Study and Analysis of the Flow Field around the Horizontal Wind Turbine Blade

Amit Kumar, Shivasheesh Kaushik, Kundan Kumar, Vikas Chauhan

Abstract— The purpose of this project is firstly to give an overview of aerodynamic performance of HAWT and description of recent developments and current status of the drag-reduction research. The wind energy is deemed as one of the most durable energetic variants of the future because the wind resources are immense. Therefore research in this field must be continuous and interdisciplinary. The objective of these investigations is to improve the wind turbines aerodynamic performance and increase their operational range by inducing complete or partial flow reattachment. Through CFD simulations for a horizontal axis wind turbine, this study is trying to analyze the flow field around the wind turbine blade. Current efforts focus on increasing their aerodynamic efficiency and operational range through passive flow control techniques. A portion of my research is focused on investigating passive flow control strategies in order to provide wind turbine manufacturers with effective approaches to enable them to increase the amount of energy that can be captured by a wind turbine. In this project we focus on exploring the way of increasing the aerodynamic performance of HAWT blade aerofoil. Current research is focused on S819 aerofoil which is used for large wind turbine blade and a new technique is used to control the flow separation.

Index Terms— Ansys Fluent, Aerofoil, Wind Turbine Blade, Coefficient of drag, Coefficient of lift, Flow separation control.

1 Introduction

The wind Turbine blades are long and slender structures where the span wise velocity component is much lower than the stream wise component, and it is therefore assumed in many aerodynamic models that the flow at a given radial position is two dimensional and that 2-D aerofoil data can thus be applied. In order to realize a 2-D flow it is necessary to extrude an aerofoil into a wing of infinite span. On a real wing the chord and twist changes along the span and the wing starts at a hub and ends in a tip, but for long slender wings, like those on modern gliders and wind turbines, Prandtl has shown that local 2-D data for the forces can be used if the angle of attack is corrected accordingly with the trailing vortices behind the wing. The reacting force F from the flow is decomposed into a direction perpendicular to the velocity at infinity Vα and to a direction parallel to Vα.

• *Amit Kumar , Research Scholar, Department of Mechanical Engineering in Uttarakhand Technical University, Dehradun, Uttrakhand, India, PH- 8273527686, E-mail:amitkumar.usn@gmail.com*
• *Shivasheesh Kaushik, Assistant Professor, Department of Mechanical Engineering in Amrapali Group of Iinstitute, Haldwani, Uttrakhand, India, PH- 9756298215, E-mail: shivasheesh.rachit.kaushik@gmail.com*
• *Vikas Chauhan, Assistant Professor, Department of Mechanical Engineering in Uttarakhand Technical University, Dehradun, Uttrakhand, India, PH- 9045230799, E-mail: chauhanvikas23@gmail.com*
• *Kundan Kumar , Research Scholar, Department of Mechanical Engineering in Uttarakhand Technical University, Dehradun, Uttrakhand, India, PH- 9801174886, E-mail: kundankmr79@gmail.com*

The former component is known as the lift, L and the latter is called the drag, D. Numerous researches have been done already on the wind turbine aerofoil analysis. Different techniques are employed to reduce the drag force and to increase the lift force so the aerodynamic performance of wind turbine blade can be increased. Some of the work done is listed below. Kianoosh Yousefi et al.[12] carried out numerical study of blowing and suction slot geometry optimization on NACA 0012 aerofoil. The effects of suction control flow on the aerodynamic characteristics of NACA 0012 aerofoil were subsequently investigated. The result show that lift to drag ratio increases dramatically but also the stall angle improves effectively from 14o to 22o. From a jet width perspective the suction jet width improvement leads to a significant augmentation in the lift to drag ratio, and separation effectively travels toward downstream. By employing the suction control flow technique the lift coefficient increased by approximately 75% and the drag coefficient decreased by 56%. Ya-lei Bai .[11] investigate the lift enhancement of aerofoil and tip flow control for wind turbine. In this paper two techniques that improve the aerodynamic performance of wind turbine airfoils are described. The flow deflector which is fixed at the leading edge is employed to control the boundary layer separation on the airfoil at a high angle of attack. The results shows that the flow deflector can suppress or postponed the flow separation, delay the stall, and enhance the lift. The results of the numerical simulation for the S809 airfoil show that the flow deflector can control the separation effectively and improve the aerodynamic characteristics of the airfoil. It is indicated that the lift coefficient of the airfoil with the flow deflector is increased by 24%, and the lift-drag ratio is increased by more than 50%. The stall of the airfoil is delayed by 2°.

