Numerical Analysis of Lift & Drag Performance of NACA0012 Wind Turbine Aerofoil

Mr. Sandesh K. Rasal[1], **Mr. Rohan R. Katwate**[2]

[1]PG student, Dr.D.Y. Patil School of Engineering Academy, Ambi-Talegaon, Maharashtra, India
[2]Professor, Department of Mechanical Engineering, Dr. D.Y.Patil School of Engineering Academy, Ambi, Maharashtra, India

***

Abstract - *Wind energy is consistent source of energy. It doesn't harm the environment. But it is more important to capture the wind energy. The shape of wind turbine blade plays very important role in capturing wind energy. In this paper, one element of this methodology i.e. aerofoil performance prediction is considered. The aerofoil has tendency to separate the flow at low Reynolds number. This has prompted the investigation of different methods to control the flow separation. In this paper an attempt is made to predict performance of aerofoil by controlling the flow separation with addition of circular dimples on upper surface of aerofoil. NACA0012 aerofoil was considered as wind turbine blade for the present work. Two models were analyzed for lift and drag performance i.e. one with regular surface and another with dimples on upper surface. The height of dimples was selected as 3 mm i.e. 1% of chord length and was placed at distance of 210 mm i.e. 70%of chord length from leading edge. Numerical analysis was done using ANSYS Fluent. The coefficient of lift to drag ratio was calculated by numerical method at various angles of attack. It was found that there was increase in lift to drag ratio after addition of circular dimple over upper surface of aerofoil.*

Key Words: aerofoil, Reynolds number, chord length, dimples, angles of attack, NACA0012.

1. INTRODUCTION

The primary application of wind turbine blade is to convert wind energy in to electrical energy. Hence, the study of aerodynamics is an important aspect of wind turbine. While setting up of these wind turbines, it needs little research before being established which aims at attaining the highest possible power output under specified atmospheric conditions. The low Reynolds number aerodynamics is 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, 7]. But the behavior of flow over aerofoil at low Reynolds number affects the aerodynamic efficiency significantly. Thus it becomes necessary to study the behavior of flow separation over aerofoil at low Reynolds number and identifying different methods to control it.

In this regard, various attempts were made by researchers in order to control the flow separation over aerofoil at different Reynolds number and angles of attack. Mohammad Mashud et al. [2] investigated experimentally using subsonic wind tunnel, the effect of partial dimples on upper surface of NACA0012 aerofoil. P. D. Gall et al. [3] had placed dynamics roughness i.e. actual humps which oscillate with unsteady motion on leading edge of aerofoil. The experimental & numerical analysis was performed to check aerodynamic efficiency. Deepanshu Srivastav [4] analyzed numerically the effect of outward and inward dimple over upper surface of aerofoil to control the flow separation. Lei Juanmian et al. [7] studied the process of flow separation over SD8020 by simulating the flow using finite volume method at low Reynolds number. A. Dhiliban et al. [8] had performed experimental and numerical analysis of NACA0018 aerofoil profile by adding triangular roughness on upper & lower surface from 55% to 90% of chord length. Agrim Sareen et al. [11] tested DU 96-W-180 airfoil with four different symmetrical V-shaped riblet sizes (44, 62, 100, and 150-µm) at three Reynolds numbers ($1x10^6$, $1.5x10^6$, and $1.85x10^6$) and at various angles of attack. Syed Hasib Akhter Faruqui et al. [12] investigated numerically the effect of bumpy surface at 80% camber for NACA4315 profile.

The results of these studies have shown that for low Reynolds number there is formation of laminar separation bubble which causes aerofoil to stall after burst leading to separation of flow near to the surface of aerofoil. This flow separation could be controlled by simple surface modifications in aerofoil profile. The flow separation could be delayed or controlled by dynamic roughness on leading edge or with bumpy surfaces on upper surface of aerofoil. Although the surface modification has proven to be effective in flow separation control but little research is carried out on type, size, location, area etc. of the modification.

2. AEROFOIL NOMENCLATURE

An aerofoil is the shape of a wing or blade (of a propeller, rotor or turbine) or sail. An aerofoil shaped body moved through a fluid produces an 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 shows the cross section of an aerofoil mentioning its basic nomenclature.

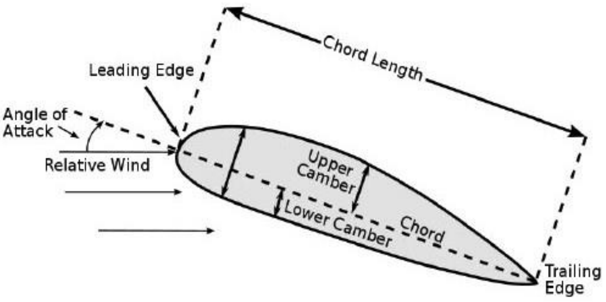

Fig -1: Basic Aerofoil Nomenclature [10]

Leading edge: The leading edge is the part of the aerofoil that first comes in contacts with 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.

