Optimizing Small-Scale HAWT Blade Performance via Compressed Fluid Dynamics

Dr. Harjit U. Pawar[1], Dr. Harshal S. Rane[2], Dr. U. S. Ansari[3], Dr. Pundlik N. Patil[4], Dr. Harshvardhan P. Ghongade[5], Dr. Anjali A. Bhadre[6]

[1]Head, Mechanical Engineering, SND COE & RC Yeola, Nashik, India
Email-harjitpawar1100@gmail.com
[2]Head, Mechatronics Engineering, SND COE & RC Yeola, Nashik, India
Email- harshal17rane@gmail.com
[3]Head, Civil Engineering, SND COE & RC Yeola, Nashik, India
Email- ansariubaids@gmail.com
[4]Associate Professor, Mechanical Engineering, KBS's COET NMKC, Jalgaon, India
[5]Assistant Professor, Mechanical Engineering, BVCOE & RI, Nashik, India
[6]Associate Professor, Computer Engineering, GHR COE&M, Pune, India

Abstract
Generation of power from wind is most common process now a days. The wind is available freely due to the uneven changes in the temperature of wind because of solar heating. It may say that wind energy is indirect source of solar energy. The device which achieves this energy conversion from wind energy to electrical power is known as wind turbine (WT).

The wind energy conversion includes wind blades, turbine and generator system. So, it is need that the maximum energy of wind is converted into electrical power. The wind energy conversion takes place by large WT blades. This conversion requires certain angle of attack of wind to produce power.

The study presented here is the CFD analysis of WT blades using different Angle of Attacks (AoA). Several models were developed in the solid modeling software and analyzed in ANSYS Fluent for CFD analysis. The various AoA are taken into consideration to find the optimum one. This work is useful in designing the table top model of WT to generate small power (Usually up to 10W). At the end, conclusion were discussed that 12 degree AoA WT blades are optimum for producing power.

1 Introduction

A wind turbine is a device that converts the kinetic energy of wind into electrical energy. As of 2020, hundreds of thousands of large turbines in installations known as wind farms generated over 650 gigawatts of power, with an annual addition of approximately 60 gigawatts. Wind turbines have become a crucial source of intermittent renewable energy, utilized globally to lower energy costs and reduce dependence on fossil fuels. A study from 2009 indicated that wind energy had the lowest relative greenhouse gas emissions, minimal water consumption, and the most favorable social impacts compared to other energy sources such as photovoltaic, hydro, geothermal, coal, and gas.
In addition to large-scale applications, smaller wind turbines serve various purposes such as charging batteries for boats or caravans and powering traffic warning signs. Larger turbines can

also contribute to domestic power supplies while allowing users to sell excess electricity back to the grid. Wind turbines come in a variety of sizes and configurations, including horizontal-axis and vertical-axis designs.Small wind turbines, often referred to as micro wind turbines, are designed for small-scale electricity generation.

Typically smaller than those used in wind farms, these turbines usually feature passive yaw systems instead of active ones. They employ direct drive generators and utilize tail fins to orient themselves into the wind, contrasting with larger turbines that often have geared powertrains actively adjusted for optimal wind capture. Small wind turbines generally produce between 500 watts and 10 kilowatts, with some models as small as 50 watts. The Canadian Wind Energy Association classifies small wind turbines as those with a capacity of up to 300 kilowatts, while the International Electrotechnical Commission (IEC) defines them as having a rotor area smaller than 200 square meters and generating voltage below 1000 volts AC or 1500 volts DC.

The primary application of wind turbines remains traditional power generation. The design of turbine blades is crucial for maximizing energy output at rated speeds, with the angle of attack (AoA) being a critical factor in optimizing blade performance. Understanding and optimizing AoA is essential for enhancing the efficiency of small-scale wind turbine designs and improving overall energy conversion from wind resources. Recent advancements in wind turbine technology have significantly improved efficiency and capacity.

Modern turbines now feature longer and lighter rotor blades, taller towers, and advanced control systems designed to optimize performance under varying wind conditions. For instance, the average rated capacity of newly installed land-based wind turbines in the U.S. reached 3.2 megawatts in 2022, marking a substantial increase from previous years. This trend reflects a broader movement towards larger turbine designs that can harness more energy from available wind resources. Moreover, innovations such as direct drive systems and permanent magnet generators are enhancing turbine reliability and efficiency by eliminating the need for traditional gearboxes. The use of advanced materials like carbon fiber composites in rotor blades is also making them lighter and more durable, further improving performance.

As land availability becomes increasingly constrained especially in urban areas enhancing turbine efficiency allows for greater energy generation per unit area. This not only reduces the number of turbines needed at a site but also lowers transportation and installation costs while minimizing operational challenges throughout the project's lifespan.

In summary, the evolution of wind turbine technology continues to play a pivotal role in meeting global energy demands sustainably. By optimizing blade design and understanding factors like AoA, we can significantly enhance the performance of small-scale wind turbines, contributing to cleaner energy solutions and supporting efforts to combat climate change.

1.1 Horizontal Axis Wind Turbine (HAWT)

Large three-bladed horizontal-axis wind turbines (HAWT) with blades positioned upwind of the tower are responsible for the majority of wind power generation worldwide. These turbines feature a main rotor shaft and electrical generator located at the top of a tall tower, necessitating that they be oriented into the wind for optimal performance. Smaller HAWTs are typically aligned using a simple wind vane, while larger models employ sophisticated yaw systems that integrate wind sensors to maintain alignment with changing wind directions. Most HAWTs are equipped with gearboxes that convert the slow rotational speed of the blades into a faster rotation suitable for driving an electrical generator.