Leonardo P. Chamorro.[10] investigate the skin friction drag reduction in large wind turbines using sharp V-grooved riblets. In this paper they focus on exploring the performance of riblets as a drag reduction strategy for wind turbine blades via a series of laboratory experiments. The experiments are being carried out in the wind tunnel and to understand the performance of riblets over a range of operating conditions and to understand the turbulent scale-to-scale interaction near the wall region. The result shows good agreement in lift to drag ratio. M. Tahar Bouzaher. [9] Carried out numerical study of flow separation control over a NACA 2415 aerofoil. This study involves numerical simulation of the flow around a NACA2415 airfoil, with an 18° angle of attack, and flow separation control using a rod; it involves putting a cylindrical rod -upstream of the leading edge- in vertical translation movement in order to accelerate the transition of the boundary layer by interaction between the rod wake and the boundary layer. The results showed a substantial modification in the flow behavior and a maximum drag reduction of 61%. U. Anand .[8] performed analysis on passive flow control over NACA 0012 aerofoil using vortex generators. Numerical simulations of turbulent flow over a NACA0012 aerofoil attached with vortex generators are carried out over a wide range of angles of attack at Re=5.5×105. Flow separation control over a NACA0012 aerofoil using vortex generators is studied by performing numerical simulations. The Stream wise vortices produced from the VGs are of sufficient strength to delay the stalling for the angle of attack range considered in this study. Jibing Lan. [16] investigate the effect of leading edge boundary layer thickness on dimple flow structure and separation control. This study concerns the effect of leading edge boundary layer thickness on dimple flow structure and separation control. A flat plate with adverse pressure gradient sufficient for separation was designed. Large eddy simulation (LES) with dynamic Smagorinsky sub grid model was validated. Dimples with R=0.378, 0.994, 1.453 (R is the ratio of leading edge boundary layer thickness to dimple depth) were investigated. The results show that the horseshoe vortex dominates the dimple flow structure. As R increases, the head of the horseshoe vortex rises further away from the wall and moves downstream. Hairpin vortices in the dimple wake are productions of the horse shoe vortex. Both the horseshoe vortex and the hairpin vortices energize the boundary layer by mixing with the free-stream fluid. As R increases, the separation control effectiveness decreases.

John C. Lin.[15] reviewed research on low-profile vortex generators to control boundary-layer separation. An in-depth review of boundary-layer flow-separation control by a passive method using low-profile vortex generators is presented. The generators are defined as those with a device height between 10% and 50% of the boundary-layer thickness. Key results are presented for several research efforts, all of which were performed within the past decade and a half where the majority of these works emphasize experimentation with some recent efforts on numerical simulations. Topics of discussion consist of both basic fluid dynamics and applied aerodynamics research. The fluid dynamics research includes comparative studies on separation control effectiveness as well as device-induced vortex characterization and correlation. The comparative studies cover the controlling of low-speed separated flows in adverse pressure gradient and supersonic shock-induced separation. The aerodynamics research includes several applications for aircraft performance enhancement and covers a wide range of speeds. Significant performance improvements are achieved through increased lift and/or reduced drag for various airfoils-low-Reynolds number, high-lift, and transonic-as well as highly swept wings. Performance enhancements for non-airfoil applications include aircraft interior noise reduction, inlet flow distortion alleviation inside compact ducts, and a more efficient over wing fairing. The low-profile vortex generators are best for being applied to applications where flow-separation locations are relatively fixed and the generators can be placed reasonably close upstream of the separation. Using the approach of minimal near-wall protuberances through substantially reduced device height, these devices can produce stream wise vortices just strong enough to overcome the separation without unnecessarily persisting within the boundary layer once the flow-control objective is achieved. Practical advantages of low-profile vortex generators, such as their inherent simplicity and low device drag, are demonstrated to be critically important for many applications as well.

