International Journal of Scientific Research & Engineering Trends
Volume 2, Issue 5, Sept.-2016, ISSN (Online): 2395-566X

A CFD Analysis of a Wind Turbine Blade Design at Various Angle of Attack and Low Reynolds Number

Raju Kumar, Priyanka Jhawar, Sanjay Kalraiya
Sri Satya Sai University of Technology & Medical Sciences, Sehore , Madhya Pradesh
Email: rajuk264@gmail.com

Abstract **– Wind-turbines extract kinetic energy from the windIn this study, two-dimensional aero foil (i.e. DU-93 and NREL-S809) CFD models are presented using ANSYS-FLUENT software.The dimensionless lift, drag and pitching moment coefficients were calculated for wind-turbine blade at different angles of attack Thesis work is aimed at designing a wind turbine for tapping the average wind speed in urban locations, The NACA 63 and 64 series is chosen as the basic group for investigation because they have good low speed and medium speed characteristics and the power curve is better in the low and medium wind speed ranges.**

Keyword **– CFD, Design, Blade Turbine, Wind Turbine, Wind Turbine.**

I. INTRODUCTION

Wind energy is a significant source of the world's energy must rank as one of the significant developments of the late 21th century. The development of the steam engine, followed by the other technology took place for converting fossil fuels to useful energy. But due to limitedquantity of fossil fuel reserved as well as of the adverse effects on environment due to burning of fuel, many people started looking for alternatives to fulfill energy demand. The other thingis it was the availability of wind energy. Wind present everywhere on the earth, and in several places with considerable wind energy density and these wind energy are being used for power development. Wind energy in India is available in high speed and low speed wind. To harness wind energy, wind turbine are install.

Airflow over a stationary air foil produces two forces, a lift force perpendicular to the air foil and a drag force in the direction of air foil, the existence of the lift force depends on laminar flow over air foil. When the air foil is move in the direction of the lift this translation will combine with the motion of the air to produce a relative wind direction (W), the air foil has been reoriented and maintain a good lift to drag ratio. The lift and drag forces can be split into components parallel and perpendicular to the direction of the undisturbed wind. The lift is perpendicular to the relative wind but is not in the direction of aerofoil translation, and these components combined to from the tangential force and axial force. The tangential force acting on the turbine rotor in the direction of translation which is available to do useful work, and allow for the blades to rotate around to horizontal axis and causes a torque that drive some load connected to the turbine rotor . The other force is axial force on the direction of the undistributed wind which must to be used in the design of air foil supports to assure structural integrity, and the tower must be strong enough to withstand this force.

Small wind turbines operating at low wind speeds regularly face the problem of poor performance due to laminar separation and laminar separation bubbles on the blades. This is due to the low Reynolds number (Re) resulting from low wind speeds and small rotor size. The use of specially designed low Reynolds number air foils it start up at lower wind speed and improving the overall performance. In this thesis work study is focused on mainly on designing the turbine blade for tapping the regions of low wind power density. This involves the selection of a suitable airfoil section for the proposed wind turbine blade. The NACA 64 series is chosen as the basic group for investigation because they have good low and medium speed characteristics and the power curve is better in the low and medium wind speed ranges. Thus used for NACA 6409 airfoil profile is considered for analysis of wind turbine blade. NACA 6409 airfoil profile is created by using the co-ordinate file generated in Java Foil and Mesh Domain for the fluid around the airfoil is created using Design Modeler in ANSYS 14.0 software. The CFD analysis is carried out various angles of attack from 1^0 to $15^0$and coefficient of lift and drag values are calculated for low Reynolds number. The airfoil NACA 6409 is analyzed to identify its suitability for its application on wind turbine blades and determine optimum angle of attack for high lift and drag ratio to give the maximum efficiency.

