DRAG REDUCTION OF WIND TURBINE BLADE TO ENHANCE AERODYNAMIC PERFORMANCE: A NUMERICAL STUDY

Mahesh Semwal, Rohit Tripathi, Neeraj Sati, Vinay Rana, Kuldeep Singh Rawat

Abstract— The scope of present work is mainly on investigating passive flow control techniques to provide wind turbine developer with efficient ways to make them to improve the energy that can be utilised by a wind turbine. In present work we aimed on searching the way of improve the aerodynamic performance of HAWT blade aerofoil. The wind energy is recognised as one of the most prominent energetic variants of the future because the wind resources are easily available. So from here we can conclude that research in this field must be necessary and interdisciplinary. The objective of this investigation is to enhance the wind turbines overall performance and increase their working capacity by doing complete flow or partial flow attachment. By doing CFD analysis for a wind turbine blade, present investigation is focused to analyse the flow around the wind turbine blade. Current efforts focus on increasing the aerodynamic efficiency of HAWT blade having S820 aerofoil using passive flow separation control technique.

Index Terms— Drag and Lift, Wind turbine, Flow separation control techniques, Aerofoil.

---

1 INTRODUCTION

Present work is focused on investigating flow control technique for horizontal axis wind turbine blade. S820 aerofoil is used for design and analysis of turbine blade. CFD analysis of aerofoil is done using ANSYS Fluent 15. CFD analysis for a wind turbine blade is done to analyse the flow around blade. In this work we aimed on exploring the method of increasing the performance of blade. Different techniques to achieve this have been shown to enhance the aerodynamics of wind turbine. Two main factors are calculated here to consider the effectiveness of an aerofoil, the lift and drag ratio and the lift coefficient. Total effect of viscous forces and negative pressure gradient results in a reverse of the flow next to the surface, which causes separation of the flow from the surface.

Flow separation is a major problem in the field of fluid mechanics. To overcome this researcher introduced many techniques. Any process which the boundary layer of a fluid flow is behaves different when the development of flow is naturally along the smooth surface. Methods of flow control to achieve transition delay, drag reduction, delay in separation, lift enhancement, etc. Boundary layer flow separation control on an aerofoil is to achieve maximum lift coefficient and minimum drag coefficient and consequently. Boundary layer controls are further divided: passive flow control, requiring no external power, and active flow control, requiring external energy expenditure.

On the skin friction drag reduction in large wind turbines using sharp V-grooved riblets. By (Leonardo P. Chamorro, Roger Arndt and Fotis Sotiropoulos) in this work they are trying to explore the performance of riblets as a drag reduction technique for wind turbine blades.

Experimental work carried out to understand the behaviour of riblets and to study the turbulent flow interaction near the wall region. In this research, wind tunnel experiments were performed to find out the drag reduction in a wind turbine aerofoil using V-groove riblet structures. In this research, wind tunnel experiments were performed to calculate the drag in a blade aerofoil using V-groove riblets. Comparison is made between CFD results and the laboratory experiments and to see the real situation.

Numerical Study of Flow Separation Control over a NACA2415 aerofoil.

By (M. Tahar Bouzaher) In this work numerical simulation of the flow around a NACA2415 aerofoil is done at an angle of 180 A rod is used to control the flow separation, upstream of the leading edge- in vertical translation movement so that it enhance the boundary layer transition. The problem is analysed by ANSYS FLUENT 13 version. The movement of rod is reproduced using the dynamic mesh method. The variation in frequency from 70 to 400 Hz and the considered amplitudes are 3%, and 5% of the chord. Frequency which is chosen is that it is equal to the frequency of separation. The results showed a maximum reduction of 61% and improve the flow behaviour.

By (U.Anand, Y.Sudhakar, R. Thileepanragu,V.T. Gopinathan, R. Rajasekar) They done Numerical analysis of turbulent flow over a NACA0012 aerofoil. Analysis is being done at different angle of attack. The 3D Reynolds average Navier-Stokes equations in addition with equations of Spalart-Allmaras turbulence model which are solved using CFD code FLUENT.