David Greenblatt. [14] Investigate control of flow separation by periodic excitation. This study presents a review of the control of flow separation from solid surfaces by periodic excitation. The emphasis is placed on experimentation relating to hydrodynamic excitation, although acoustic methods as well as traditional Boundary layer control, such as steady blowing and suction, are discussed in order to provide an appropriate historical context for recent developments. The review examines some aspects of the excited plane mixing-layer and shows how its development lays the foundation for a basic understanding of the problem. Flow attachment to, and separation from, a defected flap is then shown to be a paradigm for isolating controlling parameters as well as understanding the basic mechanisms involved. Particular attention is paid to separation control on airfoils by considering controlling parameters such as optimum reduced frequencies and excitation levels, performance enhancement, efficiency, reduction of post-stall unsteadiness, compressibility and other important features. Additional topics covered include excitation of separation bubbles, control and exploitation of diffuser flows, three-dimensional effects, the influence of longitudinal curvature and possible applications to unmanned air vehicles. The review closes with some recent developments in the control and understanding of incompressible dynamic stall, specially illustrating the control of dynamic stall on oscillating airfoils and identifying the crucial time-scale disparity between dynamic stall and periodic excitation. Kwing- So Choi.[13] presented the recent developments and current status on drag reduction. Discussions of some of the recent results on riblets, compliant coating, and drag reduction in nature, polymer additives and wall oscillation were given with an emphasis on the drag-reduction mechanisms involved. Effectiveness of riblets in turbulent boundary layers under pressure gradient was investigated at the Delft University of Technology. Through a direct measurement of drag and velocity, De Bisschop and Nieuwstadt (1996) were able to show that a greater drag reduction of up to 13% can be obtained by using riblets in a boundary layer under an adverse pressure gradient. A research group at the University of Warwick has been engaged in the study of compliant coating in delaying transition of boundary layers to turbulence (Lucey and Carpenter, 1992, Carpenter, 1993, Dixon et al., 1994, Davies and Carpenter, 1997). They demonstrated that in theory, at least, substantial transition delays are possible with Kramer's coatings. These numerical studies were supported by a series of towing tank experiments by Gaster (1987).

Recent effort has been directed towards the optimization of compliant coating for maximum reduction in skin-friction drag, such as by using multiple compliant panels and anisotropic coatings. Effects of compliant rotating disc on the boundary-layer transition were also investigated by the same group (Cooper and Carpenter, 1997). Experimental studies on live penguins were carried out by Bannasch (1997 and 1998) at the Technical university of Berlin with measurements using life-sized models in a water tank. An Axi- symmetric body based on three medium-sized penguin species was found to be an excellent low-drag laminar shape. A series of experiments were carried out by Choi et al. (1989) and Choi (1991) using a towing tank at British Maritime Technology, where a combined use of riblets with polymer coating was investigated for turbulent drag reduction. The results indicated that the rib-

IJSER

lets/polymer combination offered an overall improvement in drag-reduction characteristics over either riblets or polymer coating alone. Laadhari et al. (1994) at Ecole Centrale de Lyon carried out an experimental study to look at the problem of span wise-wall oscillation in a turbulent boundary layer. This is to experimentally confirm the results of recent direct numerical simulation suggesting that the turbulent skin-friction drag can be reduced by a wall oscillation. Joshua Yen .[20] explained enhancing vertical axis wind turbine by dynamic stall control using synthetic jets.