II. RELATED WORK

Rong Ma, Peiqing Liu.al have analyzed overall aerodynamic parameters of airfoil S1223 with various relative thicknesses and suggested that the airfoil S1223 with relative thickness of 5% can be used for the blade tip and the airfoil S1223 with relative thickness of 12.13% can be used for blade root. Moreover, further optimized design research will be carried on based on low-Reynolds-number and high-lift airfoil S1223 [1] Ravi Anant Kishore has develop small scale wind energy portable turbine (SWEPT) which can work wind speed below 5m/s. and got a very low cut in wind speed of 2.7 m/s and it produces 1.25W of mechanical power & 0.83 W of electrical power at the rated wind speed of 5 m/s. A diffuser was also designed & fabricated to study its effectiveness for the small-scale wind turbines. Study with CFD analysis revealed that the diverging part of the diffuser is stronger parameter than its converging part which influences its performance. During the study it was found that the performance of turbine with diffuser is 1.4-1.6 times more than without diffuser [2]

S. RAJAKUMAR and Ravi Chandra was Lift and Drag forces along with the angle of attack are the important parameters in a wind turbine system and calculate lift and Drag forces in various sections and the different of angle of attack on the 2 D modal in CFD software. It analyzed that angle of attack ranging from 0° to 12° and the velocity range from 3 -12 m/sec. It is found that blade with 5° angle of attack has the maximum L/D ratio [3]. Vendan is designing a wind turbine for tapping the low speed wind in urban locations. 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 C_L /C_D 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 forces 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[4]. Joao P. Monteiro was develop a horizontal axis wind turbine with a rotor diameter of 1.2 m, and Blade Element Momentum (BEM) theory is applied to get results, and then these results are analyzed and compared with those of wind tunnel tests performed on a full scale wind turbine. These rotor blades are made through wood with the help of hand tools [4]

III. WIND TURBINES

Wind turbines are rotating prime mover machines which convert wind power in to electrical power with the help of generators. . Wind power are solely depends on wind hence, free from pollution, green/clean and renewable source of energy. If the rotational power developed by rotor is directly utilized as mechanical work like water pumping, stone crushing etc. then it is called as *wind Mills*. and if power is used to generate electricity then it is termed as wind turbines. Wind turbines are classified in two ways, first according to their axis of rotation and second according to power generation. According to power generation wind turbines are classified as micro (<1 KW), mini (1KW-5KW), small (5KW-10KW), intermediate (10KW-1MW), and large (>1MW).

IV. HORIZONTAL AXIS WIND TURBINE (HAWT)

Horizontal axis wind turbine axis is parallel to direction of wind motion. The Wind sticks to the blade and blade starts rotating, the rotation is send to the gear assembly with the help of shaft. This shaft is connected to the generator which produces electricity. If changes direction wind, a motor (yaw motor) turns the nacelle so the blades are always facing the wind.

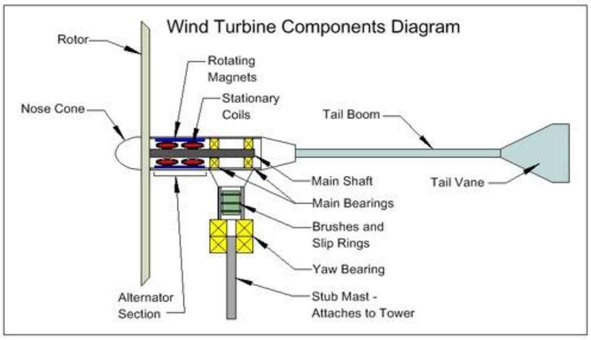

Fig. 1 horizontal axis wind turbine

V. VERTICAL AXIS WIND TURBINE (VAWT)

In VAWT's the axis of rotation is perpendicular to direction of motion of wind. The main advantage of this arrangement of VAWT is that, it need not necessarily to be pointed toward the wind flow direction. Due to These advantage it is used for sites where the wind direction is highly variable or has turbulent winds.

