International Engineering Research Journal

Experimental & Numerical Investigation of Lift & Drag Performance of NACA0012 Wind Turbine Aerofoil

Mr. Sandesh K. Rasal†, Mr. Rohan R. Katwate‡

† PG student of Department of Mechanical Engineering, Dr. DYPSOE, Ambi, Talegaon, Pune, Maharashtra
‡ Assistant Professor of Dept. of Mechanical Engineering, Dr. DYPSOE, Ambi, Talegaon, Pune, Maharashtra

Abstract

Wind is one of the competitive energy sources, dragging more and more attention as renewable and greater alternative to conventional energy sources. It is more important to capture the wind energy. The shape of wind turbine blade plays very important role in capturing wind energy. This work addresses one element of this methodology i.e. aerofoil performance prediction. The aerofoil has tendency to separate the flow at low Reynolds Number. This has prompted the investigation of separation control methods. In present work an attempt is made to improve performance of aerofoil by controlling the flow separation by adding dimples on upper side of aerofoil. NACA0012 aerofoil is considered as suitable wind turbine blade for the present work. Three models were tested for lift and drag performance i.e. two models with dimple size 1% & 2% of chord length and one without dimple at various Angle of Attack. We obtained the performance of NACA0012 at various angles of attack for regular and dimpled surface.

Keywords: *Aerofoil, chord length, dimple on surface, angle of attack*

1. Introduction

Wind energy is consistent source of energy. It can't harm the environment. India is burning 1 or 2 million tons of coal every day to generate electricity which is main source of our greenhouse gas emission. With the growth in population and with growth in consumption of electricity renewables will clearly have to fill in the gap. While setting up of these wind turbines needs little research before being established which aims at attaining the highest possible power output under specified atmospheric conditions. This depends on the shape of the wind turbine blade as the blade is responsible for the conversion of kinetic energy to mechanical energy. Low-Reynolds number aerodynamics is very important for such case. Many aerofoil applications fall into this range such as Unmanned Air Vehicles, sailplanes, jet engine fan blades, inboard helicopter rotor blades and high altitude devices [1].

The behavior of the laminar boundary layer on low-Reynolds number aerofoils would affect the aerodynamic performance of the aerofoils significantly. The predominance of viscous effects in low Reynolds Number applications would results in boundary layers rapidly growing and easily separating from surface of aerofoil [1]. If Angle of Attack (AOA) is increased gradually, lift on aerofoil will increase but beyond certain angle, the ability of aerofoil to produce lift reduces rapidly. This happens because flow starts to separate at small angle of attack but attached flow over aerofoil is dominant. As angle of attack is increased further separation region on upper surface of aerofoil increases in size results in decreased lift [4]. This angle is known as Stall angle or Critical angle of attack.

Hui Hu et al. [1] studied experimentally the behavior of laminar flow separation on a NASA low-speed GA (W)-1 aerofoil at the chord Reynolds number of 70,000. The surface pressure and PIV measurements shown that as the AOA became much higher i.e. AOA >12 deg. where the separation bubble was found to burst to causing aerofoil to stall. Lei Juanmian et al. [7] studied the trailing-edge separation of a symmetrical airfoil at a low Reynolds using Finite Volume Method. P. D. Gall et al. [3] identified dynamic roughness has ability to eliminate both the short and long separation bubbles inherent in a low Reynolds number leading edge flow operating at a moderate angle of attack, while also maintaining some physical advantages over other techniques. A. Dhiliban et al. [8] studied effects of surface roughness on flow over an aerofoil in the trailing edge in order to reduce drag. There was substantial increase of coefficient of lift when triangular roughness was placed on lower surface of the airfoil. The coefficient of lift was more than the other when the triangular shaped roughness is placed from the 60% of chord to trailing edge on the lower surface. Syed Hasib Akhter Faruqui et al. [12] undertook numerical approach to observe flow separation on aerofoil.

The purpose of this research work is to investigate effect of surface modifications over upper surface of aerofoil on efficiency of aerofoil. The surface modifications have been provided in terms of percentage of chord length. The NACA0012 aerofoil profile has been considered for this work. The National

Advisory Committee for Aeronautics (NACA) is a U.S. federal agency founded on March 3, 1915, to undertake, promote, and institutionalize aeronautical research.
This NACA aerofoil series is controlled by 4 digits NACA MPXX

- M is the maximum camber divided by 100.
- P is the position of the maximum camber divided by 10.
- XX is the thickness divided by 100.

2. Aerofoil Theory

2.1 Aerofoil

Aerofoil is defined as the cross-section of a body that is placed in an airstream in order to generate useful aerodynamic force. The component of this force perpendicular to the direction of motion is called lift. The component parallel to the direction of motion is called drag.