Chord length: The chord length is a straight line connecting the leading and trailing edges of the aerofoil. The line joining leading edge & trailing edge is known as chord line.

Angle of attack: Angle of attack (AOA) is the angle made by chord line with oncoming fluid on aerofoil.

Camber: It is deviation of surface of aerofoil from mean center line or chord line. There are two types of camber i.e. upper and lower camber depends on deviation of surface in upward and downward direction respectively from chord line.

2.1 NACA

The National Advisory Committee for Aeronautics (NACA) is a U.S. federal agency which undertakes, promote, and institutionalize aeronautical research [15]. 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

3. NUMERICAL ANALYSIS

In this paper, 2D NACA0012 aerofoil profile was considered for numerical analysis of wind turbine blade. It is a symmetrical type of aerofoil profile. The X & Y coordinates for this profile were taken from NACA data to prepare the model. The chord length of aerofoil for the study was considered as 300 mm and maximum thickness as 36 mm. Two types of models were considered for analysis
i) NACA0012 with regular surface. (Fig-2)
ii) NACA0012 with circular dimples. (Fig-3)

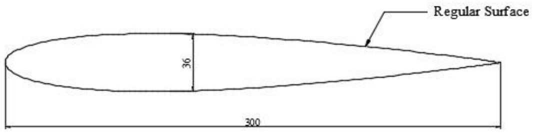

Fig-2: Geometry of NACA0012 with regular surface

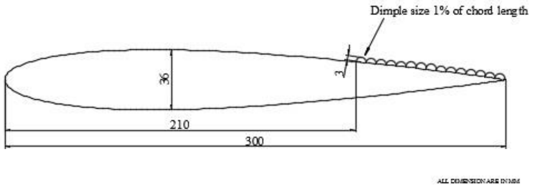

Fig-3: Geometry of NACA0012 with circular dimples

The dimesions shown in Fig-2 & Fig-3 are in mm. The dimples were added at distance of 70% of chord length i.e. 210 mm from leading edge as flow starts to separate from that point [12].

ANSYS Fluent has been used for numerical analysis of aerofoil. The Viscous-Laminar model has been used to determine coefficient of lift and drag. The tetrahedral type of meshing is used with smallest element size of 0.001 m. Fig-4 shows the structure of meshing for NACA0012 with dimple size 1% of chord length when angle of attack is 10 degrees.

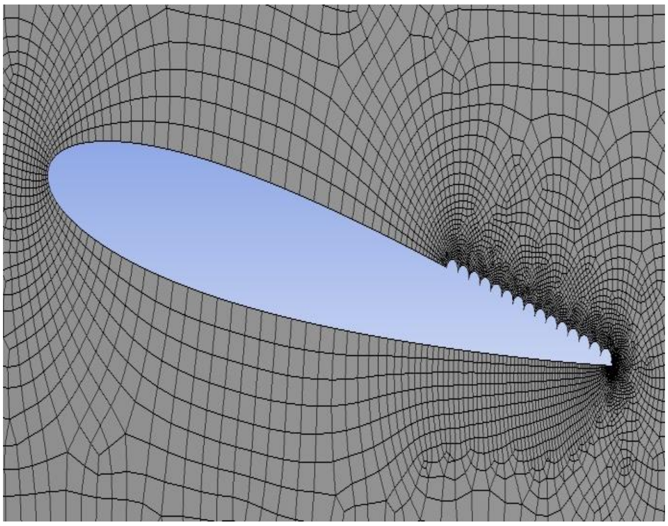

Fig-4: Structure of Tetrahedral Meshing

3.1 INPUT AND BOUNDARY CONDITION

The numerical analysis has been carried out over NACA0012 to predict lift & drag performance of aerofoil. The following boundary conditions have been used

Table -1: Input and Boundary Conditions

Sr. No.	Input	Value
1	Chord Length	300 mm
2	Maximum Thickness	36 mm
3	Velocity of fluid	6 m/s
4	Density of fluid	1.22 kg/m^3
5	Angles of Attack (in degrees)	0,10,12,15,17,19,21 and 23
6	Model	Viscous-Laminar
7	Inlet Boundary	Velocity-Inlet
8	Outlet Boundary	Pressure-Outlet

