DYNAMIC STALL INVESTIGATION OF NACA4412 AIRFOIL - A VERTICAL AXIS WINDMILL WING

SAURABH SANJAY PATHRIKAR, NIRANJAN PATIDAR, PRATIK BABASAHEB DARKUNDE &
PROF. SAVITA SHINDE

Department of Mechanical Engineering, MIT School of Engineering, MIT ADT University, Pune, Maharashtra, India

ABSTRACT

Energy vitalization is an important role in modern life, Vertical windmill is an engrossed field of design in modern renewable energy generation but due to limitation and scant knowledge in understanding the behavior of wing in dynamic stall condition selection of wing is not proposed yet. The present investigation aims to propose the behavior of NACA 4412 Airfoil in a dynamic stall condition. The creation of lift and drag coefficient on the wing is a dependent feature of the respective angle of inclination and its variable. This case study presented on NACA 4412 Airfoil is aimed of Scientists and Engineers to understand the correlation on pressure-CL & CD for the optimization and selection of wing.

KEYWORDS: Coefficient of Lift, Aerodynamics, Vertical Axis Windmill, CFD, NACA 4412, Coefficient of Drag, Pressure

Received: Oct 24, 2020; **Accepted:** Nov 14, 2020; **Published:** Dec 05, 2020; **Paper Id.:** IJMPERDDEC202014

1. INTRODUCTION

Energy plays a key role in human growth and has a significant influence on the nation's sustainability. Since conventional sources are detrimental to health, environmental standards, and global protection, the use of renewable energy worldwide must be expanded urgently. The wind turbine is a fast-growing power generation technology that contrasts with other green energy options. India is one of the leading country in wind power generation with a total capacity of 35 GW by the end of March 2019. India has set an aspiring goal to produce 60 GW of electricity from wind power by 2022 [1]. In recent years the scientist or researchers in the field of aerodynamics focuses on enhancing the aerodynamic efficiency of a wind turbine blade. The Efficiency of wind turbines is heavily based on airfoil performance. Airfoil is a cross-sectional configuration of the wind turbine blade that is responsible for producing lift & drag forces [2]. Comparison of Airfoil shapes & airfoil performance is a key function in wind turbine blade design & aerospace applications [3]. Since wind turbines operating on the concept of lift, the lift force can give energy conversion & drag is a parasitic element. Reynolds number ranging from 10^4 to 10^6 is generally called a low Reynolds number. In such a low-Reynolds-number range, the formation of separation bubbles occurs on the airfoil surface which results in increasing the drag. [4-5]. This type of physical phenomenon strongly depends on airfoil shape, Reynolds number, angle of attack, and causes non-linear lift curves as its effects on the efficiency of the airfoil. For this reason, aerodynamic forecasts must be focused on physical properties with a low Reynolds for high-performance airfoil construction. Airfoil performance depends on the angle of attack [6]. By increasing the AOA, the lift coefficient of the airfoil increases up to the maximum lift coefficient and the angle of attack at that point is known as critical/stall angle of attack and a further increase in AOA above stall AOA the separated flow breaks down results in a decrease in lift coefficient [7-8].

Aerodynamics plays a big part in changing airfoil forms. In 1992, the NACA started to investigate the properties of the air feed sequence to decide which shapes were most suitable for the particular purpose. In 1992, it began the research. CFD codes are efficient to design the airfoils for the wind turbine blades, as they correctly determine the airfoil performance. CFD is a powerful instrument for testing the turbine blade aerodynamics. In this regard, some CFD analysis on wind turbine blade has been performed [9-10-11] & it is concluded that both CFD & experimental data shows similar results. It is necessary to utilize the computational model as it shows the flow passing through the airfoil over various flow regions & it also shows flow separation and reattachment of isolated flow. Thereby it is not possible to solve such problems numerically. A turbulence model is utilized in CFD simulation to close the system of the mean flow equation. Various turbulence models are used for the simulation of the airfoil such as $k-\epsilon$ [9] for the CFD simulation of a wind turbine blade, LES turbulence model [12-13] for assessments of a dimple on airfoil & for the dynamic stall control for wind turbine airfoil, SST- $k-\omega$ turbulence model [14-15] is utilized to improve the aerodynamic performance of HAWT & to control the dynamic stall by vortex generator on the wind turbine blade. In [16] two-equation turbulence model is used to control the flow for enhancement of the aerodynamic performance, as $k-\omega$ is used near the boundary layer & $k-\epsilon$ is used outside the boundary layer to obtain the more accurate results.

As with the number of Reynolds, angle of attack & selection of the required turbulence model for aerodynamic research is important. In low Reynolds, the aerodynamic performance of airfoil decreases, resulting in a splitting of the amount wave. Trend analysis and very little research on NACA4412 as the airfoil geometry for the wind turbine blade was conducted on the fitting geometry for the air turbine blade. The main intention of the analysis is to examine the performance of NACA4412 airfoil using CFD simulation at low Reynolds number & by varying the angle of attack. In the present work, the $k-\omega$ turbulence model is used as a solver for the CFD simulation.