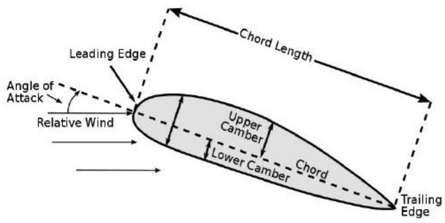

Fig.1 Aerofoil Nomenclature [9,10]

Chord length (C): The cord length is a straight line connecting the leading and trailing edges of aerofoil.

Leading edge: The leading edge is the part of the aerofoil that first contacts the air. Alternatively it is the foremost edge of an aerofoil section.

Trailing edge: The trailing edge of an aerodynamic surface such as a wing is its rear edge, where the airflow separated by the leading edge rejoins.

Angle of attack (AOA): AOA is a term used in fluid dynamics to describe the angle between a reference lines on a lifting body (often the chord line of an aerofoil) [10]

2.2 The process of flow separation

The laminar flow separation is usually found on low-Reynolds-number aerofoils as laminar boundary layers are unable to withstand any significant adverse pressure gradient. The behavior of laminar boundary layers after separating from aerofoil is responsible for affecting aerodynamic performances of aerofoils [1,3]

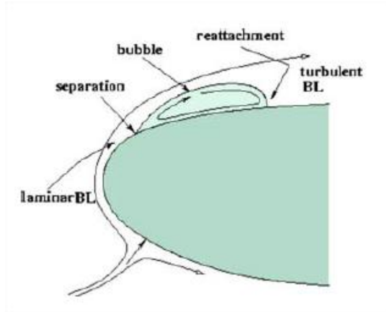

Fig.2 Laminar separation bubble formed on low Re aerofoil [1]

Laminar separation bubble may cause adverse effects, such as decreasing of lift force, increasing of drag force, reducing stability aerofoil, vibration, and noise.

2.3 *Drag and Lift on aerofoil*

The drag equation,

$$F_D = ½ C_D \rho A V^2$$

Where,

F_D is Drag force in N;
C_D is the Coefficient of drag;
ρ is density of working fluid in kg/m^3;
A is projected area of aerofoil body in m^2 and
V is velocity of air in m/s

In fluid dynamics the CD is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment such as air or water. It is used in the drag equation where a lower drag coefficient indicates the object will have less aerodynamic or drag. [5, 9]
The lift equation

$$F_L = ½ C_L \rho A V^2$$

Where,

F_L is the Lift Force on the aerofoil and
C_L is Coefficient of Lift

3. Experimental Setup

3.1 Test Aerofoils

For this work NACA0012 aerofoil profile was selected as a model. It is a symmetrical type of aerofoil. It has Chord length of 300 mm, Span 250 mm and Maximum thickness 36 mm. To investigate effect of dimples on upper surface of aerofoil two types of models are prepared. All models have been prepared by wood. The size of dimpled surface has been varied in percentage of chord length.

The models have been given numbers as follow:
Model 1: Regular surface model without any dimple
Model 2: Aerofoil with dimple size 1% of chord length
Model 3: Aerofoil with dimple size 2% of chord length

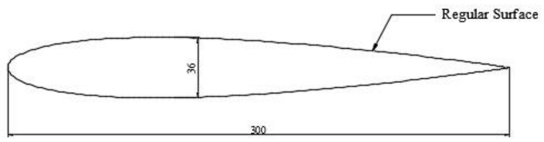

Fig.3 Model1- Regular surface model

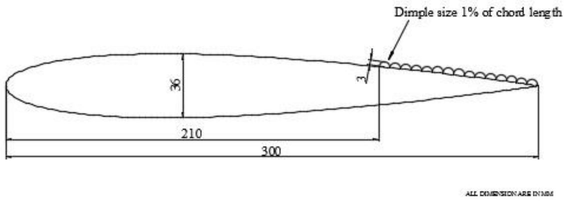

Fig. 4 1% C dimple size model

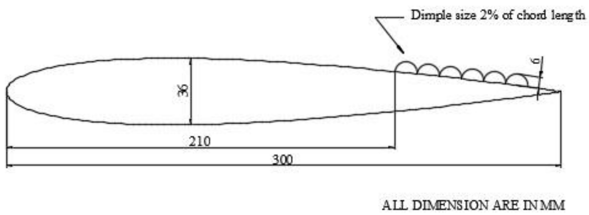

Fig. 5 2% C dimple size model

3.2 Wind Tunnel Setup

The experiments have been conducted using subsonic wind tunnel having 300 mm x 300 mm test section. Hot Wire Anemometer has been used to measure actual Wind Velocities during experimentation. The NACA0012 aerofoils with regular surface and with dimpled surface have been tested for lift and drag forces by varying following parameters:

- Free stream velocity: 6 m/s and 10 m/s
- Angle of Attack in degrees : 0, 10, 12, 15, 17, 19, 21 and 23
- Percentage of dimpled surface i.e. 1%, 2% and 3% of chord length

3.3 Numerical Analysis

NACA0012 profile is made by using X and Y coordinates from NACA data. ANSYS Fluent is used for numerical analysis of aerofoil. 2D aerofoil is considered for analysis.