Chris Kaminsky.[17] performed CFD study of wind turbine aerodynamics. The objective of the analysis was to be optimized the attack angle in order to obtain the greatest possible lift force. The drafting software that was used to model the VAWT and the far field geometries that surround it and its components was Solid Works. The analysis of the VAWT was done using STAR CCM+, developed by CD Adapco as standard commercial software. The analysis used a computational finite volume method to analyze the 2D, 3D airfoil only and 3D full assembly cases with using a segregated flow solver. Modelling of the turbulence in each of the cases was done using the 2 equation SST k-ω model. This selection for the turbulence model was made because the k-ω model offers great analysis in both fully developed flow and along the boundary layer regions. As expected, increasing the attack angle of the airfoil created larger regions of separation causing what is known as the stall effect. Increasing the wind speed caused the separation regions to be more exaggerated with a greater amount of turbulence present. Knowing the stall angle is important to the design of vertical axis wind turbines because of the lift-stall effect that was mentioned earlier in the literature review section. Having a VAWT system with very large separation regions would prove to be very inefficient because of the large amounts of drag that would exist. The results show that at higher speeds the critical angle of attack happens faster than at lower speeds. Looking at the results from the 2D airfoil analysis of the lift force versus attack angle (Figure 16), the results show that the stall condition occurred at faster wind speed at approximately 8 degrees. W. A. Timer. [18] Analyzed the aerodynamic characteristics of wind turbine blade airfoils at high angles of attack. Airfoil characteristics at deep stall angles were investigated. It appeared that the maximum drag coefficient as a function of the airfoil upwind y/c ordinate at x/c=0.0125 can be approximated by a straight line. The lift-drag ratios in deep stall of a number of airfoils with moderate lower surface thickness coincide. It was found that the lift-drag ratio of airfoils with leading edge separation is independent of aspect ratio. The lift-drag ratios of the various sections of a non-rotating and a rotating blade in deep stall coincide with the two-dimensional curve, heat transfer in Heat Exchanger usually involves convection on each side of fluids and conduction through the wall separating the two fluids. In this project a circular double pipe heat exchanger is modeled and CFD analysis is performed on it. The main application of cylindrical pipe is in heat exchanger. Heat Exchanger is a device that is used to transfer thermal energy between two or more fluid, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperature and in thermal contact. In heat exchangers, there are usually no external heat and work interactions. Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single or multi component fluid stream. In other applications, the objective may be to recover or reject heat, or sterilized, pasteurized, fractionate, distill, concentrate, crystallize, or control a process fluid. In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall. Similarly in various thermal and nuclear power-plants cylindrical pipes are used for heat exchange.

As well as with circular double pipe, the rectangular insert is also modeled and the results are compared between circular double pipe and circular double pipe with insert. Use of insert will generate a swirl flow which will help in enhancement of heat transfer characteristic of circular double pipe heat exchanger. A heat transfer analysis on circular double tube heat

IJSER

exchanger is performed and the results are compared with circular double pipe heat exchanger with insert that is also modeled. The problem is defined and solved in ANSYS FLUENT software. The heat transfer results are post processed in CFD-Post.

2 Objective

The purpose of current work is firstly to give an overview of aerodynamic performance of HAWT and description of recent developments and current status of the drag-reduction research. The wind energy is deemed as one of the most durable motivational variants of the future because the wind resources are gigantic. Therefore research in this field must be continuous and interdisciplinary. The objective of this investigation is to improve the wind turbines aerodynamic performance and increase their operational range by inducing complete or partial flow reattachment. Through CFD simulations for a horizontal axis wind turbine, this study is trying to analyze the flow field around the wind turbine blade. Current efforts focus on increasing their aerodynamic efficiency and operational range through passive flow control techniques. S819 airfoil which is used for large horizontal axis wind turbine blade is used for current analysis.

The present work is undertaken to study the following aspects of:

1. ANSYS FLUENT is used as CFD code to understand the fluid behaviour over the S819 airfoil.
2. Comparative study of S819 airfoil with slot and without slot is done to see the effect on drag and lift coefficient of airfoil for enhancement in aerodynamic performance.
3. Analysis is done at different angle of attack.

3 Governing Equation

The behavior of the flow is generally governed by the fundamental principles of the classical mechanics expressing the conservation of mass and momentum. The continuity and the momentum equations are as follows.

