IJCRT.ORG ISSN : 2320-2882
INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

CFD Analysis Carried out on Wind Turbine Blade for Different Angles of Attack and Various Reynolds Number

Subhodh S Ghadge[1], Nandakishor H Murhekar[2] , Amol L Khatode[3]
[1](Mechanical Engg. Dept, SGOI,COE,Belhe,Pune,India)
[2](Mechanical Engg. Dept, Samarth polytechnic,Belhe,Pune,India)
[3](Mechanical Engg. Dept, SGOI,COE,Belhe,Pune,India)

Abstract:
Background*: In this research work an attempt is made to find out forces on Lift and Drag by varying Reynolds number and angle of attack of wind turbine blade. For this purpose airfoil profile of NACA 0012 is considered for doing analysis of wind turbine blade . The forces on Lift and Drag are calculated at various angle of attack varying from 0^o to 80^o and Reynolds number from 10,000 to 800000 by Computational Fluid Dynamics (CFD) analysis. The validations of the work carried out is done by comparing the results obtained from analysis with that of experimental results which are obtained from the report of Sandia National Laboratories (SNL). It is found that results obtained by present CFD analysis are comparatively very closely matching with results published by SNL energy report.With the validated CFD method, the investigation of "C_L and C_D values" for lesser Reynolds number, 1000 and 5000, has been carried out. The values of C_L and C_D at lower Reynolds number is much important in analysis of Darrieus and horizontal axis type of wind turbine blade, still these values are not available in the literature that is reviewed . From CFD analysis it can be concluded that as Reynolds number increase lift forces and drag forces increases. NACA 0012 provides maximum lift and drag at higher Reynolds number.*
Materials and Methods*: In the analysis of work carried out, C type mesh and two way velocity inlet method is used. The pressure based implicit steady solver with Standard k-s model turbulence model with PRESTO second order upwind scheme is used.*
Results*: Initially CFD analysis is carried out for Reynolds number = 10000 and velocity inlet =0.17080 m/s. Map meshed is used for simulation, close view of map mesh used near the aerofoil .The obtained results are plotted as in for angle of attack 10^0, 50^0 and 70^0 respectively on velocity contour around aerofoil.*
Conclusion:*Validation of the present analysis is carried out for six different Reynolds number from 10, 000 to 800000 and with eight different angles of attack 10^o to 80^o. It is found out to be close matching in CL and CD values are obtained in the present CFD analysis in comparison with experimentally available values, which validates the present methodology of CFD analysis.*
It is also found that result of CFD analysis for C_L shows some deviations with experimental results Sandia National Laboratories energy report for lower angle of attack, however for higher angle of attack its shows close match with experimental results of Sandia National Laboratories energy report. It is also found that result of CFD analysis for C_D shows very close and good match with results of Sandia National Laboratories energy report.
The investigation for C_L& C_D values are carried out for Reynolds number 1000 & 5000 with validated CFD methodology. From present investigation it is concluded that Reynolds number 5000 provides maximum lift coefficient and drag coefficient than Reynolds number 1000.In general, it can be concluded that as Reynolds number increase lift forces and drag forces increases. Both, CFD and experimental results indicates that, NACA 0012 provides maximum lift and drag at higher Reynolds number.
Key Word*: aerofoil, angle of attack,reynolds number.*

I. Introduction

Selection of aerofoil blade shape is one of the most important phase of designing a wind turbine because blade is the one responsible for conversion of kinetic energy into mechanical energy. Aerodynamics deals with studying physical laws of the objects behavior in airflow and the forces that are produced as a result of air flows. In the earlier stage,the research on design of wind turbine blade was limited on theoretical study, field testing and wind tunnel testing which need a lot of efforts and resources.With the development of computer aided design codes, which provide another way to design and analyzed the wind turbine blades. Performance of Aerodynamic of wind turbine blades can be analyzed using Computational Fluid Dynamics (CFD), which indeed is one of the branches of fluid mechanics. Aerodynamic studies are quite mature for flows with large Reynolds number. However, there are not much analytical, numerical or experimental studies available for flows at very low Reynolds numbers. In order to extract the maximum kinetic energy from wind, researchers put much effort on the designing a good geometry of blade .Consideration of 2D geometries, i.e., airfoils, for such studies seems to be a good starting point to improve understanding of low Reynolds number flows. In this regard, some experimental studies are reported in Schmitz [1] Sunada et al.[2], Sunada et al.[3], Nazmul [4]. Numerical studies, for analysis as well as for design and experimental validations of airfoils at ultra low Reynolds number are presented in Kroo and Kunz, 2000, Kunz and Kroo,[5,6]. The effects of aerofoil profile modification on a vertical axis wind turbine performance are presented in by Ismail and vijayrahgavan [7]. The effect of Blockage-tolerant wind tunnel measurements for a NACA 0012 at high angles attack is presented by Rain bird and Peiro [8]. From literature survey, it is observed that, study of lift and drag forces on aerofoil, for very low Reynolds number at angle of attack is not explode more. So, in present work an attempt is made to study the Lift and Drag forces on a wind turbine blade for low Reynolds number and different angle of attack. In present work NACA 0012 airfoil profile is considered for analysis of wind turbine blade. The Lift and Drag forces are calculated at different angle of attack from 0º to 80º for Reynolds number from 10,000 to 800000 by computational fluid dynamics analysis.The evaluation for the work is done by comparing experimental results obtained.