The laminar model has been used as aerofoil is subjected to laminar flow because stream velocity is very small (i.e. 6 m/s and 8 m/s). The tetrahedral type of meshing is used with smallest element size 0.001 m. Fig. 5 shows the structure of meshing for NACA0012 with dimple size is 1% of chord length when angle of attack 10 degrees

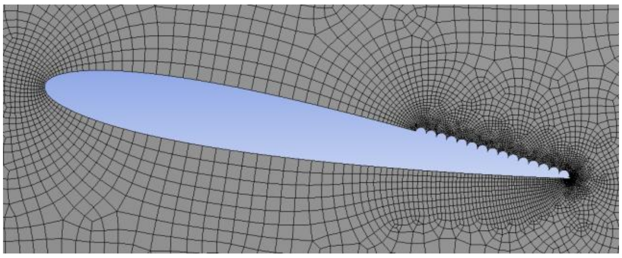

Fig. 6 Structure of meshing for NACA0012 with 1%C dimples and 10 degree angle of attack

To have realistic view the boundary over aerofoil is selected with length 600 mm and height 500 mm. The following boundary conditions have been used

- Inlet boundary set as velocity-inlet and the inlet velocity is given
- Outlet boundary set as pressure-outlet
- Aerofoil surface set as wall.

4. Results and discussion

The NACA0012 aerofoil with and without dimpled surface have been tested on wind tunnel setup for the lift and drag forces. The coefficient of performance i.e. ratio of coefficient of lift to coefficient of drag is calculated in order to determine performance of aerofoil. Fig. 7, Fig.8 and Fig.9 shows the graph of Coefficient of performance Vs AOA for Model 1, Model 2 and Model 3 respectively for 6 m/s free stream velocity

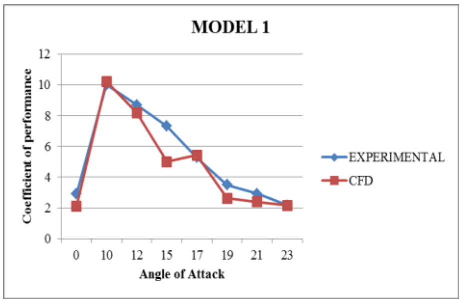

Fig. 7 Comparison of Experimental and CFD results for MODEL 1

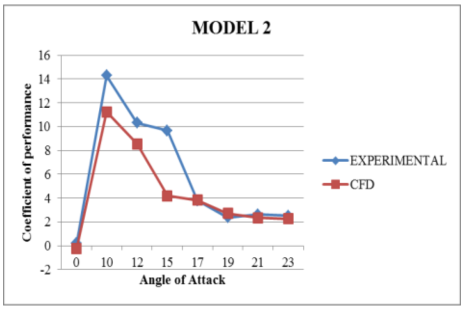

Fig. 8 Comparison of Experimental and CFD results for MODEL 2

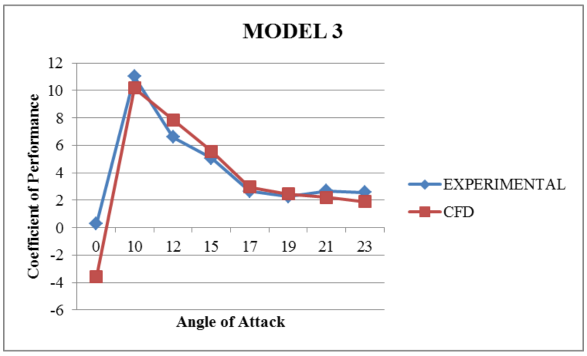

Fig. 9 Comparison of Experimental & CFD results for MODEL 3

From Fig. 7-9 it is clear that experimental and CFD results are nearly equal for all models.