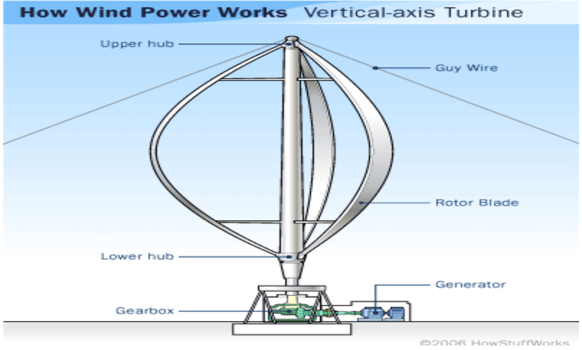

Fig. 2 Vertical Axis Wind Turbine

VI. AIRFOIL IN WIND TURBINE

The shape of the aerodynamic profile is decided for blade performance. Even minor changes in the shape of the profile can significantly affected the power curve and noise level. In order to extract the maximum kinetic energy from wind, researchers put more effort in the design of effective blade geometry. A rotor blade may have different airfoil in different sections in order to increases the efficiency, so the modern blades are more complex and efficient comparing to early wind turbine blades. A wind turbine is a complex system which consists of several components, including a rotor, a transmission system, a generator, a nacelle, a tower and other electro-mechanical subsystems. The blades are the most important components, thus transfer wind energy into mechanical power, the blade is designed as an aerodynamic geometry with nonlinear chord and twist angle distributions. The section view of a wind turbine blade is of an airfoil shape (one or more airfoils), which is expected to generate high lift and low drag forces. The shape of the blade is vital as it determines the energy captured, and the loads experienced. The study of interaction between wind flows and wind turbines is wind turbine aerodynamics which plays an important role in wind turbine design and analysis. Terms used in Airfoil are as follow

a. **Cord: -** The chord line is a straight line connecting the leading and trailing edges of the airfoil.
b. **Cord length: -** The chord length is the length of the chord line from leading edge to trailing edge and is the characteristic longitudinal dimension of the airfoil.
c. **Angle of attack: -** Angle of Attack is angle between chord line and direction of relative wind velocity.
d. **Mean camber: -** The mean camber line is a line drawn halfway between the upper and lower surfaces. The chord line connects the ends of the mean camber line. The shape of the mean camber is important in determining the aerodynamic characteristics of an airfoil section.
e. **Max. Camber: -**Maximum camber (displacement of the mean camber line from the chord line) and the location of maximum camber help to define the shape of the mean camber line. These quantities are expressed as fractions or percentages of the basic chord dimension.
f. **Thickness: -** Thickness and thickness distribution of the profile are important properties of an airfoil section. The maximum thickness and its location help define the airfoil shape and are expressed as a percentage of the chord.
g. **Leading edge radius: -** The leading edge radius of the airfoil is the radius of curvature given the leading edge shape.

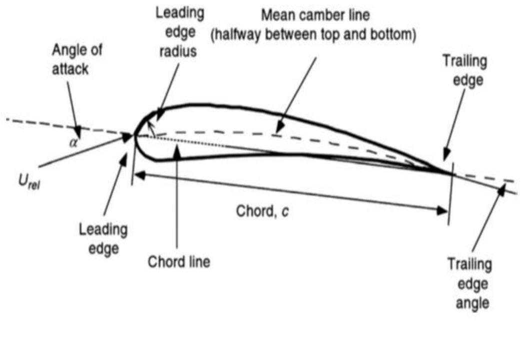

Fig.3 Show various section in of Air Foil.

VII. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics is shortly known as CFD, Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing fluid flow, heat transfer, and related phenomena such as chemical reactions. And it is combine numerical method and algorithm to solve and analyze problem that involve fluid flow. Computer is used to perform the calculation requires to simulate the interaction of liquids and gases with body. With supercomputer, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows.

For designing and analyzing the engineering systems which involve fluid flow, there are basically two approaches: experimental and numerical. The experimental approach involves a complete testing rig setup for performing experiments on a scaled model of the prototype machine, while the latter is based on either analytical or computational solution of differential equations governing the flow through the system. In the present chapter a brief introduction to computational fluid dynamics (CFD) is given. The computational approach for solving fluid flow problems is devoted to the solution of the governing equations of fluid flow through use of a computer. Today both the experimental and CFD analysis are applied and the two complement each other. For example, global properties, such as lift, drag, pressure drop, or power, may be obtained experimentally but to obtain details about the flow field, such as velocity, shear stresses, flow streamlines and pressure profiles CFD is used. In addition, for validating CFD solutions, experimental data is used by matching the computationally and experimentally determined global quantities. Then CFD is employed to shorten the design cycle through carefully controlled parametric studies, thus reducing the required amount of experimental testing. The present state of computational fluid dynamics is that it can handle laminar flows with ease, but turbulent flows of practical engineering interest are impossible to solve without considering turbulence models. Unfortunately, there is no universal turbulence model and a turbulent CFD solution is dependent on the appropriateness of the turbulence model. Despite this limitation, the standard turbulence models yield reasonable results for many practical engineering flow problems. Any numerical simulation of fluid flow problems consists of various distinguished processes. These are mentioned below:-

a. Model the geometry of problem domain
b. Choose appropriate mathematical model of the physical problem.
c. Choose a suitable discretization method.
d. Generate a grid based on problem geometry and the discretization method.
e. Use a suitable solution technique to solve the system of discrete equations.
f. Set suitable convergence criteria for iterative solution methods.
g. Prepare the numerical solution for further analysis.