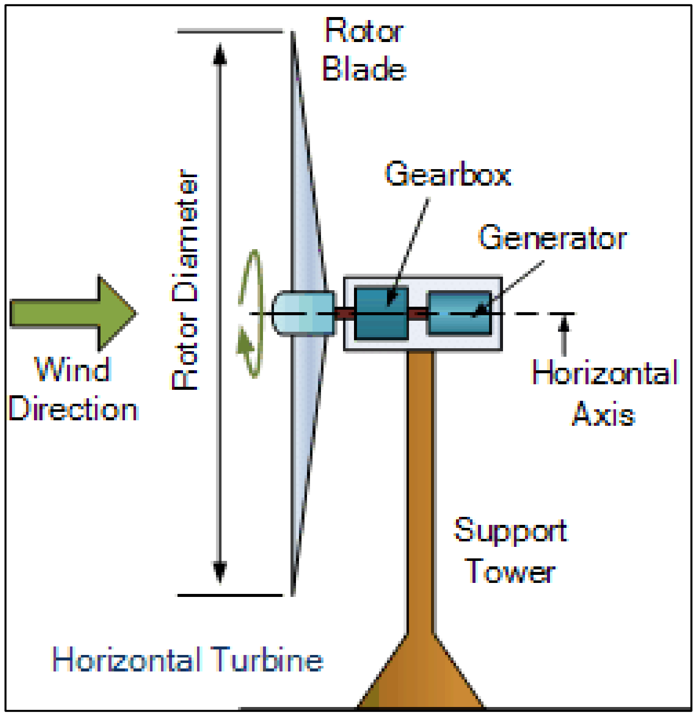


Figure 1.1 Horizontal Axis Wind Turbine (HAWT)

However, some turbines utilize direct-drive generators, which connect the rotor directly to the generator without a gearbox. While these permanent magnet direct-drive systems can be more expensive due to the rare earth materials required, they offer advantages such as reduced mechanical complexity and lower maintenance costs. Gearless turbines eliminate the gear-speed increaser, which is prone to fatigue and reliability issues, making them an appealing alternative for certain applications. Additionally, there is a pseudo direct-drive mechanism that offers advantages over traditional designs.

Typically, HAWTs have their rotors positioned upwind of the supporting tower, which helps mitigate potential damage from changing wind loads as blades pass behind the tower. Although downwind configurations do not require additional mechanisms for wind alignment and can allow blades to bend during high winds thereby reducing their swept area and wind resistance—upwind designs remain favored due to their overall reliability.

Commercially viable HAWTs are predominantly three-bladed due to their low torque ripple, which enhances operational reliability. These blades are usually painted white for visibility by aircraft and range in length from 20 to 80 meters. The trend in turbine design has been towards larger sizes, with offshore wind turbines now reaching capacities of up to 8 MW and blade lengths extending to 80 meters. Prototypes for even larger designs, ranging from 10 to 12 MW, were in development as of 2018, with plans for a "15 MW+" prototype featuring three blades each measuring 118 meters scheduled for construction in 2022.

Multi-megawatt turbines typically feature tubular steel towers ranging from 70 meters to 120 meters in height, with some extreme cases reaching up to 160 meters. The efficiency of HAWTs is influenced by various factors including rotor design, blade shape, and operational parameters such as the angle of attack (AoA). Research utilizing **Blade Element Momentum (BEM)** theory has focused on optimizing chord length and twist angle along the blade span to maximize lift-to-drag ratios and improve overall turbine performance. For example, studies have shown that optimizing these parameters can significantly enhance energy capture from wind resources. One study demonstrated that by varying airfoil shapes and optimizing external geometric parameters such as chord lengths and twist angles, a maximum power coefficient (Cp*Cp*) of nearly 47% was achieved at a fixed tip speed ratio (TSR) under low wind speed conditions of 3 m/s.
Recent advancements in turbine technology have led to improvements in efficiency and capacity. Modern turbines now feature longer and lighter rotor blades made from composite materials like carbon fiber, which enhance durability while reducing weight. This combination allows for greater energy capture while minimizing structural stress on the turbine components. Furthermore, innovations in control systems enable real-time adjustments based on wind conditions, optimizing performance further. Despite their advantages, HAWTs face environmental challenges including noise pollution, aesthetic concerns in landscapes, and risks to avian populations. Ongoing research aims to address these issues through improved designs and technologies that minimize negative impacts while maximizing energy output. For instance, dimpled structures on turbine blade surfaces have been shown to improve airflow over the blades, reducing drag and enhancing lift-to-drag ratios.
horizontal-axis wind turbines play a pivotal role in harnessing wind energy efficiently. As the industry continues to evolve with technological advancements and innovative designs, HAWTs are expected to remain at the forefront of renewable energy generation, contributing significantly to global efforts aimed at reducing carbon emissions and promoting sustainable energy solutions. The ongoing optimization of design parameter such as blade geometry and control mechanisms—will be crucial in maximizing their performance in an increasingly competitive energy landscape.

1.2 Effect of AOA on Turbine blade Performance

A wind turbine is a rotary device that extracts energy from the wind. Wind energy has been shown to be one of the most viable sources of renewable energy. With current technology the low cost of wind energy is competitive with more conventional sources of energy such as coal. Rotor blade is a key element in a wind turbine generator system to convert wind energy in to mechanical energy. Most blades available for commercial grade wind turbines incorporate airfoil shaped cross sections. These blades are found to be very efficient at lower wind speeds in comparison to the potential energy that can be extracted. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary condition.

The Angle of Attack (AOA) plays main role in power generation. Since, according to blitz limit, a maximum power generated by wind turbines is 59.3% of input power of wind. So, this power can be improved by changing the AOA for blades. The study considered here is the validation of theory that shows power generated by the turbine varies with various angle of attack. For power generation by turbine, drag force should be minimum and lift force should be maximum. Thus study investigated the performance of blades at various AOA.

2 Literature Survey

Srinivas G et al (2014) has suggested that blades play an important role in converting wind energy into electrical energy. When the Wind attack to the blade, reaction force produce in form of lift and drag forces , this Forces are Calculated by Numerical method but because of complication and also it is a tedious work we can also go for another method, That is Computation Fluid Dynamics (CFD) method for getting desired results. CFD method is based on fluid mechanics, Its Function is to use blade air-profile and consider a 2 dimensional profile Choose by Design Foil Workshop for various chords (Abbott et al Report No.824) at different Angle of Attack of air and also in Different Reynolds Number.

R. B. Gowardhanand et al (2014) has presented an aerodynamic design of blade using CFD analysis. The objective of this project is to increase the efficiency of wind turbine by reducing the drag and lift. This present work is done in designing a wind turbine blade by using CREO-ELEMENT/PRO 5.0 Software and optimizing the blade aerodynamically by using CFD analysis in ANSYS 11.0 Software. In this project a wind turbine blade is design by using CREO-ELEMENT/PRO 5.0 Software and optimize the blade aerodynamically by using CFD analysis in ANSYS 11.0 Software. The maximum value of coefficient of performance (CPmax = 0.44) was observed at velocity of air 3 m/s. and 6 rpm. This blade can generate maximum power of 64199.73 Watts at maximum CP. From analysis we concluded that ANSYS-Fluent shows a good performance in calculating the lift, drag and moment coefficients of aerofoils. The wind turbine blade designed by this method has good aerodynamic performance in low wind speed conditions.

