Flow Analysis of Vertical Axis Wind Turbine by CFD

[1]Govind Baburao Patil, [2]Siraj Ahmed Mohalkar, [3]Shubham Dattatray Choudhar, [4]Haridas Somnath Jagtap, [5]Prof Amol T Kamble

[1,2,3,4]Students, [5]Professor(Guide)
Department of Automobile Engineering,
Dhole Patil College of Engineering, Pune, India

Abstract: **In the current age of global energy crisis, the production of energy through alternate energy resources has gained a significant attention. Wind as a source of energy is a very attractive due to the fact that fuel is free of cost. This research is about the aerodynamic design of a VAWT (Vertical Axis Wind Turbines) blade using the analytical and CFD techniques. The blade design parameters and dimensions are taken aiming the required power output, and CAD models are developed to evaluate the aerodynamic forces, like lift and drag, over the surface of the blade. These forces which are very helpful for the evaluation of the structural integrity of the VAWT blade are then found to be in a close agreement with CFD results which are simulated using commercial software - ANSYS 19.0. The static CFD model is developed at a selected pitch angle during a complete 360° where the aerodynamic forces evaluated. VAWT blades has a dominant role in the generation of the aerodynamic forces and the power generation of the turbine. However, there is a significant uncertainty in determining the blade AOAs during operation due to very complex flow structures, and this limits the turbine design optimization.**

Keywords: **Wind Turbine, Wind Energy, Vertical Axis Wind Turbine, VAWT, Aerodynamic Design, Airfoil, Non-conventional Energy**

1. Overview

1.1 Overview of the Thesis

With an ever increasing energy crisis occurring in the world it will be important to investigate alternative methods of generating power in ways different than, fossil fuels. In fact, one of the biggest sources of energy is all around us all of the time, the wind. It can be harnessed not only by big corporations but by individuals using Vertical Axis Wind Turbines (VAWT). VAWT's offer similar efficiencies as compared with the horizontal axis wind turbines (HAWT) and in fact have several distinct advantages. One advantage is that unlike their HAWT counterparts, they can be placed independently of wind direction. This makes them perfect for locations where the wind direction can change daily, and do not need yaw mechanism, rudders or downwind coning. Their electrical generators can be positioned close to ground, and hence easily accessible.

This work mainly focus on more generation of power using wind energy. The Angle of Attack (AOA) of the Vertical Axis Wind Turbines blades has a dominant role in the generation of the aerodynamic forces and the power generation of the turbine. The CFD analysis is done at five different angle and constant speed to determine the optimum Angle of Attack. In blades (airfoil) of turbine, for zero degree angle of attack there is uniform distribution of pressure and there is no chance of flow separation. But for 15 degree angle of attack flow separation takes place on one end of blades and it doesn't utilizes all the flow required for lift forces thus decreasing efficiency of blades. There is an optimum angle between zero degree and 15 degree in which least flow separation occur and proper lift is gained. This optimum angle of attack is determined by CFD analysis on blade at angle between zero to 15 degree. It is intended to come out with an optimum angle for Vertical Axis Wind Turbine at which lift force obtained is more and least flow separation occur for more efficient working of Vertical Axis Wind Turbine.

1.2 Aim and Objectives

1.2.1 Aim

Fabrication and flow analysis of vertical axis wind turbine.

1.2.2 Objectives

- To analyze the effect of speed at every single point on the blade of the wind turbine.
- To determine the maximum lift angle of blades.
- To study the effects of various attack angles and wind speeds.

2. Introduction

2.1 Wind Energy

The wind is a clean, free, and readily available renewable energy source. Each day, around the world, wind turbines are capturing the wind's power and converting it to electricity. Wind power generation plays an increasingly important role in the way we power our world – in a clean, sustainable manner.

But how is wind energy created? Wind turbines allow us to harness the power of the wind and turn it into energy. When the wind blows, the turbine's blades spin clockwise, capturing energy. This triggers the main shaft of the wind turbine, connected to a gearbox within the nacelle, to spin. The gearbox sends that wind energy to the generator, converting it to electricity. Electricity then travels to a transformer, where voltage levels are adjusted to match with the grid.

2.2 Advantages And Challenge of Wind Power

Wind power is one of the fastest-growing energy sources in the world because of its many advantages. Wind power also presents inherent challenges in some regions of the world, which are being addressed through research and development (R&D) projects around the globe.

- Wind power is cost-effective in many regions. In others, wind power needs to compete with other energy sources, but global R&D efforts are working on solutions to reduce the leveled cost of electricity (LCOE) of both onshore and offshore wind power.

- Another advantage to wind power is that it is a domestic source of energy, harnessing a limitless local resource. Some viable locations for wind farms, however, are located remote areas that would present challenges in construction and electricity transmission logistics. Technology breakthrough such as two-piece blades and modular construction are helping overcome such challenges.

- An additional benefit of wind power is it is a sustainable source of energy, as wind turbine operation does not directly emit any CO_2 or greenhouse gases—helping countries meet their emission reduction targets and combating climate change. Wind energy is plentiful, readily available, and capturing its power does not deplete our valuable natural resources. In fact, an environmental benefit to wind power is its ability to counter the detrimental effects of climate change. The Global Wind Energy Outlook projects that by 2030 wind energy will offset 2.5 billion tons per year of carbon

Collecting operational wind energy data from wind turbines and transforming it into valuable information helps increase revenue, reduce costs and lower risk for wind turbine operators. At GE, we provide data-driven wind energy analytics insights, expert recommendations and advanced field services—all integrated into a single software platform that allows us to optimize 15,000+ wind turbines across 12 different OEMs.

Check out our Digital solutions for wind assets, a suite of apps that work with our hardware and services solutions to leverage data and analytics—enhancing the efficiency, cyber security, reliability and profitability of your assets.

2.3 Wind Turbines

A wind turbine, or alternatively referred to as a wind energy converter, is a device that converts the wind's kinetic energy into electrical energy.

Wind turbines are manufactured in a wide range of vertical and horizontal axis. The smallest turbines are used for applications such as battery charging for auxiliary power for boats or caravans or to power traffic warning signs. Larger turbines can be used for making contributions to a domestic power supply while selling unused power back to the utility supplier via the electrical grid. Arrays of large turbines, known as wind farms, are becoming an increasingly important source of intermittent renewable energy and are used by many countries as part of a strategy to reduce their reliance on fossil fuels. One assessment claimed that, as of 2009, wind had the "lowest relative greenhouse gas emissions, the least water consumption demands and the most favorable social impacts" compared to photovoltaic, hydro, geothermal, coal and gas.

2.4 Types of Wind Turbine

2.4.1 Horizontal Axis

Horizontal-axis wind turbines (HAWT) have the main rotor shaft and electrical generator at the top of a tower, and must be pointed into the wind. Small turbines are pointed by a simple wind vane, while large turbines generally use a wind sensor coupled with a servo motor. Most have a gearbox, which turns the slow rotation of the blades into a quicker rotation that is more suitable to drive an electrical generator.

Since a tower produces turbulence behind it, the turbine is usually positioned upwind of its supporting tower. Turbine blades are made stiff to prevent the blades from being pushed into the tower by high winds. Additionally, the blades are placed a considerable distance in front of the tower and are sometimes tilted forward into the wind a small amount [9].

Downwind machines have been built, despite the problem of turbulence (mast wake), because they don't need an additional mechanism for keeping them in line with the wind, and because in high winds the blades can be allowed to bend which reduces their swept area and thus their wind resistance. Since cyclical (that is repetitive) turbulence may lead to fatigue failures, most HAWTs are of upwind design.

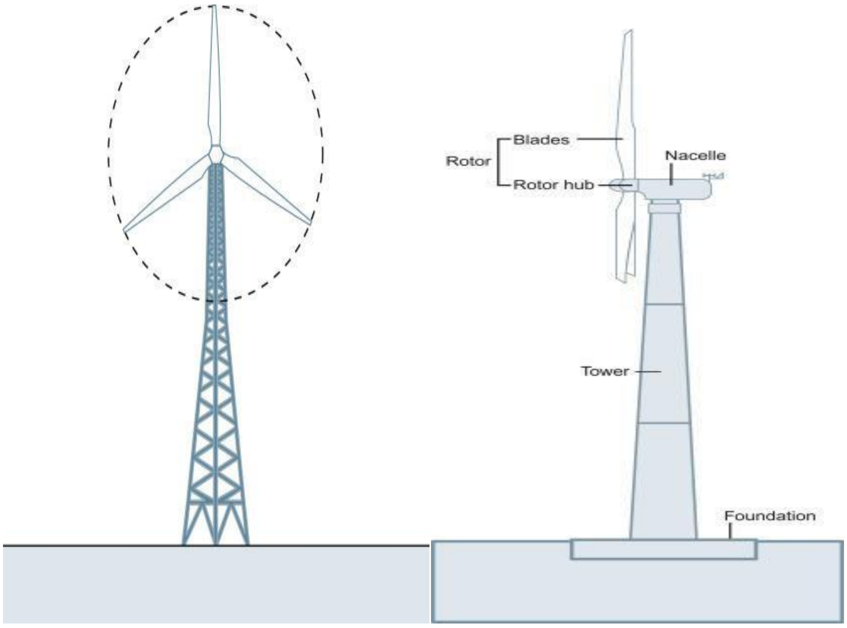

Fig. 1 - Horizontal Axis Wind Turbine

The horizontal axis wind turbine (HAWT) has the axis of the blades horizontal to the ground. On this turbine, two or three, or multiple, blades spin upwind of the tower that it sits on. Great care is taken about the design of the rotor tip because the tip of the blade moves substantially faster than the root of the blade. Blade tips have changed over time with continuing research, which is also done to study performance, since most of the torque of the rotor comes from the outer part of the blades. In addition, the airflow around the tip of rotor blades is extremely complex, compared to the airflow over the rest of the rotor blade.

2.4.2 Vertical Axis

VAWTs offer a number of advantages over traditional horizontal-axis wind turbines (HAWTs). They can be packed closer together in wind farms, allowing more in a given space. They are quiet, Omni-directional, and they produce lower forces on the support structure. They do not require as much wind to generate power, thus allowing them to be closer to the ground where wind speed is lower. By being closer to the ground they are easily maintained and can be installed on chimneys and similar tall structures. When the wind passes through the blades of a HAWT, all of them contribute to energy production. When the wind passes through a VAWT, only a fraction of the blades generates torque while the other parts merely 'go along for the ride'. The result is comparably reduced efficiency in power generation. Getting high efficiency from small scale VAWT is somewhat difficult. It is because of the performance of VAWT is very sensitive to the lift/drag ratio of the blade and it is not good in the low Reynolds number condition of small applications [10]. There are a number of obstacles in scaling VAWTs to commercial size. The first is that they aren't as sturdy by design as a HAWT.

Fig. 2 - Vertical Axis Wind Turbine

This is because of where a HAWT carries most of its stress compared to widely-used VAWT models. VAWTs' advantage sonly in niche environments. At present, VAWTs don't generate enough electricity that the full-lifecycle accounting shows them to be advantageous on a cost or materials basis over HAWTs. VAWT designs have the blades much closer to the ground than HAWTs, so they are losing significant amounts of wind. There are two main types of VAWTs called the drag driven VAWT (Savonius type) and the lift driven VAWT (Darrieus type).

The Savonius type functions similar to a water wheel that uses drag forces. On the other hand, the Darrieus type has blades similar to the HAWTs. Main rotor shaft of the VAWT is arranged vertically. The generator can be connected by using that axis shaft. The rudder is unnecessary for this type wind turbines because it accepts the wind which comes from any direction. The maximum possible efficiency of lift driven turbines is larger than the drag driven turbines, the main attention today is focused on lift driven turbines. The first turbine of this design was patented in 1931 by G. J. M. Darrieus.

2.4.2.1 Darrieus Wind Turbine

The Darrieus wind turbine is a type of vertical axis wind turbine (VAWT) used to generate electricity from wind energy. The turbine consists of a number of curved aerofoil blades mounted on a rotating shaft or framework. The curvature of the blades allows the blade to be stressed only in tension at high rotating speeds. There are several closely related wind turbines that use straight blades [10].

This design of the turbine was patented by Georges Jean Marie Darrieus, French aeronautical engineer; filing for the patent was October 1, 1926. There are major difficulties in protecting the Darrieus turbine from extreme wind conditions and in making it self-starting.

