iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: CAAA-2018 Issue: onference Month of publication: April 2018

DOI:

Computational Investigation on Aerodynamic Characteristics of a Horizontal Axis Wind Turbine Blade with various Twist and CANT Angles

Ms. Shilpa GS[1], Mr. Balaji S[2], Dr. Sankar V[3]
[1]PG Scholar, [2]Assistant Professor, [3]Head of The Department, Dept. of Aeronautical Engg. Nehru Institute of Engg. & Tech. Coimbatore, Tamilnadu.

Abstract: The demand for the renewable energy sources has increased dramatically over the past few decades, due to environmental factors like climate change and other factors. Wind energy can be harvested using both horizontal and vertical axis wind turbines. This paper mainly focuses on a horizontal axis wind turbine (HAWT) by considering a suitable blade profile (NACA 63-4411). The two different geometric parameters have been varied in the present numerical investigation. The twist angle of the blade and the CANT angle of the winglet are the major parameters studied during this analysis. A standard meshing tool (Hypermesh) is used to discretize the flow domain and a commercial CFD code (ANSYS Fluent) is used to analyze the fluid flow over the wind turbine blade. The aerodynamic characteristics like lift and drag parameters are compared for various twist angle of the wind turbine blade ranging from 0 to 45 degrees. Also the blade with optimized twist angle is taken for further analyzed with winglet of various CANT angles. The CFD results are able to give an authentic justification for the better aerodynamic characteristics of the horizontal axis wind turbine blade.
Keywords: HAWT; Winglet; CANT Angle; CFD (Computational Fluid Dynamics)

I. INTRODUCTION

The efficient use of energy resources as well as the growing production of energy from renewable sources is considered as one of the major challenges in this century. There are different forms of energy that have been explored and have been developed such as geothermal, solar, wind and hydroelectric power. The key to ensure the availability of renewable energy technologies is because of its affordability and performance [1]. Renewable energy production becomes vital in maintaining current energy demands and meeting future requirements with rise in concern over resource availability, energy prices, environmental impacts, and worldwide population growth, renewable energy production. Therefore the situation is demanding to produce green energy effectively in all respects. Wind energy has triumphed as the most cost-effective source of renewable energy production. Within the United States, energy production from wind is aimed at 20% of the total energy market by 2030 (USDOE, 2008). The rapid expansion of the wind energy market necessitates the need for understanding of wind turbine aerodynamics and wake interactions and the importance for advanced computational modeling in wind turbine blade designs. As wind turbines reach higher into the atmosphere, rotor diameters increase and wind farms can expand beyond 20 km in length. The essential part of wind farm design and optimization is to understand the flow dynamics enforced by the atmospheric boundary layer (ABL) and local turbine wake interactions. The turbine wakes not only increase fluctuations, but also decrease the downstream mean velocity resulting in power production losses, which leads to structural fatigue [2]. So wind turbine blade optimization necessitates the use of computational fluid dynamics for more production of power which gives a good parametric comparison among geometric and flow parameters. In the present work, a typical wind turbine blade is computationally investigated for various angle of twists and the aerodynamic characteristics are studies. Also the optimized twisted blade is provided with winglet of various CANT angles to increase the lift characteristics by diverting the tip vortices.

II. METHODOLOGY

A. Physical Model

The airfoil selected for the present analysis is NACA 63-4411. Figure 1 depicts the profile of this airfoil. This NACA series is selected because of its significant aerodynamic performance at higher angle of attack. For the analysis purpose a small scaled model of wind turbine blade with an overall length of 1000 mm is considered. The tip chord and root chord lengths are taken proportionally (Root Chord = 192 mm and Tip Chord = 84 mm). The height of the winglet is chosen in such a way that the height is

15 % of the overall length. The winglet attachment is made with the tip of the blade with a reasonable curvature in order to avoid the sharp edges near the attachment.

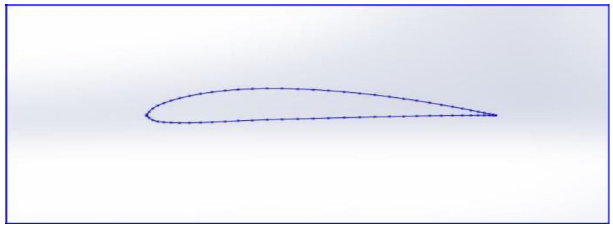

Figure 1. NACA 63-4411 Profile – Wind Turbine Blade

Figure 2 shows the wind turbine blade model with various twist angles at the root chord section. The twist angles are varied from 0º to 45º with a step of 7.5º. So there were 7 WT blade twist configurations are generated using Solidworks.