VIII. PROPOSED WORK

Blade Profile Selection: -Aerodynamics of blade is main criteria for selection of blade profile. Thus for efficient blade will have high lift to drag ratio. NACA 4- digit series will have given the good performance at low speed. Therefore we choose the NACA 6409 blade air foil for analysis. These profile given the better performance at low speed having high lift to drag ratio and good stall properties.

NACA Four-Digit Series:-First designed airfoils is
NACA 4 –digit series and it number specified as-
The first digit specifies the maximum camber (m) in percentage of the chord (airfoil length), the second indicates the position of the maximum camber (p) in tenths of chord, and the last two numbers provide the maximum thickness (t) of the airfoil in percentage of chord. For example, the NACA 2415 airfoil has a maximum thickness of 15% with a camber of 2% located 40% back from the airfoil leading edge (or 0.4c). Utilizing these m, p, and t values, we can compute the coordinates for an entire airfoil using the following relationships:

a. Pick values of x from 0 to the maximum chord c.

b. Compute the mean camber line coordinates by plugging the values of m and p into the

Following equations for each of the x coordinates.

$$Y_c = \frac{m\,[2px - x^2]}{p^2}$$

From x=0 to x = p

$$Y_c = \frac{m}{(1-p)^2}[(1-2p) + 2px - x^2]$$

From x = p to x = c

Where

x = coordinates along the length of the airfoil, from 0 to c (which stands for chord, or length)

y = coordinates above and below the line extending along the length of the airfoil, these are either y_t for thickness coordinates or y_c for camber coordinates

t = maximum airfoil thickness in tenths of chord (i.e. a 15% thick airfoil would be 0.15)

m = maximum camber in tenths of the chord

p =position of the maximum camber along the chord in tenths of chord

3. Calculate the thickness distribution above (+) and below (-) the mean line by plugging the value of t into the following equation for each of the x coordinates-

$$t = \frac{t}{0.2}[0.2969\sqrt{x} - 0.1260x - 0.3516x^2 + 0.2843x^3 - .0.1015x^4]$$

4. Determine the final coordinates for the airfoil upper surface (X_U, Y_U) and lower surface (X_L, Y_L) using the following relationships.

$$X_u = x - \ y_{t.} \sin\theta$$
$$y + y_t \cos\theta$$
$$X_l = x + \ y_t \sin\theta$$
$$Y_l = \ y_c - \ y_t \cos\theta$$

Where $\theta = \frac{dy}{dx}$

Specification of NACA 6409 low speed wind blade profile-

- Max thickness 9% at 29.3% chord.
- Max camber 6% at 39.6% chordSource

Velocity consideration:-
We are selected the velocity for investigation is 4 m/sec, 5m/sec, 6m/sec. At low speed wind turbine blade NACA 6409 to generated the power approximately one kilowatt at maximum efficiency for approximate 5 m/sec wind velocity.

ANSYS (CFD) Work Process
Discredited the governing equation put together the discretized solution iterate it to minimize residual. So which governing equations govern the fluid flow. Now here we enlist the all governing equation which governs the fluid flow. Equations governing the fluid flow: The cornerstone of CFD is the fundamental governing equations of fluid dynamics. Those are as follows:

(a) Continuity equation.
(b) Momentum equation.
(c) Energy conservation equation.
(d) Species conservation equation.
(e) Nervier-stokes equation

Geometry Modeling: -Creation of Model by using ansys cfx modeling tools for generating the geometryof the part of which we want to perform theanalysis. We selected the coordinate point of NACA 6409 airfoil in naca website and imported coordinate point in ansys cfx software and it join two different curve in upper and lower part and get the geometry of naca 6409 air foil.