Vortex generators consist of an array of small vanes which is

- *Mahesh Semwal currently pursuing B.Tech degree program in in department of mechanical engineering in shivalik college of Engineering, Dehradun,Uttarakhand, India.PH-+91-7060324631, E-mail: maheshsemwal123@gmail.com*
- *Rohit Tripathi currently pursuing B.Tech degree program in in de-partment of mechanical engineering in shivalik college of Engineering, Dehradun,Uttarakhand, India.PH-+91-8057839501, E-mail:*
- *Neeraj Sati currently pursuing B.Tech degree program in in de-partment of mechanical engineering in shivalik college of Engineering, Dehradun,Uttarakhand, India.PH-+91-8266945245, E-mail: neerajsati92@gmail.com*
- *Vinay Rana currently pursuing B.Tech degree program in in de-partment of mechanical engineering in shivalik college of Engineering, Dehradun,Uttarakhand, India.PH-+91-8650144671, E-mail: ranavinay225@gmail.com*
- *Kuldeep Rawat currently working in department of mechanical engineering in shivalik college of Engineering, Dehradun,uttarakhand, India,PH-+91-999797490, E-mail: kuldeeprawat@outlook.in*

attached perpendicularly above the surface of suction of the blades. Vortex generator increases the ability of the fluid so that it sticks to the blade surface at high angle of attack. Flow separation control above a NACA0012 aerofoil using vortex generators is analysed by performing numerical simulations. The influence of Vortex generators on bulk quantities of the flow (CL and CD) is studied and the flow field modifications of flow field are discussed.

By (Andreea BOBONEA, Mihai Leonida NICULESCU and Corneliu BERBENTE) they carried out research on active flow control with the help of blowing devices with either constant or pulsed jets. In addition with high-stored momentum air between slots in the boundary layer, they postpone separation and overcome adverse

Pressure gradients. Pulsed blowing sends short pulses in spite of a continuous jet of fluid under the boundary layer and which are significantly effective. Experimental investigations and 2D numerical simulations of active manipulation in separated flows over aerofoils has been the aim of so many researchers.

fluid domain was created around the aerofoil, length= 3000 mm, width 2500 mm.

ANSYS design modeller is used to create the geometry.

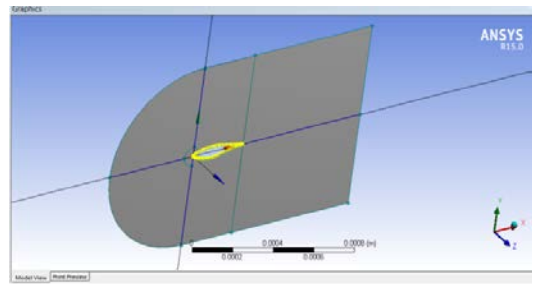


Fig: 1.2 (S820 Aerofoil without slot)

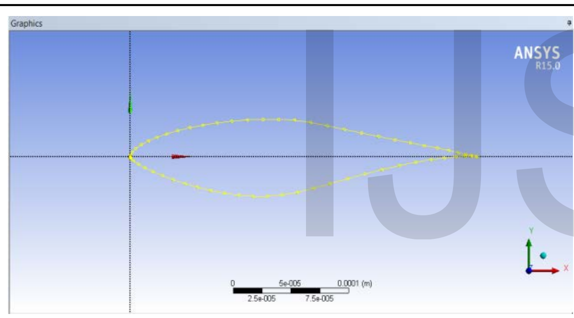


Fig: 1.1 (S820 Aerofoil)

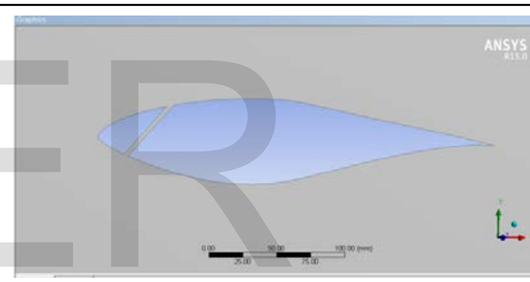


Fig: 1.3 (S820 Aerofoil with slot)

2. Methodology

To control the flow separation from top surface of the blade a hole is provided from the leading edge side of the blade, from which jet of air is passed from high pressure region to low pressure region. This will give energy to the boundary layer and delay the separation.

ANSYS Fluent 15 is used as CFD code to simulate flow field around S820 aerofoil.

For the analysis, two different geometry of aerofoil was used, the first geometry used was aerofoil with slot and the other one was aerofoil 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.

The geometry created for analysis having two different types of shapes, Aerofoil S820 with leading edge slot and without slot. Slot in aerofoil 20% from the leading edge side. Sketch for aerofoil S820 is generated through java programme. This java programme was made using co-ordinates given for S820 aerofoil by NREL (National Renewable Energy Ltd.). A

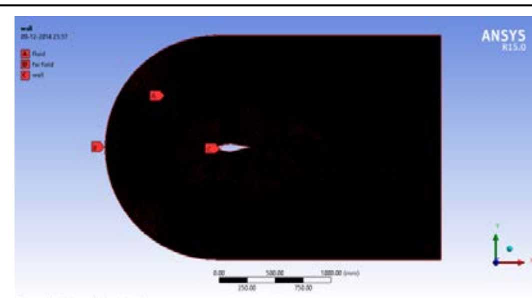
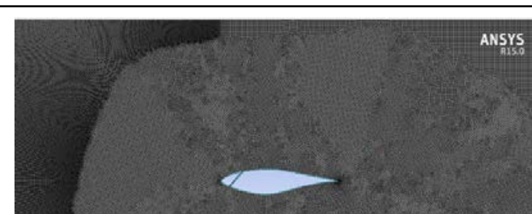


Fig: 1.4 (Fluid domain for analysis)



ISSN 2229-5518

3. Mesh Details

Meshing or discretization of fluid domain is done using ANSYS fluent 15.