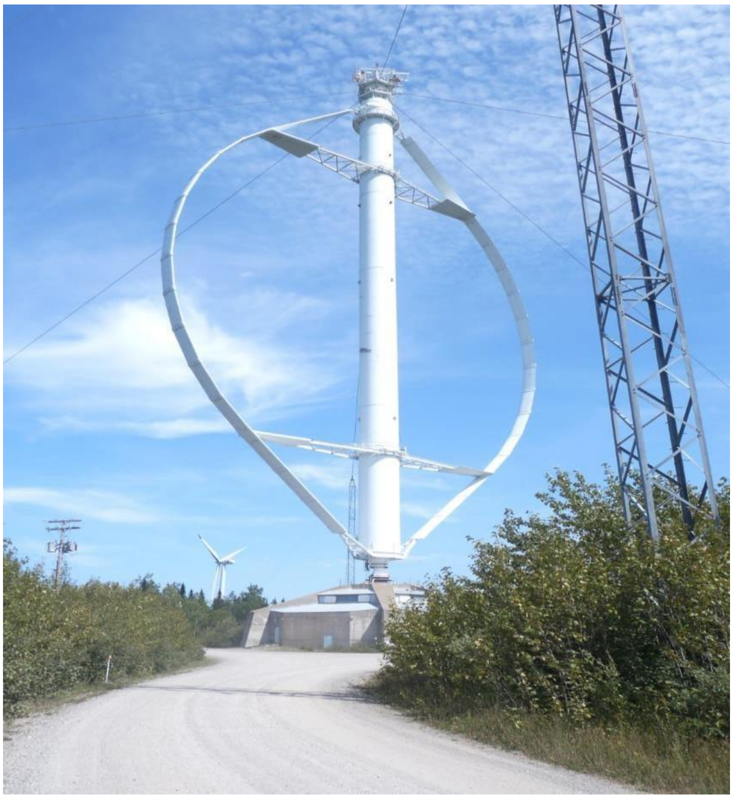

Fig. 3 - Darrieus Wind Turbine

2.4.2.2 Savonius Wind Turbine

The Savonius turbine is one of the simplest turbines. Aerodynamically, it is a drag-type device, consisting of two or three scoops. Looking down on the rotor from above, a two-scoop machine would look like an "S" shape in cross section. Because of the curvature, the scoops experience less drag when moving against the wind than when moving with the wind. The differential drag causes the Savonius turbine to spin. Because they are drag-type devices, Savonius turbines extract much less of the wind's power than other similarly-sized lift-type turbines [10]. Much of the swept area of a Savonius rotor may be near the ground, if it has a small mount without an extended post, making the overall energy extraction less effective due to the lower wind speeds found at lower heights.

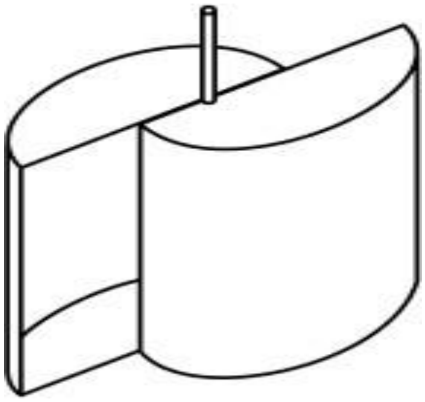

Fig. 4 - Savonius Wind Turbine

On the other hand, the situation with smaller installations with a nominal output up to approximately 10 kW can be considered to be substantially different. At this level of output, there are very many applications which up until now could only be insufficiently covered with horizontal systems. In particular, horizontal installations come up against their limits when located in high mountain areas, in regions with extremely strong or gusty winds, or in urban areas. But also in regions with relatively constant winds, that is, where the conditions are ideal for systems with a horizontal axis, a VAWT can have its advantages, at minimum if the neighbours complain about the annoyance of the noise.

There have already been reports of enraged neighbors who have settled the acoustical problems with firearms. For manufacturers of small-scale installations with a vertical axis, the issue is to concentrate upon the advantages of the vertical concept and, at the same time, to find ways of avoiding the weaknesses of the original constructions of Savonius and Darrieus to the greatest extent possible. The disadvantages of large wind turbines with vertical spindles indicated above, in particular that of the installation needing to be close to the ground, do not play any role with smaller systems.

There is no reason why a wind turbine of that sort could not be installed on a mast of any height or on a building. And the somewhat higher material expenditures, measured with respect to the system costs of a small installation, are of hardly any consequence. One great weakness of the systems originally patented by Savonius and Darrieus is, for example, the difficulty of automatic operation. However, there are already several models that combine the basic design of both concepts and thus reduce or avoid this problem. In addition, a limitation on the revolutions per minute can prove to be necessary in order to not subject the system to unnecessary loads. During extreme winds, traditional systems are either braked or turned out of the wind. According to the same principle, it is of course conceivable to regulate the blades of a so-called "H-rotor" (Grimily) according to the wind velocity. That involves, however, relatively costly and maintenance-intensive mechanics.

3. Literature Review

3.1 Vertical Axis Wind Turbine

Vertical axis wind turbine (VAWT) is a type of wind turbine in which the axis of rotation is in vertical direction. In VAWT the wind flows in perpendicular direction to the axis of rotor shaft. These types of turbines have a lot lower rotational speed, meaning higher torques involved.

Vertical Axis Wind Turbine (VAWT) is relatively simple to implement in urban areas on ground or/and building-roofs, the development of appropriate design of VAWT will open new Opportunities for the large-scale acceptance of these machines. The primary objective of this research was to design and modeling of a small-scale VAWT, which can be used to meet the power for low demand applications [6].

Horizontal Axis Wind Turbine	Vertical Axis Wind Turbine
The rotating axis of the blade is parallel to direction of wind.	The rotating axis of the blade is perpendicular to direction of wind.
The main rotor shaft runs horizontally in HAWTs.	The main rotor shaft runs vertically in HAWTs
HAWTs are generally used under streamline wind conditions where a constant stream and direction of wind is available.	VAWTs are mainly beneficial in areas with turbulent wind flow such as rooftops, coastline, cityscapes etc.
The rotor faces the wind stream to capture maximum wind energy.	The rotor can accept wind stream from any direction.
Inspection and maintenance is difficult in HAWTs.	Inspection and maintenance is easy in VAWTs
HAWTs extract more power from wind.	VAWTs extract less power from wind.
The operate fine in moderate wind speeds.	The can operate even in low speeds.

Table 1 - Horizontal and Vertical Axis Wind Turbine Differences

Recent research has shown that VAWTs can be packed significantly closer than HAWTs with less drop in efficiency, making them much better suited for wind farms. Even if individual VAWTs are less efficient than HAWT, the tighter spacing of counter rotating turbines allows VAWT farms to have higher power densities. The maintenance of VAWTs are easier and safer, placing components under the water level improves the stability of system.

3.2 NACA 0012 Airfoil

In this study we have chosen symmetrical blades because they are easier to manufacture.

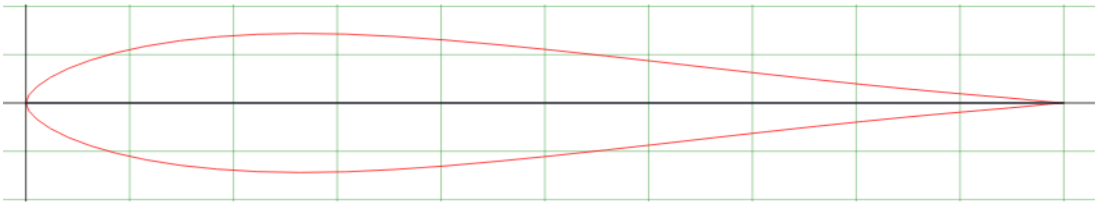

Fig. 5 - NACA00123 Airfoil

The NACA airfoils are airfoils shapes for aircraft wings developed by National Advisory Committee for Aeronautics (NACA). The shape of NACA airfoil is described using a series of digits following the word “NACA”. The parameters in the numerical code enter into the equations to precisely generate the cross section of the airfoils and calculate the properties. Another reason it was decided to choose symmetrical airfoils was that although these types of airfoils are not most efficient at producing lift, they do produce a lift stall effect that allows the system to obtain equilibrium. For six different Reynolds number from 10,000 to 8,00,000 and eight different angles of attack 10º to 80º. It is observed that, close matching in CL and CD values are obtained by present CFD analysis in comparison with experimentally available values, which validates the present methodology of CFD analysis. Results of CFD analysis for CL shows some deviations with experimental results sandia national laboratories energy report for lower angle of attack, however for higher angle of attack it shows close match with experimental results.

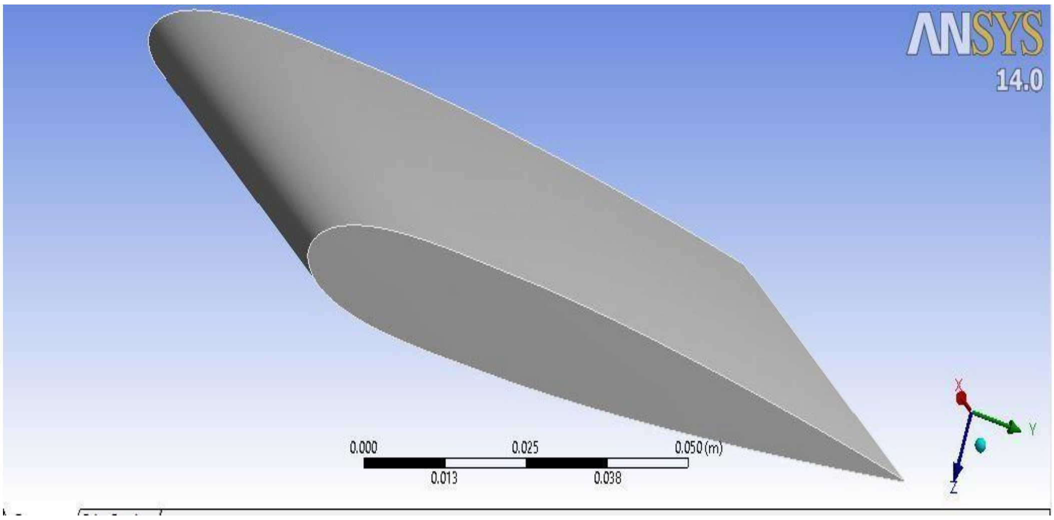

Fig. 6 - CAD Model of Airofoil

A study by Liu Shuqin of Shandong University of Chin focused on the power generation based on changing the type of blades that were used and this was used to give an insight into choosing the blade shapes for our VAWT design. The experimental setup is given below in Table 2. The results showed that self-pitch streamline symmetrical blades generated a higher power output than the others, as shown in Figure 3.

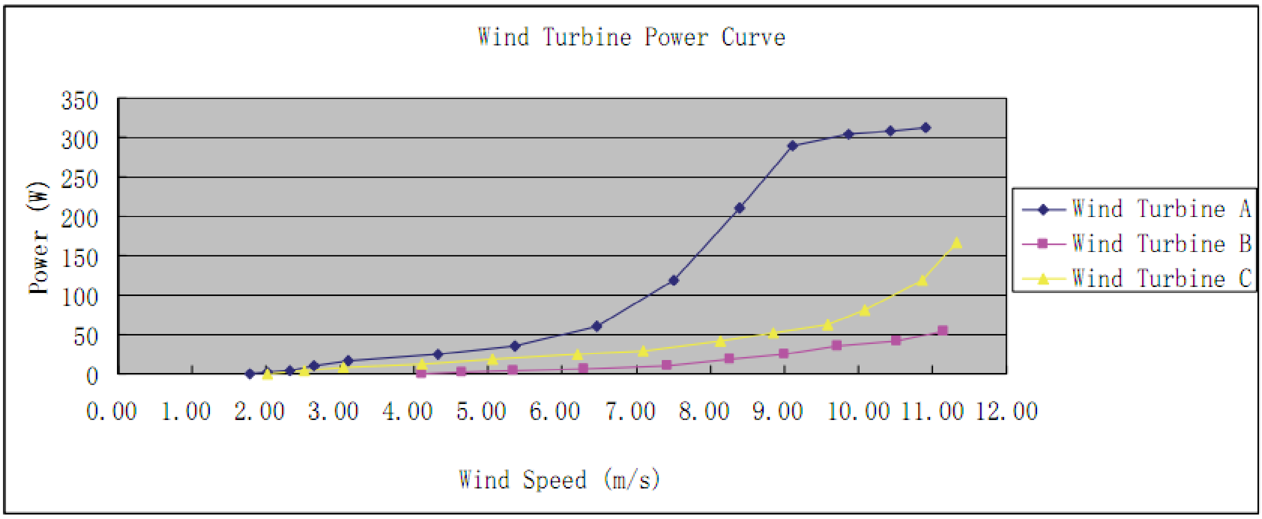

Fig. 7 - Power Generation of Various Types of Blades

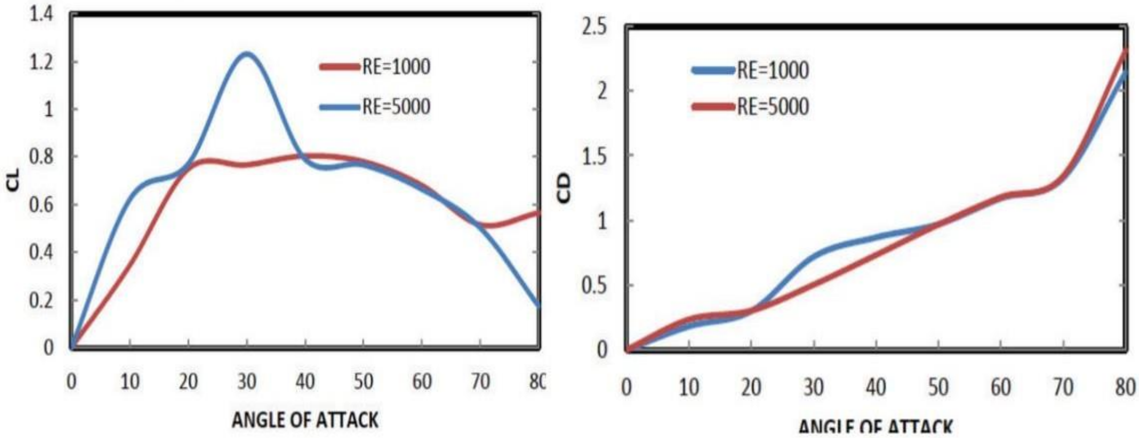

Fig. 8 - Angle of Attack vs Lift and Drag Force

From this investigation it is concluded that Reynolds number 5000 provides maximum lift coefficient and drag coefficient than Reynolds number 1000. Reynolds number increases lift force and drag force both CFD and experimental results indicates that NACA 0012 maximum lift and drag at higher Reynolds number [5].