An aerodynamic investigation of Airfoil plays a very important role to determine the optimal wing layout for gaining maximum efficiency because the construction of a windmill based on the selection of airfoil is done according to its working lift and drag characteristics on dynamic stall conditions. In recent research it has been found out there is a versatile amount of scientific evidence based on practical and CFD calculation performed on Airfoils but very scant evidence is present to determine the behavior of airfoil on dynamic stall effect for Angle of attack range from 0 to 360 Degree. The present investigation is based on the Computational Fluid Dynamic Analysis and Statistical Analysis of NACA Grade 4412 Airfoil to verify and validate results on the selection and effect of airfoil on post-Dynamic Stall Effect.

2. NUMERICAL SCHEME

2.1 Problem Description

In present study an Airfoil grade of NACA 4412 has been investigated for dynamic stall condition of Angle of attack range from 0 to 360 Degree. This study is specially proposed for Scientist and Engineering community to understand behavior of Airfoil in Dynamic stall condition as in case for Vertical Axis Windmill. Figure 1. Shows the C-Grid used for surface preparation.

Figure 1: Surface creation

2.2 CFD Investigation.

The CFD investigation is divide in 5 major steps as shown in figure below

Figure 2: CFD Steps

2.3 Mesh Creation

Figure 3: Mesh Creation

In any CFD investigation Meshing is one of the important step. Meshing is the process of discretization geometry in number of fine chunks through which information travel from the solver to the post-process result.

In present investigation the geometry of 1 M length is discretize using fine mesh technique with edge sizing of 0.05 mm near wall field. The layer of 3 refinement is giving in boundary condition to capture refine fluid flow in the solid liquid interface.

2.4 Numerical Scheme

The code used solves the total average Reynolds navigating stock Conservation equations: Equation of continuity and equation of momentum. In the context of international norms, non-steady analyses (URAND) for incompressible, isothermal, and turbulent flows with 5% intensity.

The computational fluid dynamics code ANSYS CFX was to utilize conduct the numerical examination. In the current paper with the help of unsteady three-dimensional incompressible fluid flow, the issue can be characterized by laws of mass, momentum, and energy specified by the Reynolds-averaged Navier–Stokes (RANS) equations. The details of the code are recorded in the literature. The flow is defined by the mass (continuity), momentum (Navier-Stokes) conservation laws.

In general, by adding averaged and fluctuating quantities to generate the Reynolds-averaged Navier–Stokes (RANS) equations, the turbulence model is aims to change the original unsteady Navier–Stokes equations. Only when still modeling turbulence effect, without the need for the resolutions of turbulence fluctuations, do these equations represents the mean flow quantities. All the turbulence field scales are being designed. Because of the statistical averaging technique used to obtain the equations, the turbulence model based on RANS equations is known as statistical turbulence models.

As defined, by adding average and fluctuating elements, turbulence models aim to overcome a modified set of transport equations. In the current numerical system, as seen in equation 1 and 2, the k-ε low Reynolds number is used.

For Kinetic Energy

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + 2\mu_t E_{ij} E_{ij} - \rho \varepsilon \quad \text{-----}(1)$$

For dissipation

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad \text{-----}(2)$$

Rate of change of k or ε in time + Transport of k or ε by advection = Transport of k or ε by diffusion + Rate of production of k or ε - Rate of destruction of k or ε

2.5 Boundary Condition

The initial boundary condition in the fluent solver is taken as by solving stated numerical scheme. The Velocity flow Inlet Air – 5 M/s, Outlet coupled is stated as pressure outlet, Turbulent intensity at low Reynolds number is taken at 5% coupled ratio. The convergence creation is set as 1E-06. Second order least square passed with scale factor of 1:2 is stated in Second order upwind solver. The Coefficient of Lift and Drag with vorticity is analysed for given angle of attack

Angle of Attack –

a) 0 AoA, b) 45AoA, c) 90 AoA, d) 135 AoA, e) 180 AoA, f) 225 AoA, g) 270 AoA, h) 315 AoA

3 RESULT AND DISCUSSIONS

After setting Numerical scheme with solver preparation the convergence is taken at fixed transient time scale based analysis until sinusoidal wave of residual occurs in converge creation. The time scale factor taken to converge solution is 0.05 S in capturing vortices flow field. Figure 4, Shows the Post-process result for particular Angle of attack with respect to its Pressure and velocity contours.

a) 0 AoA

b) 45 AoA

c) 90 AoA

d) 135 AoA

e) 180 AoA

f) 225 AoA

g) 270 AoA

h) 315 AoA

Figure 4: Pressure and Velocity Contours

3.2 Statistical Discussion

The CFD investigation is performed on NACA 4412 Airfoil with mesh enhancement study of discretize C-Mesh grid in which the following statistical validation is presented on basis of linear data regression.