Mayurkumar kevadiya et al. (2013) has study the aerodynamic airfoils of wind turbine blades have crucial influence on aerodynamic efficiency of wind turbine. This involves the selection of a suitable airfoil section for the proposed wind turbine blade. In this paper NACA 4412 airfoil profile is considered for analysis of wind turbine blade. Geometry of the airfoil is created using GAMBIT 2.4.6. And CFD analysis is carried out using FLUENT 6.3.26 at various angles of attack from 0° to 12. The coefficient of lift and drag values are calculated for 1×10^5 Reynolds number. And it result the coefficient of Lift and drag is calculated for this NACA 4412 series for the angle of attack 0° to 12°. The coefficient of Lift/Drag ratio increases with increase in Angle of attack up to 8°. After 8°, Lift/Drag ratio decreases with increase in Angle of attack.

Hardik Patel et al (2013) has carried out Computational Fluid Dynamics (CFD) analysis of wind turbine blade with complete drawing and details of sub-system. First the type of airfoil is used is decided. Then find out the airfoil co-ordinate for drawing the airfoil shape. Here NACA 0018 selected for the analysis. Here the maximum thickness is 18% which indicate the maximum thickness (in per cent of the chord).In this analysis, the geometry is prepared in the pro-e software package and then after it is saved in iges format. Then import this geometry in the ANSYS 12.0. Form this study they conclude as following:

1 The maximum value of coefficient of performance (CPmax = 0.277191) was observed at angle of attack 60 and 70 and the velocity of 26 m/s.
2 This blade can generate maximum power of 3374 w at maximum CP, at angle of attack 60 and velocity of air 26m/s.
3 It was observed that value of numerical power increases as angle of attack increases from 10 to 70, after 70 the value of numerical power reduced. Hence critical angle of attack for this blade is 70.

Fei-Bin Hsiao et al (2013) has suggested three different HAWT use at a different condition like Optimum Blade Shape, Optimum twist blade and Untwist blade, the HAWT blade geometry is a NACA 4418. The above this condition is to be experimented in CFD software use a k-ω SST turbulence model. We get some result in experiment work, that the
Optimum Blade (OPT) is more efficient to the untapped and optimum twist blade at a wide range of power coefficient of tip speed ratio which from 4.5 to 7 and untapered and untwist blade operates in lowest Cp value..

Farooq Ahmad Najar et al. (2013) in presented paper they done CFD analysis of NREL S809 Airfoil by selecting various numbers of solver and compare it to experimental data of wind tunnel test and conclude that k-e standard wall function is best match with experimental data than other solver like a S-A, SST, etc. NREL S809 gives a maximum performance at 140, angle of attack.

R. Mukesh et al. (2013) in presented paper they have selected a NACA 2411 airfoil as base shape for optimization process. The airfoil was described using the PARSEC parameterization scheme. The flow around the airfoil was solved using the Panel method. And finally optimized airfoil shape is validating by using an experimental validation. GA is used for optimization and at 5 of Angle of Attack airfoil analysis is carried out.

Monir Chandrala et al. (2013) in this paper, author selected NACA 0018 airfoil is designed and analyzed for different blade angle at constant wind speed 32 m/s. The CFD analysis is carried out using ANSYS CFX software. The velocity and pressure distribution at various blade angles is different. These results match with the wind tunnel experimental values. Hence the results are validated with the experimental work. The optimum value of power has been achieved at a blade angle 10° for 32 m/s wind speed. In this paper flat blade with single airfoil is considered for an analysis.

H. V. Mahawadiwar et al. (2012) in this research paper, author have an already a design of blade, which is use for CFD analysis. They modeling that in the Pro-E and then generate a mesh in GAMBIT and after then CFD analysis is carried out in fluent software. They also compute Numerical power and conclude that the numerical power is increase as AoA increase from 0° to 7°, after 7° the value of Numerical power is suddenly reduce from these effect we can say that after 7° AoA there is a stall effect is produce. Maximum value of Co-efficient of power was observed at AoA 7° and velocity of wind 8m/s. They also give a maximum power generation by blade 620W ate maximum co-efficient of power, and AoA 7° at velocity of wind 8m/s.

J. Fazil et al. (2011) in this paper, author proposed a quantic reverse engineering Bezier curve formula for producing airfoil shape. By, this formula they produce an airfoil shape in CATIA and validate with NACA four digit profile generator. They used a quantic reverse engineering Bezier curve formula for the find out the control points of the camber profile which is used to create upper and lower camber profile .By using the control points, we easily modify the shape of the profile so that to produce the cambered airfoil shape without affecting basic airfoil geometry. The objective of this work is to find a simple and accurate way to design the airfoil profile in CATIA using six camber control point position. However the proposed method is applied only for six camber control point position in the airfoil.

C. Rajendran et al (2011) had demonstrates the potential of an incompressible Navier–Stokes CFD method for the analysis of horizontal axis wind turbines. The CFD results are validated against experimental data of the NREL power performance testing activities.

Comparisons are shown for the surface pressure distributions at several conditions are show as under taken:

a) Wind Velocity is 12.5m/s.
b) Yaw Angle is 0°.
c) Rotational Speed is 25 rpm.
d) Turbulence Model is k-ω SST.

Chalothorn Thumthae et al. (2009) in this paper, author have experimental data of profile NREL's S809 and work on finding a optimal angle of attack for untwisted blade by use of CFD analysis for prediction of AoA. They analyze that the angles are slightly larger as the speeds are higher and this is consistent with the shift of the curves as the Reynolds numbers are increased.

David Hartwanger et al (2008) has aims to develop a practical engineering methodology for the CFD-based assessment of multiple turbine installations. They are constructs the 2D experimental model of wind turbine which is of NREL S809 aerofoil series and compared. Their results with 3D CFD model in XFoil 6.3 codes and two ANSYS CFX 11.0 versions. It creates the cylindrical domain whose radius 2L and length 5L where L = turbine radius. For grid generation uses ICEM-CFD (ANSYS) software. In analysis it
use k- ω turbulence model. There are two main aims for doing analysis is as under show:

- → The primary aim is to predict the lift and drag for 2D experimental wind turbine.
- → Its secondary aim is to compare the results of Lower CFD Fidelity to Higher CFD Fidelity model.