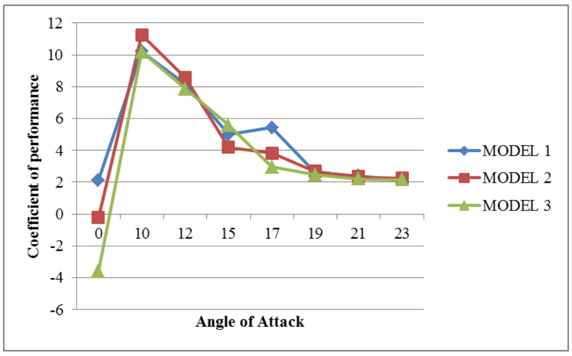

Fig. 10 CFD results for all models

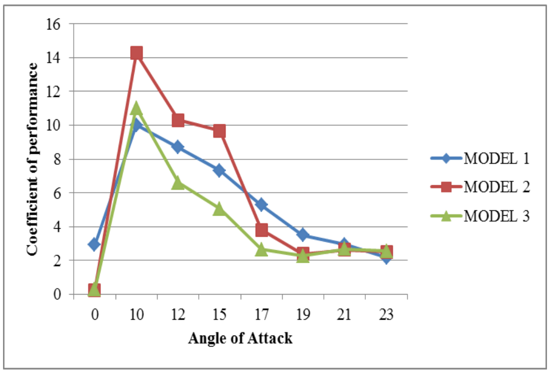

Fig.11 Experimental Results for all models

From Fig. 10 & 11 shows Coefficient of performance for all models for same AOA. It is observed that the coefficient of performance is higher for Model 2 as compared with Model 1 & Model 3.

Fig. 12 to Fig. 15 shows the velocity contour plot for regular surface aerofoil for free stream velocity 6 m/s

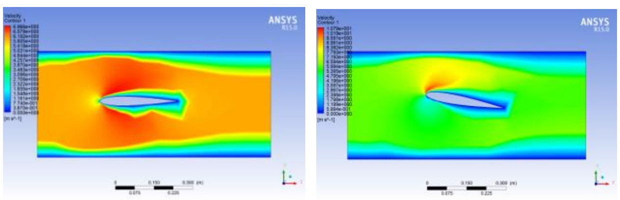

Fig. 12 Velocity contour plot at 0 & 10 deg. AOA

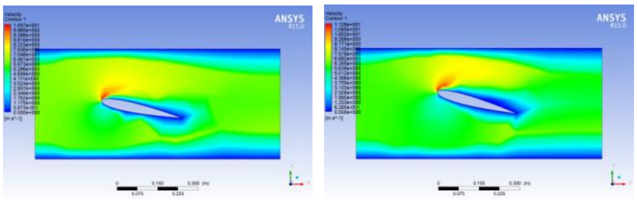

Fig. 13 Velocity contour plot at 12 & 15 deg. AOA

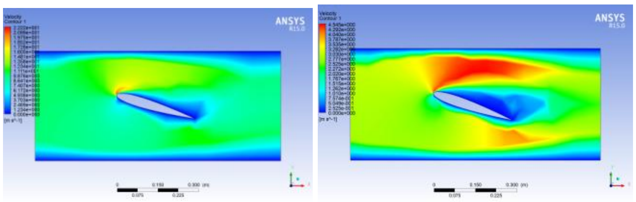

Fig. 14 Velocity contour plot at 17 & 19 deg. AOA

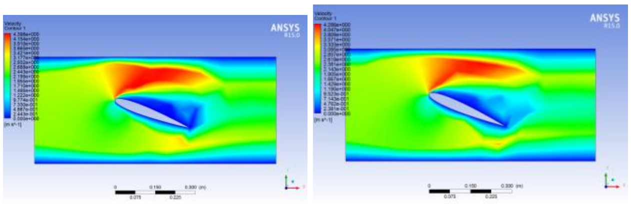

Fig. 15 Velocity contour plot at 21 deg & 23 deg AOA

Fig. 16 to Fig. 19 shows the velocity contour plot for aerofoil with dimple size 1% of chord length for free stream velocity 6 m/s.