A. Continuity Equation

Continuity Equation also called conservation of mass. Consider fluid moves from point 1 to point 2. The overall mass balance is Input – output = accumulation. Assuming that there is no storage the Mass input = mass output. However, as long as the flow is steady (time-invariant), within this tube, since, mass cannot be created or destroyed then the above equation. According to continuity equation, the amount of fluid entering in certain volume leaves that volume or remains there and according to momentum equation tells about the balance of the momentum. The momentum equations are sometimes also referred as Navier-Stokes (NS) equation. They are most commonly used mathematical equations to describe flow. The simulation is done based on the NS equations and then K-Epsilon model. Continuity equation can be expressed as:

$$\frac{\delta(\rho\bar{u})}{\delta x} + \frac{1}{r}\frac{\delta(\rho r\bar{v})}{\delta r} = 0 \tag{2.1}$$

B. Momentum Equation

- Axial component (z-component)

$$\rho\bar{v}\left[\frac{\delta\bar{u}}{\delta r} + \bar{u}\frac{\delta\bar{u}}{\delta x}\right] = \frac{\delta\bar{p}}{\delta x} + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta\bar{u}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(\mu_{eff}\frac{\delta\bar{u}}{\delta r}\right) + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta\bar{u}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(\mu_{eff}\frac{\delta\bar{u}}{\delta r}\right) \tag{2.2}$$

- Radial component (r-component)

$$\rho\left[\bar{v}\frac{\delta\bar{v}}{\delta r} + \bar{u}\frac{\delta\bar{v}}{\delta x}\right] = -\frac{\delta p}{\delta z} + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta\bar{v}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(r\mu_{eff}\frac{\delta\bar{r}}{\delta r}\right) + \frac{\delta}{\delta x}\left(\mu_{eff}\frac{\delta\bar{u}}{\delta r}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(r\mu_{eff}\frac{\delta\bar{v}}{\delta r}\right) - 2\mu_{eff}\frac{\bar{v}}{r^2} + \rho\frac{\bar{w}^2}{r} \tag{2.3}$$

- Tangential Component (Θ- component)

$$\rho\left[\bar{v}\frac{\delta\phi}{\delta r} + \bar{u}\frac{\delta\phi}{\delta x}\right] = \frac{\delta}{\delta z}\left[\mu_{eff}\frac{\delta\phi}{\delta x}\right] + \frac{1}{r}\frac{\delta}{\delta r}\left[r\mu_{eff}\frac{\delta\phi}{\delta r}\right] - \frac{2}{r}\frac{\delta}{\delta r}\left[\mu_{eff}\phi\right] \tag{2.4}$$

Here $\bar{u}$, $\bar{v}$ and $\bar{w}$ are the mean velocity components along z, r and θ directions respectively and the variable $\phi = r\bar{w}$.

The total effective viscosity of the flow is given by,

$$\mu_{eff} = \mu_l + \mu_t \tag{2.5}$$

Here μ_l and μ_t stand for molecular or laminar viscosity and eddy or turbulent viscosity respectively. The molecular or the laminar viscosity is the fluid property and the eddy viscosity or the turbulent viscosity is the flow property. By using dimensional analysis, the eddy viscosity μ_t can be expressed as,

$$\mu_t = \rho V_t l \tag{2.6}$$

Here V_t , is the turbulent velocity scale and l is the turbulent length scale. It was postulated by Prandtl and Kolmogorov and later adopted in the standard k-ε model that

$$l = \frac{\kappa^{3/2}}{\varepsilon} \tag{2.7}$$

$$V_t \sim \sqrt{k} \tag{2.8}$$

From the equation (2.6) the eddy viscosity is obtained and it is given by

$$\mu_t = \frac{\rho c \mu k^2}{\varepsilon} \tag{2.9}$$

The modelling constant, C_μ in the eddy viscosity formulation, as shown in equation (2.10), is empirically tuned for the simple shear layer. Meanwhile, there is no mechanism in the k-ε model which can either amplify the turbulent intensity or eddy viscosity in the presence of concave or convex curvature. Therefore, the expression for eddy viscosity in the standard k-ε model is considered to be inadequate to account for the streamline curvature effect. It is evident that modifications to the standard k-ε model are necessary to include the curvature effects. Therefore the constant C_μ is considered. The constant, C_μ is given by

$$C_\mu = \frac{-k_1 k_2}{\left[1+8k_1^2 \frac{k^2}{\varepsilon^2}\left(\frac{\delta U_s}{\delta n}+\frac{U_s}{R_c}\right)\frac{U_s}{R_c}\right]} \tag{2.10}$$

In the equation (2.10), $U_s = \sqrt{\overline{u}^2 + \overline{v}^2}$ and Rc is the radius of curvature of the streamline concerned (Ψ constant). The value of k1 and k2 are taken as 0.27 and 0.3334 respectively.