E. Ferrer et al (2007) have analyzed the effect of wind turbine blade tip geometry numerically using Computational Fluid Dynamics (CFD). Researcher take three different rotating blade tips are compared for attached flow conditions and the flow physics around the geometries are analyzed. For analysis they use FLUENT 6.2 version with k-ω SST turbulence model. They got pressure coefficient, thrust and torque for 3 tips with rotational speed 71.9 rpm and wind speed 7 m/s, 8.5 m/s. It results from the comparison that a better tip shape that produced better torque to thrust ratios in both forces and moments is a geometry that has the end tip at the pitch axis. The work here presented shows that CFD may prove to be useful to complement 2D based methods on the design of new wind turbine blade tips.

Dr. S. P. Vendan et al carried out in his paper a Horizontal axis wind turbine blade profile NACA 63-415 is analyzed for various angles of attack. The coefficient of Lift and drag is calculated for this NACA 63-415 for various angles of attack from 0° to 16° and the maximum ratio is achieved at 2° of angle of attack. The coefficient of Lift increases with increase in angle of attack up to 8°. After 8°, the coefficient of lift decreases and stall begins to occur. The drag force begin of dominate beyond this angle of attack. The rate of increase in lift is more for angle of attack from 0° to 8° and then it starts to decrease. The drag increase gradually until 5° angle of attack and then rapidly increases.

2.1 Problem Statement and Objectives

The table top model can be created for small power generation units. The effective blade AoA will leads to generation of power at domestic level also. The power generation by this method can be utilized for domestic applications including mobile charging, battery charging, etc.

The reviewed literature shows that there is scope for improvement in performance of WT blades by using different AoA of blades and designing as per the fan blades. Today, lots of ways are readily available in the field to improve the performance of WT blades.

To Design the WT blade in CAD software and analyze the blade for CFD analysis using different AoA. The general air velocity in the domestic region varies from 3m/s to 7 m/s. Thus, CFD analysis of WT blade is performed by using constant air velocity of 5m/s.

3 Proposed Methodology

The present work includes methodology adopted for case study is shown in figure 2. Firstly, introduction has been defined based on theoretical work presented. The data is collected from the sources to observe the performance of WT blades under varying AoA with constant wind velocity. The various objectives were defined as per the collected data. Solid model was prepared in CATIA V5 software and exported as .step file to the ANSYS 21.0 software for performing CFD analysis. the results are obtained in graphics interface and compared by tabulation method. At the end conclusion had made based on data collected and results from CFD analysis. The main purpose of this study is to check the performance of WT blades under different angle of attack. Thus, for this purpose, the blade AoAs were selected as 10^0, 12^0, 14^0, 16^0.

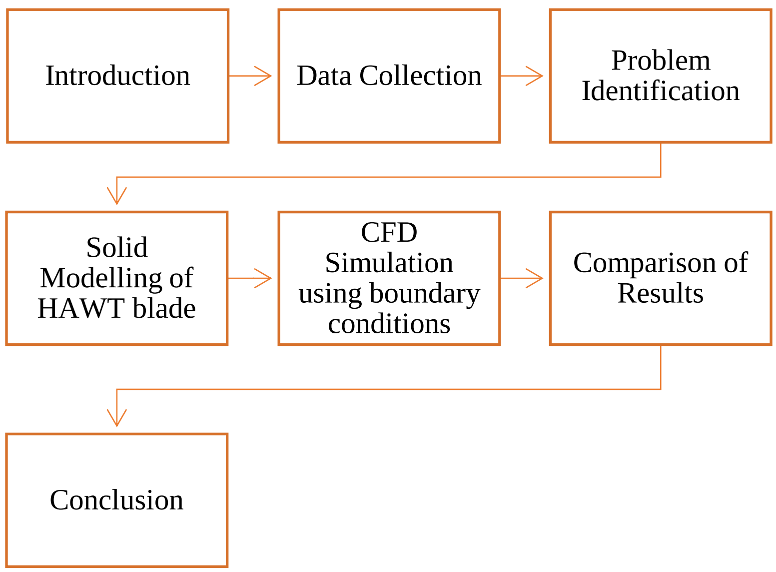


Figure 2 Proposed Methodology

4 Results and Discussion

The Figures 3-5 shows the specified dimensions of the designed WT blade for performing the ANSYS Fluent analysis.