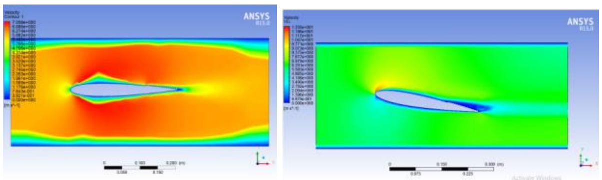

Fig. 16 Velocity contour plot for 0 & 10 deg. AOA

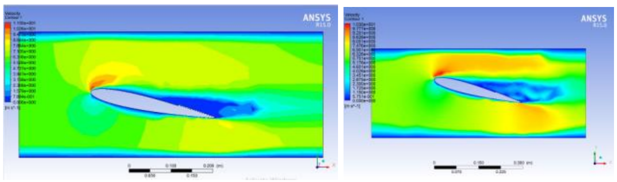

Fig. 17 Velocity contour plot for 12 & 15 deg. AOA

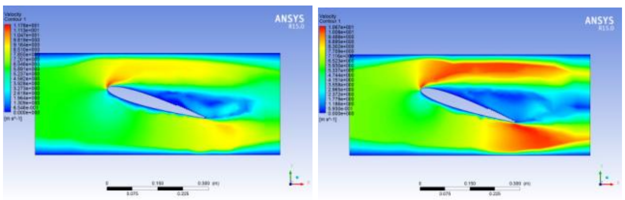

Fig. 18 Velocity contour plot for 17 & 19 deg. AOA

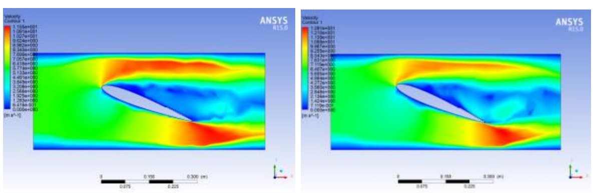

Fig. 19 Velocity contour plot at 21 & 23 deg. AOA

Conclusions

In this paper, NACA0012 models with regular shape and dimpled shape over upper surface of aerofoil were tested for lift and drag performance. The size of dimple was varied in percentage of chord length. After experimental and numerical investigation, the experimental and numerical results were compared and have shown good similarity. The results showed that addition of dimples on upper surface with size of dimple equal to 1% of chord length (Model 2) has improved the performance of aerofoil. The dimple on upper surface of aerofoil also helps to delay the flow separation.

References

Hui Hu and Zifeng Yang (2008) "An Experimental Study of the Laminar Flow Separation on a Low-Reynolds-Number Aerofoil," *Journal of Fluids Engineering ASME*, 2008, Vol. 130 / 051101-1

Mohammad Mashud, A. Al Bari and T. P. Bhowmick, (2009) " Experimental Investigation on Fluid Flow Separation Control," *International Conference on Mechanical Engineering,*

P. D. Gall, W. W. Huebsch and A.P. Rothmayer,(2010) "Dynamic Roughness as a Means of Leading edge separation flow control", *27th International Congress of The Aeronautical Sciences,*

Deepanshu Srivastav, "Flow Control over Aerofoils using Different Shaped Dimples ", IPCSIT vol.33, 2012 IACSIT Press, Singapore

Ji Yaoa, et al. "Numerical simulation of aerodynamic performance for two dimensional wind turbine aerofoils" Procedia Engineering, 2012, 80 –86

Shantanu S. Bhat, Raghuraman N. Govardhan, "Stall flutter of NACA0012 aerofoil at low Reynolds numbers," Journal of Fluids and Structures 41, 2013, 166–174

Lei Juanmian , Guo Feng, Huang Can, (2013) "Numerical study of separation on the trailing edge of a symmetrical aerofoil at a low Reynolds number," *Chinese Journal of Aeronautics,* ,26(4): 918–925

Dhiliban, P. et al.(2013) "Aerodynamic Performance Of Rear Roughness Aerofoils," *The Eighth Asia-Pacific Conference on Wind Engineering*, 10–14, , Chennai, India

P. Ghos, Akhilesh. K. Dewangan, Pratik Mitra & Dr. A. K. Rout, (2014)"Experimental study of Aero Foil with Wind Tunnel Setup", *Research Gate, Conference Paper*

Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja,(2014) "CFD Analysis of an Aerofoil", *International Journal of Engineering Research*, Volume No.3, Issue No.3, pp : 154-158,

Agrim Sareen, Robert W. Deters, Steven P. Henry,(2014) "Drag Reduction Using Riblet Film Applied to Aerofoil", *Journal of Solar Energy Engineering ASME*, Vol. 136 / 021007-1

Syed Hasib Akhter Faruqui, Md. Abdullah AlBari, MdEmran, Ahsan Ferdaus,(2014) "Numerical analysis of role of bumpy surface to control the flow separation of an aerofoil.", *Procedia Engineering* 90, 255 – 260

Izzet Şahin and Adem Acir, (2015)"Numerical and Experimental Investigations of Lift and Drag Performances of NACA 0015 Wind Turbine Aerofoil", *International Journal of Materials, Mechanics and Manufacturing* Vol. 3, No. 1,

Bhushan Patil, Hitesh Thakare, (2015) "CFD Analysis of wind turbine blades at various AOA and Low Re ", *Procedia Engineering* 127-1363 – 1369