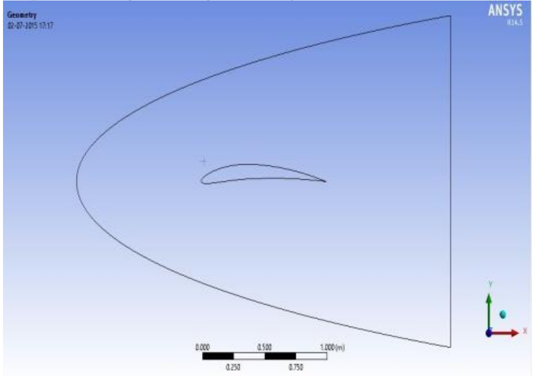

Fig 4 geometry of wind turbine blade

Meshing:-Meshing is a critical operation in CFD. In this process, the geometry isdiscretized into large numbers of small Element and nodes. The arrangement of nodes and element in space in an appropriate manner is called mesh. The CFD analysis accuracy and duration depends on the mesh size and orientations. With the increase in mesh size (increasing no. of element), the CFD analysis speed decrease but the accuracy increase. In this analysis performance node size is 0.05 used for tetragonal element. Number of node are 24244 and element is 11981

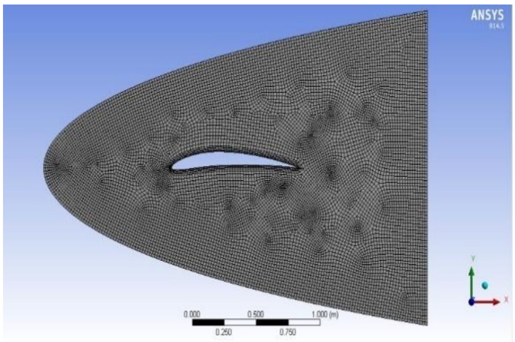

Fig.5 Meshing of NACA 6409 air foil.

Physical model: -Choose the required physical model for the problem i.e. laminar, turbulent, energy, multiphase, and. Choose the solver for the problem either Pressure based or density based solver.

Material Property: -Choose the Material property of flowing fluid.

Boundary Condition: -Define the desired boundary condition for the problem i.e. velocity, massflow rate, temperature, heat flux etc. the boundary condition are-

a. Inlet - Velocity: 4 m/s, 5m/s, 6m/s
b. Outlet - Pressure: Atmospheric
c. Rotor (Body) - Smooth: Rotating
d. Wall - No Slip and Stationary
e. luid - Air

Solution:

i) **Solution Method: -**Choose the Solution method to solve the problem should be First order, second order.
ii) **Solution Initialization**:-Initialized the solution to get the initial solution for the problem.
iii) **Run Solution: -**Run the solution by giving no of iteration for solution to converge.

Post processing: - For observing the interpretation of result. The result can be observed. In various graph, value and variation plot in over blade etc.

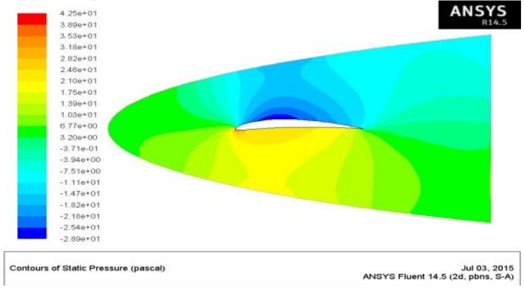

Fig.6 Pressure plot of NACA 6409 air foil.

these fig 6 show the pressure variation in air foil by different colours. Red colour show high pressure and blue coloure show low pressure thus at the upper porsion of air foil low and compare to high pressure at lower surface and at the leading edge high presure because it is stagnation point in flow field that show by red colour.

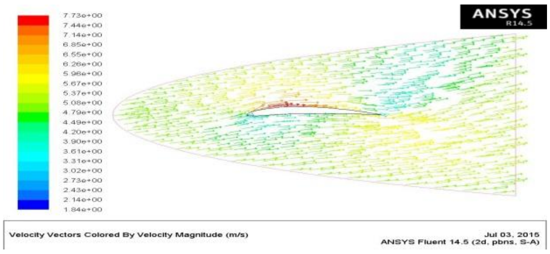

Fig.7 Velocity vector plot in NACA 6409 air foil.