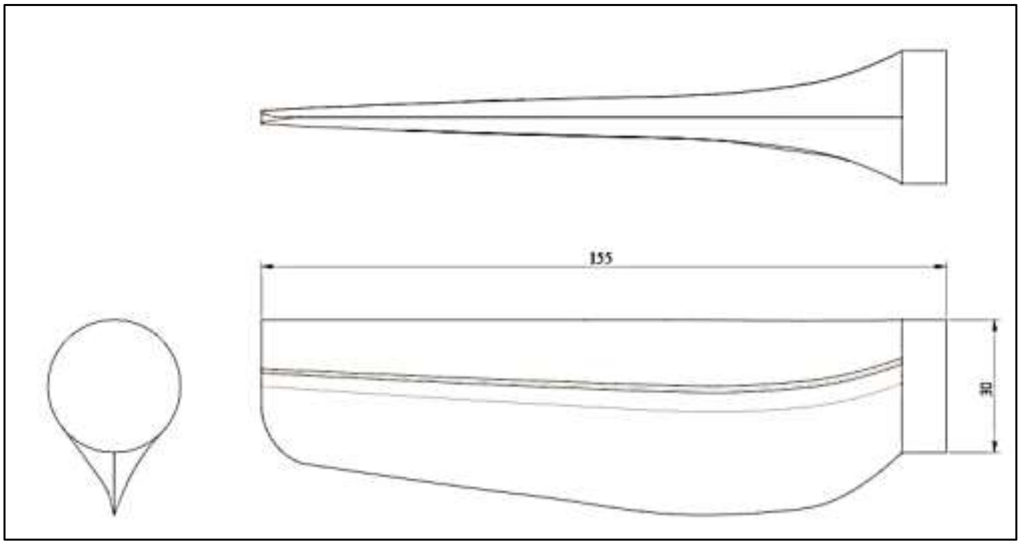


Figure 3 WT Blade Model for ANSYS

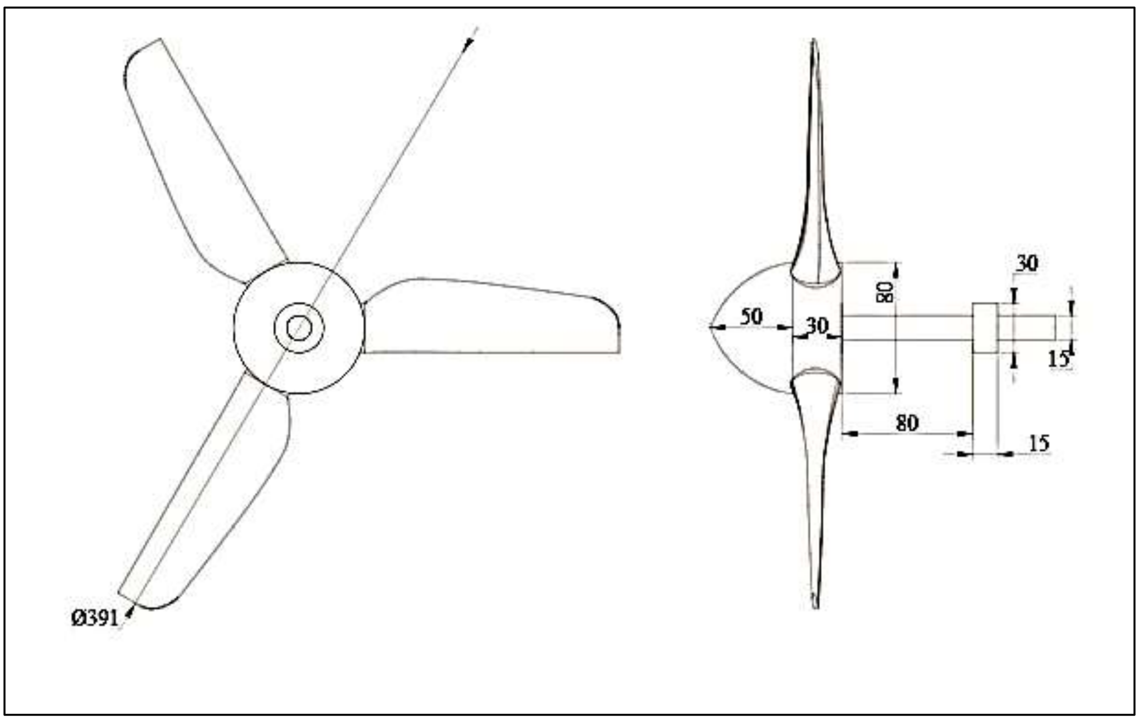


Figure 4 Assembly Model of WT Blade with Collar.

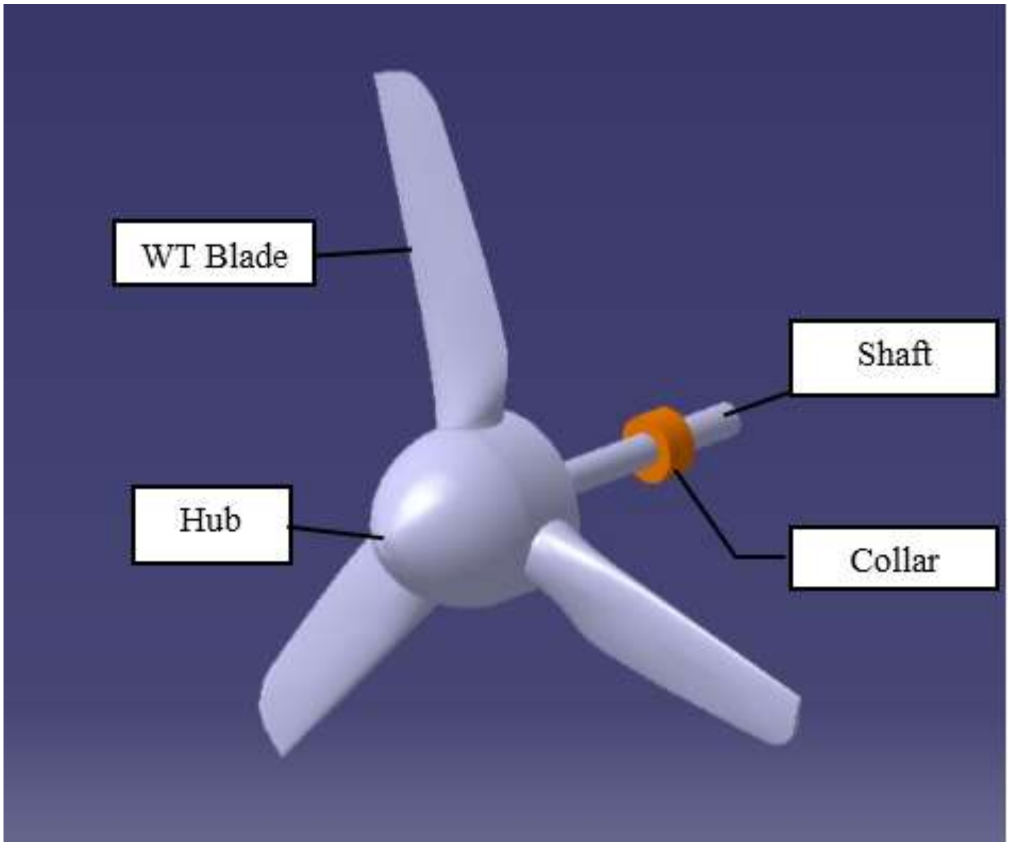


Figure 5 3D view of developed WT blade model

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels. In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, heat transfer, industrial system design and analysis, biological engineering and fluid flows, and engine and combustion analysis.