C. The Turbulent Modeling:

1. K-ω and SST Model:

K-omega (k-ω) turbulence model is a common two-equation turbulence model in computational fluid dynamics that is used as a closure equation of the Reynolds-averaged Navier–Stokes equations. The model attempts to predict turbulence by two partial differential equations with the first variable being the turbulence kinetic energy while the second being the specific rate of dissipation.

K − Equation

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k \frac{\rho k}{\omega}\right)\frac{\partial k}{\partial x_j}\right] \tag{2.11}$$

ω- Equation

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma\omega}{k}P - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega \frac{\rho k}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \frac{\rho\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{2.12}$$

SST (Menter's shear stress transport model)

The SST two equation turbulence models was introduced in 1994 by F.R. Menter to deal with the strong free stream sensitivity of the k-omega turbulence model and improve the predictions of adverse pressure gradients.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k \mu_t)\frac{\partial k}{\partial x_j}\right] \tag{2.13}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma}{\nu_t}P - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega \mu_t)\frac{\partial \omega}{\partial x_j}\right] + 2(1-F_1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{2.14}$$

2. Performance Parameters:

This section describes how lift coefficient and drag coefficient are characterized. Lift and Drag are two main parameters for wind turbine blade in terms of aerodynamic performance. Drag depends on the properties of the fluid and on the size, shape, and speed of the object.

$$F_D = \tfrac{1}{2}\rho v^2 C_D A$$

Where

- FD is the drag force, ρ is the density of the fluid, v is the speed of the object relative to the fluid, A is the cross sectional area, C_D is the drag coefficient.

  Lift is a result of pressure differences and depends on angle of attack, airfoil shape, air density, and airspeed

$$L = \tfrac{1}{2}\rho v^2 A C_L$$

L is lift force, ρ is air density, C_L is the lift coefficient at the desired angle of attack.

4 MATERIAL AND METHEDOLOGY

This research paper describes the CFD analysis of S819 airfoil. For the analysis, two different geometry of airfoil was used, the first geometry used was airfoil with slot and the other one was airfoil without slot, the boundary conditions applied to the domain, the assumptions made, the equations used, the results obtained after calculation and then the results were compared with each other.

A. GEOMETRICAL DESCRIPTION

The geometry created for analysis having two different types of shapes, Airfoil S819 with leading edge slot and without slot. Slot in airfoil 30% from the leading edge side and 70% from trailing edge side. Sketch for airfoil S819 is generated through java programme. This java programme was made using co-ordinates given for S819 airfoil by NREL (National Renewable Energy Ltd.)

B. MESH

A quadrilateral mesh element was generated through meshing done in Ansys Fluent 15.0.

Following are the details for computational grid.