Following parameters was taken during meshing.

Length of computational Domain- 3000 mm

Width of computational Domain- 2000 mm

No of Nodes- 250000

No. of Elements- 249000

Type of Mesh- Quadrilateral

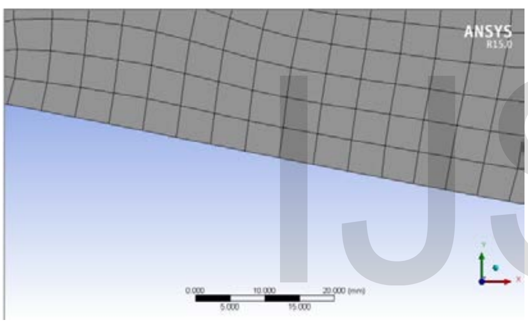


Fig: 1.6 Quadrilateral mesh generated in ANSYS Fluent

4. Analysis and Solution

The governing equations were solved using the commercial CFD package FLUENT with the following simplifying conditions:

Solver- Density based solver

Equation used- k-ω, SST

Fluid- Air

Parameters monitor- Coefficient of drag and lift

2D analysis of aerofoil is done using above parameter. Analysis is done at different angle of attack and results are obtained.

5. Results and Discussion

Analysis has been carried out in ANSYS 15.0 FLUENT by changing angle of attack on S820aerofoil with slot and without slot. By using leading edge slot upper layer of boundary layer is going to energize and try to pull down flow separation towards trailing edge.

Results obtained from S820 aerofoil without slot will be compared with S820 aerofoil having leading edge slot to see the changes in drag and lift and hence improvement in aerodynamic performance. The results obtained at different angle of attack from 0˚ to 30˚.

5. Results and Discussion

Analysis has been carried out in ANSYS 15.0 FLUENT by changing angle of attack on S820aerofoil with slot and without slot. By using leading edge slot upper layer of boundary layer is going to energize and try to pull down flow separation towards trailing edge.

Results obtained from S820 aerofoil without slot will be compared with S820 aerofoil having leading edge slot to see the changes in drag and lift and hence improvement in aerodynamic performance. The results obtained at different angle of attack from 0˚ to 30˚.

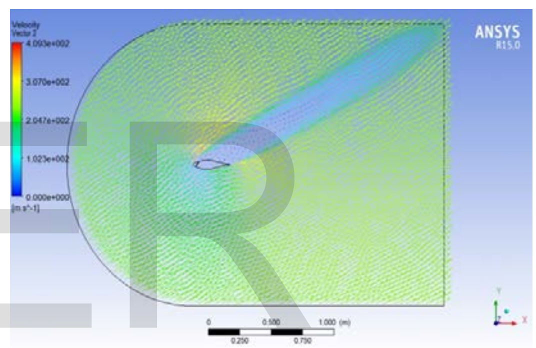


Fig: 1.7 (Velocity vectors of S820 aerofoil with slot at AOA 30o)

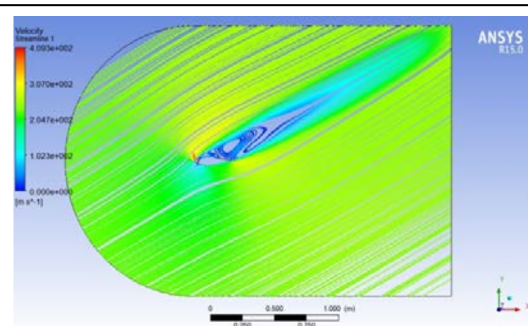
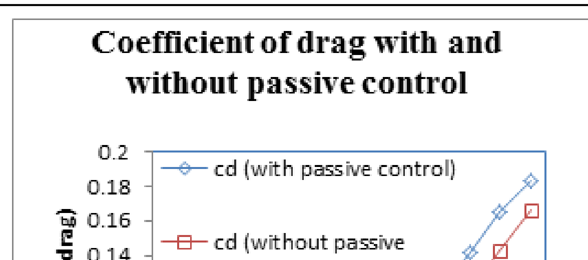