4. RESULTS & DISCUSSION

The coefficient of lift (CL) and coefficient of drag (CD) was determined using ANSYS Fluent for both geometries and their performance is represented in Chart-1 & Chart-2

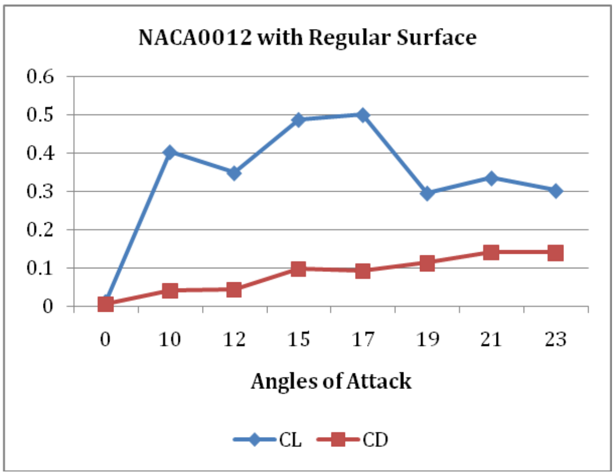

Chart-1: CL, CD Vs Angles of attack for regular surface aerofoil

Chart-1 shows the lift and drag performance of NACA0012 with regular surface i.e. without dimples for different angles of attack from 0^0 to 23^0. From Chart-1 it is clear that for NACA0012 aerofoil with regular surface, coefficient of drag (CD) increases as angle of attack goes on increasing. The value of coefficient of lift (CL) increases initially with increase in angles attack, but after reaching 17^0 the value of coefficient of lift is suddenly decreasing and goes on decreasing as per increase in angles of attack. The condition at which value of coefficient of lift starts reducing is known to be stalling condition.

Chart-2 shows the lift and drag performance of NACA0012 aerofoil with dimples on upper surface for different angles of attack from 0^0 to 23^0.

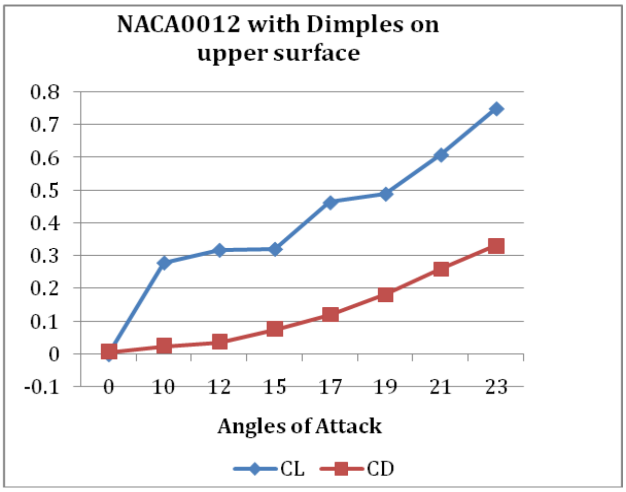

Chart-2: CL, CD Vs Angles of attack for aerofoil with dimples on upper surface

Chart-2 indicates that the for NACA0012 aerofoil with dimples on upper surface, coefficient of drag goes on increasing as angle of attack increases whereas graph for coefficient of lift is also showing the same nature. Here, for this aerofoil, the values of coefficient of drag are higher as compared with that of Chart-1 which is responsible for reducing the aerodynamic efficiency. Although the coefficient of lift is increasing with angle of attack, the net effect is reduced efficiency for higher angles of attack. Hence another parameter is considered in order to know the performance i.e. lift to drag ratio also known as coefficient of performance.

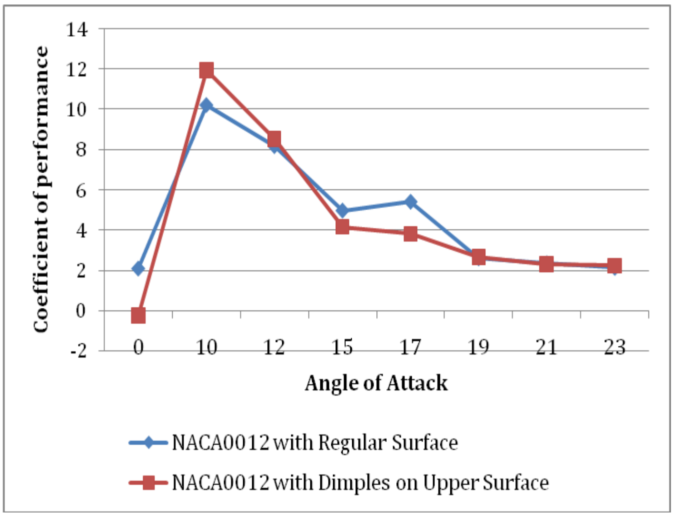

Chart-3: Coefficient of performance Vs angles of attack

Chart-3 shows the coefficient of performance for NACA0012 aerofoil with regular surface and dimples for different angles of attack. The coefficient of performance graph clearly indicates that NACA0012 aerofoil with dimples on upper surface have higher value as compared to NACA0012 aerofoil with regular surface at 10^0 angle of attack.