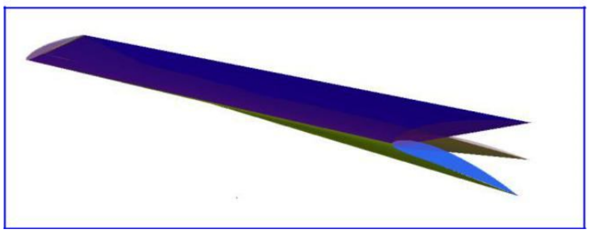

Figure 2. Wind Turbine Blade with Different Twist angle at Root Section of the Blade

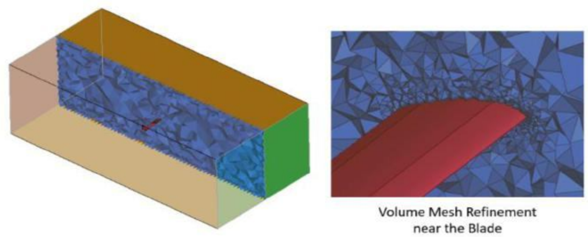

Figure 5. Volume Mesh Cut Sectional View – Refinement near the Blade

Figure 3 shows the wind turbine blade model with optimized twist angle along with winglet. The winglet profile is given with the same NACA 63-4411 series [3].

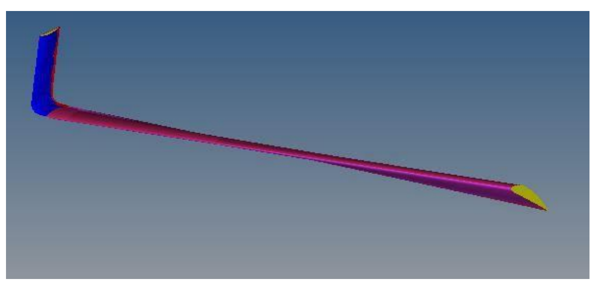

Figure 3. Wind Turbine Blade Model with optimized Twist and Winglet

B. Computational Model

The computational domain and Blade surfaces are meshed with unstructured triangular elements. Mesh refinements have been carried out near the tip surfaces, leading edges and trailing edges of the blade. Figure 4 shows the discretized computational domain with blade. The minimum size of the element used is around 0.3 mm and the maximum element size is 8 mm over the blade surfaces.

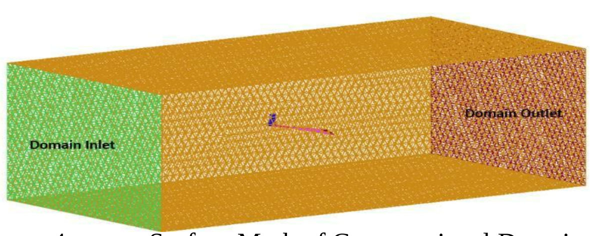

Figure 4. Surface Mesh of Computational Domain

Figure 5 shows the cut sectional view of volume mesh. The volume mesh is generated with tetra elements. The tetra growth is carried out with a growth rate of 1.2 from blade surface to domain.
In order to capture the wall shear effects a boundary layer mesh is generated over the wall surfaces of blade. Penta elements are formed in a structured manner with a growth rate of 1.3. The total mesh count of the computational domain is around 3.4 million.

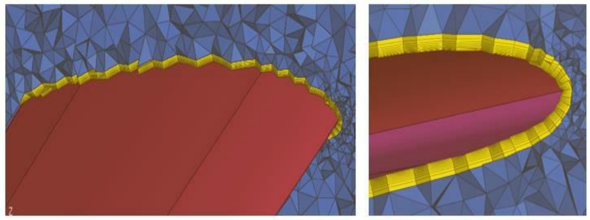

Figure 6. Boundary Layer Mesh over the Blade Surface

C. Physics Definitions

The discretized domain is given with appropriate boundary conditions in Ansys Fluent. The inlet of the domain is mentioned with a 'velocity inlet' boundary conditions with a magnitude of 6 m/s. The inlet is specified with a turbulent intensity of 5 % along with hydraulic diameter. The outlet of the domain is given with 'Pressure Outlet' boundary condition where the gauge pressure is given as 0 Pa. The surfaces of the blade is mentioned with 'Standard Wall' boundary condition without slip velocity. The fluid used for the analysis is air with standard atmospheric conditions (P=1.01325 bar; T = 300 K). Standard K-Epsilon model is used to solve the turbulent quantities.