3.3 Lift And Drag Force

The lift that is created from the wind passing over the airfoil is eventually lost and the blade goes into its stall condition, but the other blades pick up more lift and keep the cycle going. This in turn regulates the entire VAWT [2].

To analyze the effectiveness of a VAWT, methods of Computational Fluid Dynamics (CFD) were used to simulate various airflows and directions. The first part of CFD analysis analyzed the 2D flow over chosen airfoil. The numerical simulation can accomplished by means of CFD simulation. CFD simulation of VAWT is performed by solving Unsteady Reynolds Averaged Navier Stokes (URANS) equation[4]. In the case of VAWT simulation, away to validate the chosen numerical method is to compare the obtained data with experimental one in terms of power coefficient.

Some researcher had tested their CFD methods with respect to small scale turbines. For two dimensional 2D simulation a large discrepancy could be seen. For 3D case, although the level of discrepancy is lower than 2D, the results is not that satisfactory. Drag is the force experienced by an object representing the resistance to its movement through a fluid. Sometimes called wind resistance or fluid resistance, it acts in the opposite direction to the relative motion between the object and the fluid. The example opposite shows the aerodynamic drag forces experienced by an aerofoil or aircraft wing moving through the air with constant angle of attack as the air speed is increased.

The Total Aerodynamic Drag is the sum of the following components:

- **Induced Drag** - Due to the vortices and turbulence resulting from the turning of the air flow and the downwash associated with the generation of lift. Increases with the angle of attack. Inversely proportional to the square of the air speed. Decreases as aircraft speed increases and the angle of attack is reduced. Induced drag associated with the high angle of attack needed to maintain the lift is dominant at low air speeds.

- **Form Drag or Pressure Drag** - Due to the size and shape of the aerofoil. Increases with the square of air speed. Streamlined shapes designed to reduce form drag.

- **Friction Drag** - Arises from the friction of the air against the "skin" of the aerofoil moving through it. Increases with the surface area of the aerofoil and the square of air speed.

- **Profile Drag or Viscous Drag**- The sum of Friction Drag and the Form Drag.

- **Parasitic Drag or Interference Drag** - Incurred by the non-lifting parts of the aircraft such as the wheels, fuselage, tail fins, engines, handles and rivets. Increases with the square of air speed. Parasitic drag becomes dominant at higher air speeds.

- **Wave Drag** - Due to the presence of shock waves occurring on the blade tips of aircraft and projectiles. Associated with passing the sound barrier it is a sudden and dramatic increase in drag which only comes into play as the vehicle increases speed through transonic and supersonic speeds. Independent of viscous effects.

Different aerodynamic force can be generated on the sectional profile of the blades due to air flow stream through the turbine rotors. Mainly those forces were called drag and lift forces: the drag force FD in the direction of the air flow and lift force FL perpendicular to the air flow [3]**.** Aerodynamically, these forces have deep relations with angle of attack of the wind.

Lift Force: $FL = 0.5\ CL\rho Av^2$, Drag Force: $FD = 0.5\ Cd\rho Av^2$

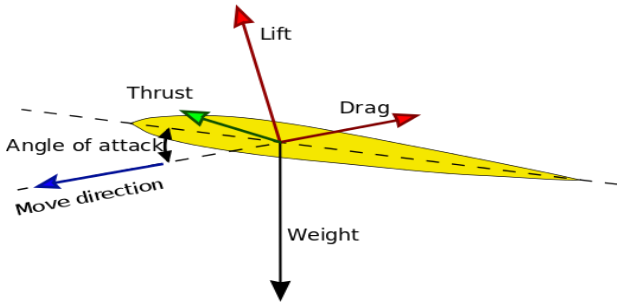

Fig. 9 - Angle of Attack Demonstration

3.4 Angle of Attack

The angle of attack of a turbine blade is the angle between the direction of the apparent or relative wind and the chord line of the blade. For an aircraft wing, it is the angle between the direction of motion of the wing and the chord line of the wing.

At very low angles of attack, the airflow over the aerofoil is essentially smooth and laminar with perhaps a small amount of turbulence occurring at the trailing edge of the aerofoil. The point at which laminar flow ceases and turbulence begins is known as the separation point.

Increasing the angle of attack increases the area of the aerofoil facing directly into the wind. This increases the lift but it also moves the separation point of laminar flow of the air above the aerofoil part way up towards the leading edge and the result of the increased turbulent flow above the aerofoil is an increase in the drag.

Maximum lift typically occurs when the angle of attack is around 15 degrees but this could be higher for specially designed aerofoil. Above 15 degrees, the separation point moves right up to the leading edge of the aerofoil and laminar flow above the aerofoil is destroyed. The increased turbulence causes the rapid deterioration of the lift force while at the same time it dramatically increases the drag, resulting in a stall.

The angle of attack of a VAWT blades has a dominant role in the generation of aerodynamic forces and the power generation of the turbine. However, there is a significant uncertainty in determining the blade angle of attack during operation due to very compress flow structure and this limits the turbine design optimization. In this study a fast and accurate method for calculation of constantly changing angle of attack based on the velocity flow field data at two reference points upstream the turbine blades. The interactions between the wind and the VAWT rotations lead to very complex time variant aerodynamic phenomena around the spinning blades. Although several studies have analyzed the instantaneous power and torque generation over one rotating cycle. A range of different fidelity analyses has been used to investigate both fixed and variable pitch VAWTs and the estimations of AOA (angle of attack).

In fluid dynamics, a stall is reduction in the lift coefficient generated by airfoil as an angle of attack increases this occurs when the critical angle of attack of the foil is exceeded. The critical of attack is near about 10-15 degrees but it may vary significantly depending upon the type of fluid and Reynolds number.

In the airfoil, according to Newton's law the lift force involves attachment of the boundary layer of air on the top of airfoil which results in downwash of air behind the airfoil. If the airfoil gives the air a downward force, then by Newton's third law, the wing experiences a force in opposite direction a lift. Increasing the angle of attack gives a larger lift from upward component of pressure on the bottom of the wing. The lift force can be considered as Newton's third law reaction force to the force exerted downward on the air by wing. At too high angle of attack, turbulent flow increases the drag dramatically and will stall the airfoil.

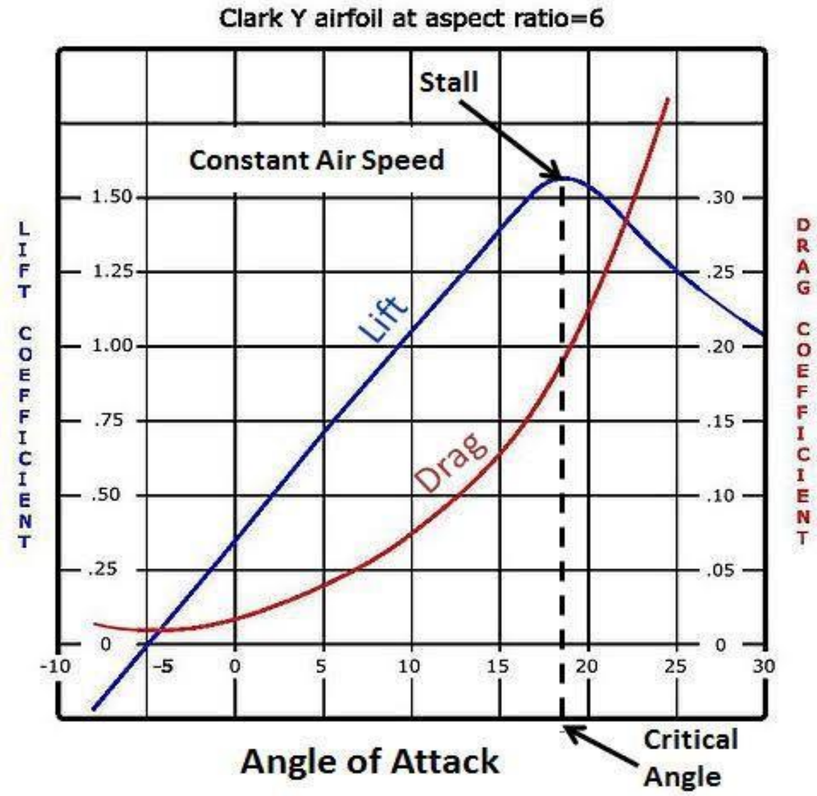

Fig. 10 - Variation of Lift and Drag vs Angle of Attack

.
Whenever there is a relative moment between a fluid and a solid surface, weather externally around a body, or internally in an enclosed passage, a boundary layer exists with a viscous forces present in the layer of fluid closed to the surface boundary layer can either be laminar or turbulent. The wings provide lift by creating a situation where the pressure above the wing is lower than the pressure below the wing. Since the pressure below the wing is higher than the pressure above the wing, there is a net force upwards [7].

Viscosity is essential in generating lift the effects of viscosity lead to the formation of starting vortex which in turn is responsible for producing a proper condition for lift. Flow separation or boundary layer separation or boundary layer separation is a detachment of boundary layer from a surface into a wake. Separation occurs in a flow that is slowing down, with pressure increasing after passing a thickest part of a steam line body or passing through a widest passage for example flowing against an increasing pressure is flowing in a an adverse pressure gradient. The boundary layer separates when it has travelled far enough in a adverse pressure gradient that the speed of boundary layer relative to the surface has stopped and reversed direction. The flow becomes detached from the surface, and the instead takes the form of eddies and vortices. The fluid exerts a constant pressure on the surface it has separated instead of continued increasing pressure if still attached. In aerodynamics flow separation results in reduced lift and increased pressure drag, caused by the pressure differentials between the front and rare surfaces of the object.

3.5 Flow Separation

When the boundary layer separates, its remnants form a shear layer and the presence of a separated flow region between the shear layer and surface modifies the outside potential flow and pressure field. In the case of airfoils, the pressure field modification results in an increase in pressure drag, and if severe enough will also result in stall and loss of lift, all of which are undesirable. For internal flows, flow separation produces an increase in the flow losses, and stall-type phenomena such as compressor surge, both undesirable phenomena.

Fig. 11 - Flow Separation

Another effect of boundary layer separation is regular shedding vortices, known as a Kármán vortex street. Vortices shed from the bluff downstream surface of a structure at a frequency depending on the speed of the flow. Vortex shedding produces an alternating force which can lead to vibrations in the structure. If the shedding frequency coincides with a resonance frequency of the structure, it can cause structural failure. These vibrations could be established and reflected at different frequencies based on their origin in adjacent solid or fluid bodies and could either damp or amplify the resonance.

Separation of flow or stall is a very important viscous fluid phenomenon which causes energy loss accompanied by deviation of stream lines and reverse flow[8]. The definition of stall may be referred to any reverse flow at a wall and stall may be viewed as a spectrum of states with many possible geometries and transient elements. For external flow, the separation of flow increases drag and decreases lift; for internal flow, stall reduces efficiency. Since malfunctions of downstream components are caused due to separation of flow and the maximum pressure recovery can be obtained at or near stall, it is essential from a practical point of view to understand and predict stall in order to design optimum fluid devices. However, the problem of stall is very complex and despite a great deal of research efforts, mainly based on boundary- layer theory and experiments, stall remains the most common unsolved problem of fluid mechanics whose solution can be applied to engineering design.

3.6 Components of Wind Turbine

The principle behind wind turbines is very simple: the energy in the wind turns the blades around a rotor. The rotor is connected to the shaft, which spins a generator to create electricity. Wind turbines are mounted on a tower to capture the energy from the wind. The higher the blades are, the more they can take advantage of faster and less turbulent wind. A simple wind turbine consists of three main parts, the blade, shaft and generator [11].