II. Material And Methods

Geometry and Mesh Generation

In the analysis of work carried out, C type mesh and two way velocity inlet method is used. The close-up view of mesh and its boundary condition used for analysis is as shown in fig.2 and fig.1 respectively. For analysis purpose the pressure based implicit steady solver with Standard k-s model turbulence model with PRESTO second order upwind scheme is used.

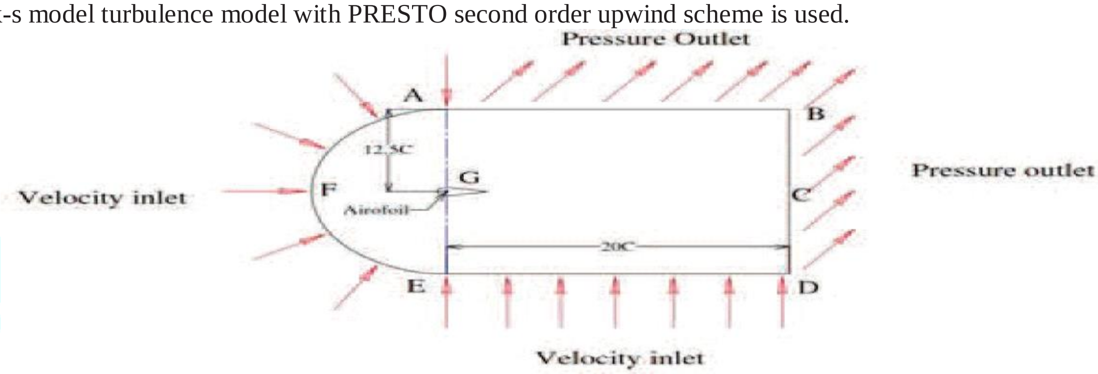


Fig.no. 1 Close-up view of Geometry with its boundary conditions.

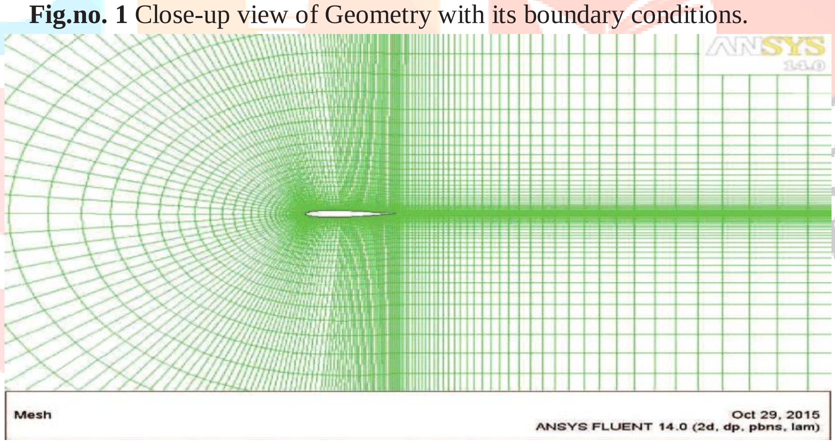


Fig. No.2 Close-up view of C type mesh

III. Result

Results and discussion

Initially CFD analysis is carried out for Reynolds number = 10000 and velocity inlet =0.17080 m/s. The geometric details are as in fig.no. 1. Map meshed is used for simulation, close view of map mesh used near the aerofoil is as shown in fig.no.2.The obtained results are plotted as in fig.no.3 , 4 and 5 for angle of attack 100, 500 and 700 respectively on velocity contour around aerofoil.