4.1 VELOCITY CONTOUR PLOTS

The velocity contour plots have been drawn for NACA0012 for various angles of attack (AOA). Fig-5 to Fig-8 shows the velocity contour plot for NACA0012 with regular surface at velocity of 6 m/s.

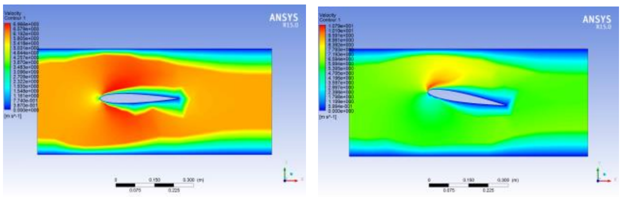

Fig-5: Velocity contour plot at 0 & 10 deg. AOA

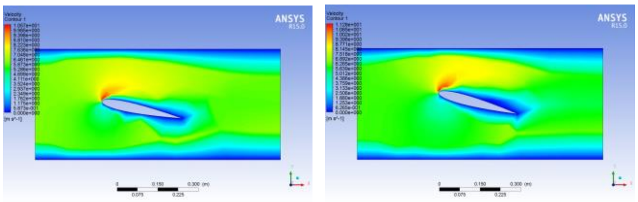

Fig-6: Velocity contour plot at 12 & 15 deg. AOA

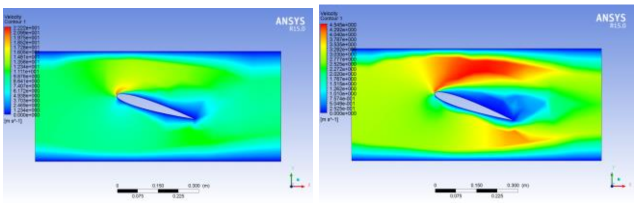

Fig-7: Velocity contour plot at 17 & 19 deg. AOA

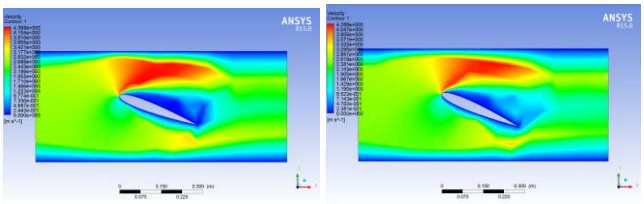

Fig-8: Velocity contour plot at 21 deg & 23 deg AOA

Fig-9 to Fig-12 shows the velocity contour plot for NACA0012 aerofoil with dimples on upper surface at velocity of 6 m/s.