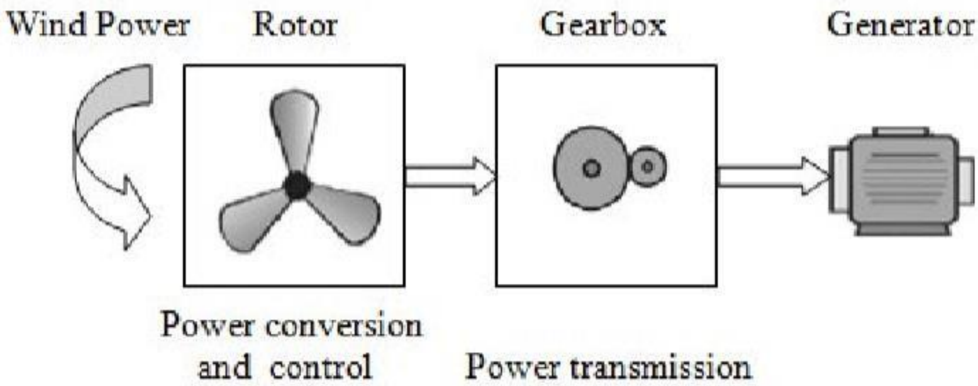

Fig. 12 - Mechanical Components of Wind Turbine

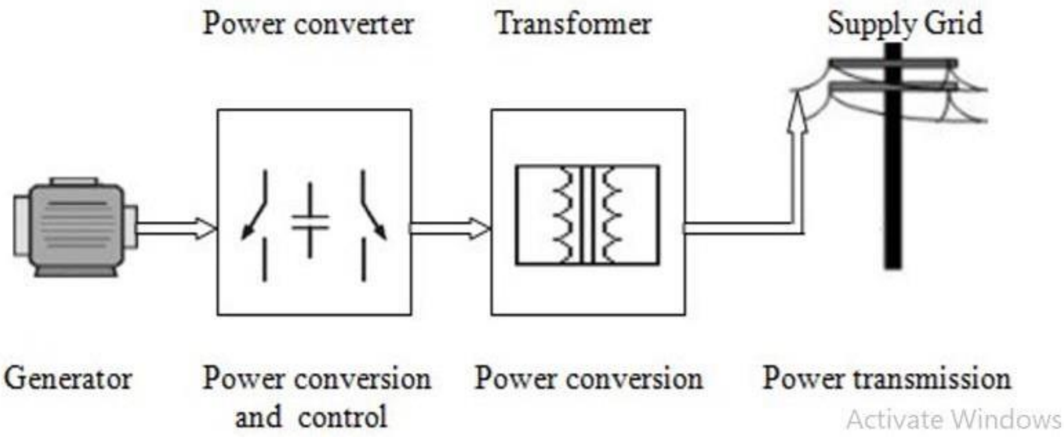

Fig. 13 - Electrical Components of wind turbine

1. **Blades:** The blades acts as barriers to the wind. When the wind force the blade to move, some of the wind energy is transferred to the rotor.

2. **Shaft:** When the rotor spins, the shaft also spins, and transfers the Mechanical energy into rotational energy.

3. **Gearbox:** It is typically used in a wind turbine to increase rotational speed from a low speed rotor to a higher speed electrical generator. A common ratio is about 90:1with rate 16.7 rpm input from the rotor to 1500rpm output for the generator.

4. **Generator:** A generator uses the difference in the electrical charge to produce a change in voltage. Voltage is actually electrical pressure, the force that moves an electrical current. The voltage drives the electrical current (alternating current power) through power lines for distribution.

5. **Power Converter and Control:** Power converter is an electronic application of solid state electronic to control and convert one form of electrical power to another form of electrical power to another form such as converting between AC and DC or changing the magnitude and phase of voltage and current or frequency or combination of these.

6. **Transformer:** It acts as a link between wind turbines and distribution grid. It steps up the low output voltage from the generator to higher distribution voltage level. Wind turbine transformers are considered to be one of the sensitive and weak components in a wind farm.

7. **Supply Grid:** It is an interconnected network for delivering electricity from producers to consumers.

The wind turbines are arranged in such manner that one turbine does not take the wind away from another. However other factors such as environmental considerations, visibility and grid connection requirements often take precedence oer the optimum wind capture layout [12].

4. Mathematical Modeling

New and powerful methods of characterizing existing and new airfoil geometries with mathematical equations are presented. The methods are applicable to a wide range of airfoil shapes representing traditional, cusped, reflexed, flat-bottom, laminar, transonic, and supersonic designs. With the emphasis on low-speed airfoils, several existing airfoils are first closely matched with the math-modeling methods. Then, to support the design of new airfoil geometries, a new interpretation of Theodore's potential flow method is outlined for the calculation and presentation of surface velocity in inviscid flow. Also, a vector approach is introduced for the calculation of pitching moment. Finally, new math-modeled airfoils are proposed for conventional and unique aircraft configurations.

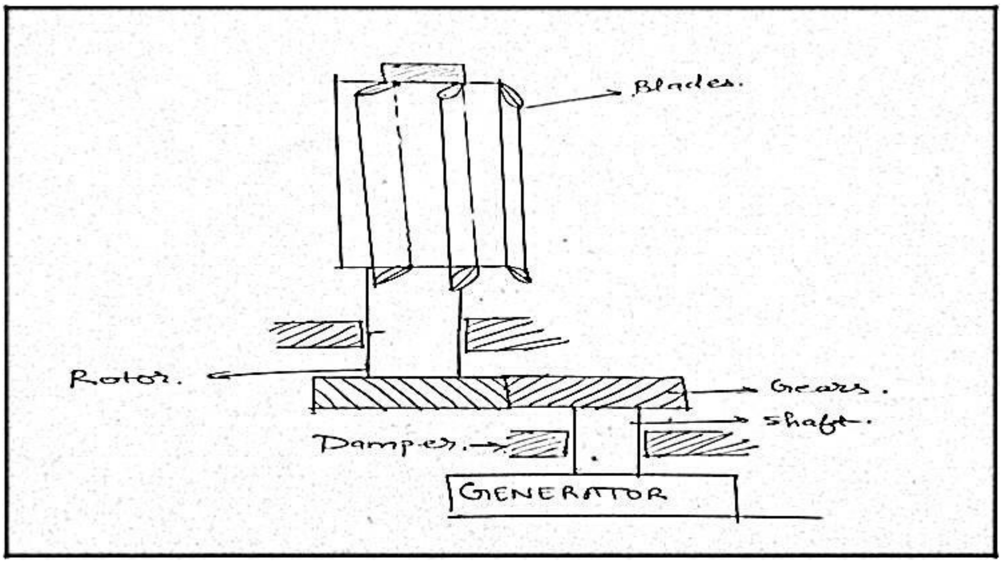

Fig. 14 - Model of Wind Turbine System

In the given figure we know that according to modeling there are different systems like rotor, gears, dampers, shaft, and generator. This system helps to find the mathematical equations.

J - Polar moment of Inertia of shaft B1, B2-Damping coefficient
T - Torque
Tm - Motor torque T1-Torque after D1
T2 - Torque after D2
Tg - Torque generated

$J1\varnothing2 + B1\varnothing1 + T1 = Tm$ ____ a
$J2\varnothing2 + B2\varnothing1 + Tg = T2$ ____ b

$m = \frac{d}{t}$, r1 = t1 , r2 = t2

$$\frac{r1}{r2} = \frac{t1}{t2}$$

$T1\varnothing = T2\varnothing$

$$\frac{T1\varnothing}{T2\varnothing} = \frac{r1}{r2}$$

(Where t1 and t2 are the teeth)

from 2,

$$\frac{T2}{T1} = \frac{t2}{t1}$$

$\frac{t2}{t1}T1 = J2\varnothing2 + B2\varnothing2 + Tgt1$

$T1 = (J2\varnothing2 + B2\varnothing2 + Tg)\frac{t2}{t1}$

Converting into Laplace equation a and b
$J1S^2\varnothing1(S) + B1S\varnothing1(S) + T1(S)$
$= Tm(S)\ J2S^2\ \varnothing2(S) + B2S\varnothing2(S) + Tg(S)$
$= T2(S)\ (J1S^2 + B1S)\ \varnothing1(S)$
$= Tm(S) - T1(S)\ (J2S^2 + B2S)\ \varnothing2(S)$
$= T2(S) - Tg(S)$

$$\frac{\varnothing1(S)}{\varnothing2(S)} = \frac{Tm(S) - T1(s)}{T2(S) - Tg(s)} \text{X} \frac{J2S^2 + B2S}{J1S^2 - B1S}$$

Transfer function of the system

5. Methodology

5.1 Software to be Used

5.1.1 CATIA (For Model Design)

- CATIA is a 3D modeling software from dassault systems.
- It can produce models in 3D of almost everything, focusing on the surfaces and curves modeling, making it far easier than Solid works for that kind of job.
- CATIAV5 Student edition. CATIA is the world's engineering and design leading software for product 3D CAD design excellence.
- It is used to design, simulate, analyze, and manufacture products in a variety of industries including aerospace, automotive, consumer goods, and industrial machinery.
- We have to design an streamline blade with size 120 cm (L) * 20 cm (W) * 2.5 cm (T), Diameter of arm rotation 2 m with 300 W permanent magnet generator.

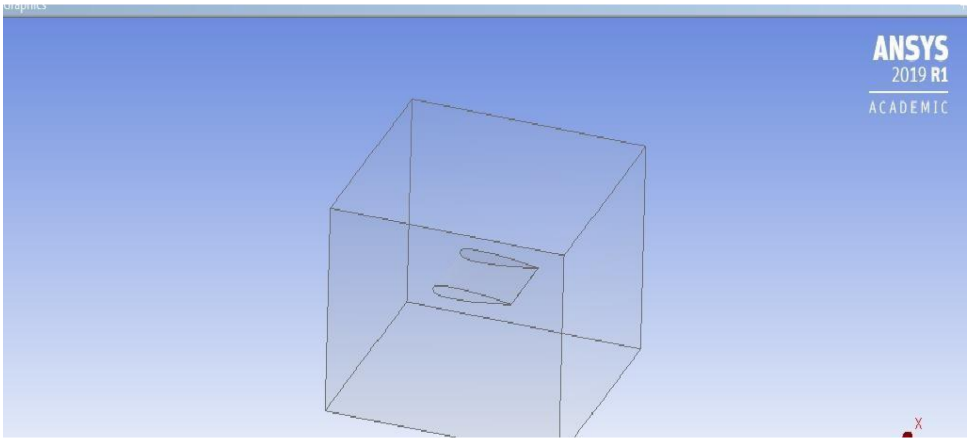

Fig. 15 - Enclosure of Blade Model

5.1.2 Fluent (For Flow Analysis)

Fluent software contains the broad, physical modeling capabilities needed to model flow, turbulence, heat transfer and reactions for industrial applications. These range from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing and from clean room design to waste-water treatment plants. Fluent spans an expansive range, including special models, with capabilities to model in-cylinder combustion, aero- acoustics, turbo machinery and multiphase systems.

Fluent also offers highly scalable, high-performance computing (HPC) to help solve complex, large-model computational fluid dynamics (CFD) simulations quickly and cost-effectively. Fluent set a world supercomputing record by scaling to 172,000 cores.

5.2 Meshing

- Meshing is the initial step in doing analysis of any geometry. It is done in ICEM CFD software of Ansys. After doing the meshing, analysis is done in Ansys Workbench. Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain.

- The mesh influences the accuracy, convergence and speed of the simulation. Furthermore, since meshing typically consumes a significant portion of the time it takes to get simulation results, the better and more automated the meshing tools, the faster and more accurate the solution.

- Ansys provides general purpose, high-performance, automated, intelligent meshing software which produces the most appropriate mesh for accurate, efficient multiphysics solutions from easy, automatic meshing to highly crafted mesh.
- Methods available cover the meshing spectrum of high-order to linear elements and fast tetrahedral and polyhedral to high-quality hexahedral and Mosaic.

- The material selected for the design of streamline blade is aluminum alloy and the property of material are as follow
  - Density: 2719 Kg/m^3
  - Young's modulus: 69 Mpa
  - Yield strength: 276 Mpa
  - Cost: 170 Rs/Kg

- Nodes and Elements are the very backbones of Finite Element Analysis. You will use them in every analysis you will perform in FEA, what are Nodes and Elements in Finite Element Analysis?

- In FEA, you divide your model into small pieces. Those are called Finite Elements (FE). Those Elements connect all characteristic points (called Nodes) that lie on their circumference. This "connection" is a set of equations called shape functions.

- Each FE has its own set of shape functions that connect all of the Nodes of that Element). Adjacent Elements share common Nodes (the ones on the shared edge). This means that shape functions of all the Elements in the model are "tied" thanks to those common nodes.

- Those Elements connect all characteristic points called Nodes, that lie on their circumference. This "connection" is a set of equations called shape functions. Each function element has its own set of shape functions, that connect all of the Nodes of that Element. Adjacent Elements share common Nodes (the ones on the shared edge).