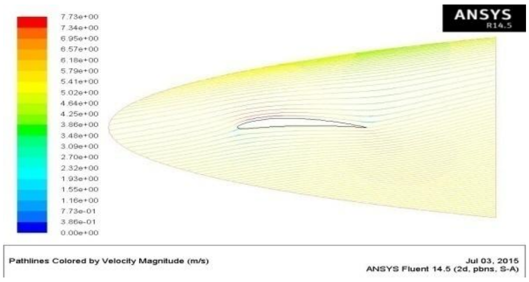

Fig.8 velocity path line over the NACA 6409 air foil.

Fig. 7 and fig 8 sho that the directon of velocity vector and path line of flow over air foil and different coloure show different magnitude of velocity of flow.

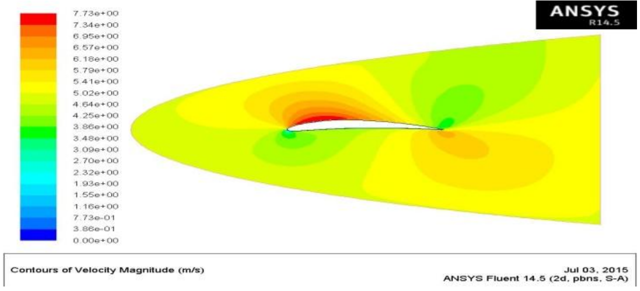

Fig.9 velocity magnitude over NACA 6409 air foil.

these fig. 9 show that velocity variation in air foil by different coloure. Red colour show high velocity and blue coloure show low velocity thus at the upper porsion of air foil velocity is high and lower porsion compare to velocity low and at the leading edge of airfoil aproximate zero velocity because it is stagnation point in flow field that show by blue coloure

IX. RESULT AND DISCUSSION

Lift and Drag Calculation: - In this paper work Horizontal axis wind turbine blade with .NACA 6409 performance has been investigated for different angle of attack and various low velocity.The airfoil NACA 6409 is chosen for investigation as shown in various figure. The airfoil profile and boundary circumstances are formed. NACA 4-series profiles are obtained from Java foil. A Mesh Domain for the fluid around the airfoil is formed using Design Modeler in ANSYS software. A fine mesh is formed for the fluid flow area of the domain in ANSYS cfx software:-

The Lift and Drag forces are calculated for NACA 6409 air foil at the various angle of attack from 1 to 15 degree for the velocity ranges from 4 m/sec in the figure10 –