Length of computational Domain- 3000 mm

Width of computational Domain- 2000 mm

No of Nodes- 250613

No. of Elements- 249703

Type of Mesh- Quadrilateral

Min. face size- 2 mm

Max. Face size- 5 mm

Mesh growth rate- 1.20

Minimum orthogonal quality of mesh- 0.382559423645068

Maximum skewness- 0.83796842115418

Minimum Aspect ratio- 1

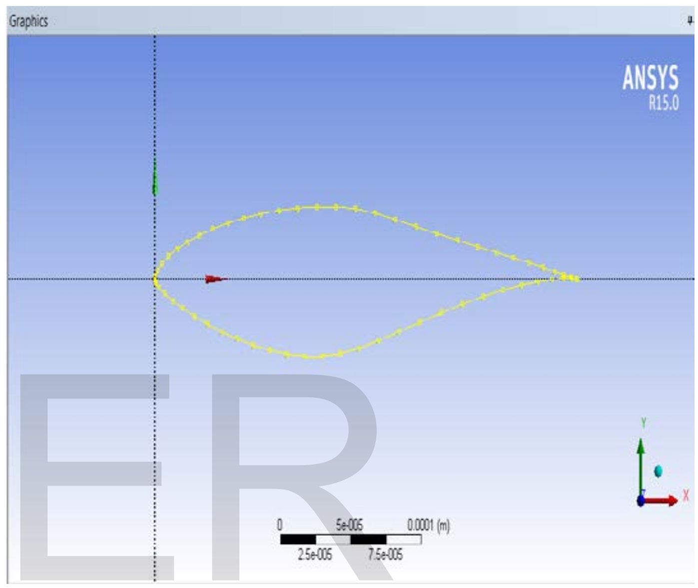

Fig. 4.1 Sketch for Airfoil

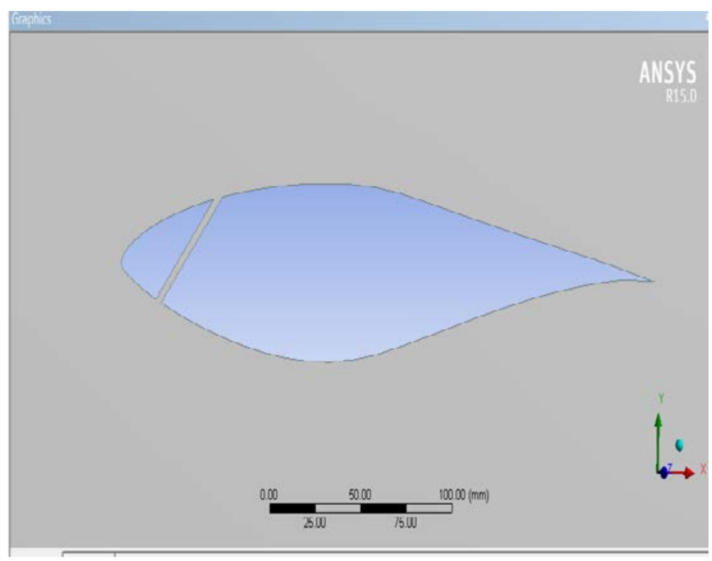

Fig. 4.2 Airfoil with slot

5 RESULT AND DISSCUSSION

Analysis is done for airfoil S819 with slot and without slot and

IJSER

ISSN 2229-5518

results are obtained at different angle of attack. After that comparison is made between them to see the effect on lift and drag coefficient. A result shows that airfoil S819 with slot gives more lift.

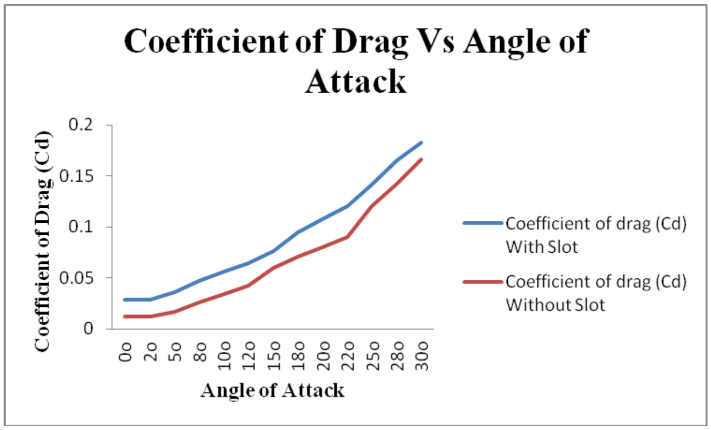

Table- 5.1 Comparison in Results between S819 Aerofoil with Slot and Without Slot for Cofficient of Drag with varying angle of attack