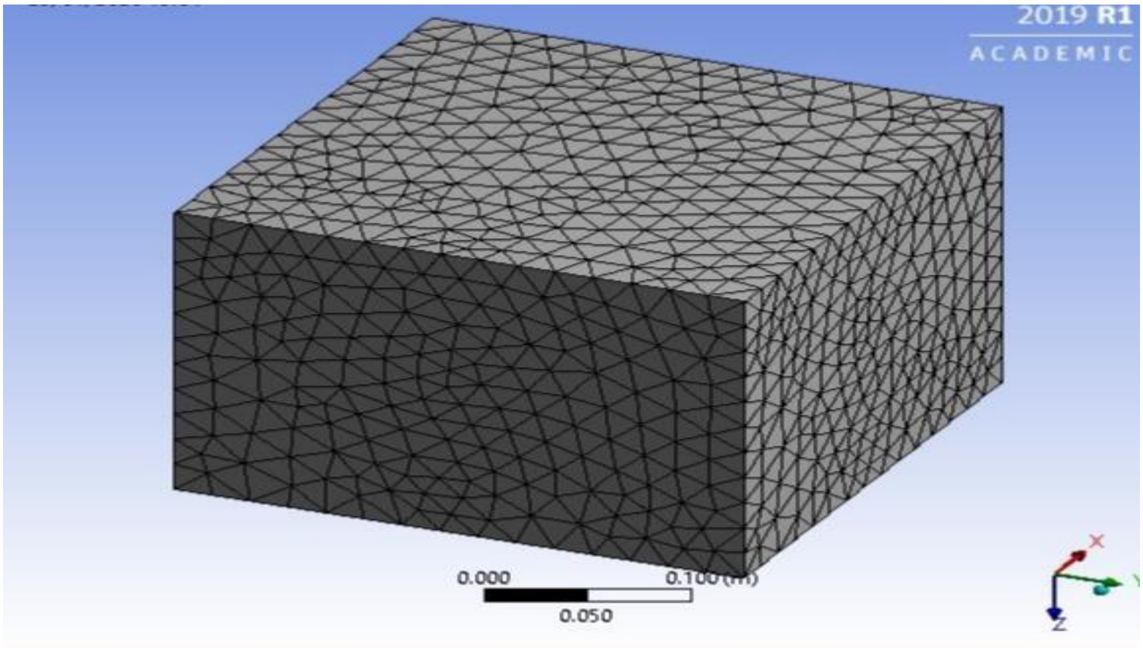

Fig. 16 - Number of Nodes Used in Streamline Blade is 166913 and 916697 Elements

5.3.1 Computational Fluid Dynamics (CFD)

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are:

- **Aerodynamics of aircraft and vehicles:** lift and drag hydrodynamics of ships
- **Power plant:** combustion in internal combustion engines and gas turbines
- **Turbo machinery:** flows inside rotating passages, diffusers etc.
- **Electrical and electronic engineering:** cooling of equipment including microcircuits
- **Chemical process engineering:** mixing and separation, polymer molding
- **External and internal environment of buildings:** wind loading and heating/ventilation
- **Marine engineering:** loads on off-shore structures
- **Environmental engineering:** distribution of pollutants and effluents
- **Hydrology and oceanography:** flows in rivers, estuaries, oceans
- **Meteorology:** weather prediction
- **Bio-medical engineering:** blood flows through arteries and veins

5.3.2 Aim of CFD

The ultimate aim of developments in the CFD field is to provide a capability comparable with other CAE (computer-aided engineering) tools such as stress analysis codes. The main reason why CFD has lagged behind is the tremendous complexity of the underlying behavior, which precludes a description of fluid flows that is at the same time economical and sufficiently complete. The availability of affordable high-performance computing hardware and the introduction of user- friendly interfaces have led to a recent upsurge of interest, and CFD has entered into the wider industrial community since the 1990s.

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (i) a pre-processor, (ii) a solver, and (iii) a post-processor. We briefly examine the function of each of these elements within the context of a CFD code.

5.3.3 How does CFD Code Works?

5.3.3.1 Pre-Processor

Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

- Definition of the geometry of the region of interest: the computational domain
- Grid generation: the sub-division of the domain into a number of smaller, non-overlapping sub-domains, a grid (or mesh) of cells (or control volumes or elements)
- Selection of the physical and chemical phenomena that need to be modeled
- Definition of fluid properties
- Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary

The solution to a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in the grid .In general, the larger the number of cells, the better the solution accuracy. Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often non-uniform: finer in areas where large variations occur from point to point and coarser in regions with relatively little change. Efforts are under way to develop CFD codes with a (self-) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variations. A substantial amount of basic development work still needs to be done before these techniques are robust enough to be incorporated into commercial CFD codes. At present it is still up to the skills of the CFD user to design a grid that is a suitable compromise between desired accuracy and solution cost.

Over 50% of the time spent in industry on a CFD project is devoted to the definition of the domain geometry and grid generation. In order to maximize productivity of CFD personnel all the major codes now include their own CAD-style interface and/or facilities to import data from proprietary surface modelers and mesh generators such as PATRAN and I-DEAS. Up-to-date pre-processors also give the user access to libraries of material properties for common fluids and a facility to invoke special physical and chemical process models (e.g. turbulence models, radioactive heat transfer, combustion models) alongside the main fluid flow equations.

5.3.3.2 Solver

There are three distinct streams of numerical solution techniques: finite difference, finite element and spectral methods. We shall be solely concerned with the finite volume method, a special finite difference formulation that is central to the most well-established CFD codes: CFX/ANSYS, FLUENT, PHOENICS and STAR-CD. In outline the numerical algorithm consists of the following steps:

- Integration of the governing equations of fluid flow over all the (finite) control volumes of the domain
- Discretization - conversion of the resulting integral equations into a system of algebraic equations
- Solution of the algebraic equations by an iterative method

The first step, the control volume integration, distinguishes the finite volume method from all other CFD techniques. The resulting statements express the (exact) conservation of relevant properties for each finite size cell. This clear relationship between the numerical algorithm and the underlying physical conservation principle forms one of the main attractions of the finite volume method and makes its concepts much more simple to understand by engineers than the finite element and spectral methods. The conservation of a general flow variable φ , e.g. a velocity component or enthalpy, within a finite control volume can be expressed as a balance between the various processes tending to increase or decrease it.

5.3.3.3 Post-Processor

As in pre-processing, a huge amount of development work has recently taken place in the post-processing field. Due to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools. These include:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)
- Colour PostScript output

More recently these facilities may also include animation for dynamic result display, and in addition to graphics all codes produce trusty alphanumeric output and have data export facilities for further manipulation external to the code. As in many other branches of CAE, the graphics output capabilities of CFD codes have revolutionized the communication of ideas to the no specialist.

5.3.4 Scope of CFD

Computational fluid dynamics (CFD) is a science that, with the help of digital computers, produces quantitative predictions of fluid-flow phenomena based on the conservation laws (conservation of mass, momentum, and energy) governing fluid motion. CFD has increased in importance and in accuracy; however, its predictions are never completely exact. Because many potential sources of error may be involved, one has to be very careful when interpreting the results produced by CFD techniques.

The most common sources of error are mentioned in the chapter. The key to various numerical methods is to convert the partial different equations that govern a physical phenomenon into a system of algebraic equations. Different techniques are available for this conversion. CFD is merely a tool for analyzing fluid-flow problems. If it is used correctly, it can provide useful information cheaply and quickly. This chapter presents the basics of the finite-difference and finite-element methods and their applications in CFD. There are other kinds of numerical methods, for example, the spectral method and the spectral element method, which are often used in CFD. They share the common approach that discretizes the Navies-Stokes equations into a system of algebraic equations.

5.3.5 Governing Equations of Fluid Flow and Heat Transfer

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics:

- The mass of a fluid is conserved
- The rate of change of momentum equals the sum of the forces on a fluid particle (Newton's second law)
- The rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle (first law of thermodynamics)

5.3.5.1 Mass Conservation in Three Dimensions

The first step in the derivation of the mass conservation equation is to write down a mass balance for the fluid element:

rate of increase of mass in fluid element = net rate of flow of mass into fluid element

The rate of increase of mass in the fluid element is

$$\frac{\delta}{\delta t}\rho\delta x\delta y\delta z = \frac{\delta\rho}{\delta t}\delta x\delta y\delta z$$

Next we need to account for the mass flow rate across a face of the element, which is given by the product of density, area and the velocity component normal to the face. it can be seen that the net rate of flow of mass into the element across its boundaries is given by

$$(\rho u - \frac{\partial(\rho u)1\delta x}{\partial x2})\ \delta y\delta z - (\rho u + \frac{\partial(\rho u)1\delta x}{\partial x2})\ \delta y\delta z$$

$$+ (\rho v - \frac{\partial(\rho v)1\delta y}{\partial y2})\ \delta x\delta z - (\rho v + \frac{\partial(\rho v)1\delta y}{\partial y2}\delta y)\ \delta x\delta z$$

$$+ (\rho w - \frac{\partial(\rho w)1\delta z}{\partial z2}\delta z)\ \delta y\delta x - (\rho w + \frac{\partial(\rho w)1\delta z}{\partial z2}\delta z)\ \delta y\delta x$$

Flows which are directed into the element produce an increase of mass in the element and get a positive sign and those flows that are leaving the element are given a negative sign.

5.3.5.2 Momentum Equation in Three Dimensions

Newton's second law states that the rate of change of momentum of a fluid particle equals the sum of the forces on the particle:

Rate of increase of momentum of fluid particle = Sum of forces on fluid particle

The rates of increase of x-, y- and z-momentum per unit volume of a fluid particle are given by

$$\rho\frac{Du}{Dt}\rho\frac{Du}{Dt}\rho\frac{Du}{Dt}$$

We distinguish two types of forces on fluid particles:

- surface forces – pressure forces – viscous forces – gravity force
- body forces – centrifugal force – Coriolis force – electromagnetic force

It is common practice to highlight the contributions due to the surface forces as separate terms in the momentum equation and to include the effects of body forces as source terms. The state of stress of a fluid element is defined in terms of the pressure and the nine viscous stress components shown in Figure 2.3. The pressure, a normal stress, is denoted by p. Viscous stresses are denoted by τ. The usual suffix notation τ ij is applied to indicate the direction of the viscous stresses. The suffices i and j in τ ij indicate that the stress component acts in the direction on a surface normal to the i-direction.

5.3.5.3 Energy Equation in Three Dimensions

The energy equation is derived from the first law of thermodynamics, which states that the rate of change of energy of a fluid particle is equal to the rate of heat addition to the fluid particle plus the rate of work done on the particle:

Rate of increase of energy of fluid particle	=	Net rate of heat added to fluid particle	+	Net rate of work done on fluid particle

As before, we will be deriving an equation for the rate of increase of energy of a fluid particle per unit volume, which is given by

$$\rho$$

Work done by surface forces The rate of work done on the fluid particle in the element by a surface force is equal to the product of the force and velocity component in the direction of the force. The work done by these forces is given by

$$\left[\left(pu - \frac{\partial(pu)}{\partial x}\frac{1}{2}\delta x\right) - \left(\tau_{xx}u - \frac{\partial(\tau_{xx}u)}{\partial x}\frac{1}{2}\delta x\right)\right.$$
$$\left. - \left(pu + \frac{\partial(pu)}{\partial x}\frac{1}{2}\delta x\right) + \left(\tau_{xx}u + \frac{\partial(\tau_{xx}u)}{\partial x}\frac{1}{2}\delta x\right)\right]\delta y\delta z$$
$$+ \left[-\left(\tau_{yx}u - \frac{\partial(\tau_{yx}u)}{\partial y}\frac{1}{2}\delta y\right) + \left(\tau_{yx}u + \frac{\partial(\tau_{yx}u)}{\partial y}\frac{1}{2}\delta y\right)\right]\delta x\delta z$$
$$+ \left[-\left(\tau_{zx}u - \frac{\partial(\tau_{zx}u)}{\partial z}\frac{1}{2}\delta z\right) + \left(\tau_{zx}u + \frac{\partial(\tau_{zx}u)}{\partial z}\frac{1}{2}\delta z\right)\right]\delta x\delta y$$

The net rate of work done by these surface forces acting in the x-direction is given by

$$\left[\frac{\partial(u(-p + \tau_{xx}))}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z}\right]\delta x\delta y\delta z$$

Surface stress components in the y- and z-direction also do work on the fluid particle. A repetition of the above process gives the additional rates of work done on the fluid particle due to the work done by these surface forces:

$$\left[\frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v(-p + \tau_{yy}))}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z}\right]\delta x\delta y\delta z$$

$$\left[\frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w(-p + \tau_{zz}))}{\partial z}\right]\delta x\delta y\delta z$$

The total rate of work done per unit volume on the fluid particle by all the surface forces is given by the sum of divided by the volume δ x δ y δ z. The terms containing pressure can be collected together and written more compactly in vector form.

$$-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}=-\mathrm{div}(p\mathbf{u})$$

This yields the following total rate of work done on the fluid particle by surface stresses:

$$[-\mathrm{div}(p\mathbf{u})]+\left[\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}\right.$$
$$\left.+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}\right]$$

5.3.6 What is Turbulence?

First we take a brief look at the main characteristics of turbulent flows. The Reynolds number of a flow gives a measure of the relative importance of inertia forces (associated with convective effects) and viscous forces. In experiments on fluid systems it is observed that at values below the so called critical Reynolds number *Recrit* the flow is smooth and adjacent layers of fluid slide past each other in an orderly fashion. If the applied boundary conditions do not change with time the flow is steady. This regime is called laminar flow. At values of the Reynolds number above *Recrit* a complicated series of events takes place which eventually leads to a radical change of the flow character. In the final state the flow behavior is random and chaotic. The motion becomes intrinsically unsteady even with constant imposed boundary conditions. The velocity and all other flow properties vary in a random and chaotic way. This regime is called turbulent flow.