The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define many single-phase (gas or liquid, but not both) fluid flows. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

4.1 CFD Analysis of WT Blade Using Fluent

CFD analysis has been performed in ANSYS Fluent 21.0.0 software. Initially, a geometry had modelled in geometry modeler (CATIA V5). The dimensions taken for designing the WT Blade are as follows:

Blade Length = 155 mm,
Blade Diameter at Hub = 30 mm,
WT Diameter = 391 mm,
Hub Diameter = 80 mm,
Collar Diameter = 30 mm,
Collar Length = 15 mm,
Shaft Diameter = 15 mm,
Velocity of Air = 5 m/s

A fluid domain has been added in geometry by considering volume enclosed by entire WT Assembly. The K-Epsilon model is considered for the Fluent Analysis. The inlets and outlets are specified and boundary conditions are specified. After that meshing has been done (figure 6). The mesh size is selected as 20 mm and mesh sizing is performed with face meshing to obtain accurate results. The materials are specified as given in table 1.

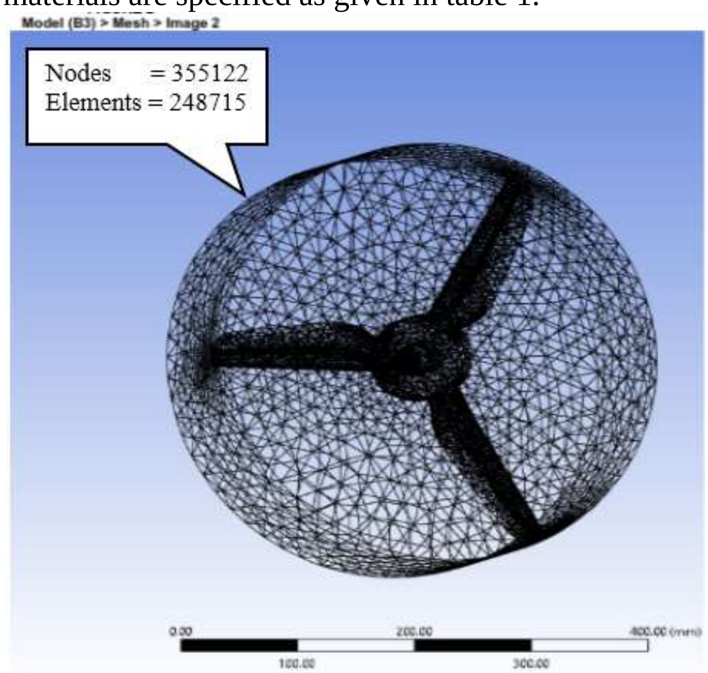


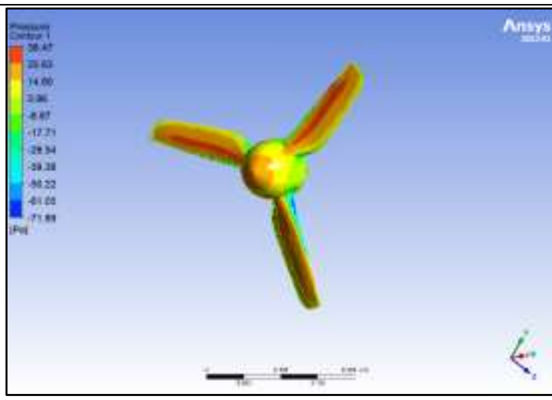
Figure 6 Meshed Model of WT Blade and Fluid Domain

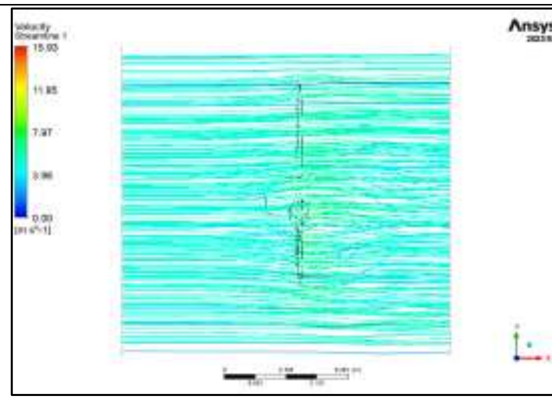
Table 1 Material Properties for ANSYS Fluent

Sr. No.	Property	Air	Aluminium
1	Density (Kg/m3)	1.225	7135
2	Molar Mass (Kg/Kg-mole)	28.966	26.98
3	Specific Heat (KJ/KgK)	1.006	0.887
4	Thermal Conductivity (w/mK)	100	120

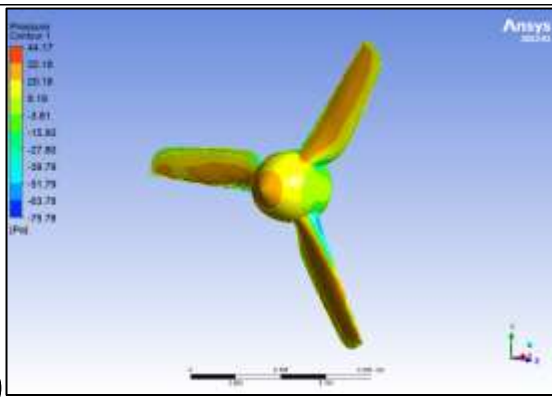
5	Thermal Expansion (/C)	34e-4	34e-6

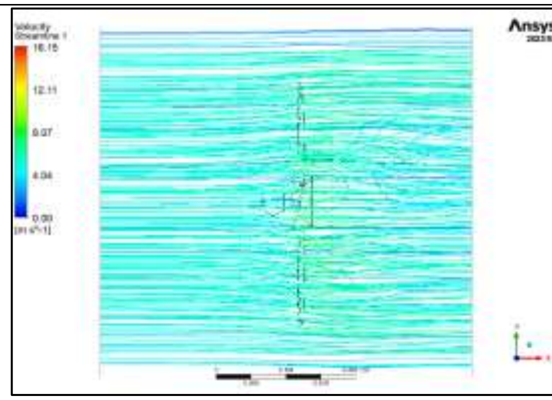
a)