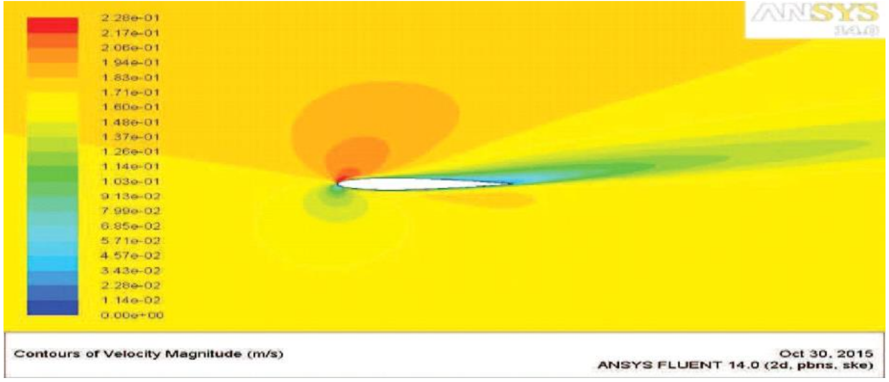
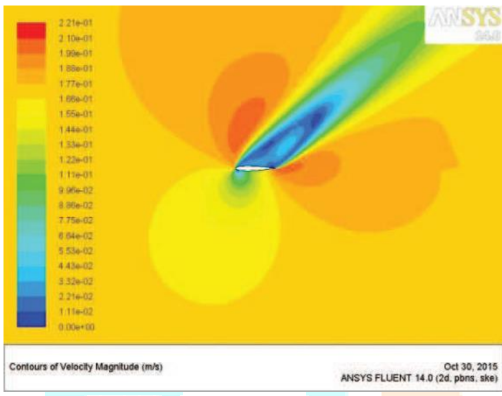


Fig. No.3. Angle of attack 10º close view



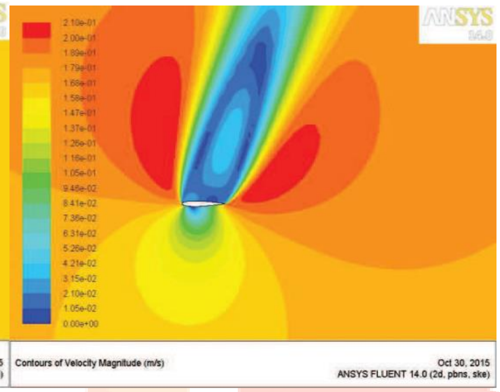
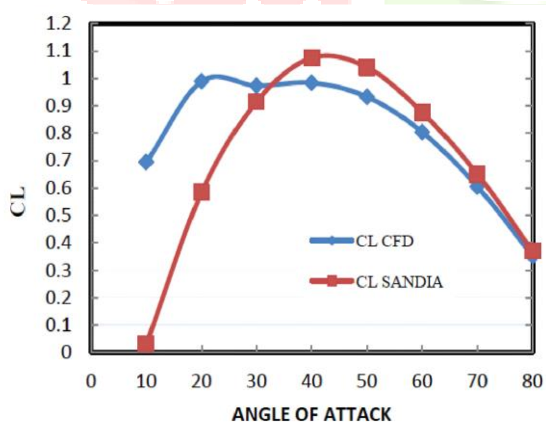


Fig.no. 4. Angle of attack 40º close view **Fig.no.5.** Angle of attack 70º close view

Validation of Analysis by Comparisons of Experimental Results with CFD Results

The results which are obtained by simulation is shown in following figures (fig.6 to fig. 17) for showing variation of lift coefficient with reference to angle of attack for various Reynolds number. The Reynolds number is varied by changing free stream velocity at inlet. The obtained results are validated with experimental results available with sandia national laboratory energy report. Results obtained by CFD simulation is closely matched with results available in sandia national laboratory, which validates the procedure and results obtained by CFD simulation