5.3.7 The *K*---E Model

In two-dimensional thin shear layers the changes in the flow direction are always so slow that the turbulence can adjust itself to local conditions. In flows where convection and diffusion cause significant differences between production and destruction of turbulence, e.g. in recirculating flows, a compact algebraic prescription for the mixing length is no longer feasible. The way forward is to consider statements regarding the dynamics of turbulence. The k–ε model focuses on the mechanisms that affect the turbulent kinetic energy. Some preliminary definitions are required first. The instantaneous kinetic energy $k(t)$ of a turbulent flow is the sum of the mean kinetic energy $K = 1/2\ (U2 + V2 + W2)$ and the turbulent kinetic energy.

5.4 Boundary Conditions

5.4.1 What is Boundary Condition?

- Something that indicates bounds or limits; a limiting or bounding line. Also called frontier. Mathematics. The collection of all points of a given set having the property that every neighborhood of each point contains points in the set and in the complement of the set.

- Some data for velocity, pressure and temperature from all over the country.

	Rajasthan (Jaisalmer)	**Tamilnadu (Kanyakumari)**	**Maharashtra (Dhule)**
Velocity	32 Kmph	32 Kmph	34 Kmph
Pressure	1.01325 bar	1.01325 bar	1.01325 bar
Temperature	8 ˚C - 40 ˚C	22 ˚C - 35 ˚C	22 ˚C

Table 2 - Boundary Conditions

- Velocity inlet condition, inlet velocity 100m/sec

6. CFD Results and Discussion

The full analysis of the report involves the effect varying angle of attack of wind turbine blades at wind speed of 100m/s. the angle of attack is based on the wind tunnel environment as the angle between the chord line and the wind velocity vector, which has the same direction as the uniform flow along the wind surrounding walls. In this study we have done the analysis on 5 different angles i.e 0°, 2°, 4°, 6°, 8°.

The velocity, pressure distribution, lifts force and drag force at different angle of attack is studied in this case. As expected from literature review , the increasing the angle of attack of the airfoil created the larger region of separation causing what is known as stall effect.

Also the stall angle plays a crucial role in design of vertical axis wind turbine. Having the VAWT system with very large separation region would prove to be very inefficient because of the large amounts of drag that would exist. From the above different degrees we have to find critical angle of attack at which there is minimum flow separation and maximum lift force which will help to generate maximum torque on rotor.

6.1 Pressure Distribution

The different pressure distributions are as follows:

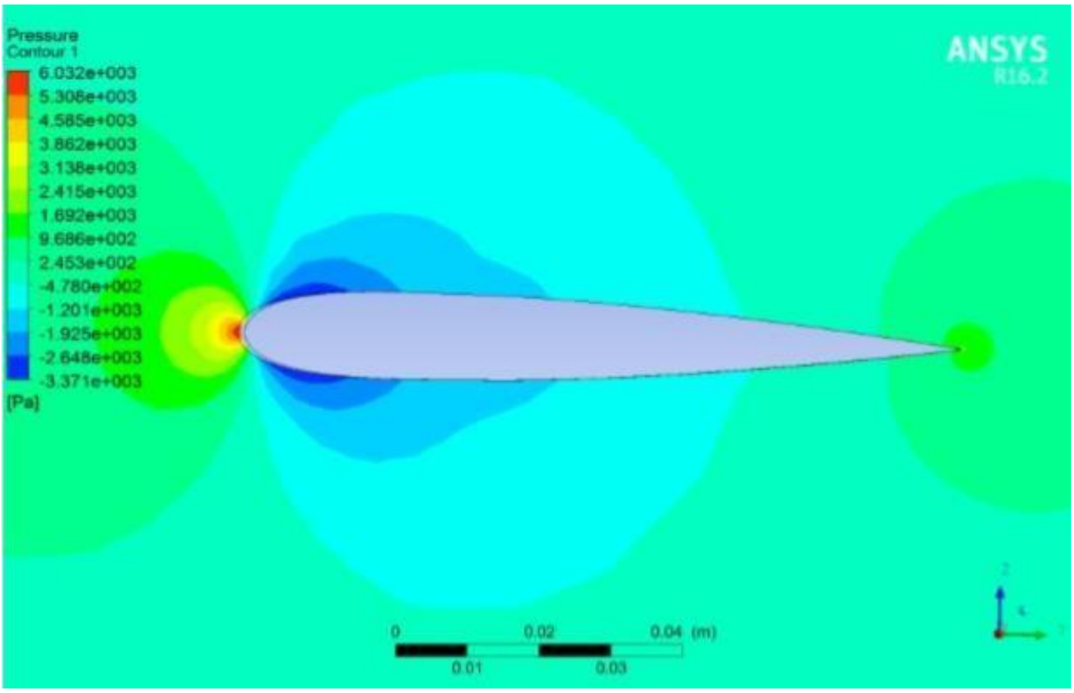

Fig. 17 - Pressure Distribution at 0° Angle of Attack

Fig 17 shows the result of pressure distribution of blade at 0° angle of attack and it shows that almost equal pressure distribution on both the top and bottom of the blade. This is due to the symmetry of airfoil which is NACAs 0012.In this we came to know that lift force is not that much, also there is flow separation. But due to effect of circulation and vortices on the trailing edge of an airfoil, high pressure region is formed near the trailing edge on the upper surface of an airfoil.

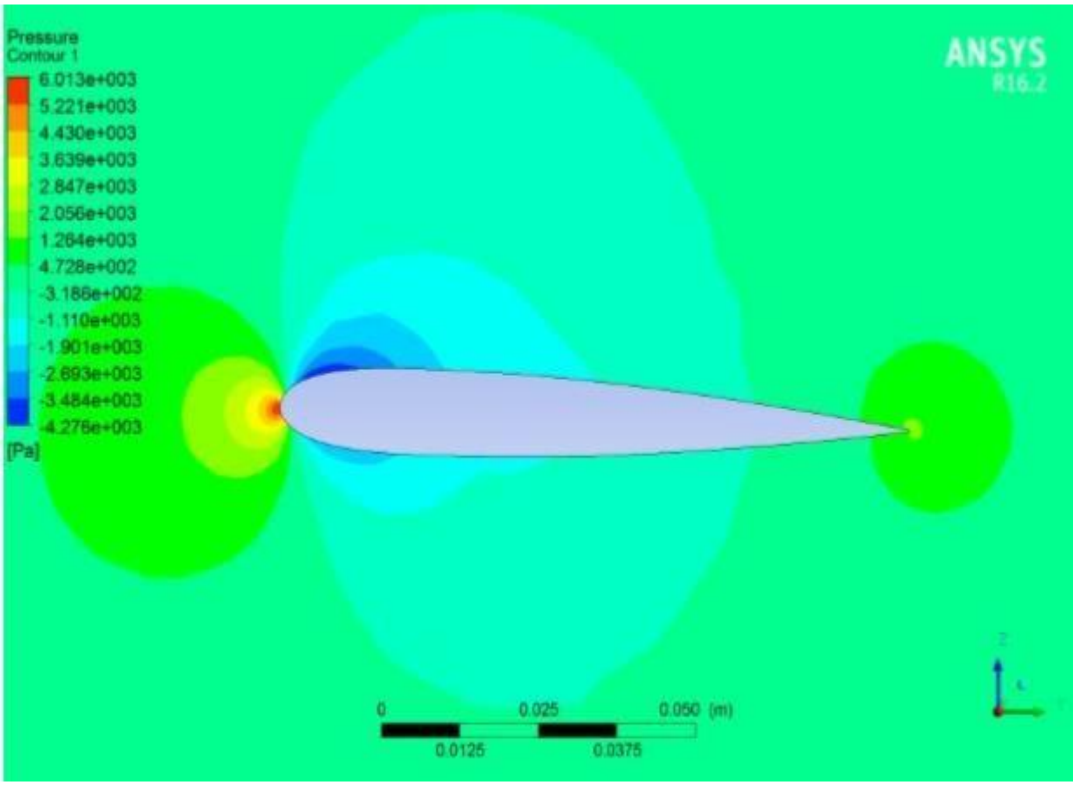

Fig. 18 - Pressure Distribution at 2° Angle of Attack

In fig 18 flow separation at upper surface of airfoil decreased than lower surface. Also at leading edge and trailing edge the pressure is reduced as compared to 0° angle of attack. But according to pressure reduction there is increase in lift and drag force is seen in table.

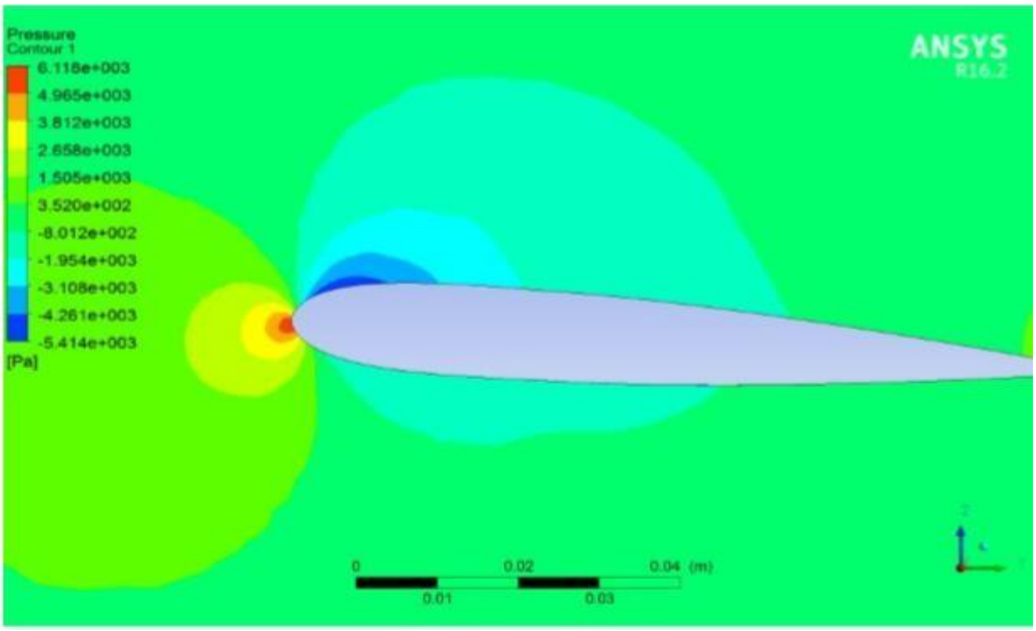

Fig. 19 - Pressure Distribution at 4° Angle of Attack

In fig 19 it shows again that decrease in pressure at upper side of blade is seen which increases the lift and drag force. Also at leading edge it is observed that the circle of pressure has increased, because the pressure is increased at leading edge.

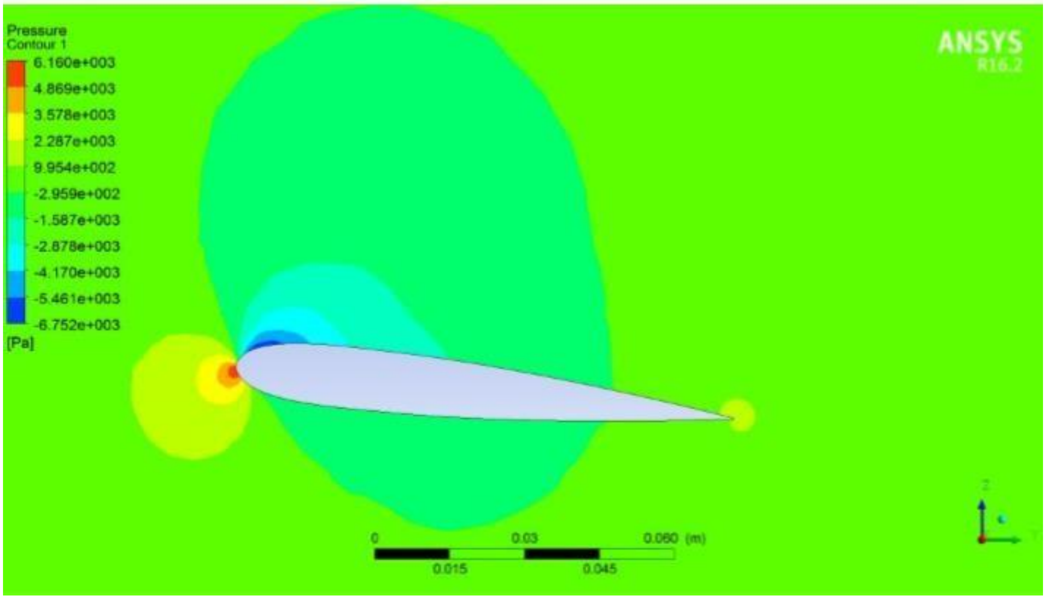

Fig. 20 - Pressure Distribution at 6° Angle of Attack

In fig 20 some different are figured out , because in this pressure is decreased at upper surface as well as pressure at leading and trailing edge there is increase in pressure which means there is no negative pressure at edges so flow separation is minimum. Also increase in lift and drag force is seen at 6 ° angle of attack.