III. RESULTS AND DISCUSSIONS

The solver deck file prepared in Ansys Fluent is solved using Pressure Based Navier Stokes (PBNS) solver. The flow equations (Continuity, Momentum & Turbulence) are solved by setting the convergence criteria to 1*E-04. The results are presented below.

A. Effect of Twist Angles

As a first cut analysis, the wind turbine blade model is given with various angle of twist at root section of the blade and the analysis is carried out with a free stream wind speed of 6 m/s. The aerodynamic characteristics like lift co-efficient and drag co-efficient with respect to angle of twist are analyzed.

Figure 7 shows the effect of twist on static pressure variations. It is observed that a significant static pressure rise on pressure side achieved at the twist angle of 22.5 degree itself. It reached the maximum at 30º twist angle. Further increment in twist angle is not having significant influence on pressure rise on the bottom side.

4

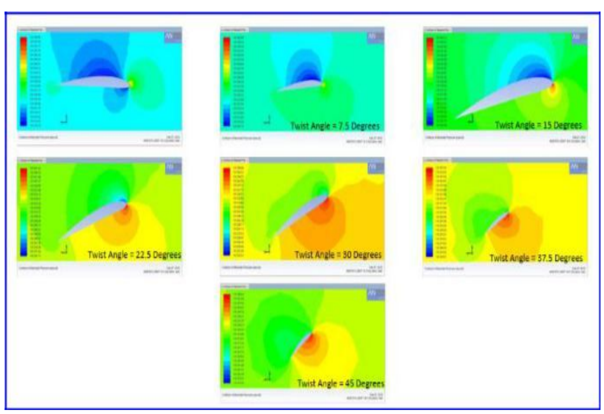

Figure 7. Static Pressure Variations with Various Twist Angles

Figure 8 shows the variations of dynamic pressure at various twist angles which directly represents the kinetic energy of the flow. So the flow separations and wake formations are clearly visible in this contour. It is noticed that the wake formation behind the wing is high afterr 30 degrees of twist.

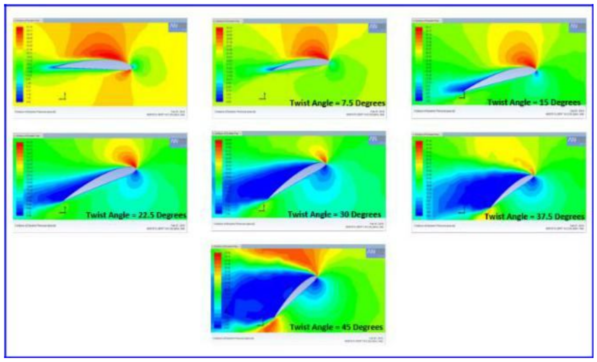

Figure 8. Variation of Dynamic Pressure – Vairous Twist Angles

Figure 9 depicts the vector plots around the blade with various twist angles. The vectors clearly shows the flow detachement at higher twists due to larger angle of attack.

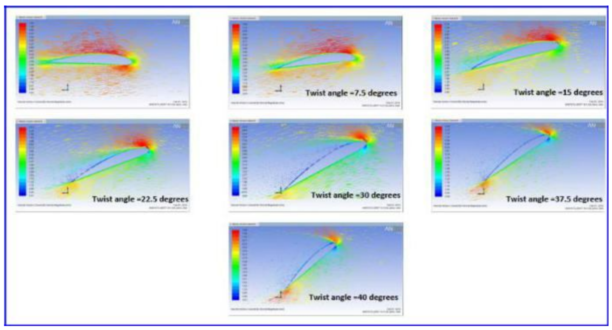

Figure 9. Velocity Vectors showing the Flow Separations and Wake Formations

Figure 10 shows the aerodynamic lift characteristics of the wind turbine blade with different angle of twists. It is observed from the curve that it follows general angle of attack trend as the twist angle varies. It can be noted from the graph that the lift co-efficient is maximum at the twist angle of 30º.