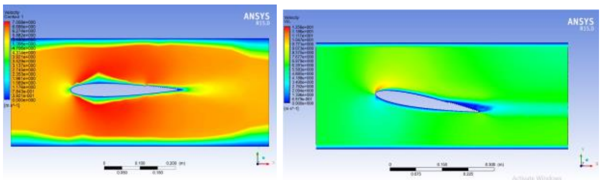

Fig-9: Velocity contour plot for 0 & 10 deg. AOA

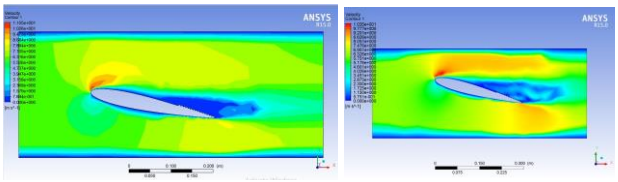

Fig-10: Velocity contour plot for 12 & 15 deg. AOA

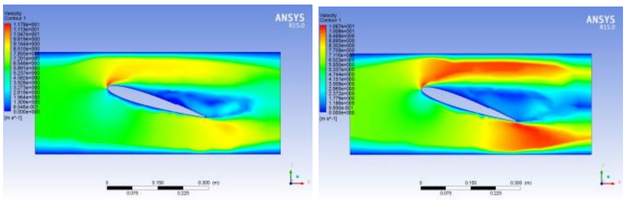

Fig-11: Velocity contour plot for 17 & 19 deg. AOA

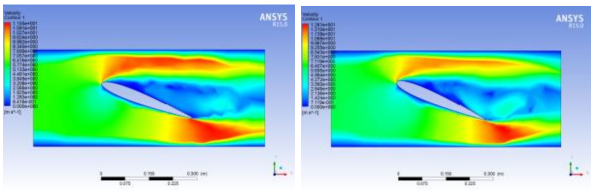

Fig-12: Velocity contour plot at 21 & 23 deg. AOA

The velocity contour plot clearly indicates that, for NACA0012 aerofoil with dimples on upper surface has the higher velocity regions over upper surface of aerofoil as compared to aerofoil with regular surface for higher angles of attack. Hence for aerofoil with dimples on upper surface has a low pressure region result in high lift coefficient.

5. CONCLUSIONS

In this paper, NACA0012 models with regular surface and with dimples on upper surface were analyzed for lift and drag performance at velocity of 6 m/s. The size of dimple was considered as 1% of chord length. The numerical analysis shows that

1. The addition of dimples increases the value of coefficient of lift but at the same time it also increases the coefficient of drag with increase in angles of attack.
2. The lift to drag ratio has shown higher value for aerofoil with dimple on upper surface as compared to regular surface aerofoil.
3. Addition of outward dimples over upper surface of aerofoil results in increased aerodynamic performance.
4. The addition of circular dimples over upper surface has produced high velocity region in vicinity of upper surface as compared to regular surface model.

REFERENCES

[1] Hui Hu and Zifeng Yang "An Experimental Study of the Laminar Flow Separation on a Low-Reynolds-Number Aerofoil," Journal of Fluids Engineering, ASME, 2008, Vol. 130 / 051101-1
[2] Mohammad Mashud, A. Al Bari and T. P. Bhowmick, "Experimental Investigation on Fluid Flow Separation Control," International Conference on Mechanical Engineering, 2009
[3] P. D. Gall, W. W. Huebsch and A.P. Rothmayer, "Dynamic Roughness as a Means of Leading edge separation flow control", 27th International Congress of The Aeronautical Sciences, 2010
[4] Deepanshu Srivastav, "Flow Control over Aerofoils using Different Shaped Dimples ", IPCSIT, IACSIT Press, Singapore, vol.33, 2012
[5] Ji Yaoa, et al. "Numerical simulation of aerodynamic performance for two dimensional wind turbine aerofoils" Procedia Engineering, 2012, 80 –86
[6] Shantanu S. Bhat, Raghuraman N. Govardhan, "Stall flutter of NACA0012 aerofoil at low Reynolds numbers," Journal of Fluids and Structures, 41, 2013, 166–174
[7] Lei Juanmian , Guo Feng, Huang Can, "Numerical study of separation on the trailing edge of a symmetrical aerofoil at a low Reynolds number," Chinese Journal of Aeronautics, 2013,26(4): 918–925
[8] Dhiliban, P. et al. "Aerodynamic Performance Of Rear Roughness Aerofoils," The Eighth Asia-Pacific Conference on Wind Engineering, December 10–14, 2013, Chennai, India
[9] P. Ghos, Akhilesh. K. Dewangan, Pratik Mitra & Dr. A. K. Rout, "Experimental study of Aerofoil with Wind Tunnel Setup", Research Gate, Conference Paper, January 2014
[10] Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, "CFD Analysis of an Aerofoil", International Journal of Engineering Research, Volume No.3, Issue No.3, pp : 154-158, 2014
[11] Agrim Sareen, Robert W. Deters, Steven P. Henry, "Drag Reduction Using Riblet Film Applied to Aerofoil", Journal of Solar Energy Engineering ASME MAY 2014, Vol. 136 / 021007
[12] Syed Hasib Akhter Faruqui, Md. Abdullah AlBari, MdEmran, Ahsan Ferdaus, "Numerical analysis of role of bumpy surface to control the flow separation of an aerofoil.", Procedia Engineering 90, 2014, 255 – 260
[13] Izzet Şahin and Adem Acir, "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, February 2015
[14] Bhushan Patil, Hitesh Thakare, "CFD Analysis of wind turbine blades at various AOA and Low Re ", Procedia Engineering 127 (2015) 1363 – 1369
[15] Anshuman Yadav et al., "Design, development &fabrication of Horizontal Axis Wind Turbine", 2016 IEEE Students' Conference on Electrical, Electronics and Computer Science,2016
[16] Ankan Dash, "CFD Analysis of Wind Turbine Airfoil at Various Angles of Attack", IOSR Journal of Mechanical and Civil Engineering, e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 13, Issue 4 Ver. II, Jul. - Aug. 2016.