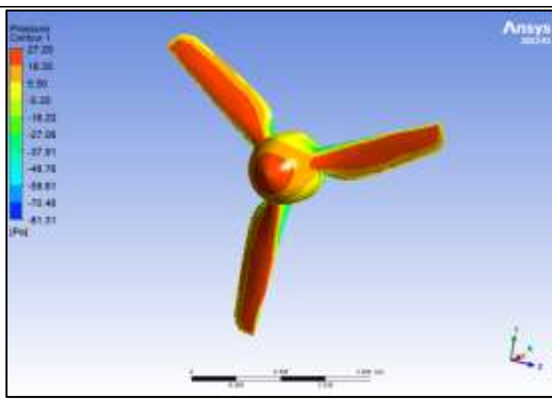


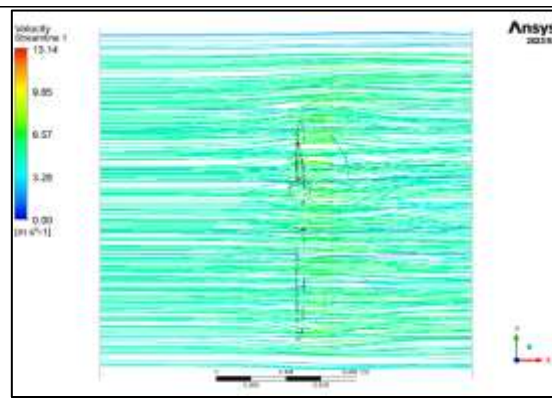
b)





c)





d)

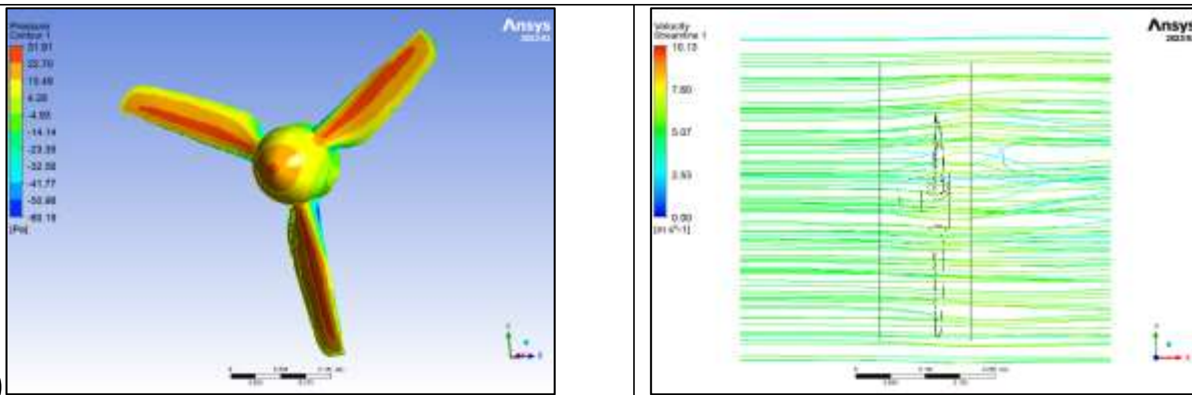


Figure 7 CFD Results (for a=10^0, b=12^0, c=14^0, d=16^0)

Table 2 Results of ANSYS Fluent for Different AOA of WT Blade

Sr. No.	Angle of Attack	Time Step	Drag Force (N)	Lift Force (N)	Moment	Resultant Force (F)	Torque	Wind Power	N at Input Power	N=C	Out Power	Eff.	LDR
1	10 Deg. AoA	6	0.5614	-0.0020	0.0010	0.5614	0.109	9.19	802	455	5.22	57	-0.4
2		12	0.5786	-0.0062	0.0011	0.5786	0.113	9.19	778	455	5.38	59	-1.1
3		18	0.6330	0.0115	-0.0027	0.6331	0.123	9.19	711	455	5.88	64	1.8
4		24	0.5552	-0.0050	0.0004	0.5552	0.108	9.19	811	455	5.16	56	-0.9
5		30	0.5841	-0.0038	0.0002	0.5841	0.114	9.19	771	455	5.43	59	-0.6
6		36	0.5554	-0.0066	-0.0002	0.5555	0.108	9.19	810	455	5.16	56	-1.2
7		42	0.5479	-0.0047	-0.0005	0.5479	0.107	9.19	822	455	5.09	55	-0.9
8		48	0.6500	-0.0068	0.0061	0.6500	0.127	9.19	693	455	6.04	66	-1.0
11	12 Deg. AoA	6	0.5750	-0.0050	-0.0004	0.5750	0.112	9.19	783	455	5.34	58	-0.9
12		12	0.7546	0.0023	-0.0003	0.7546	0.147	9.19	597	455	7.01	76	0.3
13		18	0.5722	-0.0026	-0.0007	0.5722	0.112	9.19	787	455	5.32	58	-0.5
14		24	0.7048	0.0251	0.0039	0.7053	0.138	9.19	638	455	6.55	71	3.6
15		30	0.5796	-0.0213	-0.0016	0.5800	0.113	9.19	776	455	5.39	59	-3.7
16		36	0.5889	0.0322	0.0013	0.5898	0.115	9.19	763	455	5.48	60	5.5
17		42	0.5944	0.0074	-0.0040	0.5945	0.116	9.19	757	455	5.52	60	1.2
18		48	0.5764	0.0083	0.0021	0.5765	0.112	9.19	781	455	5.36	58	1.4
21	14 Deg. AoA	6	0.5462	-0.0110	0.0004	0.5463	0.107	9.19	824	455	5.08	55	-2.0
22		12	0.5903	0.0110	0.0008	0.5904	0.115	9.19	762	455	5.49	60	1.9
23		18	0.5586	0.0054	0.0014	0.5587	0.109	9.19	806	455	5.19	56	1.0
24		24	0.5882	-0.0029	-0.0009	0.5883	0.115	9.19	765	455	5.47	59	-0.5
25		30	0.5513	0.0007	0.0002	0.5513	0.108	9.19	816	455	5.12	56	0.1
26		36	0.6468	0.0036	0.0052	0.6468	0.126	9.19	696	455	6.01	65	0.6
27		42	0.5514	-0.0020	-0.0016	0.5514	0.108	9.19	816	455	5.12	56	-0.4
28		48	0.6031	-0.0073	0.0000	0.6032	0.118	9.19	746	455	5.60	61	-1.2
31	16 Deg. AoA	6	0.5432	-0.0138	-0.0001	0.5433	0.106	9.19	828	455	5.05	55	-2.5
32		12	0.6086	-0.0090	-0.0018	0.6086	0.119	9.19	740	455	5.65	62	-1.5
33		18	0.5675	-0.0091	-0.0011	0.5676	0.111	9.19	793	455	5.27	57	-1.6
34		24	0.5692	-0.0154	-0.0004	0.5694	0.111	9.19	791	455	5.29	58	-2.7