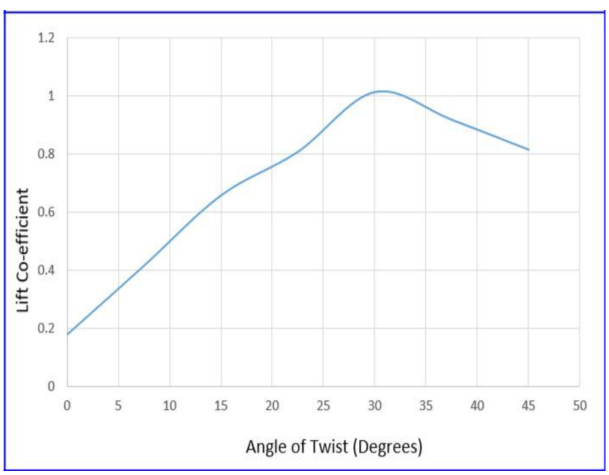

Figure 10. Lift co-efficent Vs Angle of Twist

B. Effect of CANT angles of Winglet

The second phase of analysis is carried out by adding the winglet at the tip section for the wind turbine blade with 30 degrees twist. Figure 11 shows the variation of lift co-efficient with respect to CANT angle. It is observed from the figure that the lift force produced by the wind turbine blade increases as the CANT angle increases. The maximum lift is achieved at an angle of 90 degree winglet. The variation in the lift value produced is very much significant and there exists.

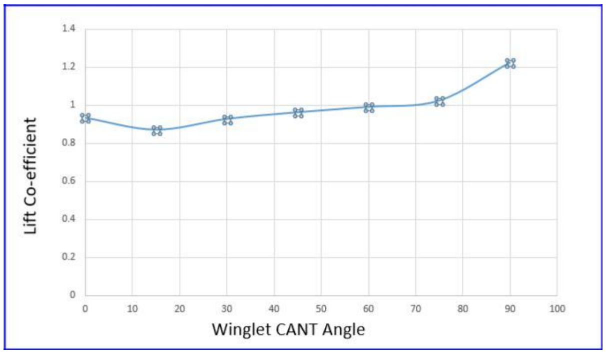

Figure 11. Lift co-efficent Vs Winglet CANT Angle

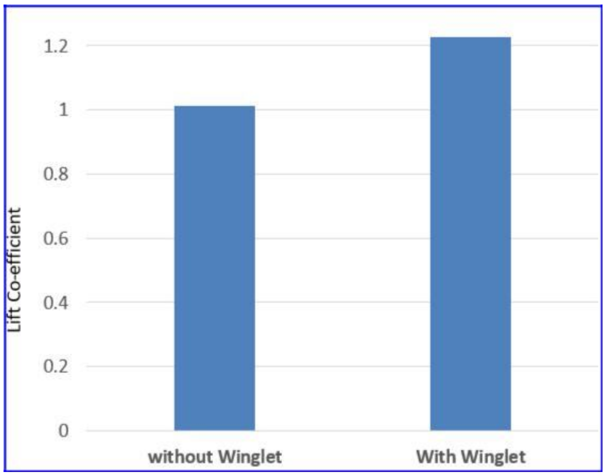

Figure 12. Comparison of Lift Co-efficient – Effect of Winglet

IV. CONCLUSION

This present study indicates the effect of changes in twist angle and cant angle over wind turbine blade lift characteristics.

A. As the angle of twist is increased, lift coefficient of the blade starts to increase. It can be seen from the above results that the flow begins to separate with high turbulence intensity over the upper surface for the twist angle beyond 30 degrees. It is evident from the simulation results that the blade profile with 30 degree twist provides better lift coefficient than other profiles.

B. The lift coefficient increases as the cant angle is increased and it reaches a maximum at the design with 90 degree cant angle. It can be realised from the results shown in the graph that the blade generates more lift for the design with 30 degree twist and 90 degree cant angle. It can be concluded that, in the above design, the drag induced by the spanwise component of flow can be significantly reduced and the performance of the wind turbine can be highly augumented to produce power efficiently even at lower wind speeds.

REFERENCES

[1] Rinaldo Gonzalez Galdamez, Diego Moreno Ferguson,and Juancarlo Rodriguez Gutierrez, “Winglet Design and Analysis for Wind Turbine Rotor Blades 100% Report, ” pp.10 , November 30, 2011.

[2] Cole J. Davis, “Computational Modeling Of Wind Turbine Wake Interactions,” Fort Collins, Colorado, Spring 2012.

[3] Jeppe Johansen and Niels N Sorensen, “ Numerical analysis of winglets on wind turbine blades using CFD , Wind Energy Department,Riso National Laboratory,Technical University Of Denmark, Dept Of Civil Engineering, Aalbory University,Denmark.

[4] John D Anderson, “Fundamentals of aerodynamics”, McGraw Hill Series In Aeronautical And Aerospace Engineering, Fifth Edition.