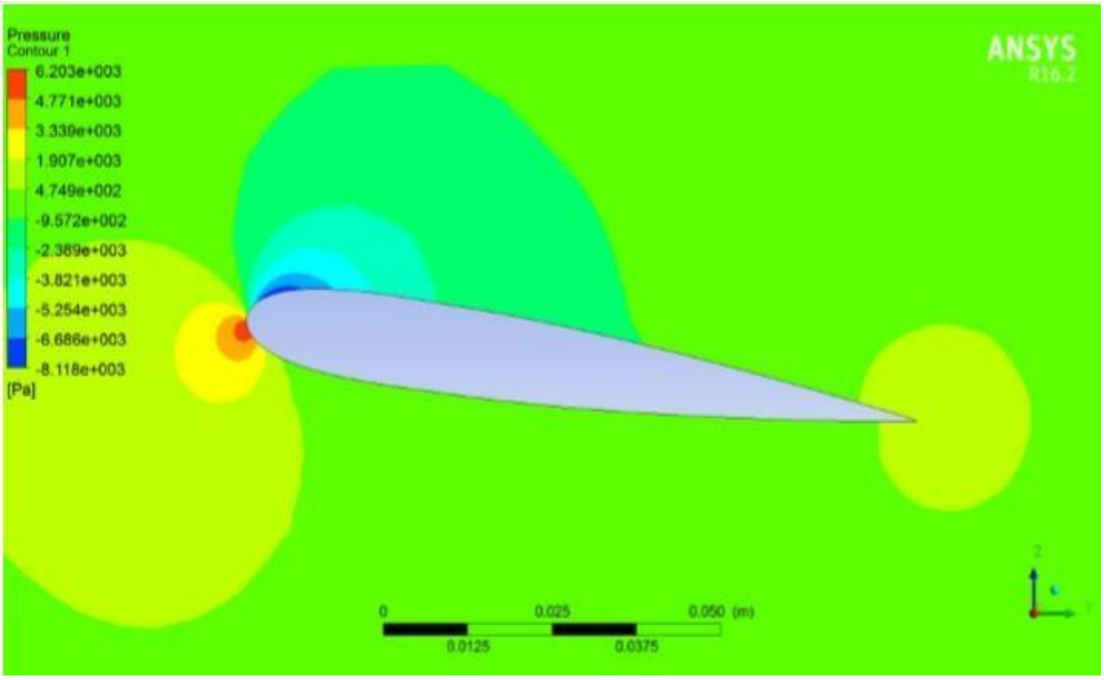

Fig. 21 - Pressure Distribution at 8° Angle of Attack

In this fig 21 it is seen that pressure is increased at leading and trailing edge so flow separation will decrease. Also the pressure at upper surface has increased very little but there is negative pressure only.

Degree	0	2	4	6	8
Lift (N)	-0.28560303	3.50744510	7.17387140	11.30568900	15.58068500
Drag (N)	0.36634305	0.43000640	0.62972917	1.00064690	1.54936050

Table 3 - Values of Lift and Drag Force

From the above fig 21 we know that lift and drag force increases with increase in angle of attack blade. Greater the lift force greater will be the torque produced on the rotor shaft. As the angle of attack increases the air is deflected through a larger angle and the vertical component of the airstream velocity increases, resulting in more lift. As the angle of attack increases, the lift reaches a maximum at some angle; increasing the angle of attack beyond this critical angle of attack causes the upper surface flow to separate from the wing; there is less deflection downward so the airfoil generates less lift.

It is possible to apply potential flow theory with no circulation to an airfoil, leading to the flow pattern. It is apparent from the figure that the flow pattern has some peculiar features.

There exists a stagnation point on the upper surface of the foil just forward of the trailing edge and the flow travels from the lower side to the upper side around the trailing edge.

Consider the pressure distribution associated with this flow. Consider the pressure distribution associated with this flow. Recall that for both potential and viscous flow, and deducing from the centrifugal forces acting on a particle moving in a curved path, the pressure gradient normal to a streamline of radius *r* is given by

$$\frac{\partial p}{\partial n} = \frac{\rho v^2}{r}$$

This indicates that large pressure gradients are associated with small radii of curvature (recall flow around a corner). The Bernoulli equation shows that such rapid changes of pressure are accompanied by corresponding rapid changes in velocity and that the velocity increases with diminishing radius (and pressure) and theoretically reaches an infinite value at a corner(radius = 0). Therefore, we can conclude that the flow pattern indicates infinite velocities at the trailing edge.

The velocity on the top surface is lower than that on the lower surface. A pressure distribution is developed around the foil. The pressure changes and the velocity changes are a result of a non-zero circulation around the foil. Notice that even in a viscous flow, a symmetrical foil at a zero angle of attack will not produce lift. Circulation (and therefore lift) is generated when an asymmetry is introduced, either by introducing a camber or an angle of attack.

The result at the trailing edge is two streams traveling at different velocities. This causes a starting vortex at the trailing edge. One particularly interesting feature is that a reduced pressure exists near the trailing edge that acts to deflect the flow from the underside to the upper surface of the foil. The vortex is eventually shed and extends to infinity, leaving a positive pressure at the lower surface and a negative pressure at the upper surface.

Once the flow pattern has been established, the effect of these friction forces are confined to the boundary layers on the surface of the foil and to the thin vortex sheet in the wake of the foil and at the trailing edge. In the remainder of the flow, velocity gradients are small and the flow is smoothly curved, therefore it may be expected that potential flow theory would be a satisfactory tool for analyzing the airfoil performance.

There is one situation in which an infinite ideal fluid moving at a uniform velocity can exert a force perpendicular to its general direction of motion on a body immersed in it. This arises when circulation exists about the foil.

By the principle of conservation of angular momentum, the formation of the vortex must have resulted in the development of a rotary motion of equal angular momentum, but in the opposite direction.

Since the foil is symmetric, pressure readings are only taken on a single side of the foil. The pressure profile can be observed on both sides of the foil by making use of positive and negative angles of incidence.

The initial test shows that a negative pressure, a suction pressure, exists on the upper surface, and a smaller positive pressure exists on the lower surface (See figure 15). The suction pressure in fact contributes about three quarters of the lift force.

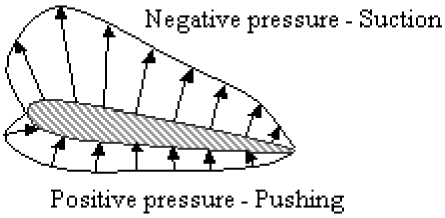

Fig. 22 - Pressure Distribution of an Airfoil

According to Bernoulli, the corresponding static pressure on the upper surface is diminished and the pressure on the lower surface is increased. At the same time, the proportion of the fluid stream flowing above the airfoil increases, the flow below the airfoil decreases and the position of the forward stagnation point is displaced downwards. The velocity at the trailing edge is no longer infinite. The condition of finite velocity at the trailing edge is known as the Kutta condition.

6.2 Velocity Distribution

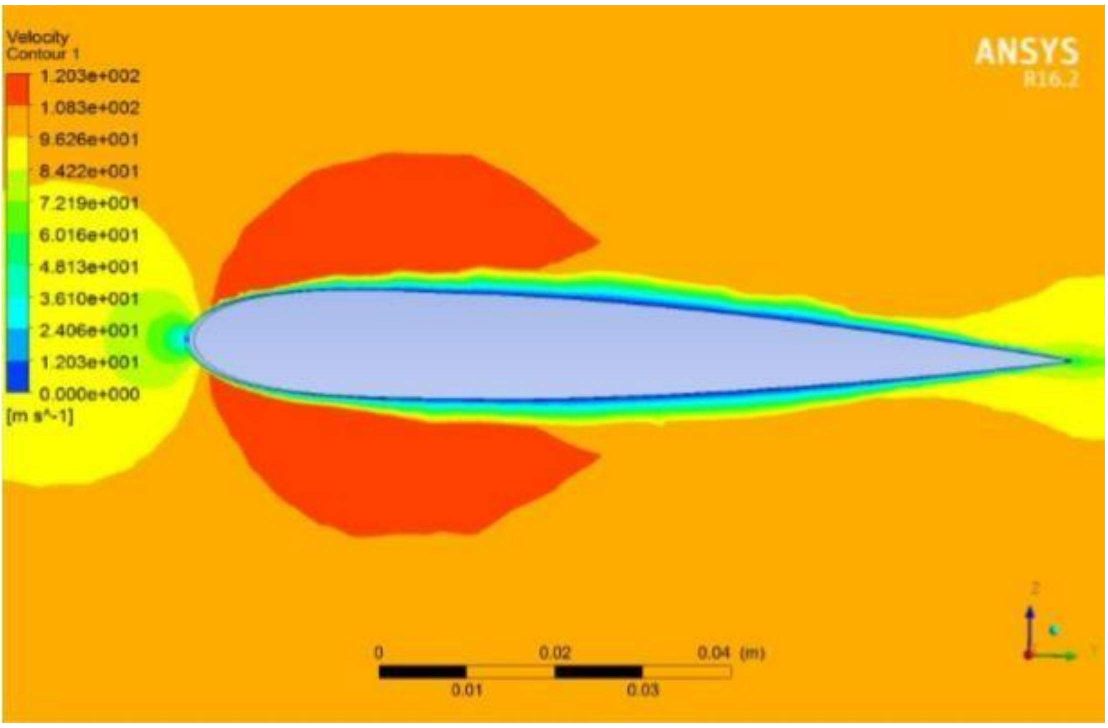

Fig. 23 - Velocity Distribution at 0° Angle of Attack

In fig 23 we have seen that perfect symmetry has be observed on velocity profile due to airfoil shape, and it gives that the maximum velocity of air is at upper and lower surface of blade. Also due to obstruction of blade the minimum velocity is at the leading edge of blade. The flow of air has proper streamline along the shape, which helps to swivel the blade because the equal force is generated at both upper and lower surface.

Due to friction the velocity of air gets slower. Usually zero angle of attack is not preferred because of low lift force as seen in the table. As velocity and pressure distribution along the both surfaces would have been cancelling each other to a resulting total lift force to zero, but here in the calculation we got negative value of force as this is limitation of software used or any other technical fault during design. The calculation of the forces can be performed by summing (math integrating) the pressure distribution of a plot of Cp versus chord length.

In general an airfoil does not only create a lifting force, it also creates a moment, which tries to rotate the airfoil to a different angle of attack. For comparisons it is generally assumed that the lift force and his torque moment are acting at a fixed point, which is located 25% behind the leading edge on the x axis (called as chord point).the moment acting at ¼ of chord point is nearly independent of angle of attack, as long as the flow stays attached to the airfoil.

If no separation or compressibility effects are present which are mentioned in pressure distribution point, the pressure field around a two dimensional airfoil is creates lifting force only, no drag. The drag of a two dimensional airfoil is created by the friction o air particles moving close to surface, the airfoil is surrounded by a boundary layer, which forms a thin sheet adjacent to the wall where the velocity is reduced form the free stream value down to zero on the wall. For the theoretical treatment of lift, the boundary layer effect is so small that it can be neglected as long as no separation occurs; this applies also to the moment of the airfoil. But the boundary layer is the main source for the drag in two dimensional flow.

The lift ,drag and moment of awing depends not only on the airfoil shape and its associated velocity distribution, but also on wing platform and on the wing area. The forces and moments also depend on the density of the air and on shape of wing.

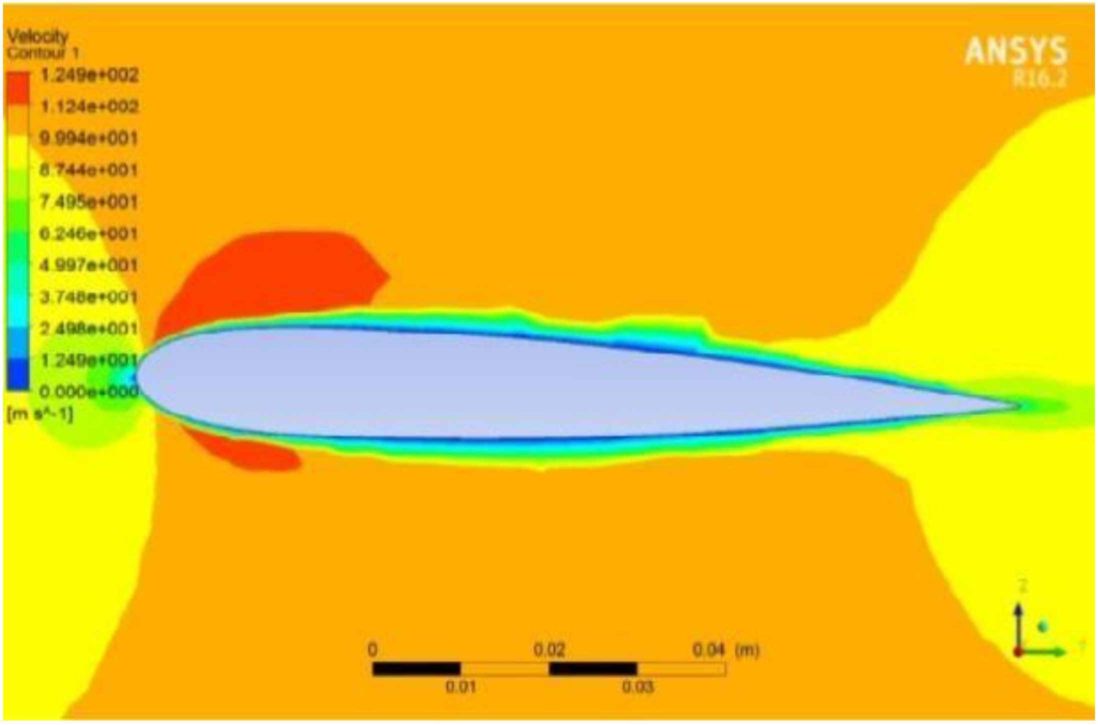

Fig. 24 - Velocity Distribution at 2° Angle of Attack

In this fig 24 we have increased to 2° angle of attack in which the velocity is increased at the upper surface and at lower surface there is less velocity of air, hence the lift force acting on blade increases. Here the final and total velocity has been increased as compared to 0° angle of attack. The air is formed as layers and the velocity of is high at top layer whereas at bottom there is low velocity. The stream flow across the blade is turbulent which generates force to lift.