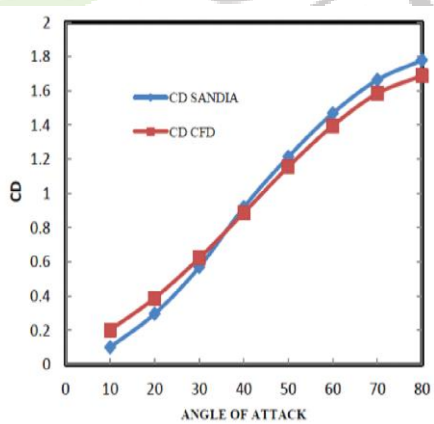


Fig.no. 6. C_L Vs AOA at RE =10000 **Fig.no. 7**. C_D Vs AOA at RE =10000

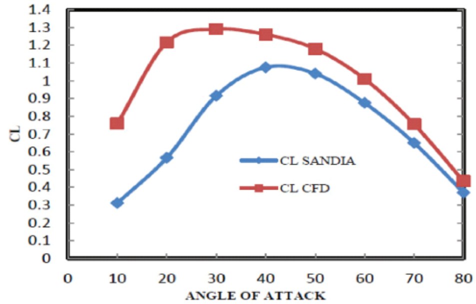


Fig.no. 8. C_L Vs AOA at RE =20000

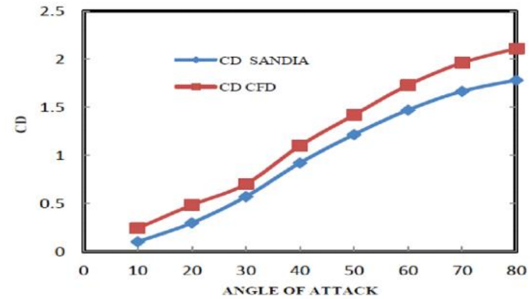


Fig.no. 9. C_D Vs AOA at RE =20000

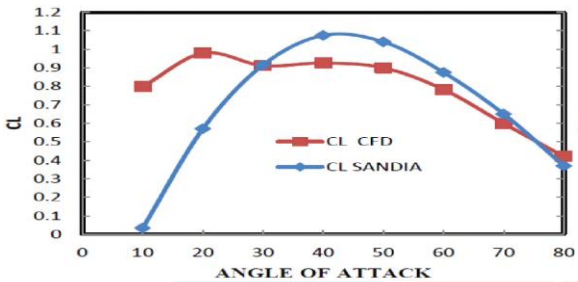


Fig.no. 10. C_L Vs AOA at RE =40000

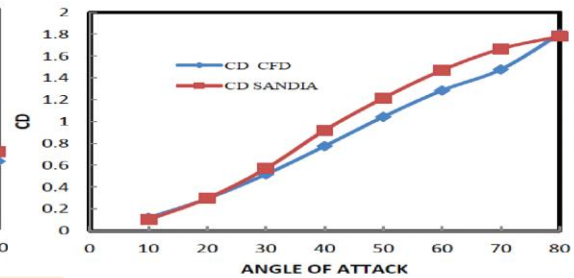


Fig.no. 11. C_D Vs AOA at RE =40000

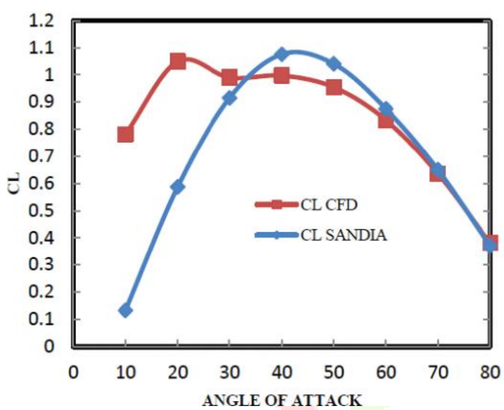


Fig.no. 12. C_L Vs AOA at RE =160000

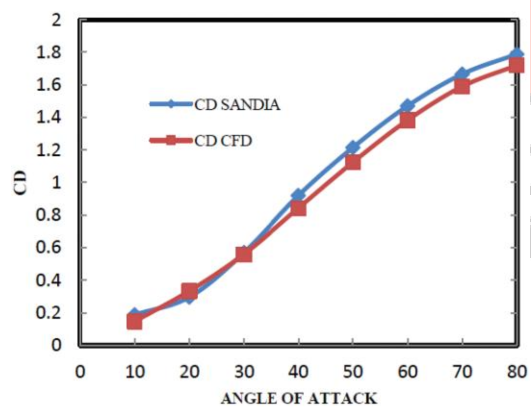


Fig.no. 13. C_D Vs AOA at RE =160000

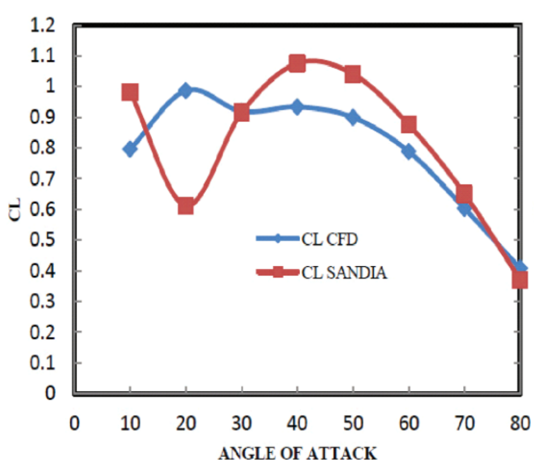


Fig.no. 14. C_L Vs AOA at RE =360000

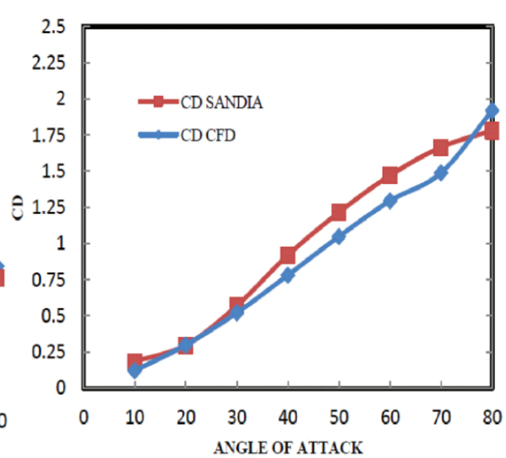


Fig.no. 15. C_D Vs AOA at RE =360000

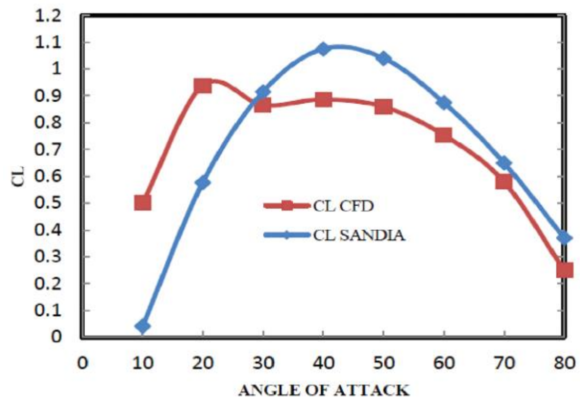


Fig.no. 16. C_L Vs AOA at RE =800000

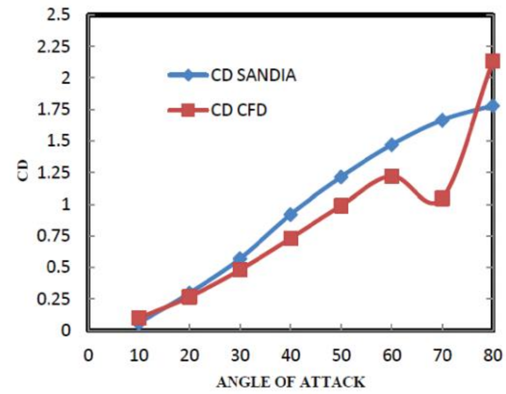


Fig.no. 17. C_D Vs AOA at RE =800000

From above graphs it can be conclude that, for lower values angle of attack (0^0 to 30^o), results obtained from CFD simulations indicates much diversified values compare to experimental values. However for angle of attack in between 30^o to 80^o the value of C_L matches closely with the experimental results of Sandia laboratory. However,C_D values obtained by CFD simulations closely matches with experimental results of Sandia laboratory. The sudden decrease in C_D value is obtained for Reynolds number 8, 00,000 by CFD simulations at some angle of attacks as shown in figures.The maximum value of C_L for CFD is obtained at angle of attack 20^o for all Reynolds number but it is maximum for angle of attack 20^o and 40^o at Reynolds number 10,000.Also the maximum values of C_L for CFD simulations are vary in between 0.95 to 1.3, but for Sandia it is constant 1.1.

Investigation of C_L and C_D for low Reynolds number

In Sandia National Laboratories energy reports values of C_L and C_D are given for higher Reynolds number from 10000 to 80000 and more than that but for Darrieus turbine and axial flow turbine, for some angle of attack and at specific location of blade, the Reynolds number becomes too low even less than 10000.There is no standard data available of C_L and C_D values for Reynolds number below 10000. In this concern, investigation has been performed for Reynolds number 1000 and 5000 for values C_L and C_D. The obtained results are shown in fig. 18 and fig. 19 for Reynolds number 1000 and 5000.