35		30	0.5348	-0.0072	-0.0009	0.5348	0.104	9.19	842	455	4.97	54	-1.3
36		36	0.5508	-0.0029	-0.0007	0.5508	0.107	9.19	817	455	5.12	56	-0.5
37		42	0.6143	-0.0016	-0.0017	0.6143	0.120	9.19	733	455	5.71	62	-0.3
38		48	0.5282	-0.0045	0.0004	0.5282	0.103	9.19	852	455	4.91	53	-0.9

Conclusion

This study investigated the performance of wind turbine (WT) blades at various **Angles of Attack (AoA)** using aluminum as the material for both the WT blade and collar, with air serving as the fluid medium. The analysis was conducted at a flowing air velocity of **5 m/s**, utilizing **ANSYS Fluent** to evaluate the performance metrics of the WT blades across different AoAs.The results indicate that the performance of the WT blade is influenced by several factors, including the angle of attack, mass flow rates, and the properties of air. Variations in these parameters lead to significant changes in performance outcomes. While the ANSYS method effectively provided results, it was noted for its complexity and time-consuming nature.Key findings from the analysis include:

- The **12-degree AoA** exhibited superior overall performance compared to other angles tested, demonstrating higher lift-to-drag ratios and better aerodynamic efficiency.
- From a drag perspective, the **16-degree AoA** blade showed lower drag coefficients, making it suitable in applications where minimizing drag is critical. However, the difference in drag between the 12-degree and 16-degree blades was minimal and can be considered negligible.
- Performance metrics such as lift, drag, moment, pressure distribution, and velocity streamlines were thoroughly analyzed to assess the aerodynamic characteristics of each configuration.

In conclusion, this study demonstrates that a **12-degree angle** is optimal for designing small wind turbine blades, balancing both lift generation and drag reduction effectively. These findings underscore the importance of optimizing AoA in enhancing turbine efficiency and performance. Future research may further explore additional factors such as varying wind speeds and environmental conditions to refine these results and improve small-scale wind turbine designs.

References

[1] Srinivas G , Mahesha G T, Chethan K N and Arjun N, 2014, Analysis of Wind Turbine Blade, International Journal of Current Engineering and Technology E-ISSN 2277-4106,P-ISSN 2347-5161.

[2] R. B. Gowardhan and, Mr. Ritesh k. Bais (2014) IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 45-47.

[3] Mayurkumar kevadiya, 2D Analysis of NACA 4412 AIRFOIL International Journal of Innovative Research in Science, Engineering & Technology Vol. 2, Issue 5, May 2013.

[4] HardikPatel, Sanat Damania, Performance Prediction of Horizontal Axis Wind Turbine Blade, International Journal of Innovative Research in Science, Engineering & Technology, Vol. 2, Issue 5, May 2013.

[5] Fei-Bin Hsiao, Chi-JengBai and Wen-Tong Chong, The Performance Test of Three Different Horizontal Axis Wind Turbine (HAWT) Blade Shapes Using Experimental and

Numerical Methods, www.mdpi.com/journal/energies, Energies 2013, 6, 2784-2803; doi:10.3390/en6062784.
[6] Farooq Ahmad Najar, G A Harmain, Blade Design & Performance Analysis of Wind Turbine, International Journal of Chem Tech Research, Vol.5, No.2, pp 1054-1061, April-June 2013.
[7] R. Mukesh, K. Lingadurai, U. Selvakumar, Airfoil shape optimization using non-traditional optimization technique and its validation, Journal of King Saud University – Engineering Sciences (2014) 26, 191–197, http://dx.doi.org/10.1016/j.jksues. 2013.04.003.
[8] Monir Chandrala, Abhishek Choubey, Bharat Gupta, CFD analysis of horizontal axis wind turbine blade for optimum value of power, International Journal Of Energy And Environment, Volume 4, Issue 5, Pg. No. 825-83, 2013.
[9] H. V. Mahawadiwar, V.D. Dhopte, P.S.Thakare, Dr. R. D. Askhedkar, CFD Analysis of Wind Turbine Blade, International Journal of Engineering Research and Applications (IJERA), Vol. 2, Issue 3, Pg. No. 3188-3194, 2012.
[10] J. Fazil and V. Jayakumar, Investigation of Airfoil Profile Design Using Reverse Engineering Bezier Curve, Asian Research Publishing Network (ARPN), VOL. 6, No. 7, 2011.
[11] Syed umaid ali, Rafiullah khan, Syed Athar Masood, Analysis of wind energy potential and optimum wind blade design for jamsharo wind corridor, Mehran university research journal of engineering and technology, 36(4), 2017.
[12] Hani Muhsen, Wael al Kouz, waqar khan, Small Wind turbine blade design and optimization, Symmetry, 12(18), 2020.
[13] karthikeyan k, R. Niranjan, Kumaravel. M, Power, lift and drag coefficients analogy of wind turbine blade from aerodynamics characteristics of NACA2412 & NACA0012, Turkish Journal of Computer and Mathematics Education Vol.11 No.03 (2020), 1305-1311.
[14] P.K. Nigam, Nitin Tenguria, M.K. Pradhan, Analysis of horizontal axis wind turbine blade using CFD, International Journal of Engineering, Science and Technology Vol. 9, No. 2, 2017, pp. 46-60
[15] Yin Yin Htay Aunga, Myat Myat Soe, Aung Myat Thu, Effect of Attack angle on Aerodynamics Analysis of Different Wind Turbine Wings using Numerical Simulation, American Scientific Research Journal for Engineering, Technology, and Sciences (ASRJETS) (2016) Volume 26, No 4, pp 319-329
[16] Irshadhussain I. Master, Azim Aijaz Mohammad, Ratnesh T. Parmar, Aerodynamic Performance Evaluation of a Wind Turbine Blade by Computational and Experimental Method, International Journal of Engineering Research & Technology (IJERT), Vol. 3 Issue 6, June – 2014, 73-81.
[17] Mustafa Alaskari, Oday Abdullah, Mahir H. Majeed, Analysis of Wind Turbine Using QBlade Software, IOP Conf. Series: Materials Science and Engineering 518 (2019) 032020
[18] A S Samosir, and A Riszal, The effect analysis of wind speed variation to the horizontal axis wind turbine design with Q-blade, IOP Conf. Series: Materials Science and Engineering 1173 (2021) 012009