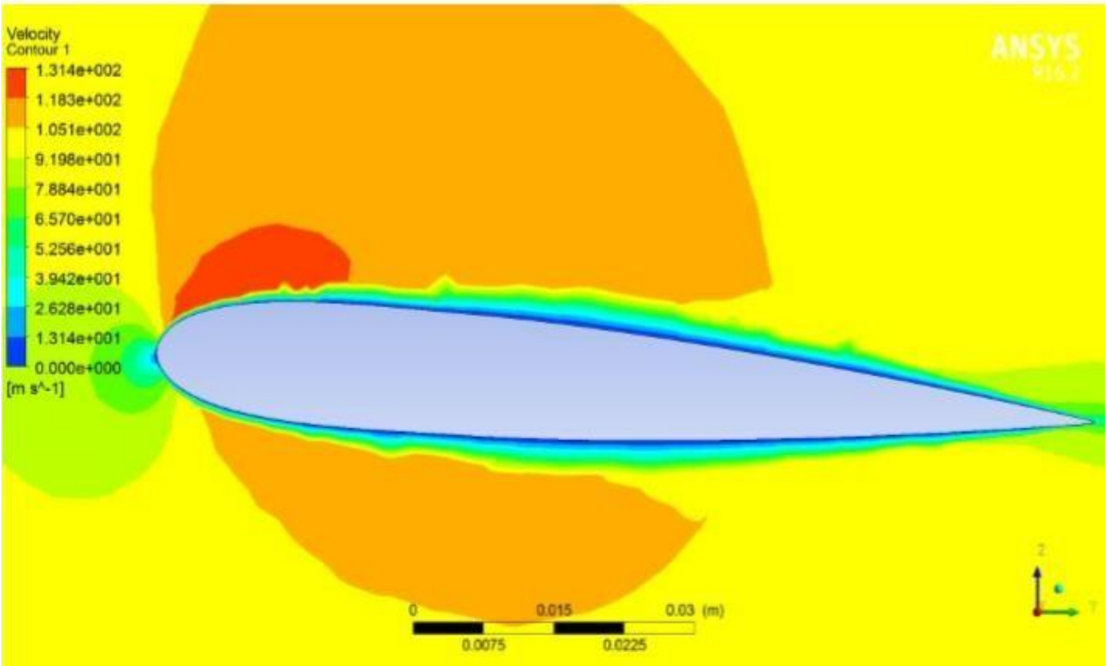

Fig. 25 - Velocity Distribution at 4° Angle of Attack

In this above fig 25 we have seen that angle is changed to 4° angle of attack and hence the velocity magnitude is also changed the velocity at the top of the blade surface is more than the velocity at the bottom of blade. Also there is increase in lift force, and the drag force is also increased. The velocity at leading edge is reduced or lower, because of collision with blade surface.

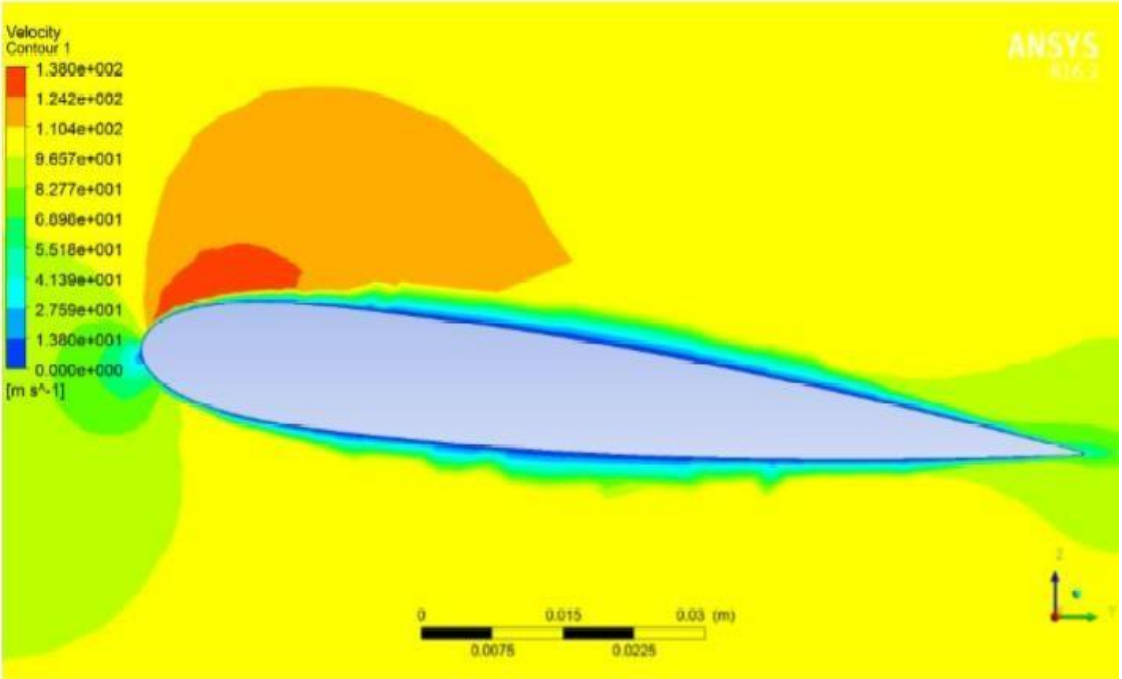

Fig. 26 - Velocity Distribution at 6° Angle of Attack

In this above fig 26 we got the velocity at 6° angle of attack in this we got velocity more which generates more lift force. The lift force at this angle is near to the greatest value from the total number of results are produced. Moreover we are taking this angle of attack because the lift force is maximum and in above pressure distribution of 6° there is low flow separation. The drag force is less as compared to 8° shown in table 3.

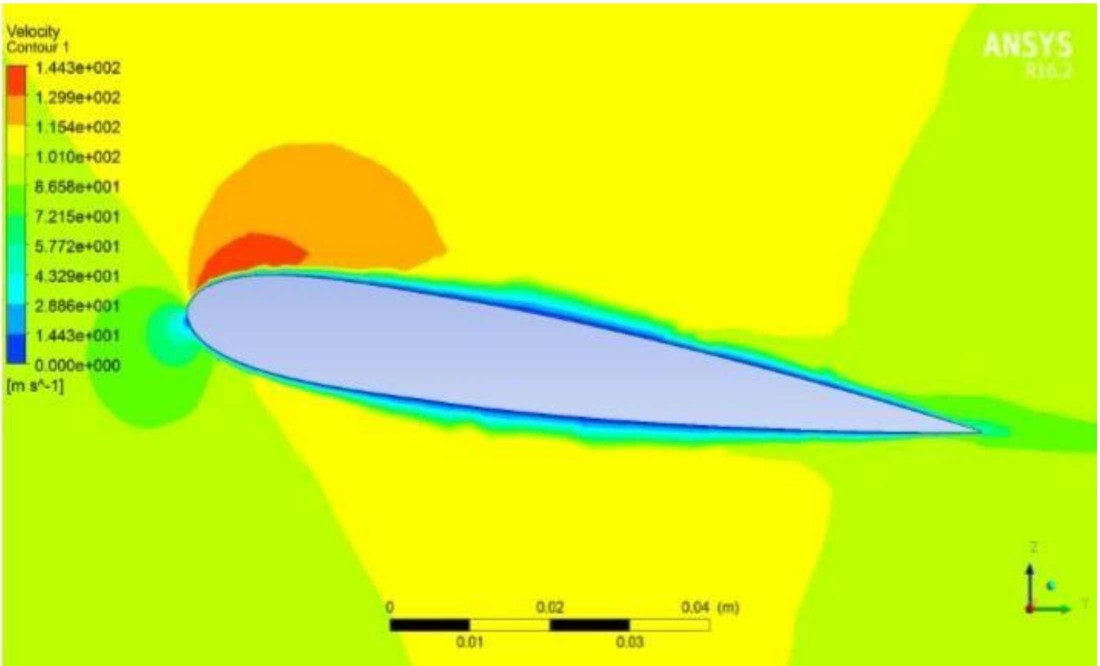

Fig. 27 - Velocity Distribution at 8° Angle of Attack

In this we got highest drag force as well as high lift force; also there is more flow separation in this angle. So we may not choose this angle of attack as the net force will not be more to generate torque around the rotor.

7. Conclusion

Vertical axis wind turbine (VAWT) has been working around for generations, despite of its several dominances this automation has not attracted more attention as compared to horizontal axis wind turbine (HAWT) but in this study we have acquired some real-world fulfillment.

If we place a wind turbine at an optimum angle to obtain better lift force in order to minimize the flow separation because it opposes the lift force also maximum lift is essential to increase the torque of rotor shaft, ultimately increasing the generation of electricity. We executed a CFD analysis of the blade of VAWT in which we carried out pressure distribution and velocity distribution at different angles that is 0°, 2°, 4°, 6°and 8° which can be seen in result.

And we conclude that 6° was the foremost angle to procure supreme lift force and minimum drag force that opposes the direction of the oncoming flow as well as the flow separation is also minimum.

8. Future Scope

We have examined VAWT at 100 Km/hr and thus we got the result, at 6° angle of attack. But this velocity of air is not possible at normal condition. Due to shortage of time we are not able to calculate the results at 30 km/hr. but we conclude that both results will be approximately same according to research paper. VAWT has a wider range of future scope, wind energy is available without any cost and it does not emit any greenhouse gases. VAWT will continue to generate a minuscule fraction of a % of electricity generated from wind. We know that we have limited amount of fossil fuel left and after same time it will be finished. Then VAWT will be used for generating electricity. VAWT can be design act smaller level for eg:

1) Charging mobiles
2) Boiling water

References

[1] Chris Kamisky, Austin Filush, Paul Kasprzak and WaelMokhtar. A CFD Study of Wind Turbine Aerodynamics. North Central Section Conference, American Society for Engineering Education, 2012
[2] Ragheb M. Vertical Axis Wind Turbines. University of Illinois at Urbana-Champaign. 1 Aug, 2011, Web: 7 Dec, 2011
[3] Shafiee Shahriar, Erkan Topal. When will fossil fuel reserves be diminished? Energy Policy 37. 1 Jan, 2009, 181, Science Direct. Web: 1 Feb, 2012
[4] Riegler Hannes. HAWT versus VAWT : Small VAWTs find a clear niche. July/August, 2003
[5] Sumedha Singh Rathore, Rushabh Dalmia, Karan Tamakuwala, Sreekanth Manavalla. Design Fabrication and Testing of a Low Cost Vertical Axis Wing Mill for Low End Power Generation. International Research Journal Of Engineering And Technology, Vol. No. 3 (5), pp 1524-1528, 2016
[6] Design And Construction Of Vertical Axis Wind Turbine, Volume 5, Issue 10, October 2014
[7] Robert E Akins, Dale E Berg, W Tait Cyrus. Measurement And Calculation Of Aerodynamic Torque For VAWT. October 1987
[8] Xiaomin Chen. Optimization Of Wind Turbine Airfoils/Blades And Wind Farm Layouts. 20 January 2014

Acknowledgement

We would like to express deep sense of gratitude to our Project Guide, **Amol T Kamble, Assistant Professor, Department of Automobile Engineering**, for being the cornerstone of our project. It was his incessant motivation and guidance during periods of doubts and uncertainties that has helped us to carry on with this project.

We would like to thank **Mr. Siddharaj Allurkar**, **Head of Department**, **Automobile Engineering** for providing necessary guidance, support, motivation and inspiration without which this project would not have been possible.

We would like to extend our sincere thanks to the **Management of Dhole Patil College of Engineering** for providing all the necessary infrastructure and laboratory facilities.

We also like to acknowledge the help extended by the **faculty members and non-teaching staff** of Automobile Engineering Department for successful completion of our project. Last but not the least, special thanks to our family members, friends and colleagues for their continuous support.