Coefficient of Lift –

Statically plot of Coefficient of Lift

Statically plot of Coefficient of Drag

The lift coefficient is validly depended function of pressure and Angle of attack. The NACA 4412 is Non-Symmetrical Airfoil in which there is a creation of negative lift as per the increasing angle of rotation from 0 to 100. The creation of lift gets positive when streamline body gets attached to the vertex of axial inlet motion. The Coefficient of lift again tends to be negative as such the angle of attack is dynamic stall increase and it again tends to increasing positive value of lift as the value increase from 250 AoA to 360 Streamline AoA. The value of C_l and C_d is inverse proportion if Drag increase the amount of lift Decreases.

CL and CD Dominance as per the Angle of Inclination in Dynamic Stall

4. CONCLUSIONS

Investigating Air foils for dynamic stall effect is engrossed research area to find out optimal consideration of wing in designing of vertical axis windmill. This investigation is processed by using CFD based commercial coded software to find out the effect of Dynamic stall of the Wing of windmill. The investigation is processed using Ansys fluent software and validated using literature based survey on creation of mesh and work domain space. The following points are drawn through this investigation.

- In Dynamic stall condition the characteristic of lift coefficient is based on the dominance of Angle of inclination.
- There is negative lift coefficient in the angle of 5 to 100 then again 160 to 270. This creation of negative lift is occurred because dominance of negative pressure increase at trailing edge.
- The spiral plot for dominance of inverse proportion lift and drag characteristics is also proposed in comparative data driven analysis.

REFERENCES

1. Offshore wind energy report in India, <https://mnre.gov.in/img/documents/uploads/2e423892727a456e93a684f38d8622f7.pdf>, (2019), Assess date: 4 November 2020.
2. Alex E. Ockfen & Konstantin I. Matveev. Aerodynamic characteristics of NACA-4412 airfoil section with flap in extreme ground effect, *Inter J Nav Archit Oc Engng* (2009) 1:1~12
3. Mohamed A. Sayed, Hamdy A. Kandil, Ahmed Shaltot. Aerodynamic analysis of different wind-turbine-blade profiles using finite-volume method, *Energy Conversion and Management* xxx (2012) xxx-xxx.

4. Juanmian Lei, Jiawei Zhang, Jianping Niu. *Effect of active oscillation of local surface on the performance of low Reynolds number airfoil*, *Aerospace Science and Technology*, 2020.
5. Hua Shan a,*, Li Jiang a, Chaoqun Liu a, Michael Love b, Brant Maines b. *Numerical study of passive and active flow separation control over a NACA0012 airfoil*, *Computers & Fluids* 37 (2008) 975–992.
6. Mustafa Serdar Genç, b,*, Halil Hakan Açıkela, Kemal Kocaa. *Effect of partial flexibility over both upper and lower surfaces to flow over wind turbine airfoil*, *Energy Conversion and Management* 219 (2020) 113042.
7. Chengyong Zhu, Jie Chen, et.al. *Effect of partial flexibility over both upper and lower surfaces to flow over wind turbine airfoil*, 2020.
8. Guangqiang Di, Zhaochun Wu, et.al. *The research on active flow control method with vibration diaphragm on a NACA0012 airfoil at different stalled angles of attack*, 2017.
9. Bhushan S. Patil, Hitesh R. Thakre. *Computational fluid dynamics analysis of Wind turbine blade at various Angle of attack and different Reynolds number*, *Procedia Engineering* 127 (2015) 1363 – 1369.
10. K. A. Fagbenro, M.A. Mohamed, D.H. Wood. *Computational modeling of the aerodynamics of windmill blades at high solidity*, *Energy for Sustainable Development* (2014).
11. Xin Jina, b, Yaming Wanga, Wenbin Jua, Jiao Hea, Shuangyi Xiea. *Investigation into Parameter Influence of Upstream*
12. *Deflector on Vertical Axis Wind turbines output power via Three-dimensional CFD Simulation*, *Renewable Energy* (2017), doi: 10.1016/j.renene.2017.08.012.
13. Valerio D'Alessandro, Giacomo Clementi, Luca Giammichele, Renato Ricc. *Assessment of the dimples as passive boundary layer control technique for laminar airfoils operating at wind turbine blades root region typical Reynolds numbers*, *Energy* 170 (2019) 102–111.
14. Li Guoqiang a, b, *, Yi Shihe a. *Large eddy simulation of dynamic stall flow control for wind turbine airfoil using plasma actuator*, *Energy* 212 (2020) 118753.
15. Mohammad Moshfeghia, b, Shahrokh Shamsb, Nahmkeon Hura, c,. *Aerodynamic performance enhancement analysis of horizontal axis wind turbines using a passive flow control method via split blade*, *Journal of Wind Engineering & Industrial Aerodynamics* 167 (2017) 148–159.
16. Chengyong Zhu, Jie Chen, Jianghai Wu, Tongguang Wang. *Dynamic stall control of the wind turbine airfoil via single-row and double-row passive vortex generators*, C. Zhu et al. / *Energy* 2019.
17. Xuyang Shi, Shuai Xu, Li Ding, Dianguai Huang. *Passive flow control of a stalled airfoil using an oscillating micro-cylinder*, <https://doi.org/10.1016/j.compfluid.2018.08.012>.