Fig: 1.8 (Streamline of S820 aerofoil with slot at AOA 30o)



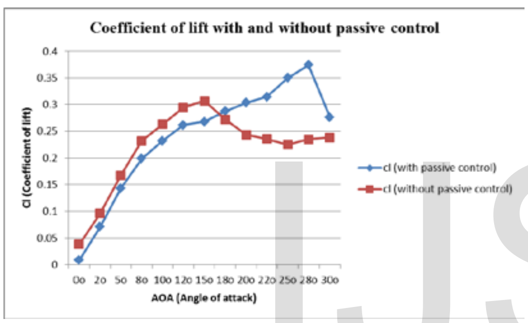


Fig: 1.10 (Coefficient of lift for aerofoil with and without passive control)

CONCLUSION

To utilise maximum energy from wind, it is mandatory to make turbine blade more effective. Flow separation is a serious problem in wind turbine blades, to overcome this we need to make blades more aerodynamic.

By inducing velocity jet at the trailing edge, the flow is nicely attached to the profile or the boundary layer is delay.

In future different techniques especially passive flow control techniques can be used to enhance the aerodynamic performance of wind turbines.

Following results are obtained from analysis:

1. For given parameter and physics stall condition for aerofoil without slot occur at 15o.
2. Maximum lift for aerofoil without slot occurs at 15o.
3. Slotted aerofoil gives higher lift and lower drag then aerofoil without slot.
4. Stall condition occur at higher angle of attack in slotted aerofoil compare to aerofoil without slot.
5. Stall angle in slotted aerofoil is 28o, which shows improvement in lift force.

REFERENCE

[1] Numerical investigation of flow control over an aerofoil using synthetic jets and its optimization .International Aerospace Conference, Turkey (2009). E. Akcayoz and I. H. Tuncer

[2] Numerical investigation of leading edge blowing and optimization of the slot geometry for a circulation control aerofoil, Notes on Numerical Fluid Mechanics and Multidisciplinary Design.112 (2010) 183-190. C. Jensch, K. C. Pfingsten and R. Radespiel

[3] Drag minimization using active and passive flow control techniques. Aerospace Science and Technology, 17 (1) (2012) 21-31. B. Yagiz, O. Kandil and Y. V. Pehlivanoglu

[4] 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. M. Goodarzi, R. Fereidouni and M. Rahim

[5] Experimental study of aerofoil performance with vortex generators. Bragg, M.B., Gregorek, G.M., 1987

[6] Aerodynamics of Wind Turbines, China Power Press, Beijing (2009), Hansen, M. O. L.

[7] Numerical Investigation of Flow Control Over an aerofoil Using Synthetic Jets and its Optimization, International Aerospace Conference, Turkey, 2009 E. Akcayoz and I.H. Tuncer

[8] Investigation of Boundary layer Suction on a Wind Turbine aerofoil using CFD, Master Thesis, Technical University of Denmark, Denmark, 2010 A.T. Piperas

[9] Drag Minimization Using Active and Passive Flow Control Techniques, Aerospace Science and Technology, Vol. 17, pp. 21-31, 2011 B Yagiz, O. Kandil and Y. V. Pehlivanoglu

[10] Separation control by tangential blowing inside the bubble. Exp. Fluids 29: 96–102 Viswanath P R, Ramesh G, Madhavan K T

[11]A review of recent developments in flow control. Aeronautics J.109: 205–232, Ashill P R, Fulker J L, and Hackett K C 2005

[12] Modern developments in flow control. Appl. Mech. Rev. 48: 365–379, Gad-el-Hak M 1996

[13] Boundary-layer and flow control (Oxford: Pergamon) vol. 1 & 2

[14] Injection slot location for boundary layer control in shock-induced separation. Viswanath P R, Sankaran L, Sagdeo P M, Narasimha R, Prabhu A 198,

[15] Separation control by tangential blowing inside the bubble. Exp. Fluids 29: 96–102, Viswanath P R, Ramesh G, Madhavan K T

[16] Passive flow control over NACA 0012 aerofoil using vortex generator.U. Anand

[17] Numerical study of flow separation control over a NACA 2415 aerofoil. M. Tahar Bouzaher

[18] On the skin friction drag reduction in large wind turbines using sharp V-grooved riblets. Leonardo P. Chamorro, Roger Arndt and Fotis Sotiropoulos

[19] Lift enhancement of aerofoil and tip flow control for wind turbine.Ya-lei BAI, Xing-yu MA, Xiao MING
[20] Numerical study of blowing and suction slot geometry optimization on NACA 0012 aerofoil. Kianoosh Yousefi, Reza Saleh and Peyman Zahedi