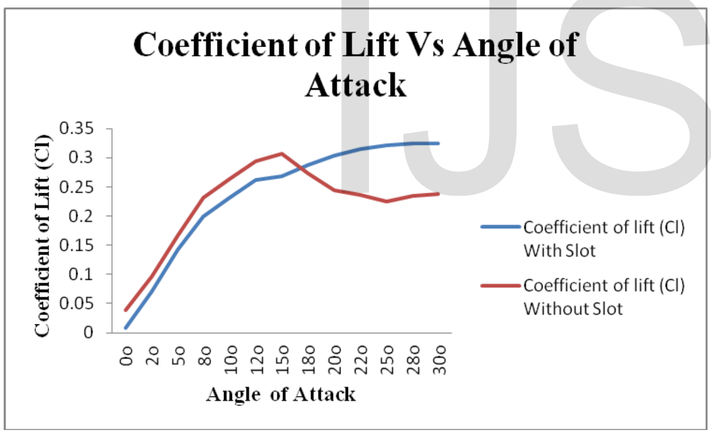

Table- 5.2 Comparison in Results between S819 Aerofoil with Slot and Without Slot for Cofficient of Lift Drag with varying angle of attack

CONCLUSION AND FUTURE SCOPE

Analysis has been carried out in ANSYS 15.0 FLUENT by changing angle of attack on S819 airfoil with slot and without slot. By using leading edge slot upper layer of boundary layer is going to invigorate and try to decrease flow separation towards the trailing edge.

1. As angle of attack increases flow separation is moving towards leading edge.

2. For given parameter and physics stall condition for airfoil without slot occur at 20o.

3. Maximum lift for airfoil without slot occurs at 15o.

4. Slotted airfoil gives higher lift and lower drag then airfoil without slot.

5. Stall condition occur at higher angle of attack in slotted airfoil compare to airfoil without slot.

The present study deals with the lift improvement by the use of slot on S819 airfoil. Current research can be implemented on different aero foils, which are used in wind turbine blade.

- Analysis can be performed experimentally.
- The active and passive methods can be used simultaneously or compound methods can be used for lift improvement in wind turbine blade.
- Airfoil S819 is used for large wind turbine blade and due to good lift to drag ratio the research on this field is mandatory for further improvement in wind turbine efficiency.

IJSER

References

[1] E. Akcayoz and I. H. Tuncer, Numerical investigation of flow control over an airfoil using synthetic jets and its optimization. International Aerospace Conference, Turkey (2009).
[2] C. Jensch, K.C. Pfingsten and R. Radespiel, Numerical investigation of leading edge blowing and optimization of the slot geometry for a circulation control airfoil, Notes on Numerical Fluid Mechanics and Multidisciplinary Design.112 (2010) 183-190.
[3] B. Yagiz, O. Kandil and Y.V. Pehlivanoglu, Drag minimization using active and passive flow control techniques. Aerospace Science and Technology, 17 (1) (2012) 21-31.
[4] M. Goodarzi, R. Fereidouni and M. Rahim, Investigation of flow control over a NACA0012 aerofoil by suction effect on aerodynamic characteristics, Canadian Journal on Mechanical Sciences and Engineering, 3 (3) (2012)102-109.
[5] Bragg, M.B., Gregorek, Experimental study of airfoil performance with vortex generators, 1987.
Hansen, Aerodynamics of Wind Turbines, China Power Press, Beijing (2009).
[6] U. Anand, Passive flow control over NACA 0012 aerofoil using vortex generators.
[7] M. Tahar Bouzaher, Numerical study of flow separation control over a NACA 2415 aerofoil.
[8] Leonardo P. Chamorro, Roger Arndt and Fotis Sotiropoulos, on the skin friction drag reduction in large wind turbines using sharp V-grooved riblets.
[9] Ya-lei Bai, Xing-yu Ma, Xiao Ming, Lift enhancement of aerofoil and tip flow control for wind turbine.
[10] Kianoosh Yousefi, Reza Saleh and Peyman Zahedi, "Numerical study of blowing and suction slot geometry optimization on NACA 0012 aerofoil"
[11] Kwing-So Choi, European drag reduction research- recent development and current status.
[12] David Greenblatt, The control of flow separation by periodic excitation.

[13] John C. Lin, Review of research on low profile vortex generators to control boundary layer separation.
[14] Jibing Lan, Effect of leading edge boundary layer thickness on dimple flow structure and separation control.
[15] Chris Kaminsky, A CFD study of wind turbine aerodynamics.
[16] W.A. Timmer, Aerodynamic characteristics of wind turbine blade airfoils at high angles of attack.
[17] P.B. Makwana, Numerical simulation of laminar flow over slotted airfoil.
IJSER