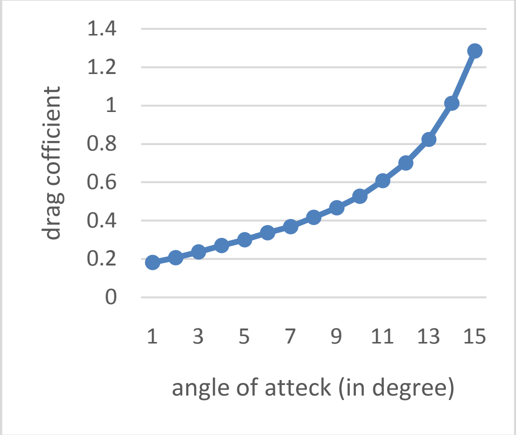

Fig.10 Increase in drag for various AOA

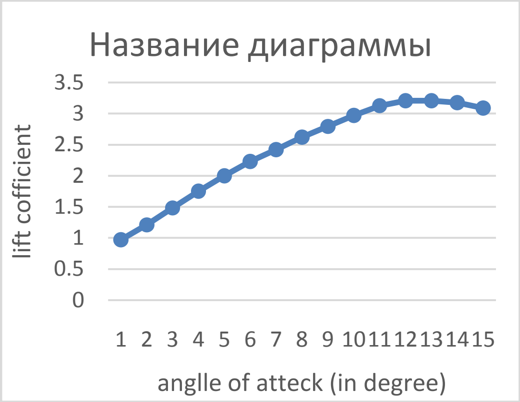

Fig.11 Increase in lift for various AOA

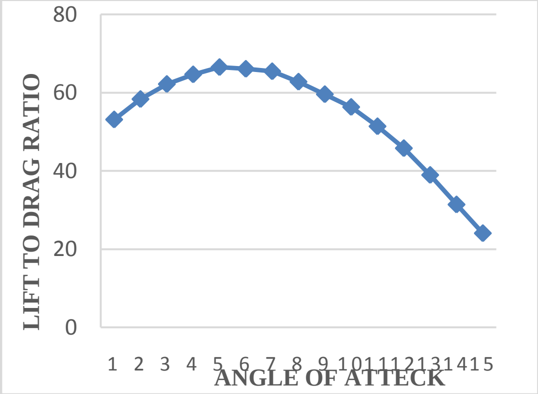

Fig.12 Correlation between L/D ratio and angle of attack

The coefficient of Lift and drag is calculated for this NACA 6409 series for the angle of attack 1° to 15° in the velocity of 4 m / second. The lift force at varies lengths from hub to tip is analyzed and it is clear that lift force increases from hub to tip for all range of angle of attack. In the first graph plot no. 6.1 willbe lift coefficient increases with increase in angle of attack up to 12^{o}at 4 m/sec and it starts to decreasegradually after 12^{o}angle of attack.

The drag forces begin gradually increasing up to 10^{0} of attack angel and then it growth of is rapidly. The rate of increase in lift is more for angle of attack from $1^{o}$to $10^{o}$and between 10^{o} to 15^{o} the rise in lift force is less.

Lift coefficient to drag coefficient ratio increases linearly 1^{0} to 6^{0} but after 6^{0} to 15^{0} will be decreases. Because of after the 6^{0} lift force is increases less compare to increases to drag force. Lift to drag ratio maximum at 6^{0} of angle of attack. Thus blade angle of attack are design at 6^{o}at 4 m/ sec velocity because maximum lift to drag ratio at these angle of attack.

X. CONCLUSION

In this paper Low speed wind turbine blade are investigated at various angle of attack and comparison and examine of the overall aerodynamic performance of airfoil NACA 6409 with various angle of attack, it can see that aerodynamic parameters (especially lift-to-drag ratio) are greatly affected by Reynolds number and angle of attack. In future we will try to Designing, and development and installation of low wind speed turbines large scope in urban and rural locations in low power requirement.

REFERENCES

[1] Rong Ma, Peiqing Liu “Numerical Simulation Of Low-Reynolds-Number And High-Lift Airfoil S1223” Journal Of Wind Engineering 2009

[2] Ravi Anant Kishore & Thibaud Coudron, Shashank Priya, “Small-Scale Wind Energy Portable Turbine (Swept)” Journal Of Wind Engineering And Industrial Aerodynamics, 2013

[3] S. Rajakumar & Dr. D.Ravindran, “Computational Fluid Dynamics Of Wind Turbine Blade At Various Angles Of Attack And Low Reynolds Number.” 2010

[4] S. P. Vendan, S. Aravind Lovelin, M. Manibharathi And C. Rajkumar “Analysis Of A Wind Turbine Blade Profile For Tapping Wind Power At The Regions Of Low Wind Speed.” Journal of Industrial Aerodynamics 2009

[5] Joao P. Monteiro & Miguel R. Silvestre & Hugh Piggott, Jorge C. Andre, “Wind Tunnel Testing Of A Horizontal Axis Wind Turbine Rotor And Comparison With Simulations From Two Blade Element Momentum Codes”, Elsevier Journal Of Wind Engineering And Industrial Aerodynamics,2013

[6] Ronit K. Singh & M. Rafiuddin Ahmed, “ Blade Design And Performance Testing Of A Small Wind Turbine Rotor For Low Wind Speed Applications”, Elsevier Renewable Energy ,2013.

[7] Joao P. Monteiro & Miguel R. Silvestre & Hugh Piggott, Jorge C. Andre, “ Wind Tunnel Testing Of A Horizontal Axis Wind Turbine Rotor And Comparison With Simulations From Two Blade Element Momentum

Codes", Elsevier Journal Of Wind Engineering And Industrial Aerodynamics,2013

[8] Clausen Pd, Reynal F, Wood Dh " Design, Manufacture And Testing Of Small Wind Turbine Blades". In: Brondsted P, Nijssen Rpl, Editors. Advances In Wind Turbine Blade Design And Materials, Woodhead Publishing.,2013

AUTHOR'S PROFILE

Raju Kumar
pursuing M.Tech. in Thermal Engineering, Department of Mechanical engineering in Sri Satya Sai University Of Technology And Medical Sciences Bhopal. Received B.E Degree in Mechanical Engineering From Agnos College Of Technology Bhopal In 2014.