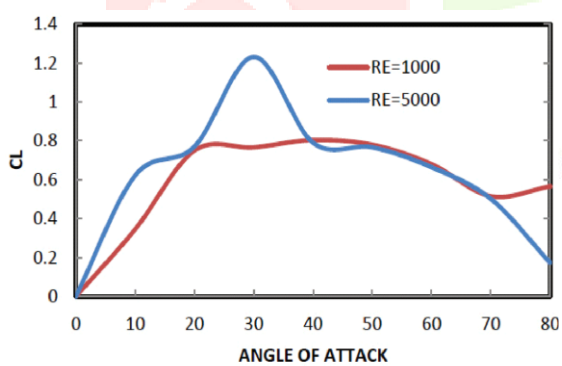


Fig.no. 18. Comparisons of C_L Vs AOA at RE =1000,5000

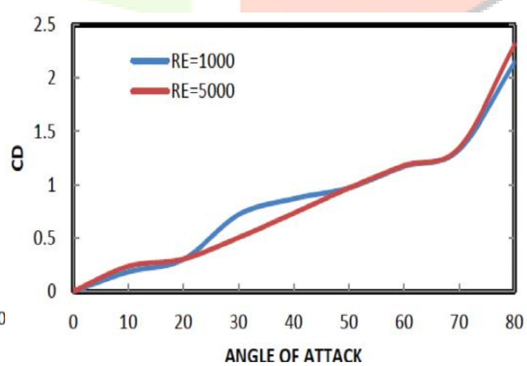


Fig.no. 19.Comparisons of C_D Vs AOA at RE =1000,5000

IV. Conclusion

- Validation of the present analysis is carried out for six different Reynolds number from 10, 000 to 800000 and with eight different angles of attack 10^o to 80^o. It is found out to be close matching in CL and CD values are obtained in the present CFD analysis in comparison with experimentally available values, which validates the present methodology of CFD analysis.
- It is also found that result of CFD analysis for C_L shows some deviations with experimental results Sandia National Laboratories energy report for lower angle of attack, however for higher angle of attack its shows close match with experimental results of Sandia National Laboratories energy report. It is also found that result of CFD analysis for C_D shows very close and good match with results of Sandia National Laboratories energy report.
- Theinvestigation for C_L& C_D values are carried out for Reynolds number 1000 & 5000 with validated CFD methodology. From present investigation it is concluded that Reynolds number 5000 provides maximum lift coefficient and drag coefficient than Reynolds number 1000.
- In general, it can be concluded that as Reynolds number increase lift forces and drag forces increases. Both, CFD and experimental results indicates that, NACA 0012 provides maximum lift and drag at higher Reynolds number.

References

[1]. Schmitz F. W. (1967 Aerodynamics of the model airplane. Part I. Airfoil measurements, NACA TM X-60976.

[2]. Sunada S., Sakaguchi A., Kawachi K. (1997). Airfoil section characteristics at a low Reynolds number. Journal of Fluids Engineering, 119, 129-135.

[3]. Sunada S., Yasuda T., Yasuda K., Kawachi K., (2002) Comparison of wing characteristics at an ultralow Reynolds number. Journal of Aircraft, 39, 331-338.

[4]. Nazmul Haque, Mohammad Ali, Ismat Ara, "Experimental investigation on the performance of NACA 4412 aerofoil with curved leading edge plan form", Procedia Engineering 105 (2015) 232 – 240, Elsevier

[5]. Kroo I., Kunz P. J., (2000) Meso-scale ight and miniature rotorcraft development, Proceedings of the Conference on Fised, Flapping and Rotary ,Vehicles at very Low Reynolds Numbers, Univ. of Notre Dame, Notre Dame, IN, 184-196.

[6]. Kunz P. J., Kroo I., (2000) Analysis, design and testing of airfoils for use at ultra-low Reynolds numbers. Proceedings of the Conference on Fised, Flapping and Rotary Vehicles at very Low Reynolds Numbers, Univ. of Notre Dame, Notre Dame, IN, 349-372.

[7]. Md Farhad Ismail, Krishna Vijayaraghavan, "The effects of aerofoil profile modification on a vertical axis wind turbine performance", Energy 80 (2015) 20e31, Elsevier.

[8]. J.M. Rainbird, J. Peirσ, J.M.R. Graham, "Blockage-tolerant wind tunnel measurements for a NACA 0012 at high angles of attack", J. Wind Eng. Ind. Aerodyn. 145 (2015) 209–218, Elsevie