis Wind Turbine Rotors.
- [11]. N. Dibakara Reddy , Surender Singh, N.Tejeswar Reddy – CFD analysis on a savonious rotor wind turbine made using mild steel.
- [12]. Mayank D . Patel, Tushar. P. Gundarneeya – Performance analysis of Horizontal wind axis turbine with comparison of CFD and experimental analysis.

- [13]. Kadhim Suffer, Ryspek Usubamatov, Ghulam Quadir , Khairul Ismail - Modeling and Numerical Simulation of a Horizontal Axis Wind Turbine Having Cavity Vanes ,2014.
- [14]. Dr.P.M.Ghanegaonkar , Ramesh K.Kawade , Sharad Garg - Conceptual Model of Horizontal Axis Wind Turbine and CFD analysis, May 2014.
- [15]. Kyung Chun Kim, Ho Seong Ji, Yoon Kee kim, Qian Lu, Joon Ho Back, Rinus Mieremet- Experimental and Numerical study of the Aerodynamic characteristics of an Archimedes Spiral Wind turbine Blade, 2014.