ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 4, August 2016

Effect of Turbulence Model and Wind Velocity on Aerodynamic Performance of Wind Turbine Blade

[1]Saurabh V. Patil, [2] Dr. Shailesh Nikam
[1] M.Tech, [2] Asst. Professor
K. J. Somaiya College of Engineering, Mumbai

Abstract: **-- Present paper reports, effect of turbulence model and wind velocity on aerodynamic performance of an airfoil, NERL S809. The computational simulations are done by using RANS steady equations. Among four turbulence model (standard k–ε, Spalart–Allmaras, k–ω and k–ω SST) the best model has been selected on the basis of comparison with experimental results from available literature. The pressure coefficient, drag coefficient and lift coefficient are compared at different angle of attack, wind velocity using different solver. This computational simulation is carried out using Ansys-Fluent (14.0) software. The accurate aerodynamic load acting on blade of wind turbine is obtained by using, k–ω SST turbulence model for unsteady flow behavior.**

Index Terms**— Flow over an airfoil, wind velocity, pressure coefficient, Angle of attack, CFD analysis.**

I. INTRODUCTION

Aerodynamic performance of wind turbine blade plays an important role for deciding overall performance of the wind turbine. The parameters deciding the aerodynamic performance of the blade are wind velocity, angle of attack and blade geometry [4]. As experimental study is very costly and time consuming for different iteration, computational tool can be used very effectively [5] to optimize these parameters and to maximize wind turbine performance. Selection of turbulence model [6], [7] for unsteady flow analysis is important aspect for accurate prediction of the result. Turbulence model available for the computational analysis are, k-ε standard, k-ω standard, k-ω sst and Spalart–Allmaras.The prediction of dynamic stall of wind turbine blade for different turbulence model around a rotor blade of horizontal axis wind turbine (HAWT) is aim of the present work.

II. METHODOLOGY

Computational method is used for aerodynamic analysis of an airfoil for different turbulence modal and different wind velocity. Analysis is done using ANSYS-FLUENT (14.0) software. The finite volume method is used in FLUENT, as it gives large generality in its formalism. It has conservative treatment which is adaptive for physical problem.

A. Geometry & Computational domain

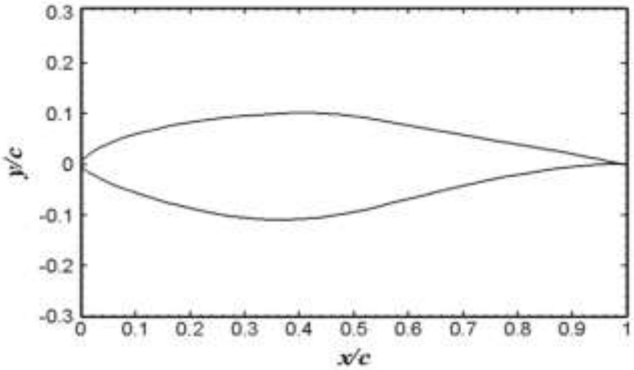

Fig.1 Geometry of NERL S809 Airfoil

The NERL (national renewable energy laboratory) s809 airfoil is selected for present study having chord length of 1m. The thickness of airfoil is 21% of chord length. The detailed geometry of airfoil is shown in fig.1. The computational domain size is shown in fig.2. The surface of an airfoil is treated as a stationary wall with no slip shear. The velocity-inlet is given as inlet boundary condition and for exist boundary condition, pressure-outlet is given.

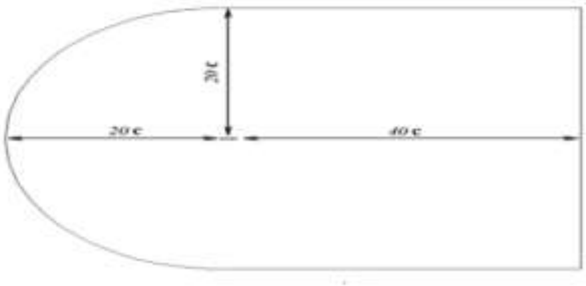

Fig.2 Computational domain

B. Mesh

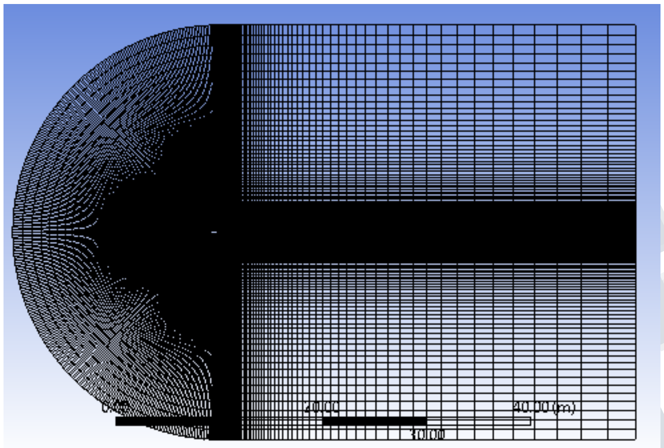

Fig.3 C-Type Structured Mesh

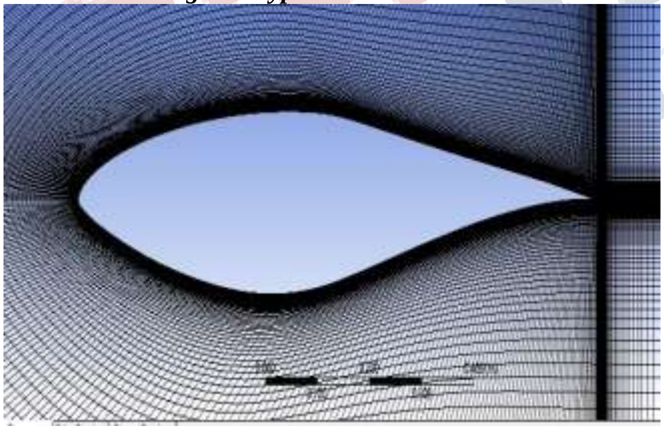

Fig.4 Grids Points Over An Airfoil

There are near about 90,000 quadrilateral cells consists in mesh geometry as shown in Fig.3. The C-type grid is used. This helps to capture various gradient acting in the boundary layer [3]. The C-type mesh helps to generate large number of grid points closer to airfoil surface as shown in fig.4. The wall functions model is used to adjust the thickness of cells nearer to blade surface. For k-ε standard model, 1×10-3 m value is used to satisfy y+ ≥30 wall function and for k–ω, k–ω SST and Spalart–Allmaras (S-A), 1×10-5 m value is used to satisfy y+ ≥30 wall function. This is done in order to satisfy y+≈ 1 as y+ is a characteristic dimensionless distance from wall [9].

C. Turbulence modeling

2D computational simulation is done on the NERL S809 airfoil for comparing the computational result. In present study, four turbulence models (i.e. k–ε, k–ω standard, k–ω SST and Spalart–Allmaras) were compared. The k–ε turbulence model is obtained from the Navier–Stokes equations by doing mathematical a technique which is known as „the Re-Normalization Group (RNG)" methods [1]. This RNG theory provides an effective viscosity for low Reynolds number effects [2]. In k–ω SST model, the SST stands for shear stress transport. The k–ω model is used for effective Reynolds number nearer to wall region and for far-field region k–ε model is used.

III. RESULT AND DISCUSSION

Computational results obtained for different model are compared with the experimental results from the literature to determine effective turbulence model in present computational setup. The wind velocity considered is 24.34m/s. Simulation is performed at two angles of attack i.e. 14.24° and 20.15° for pressure coefficient.

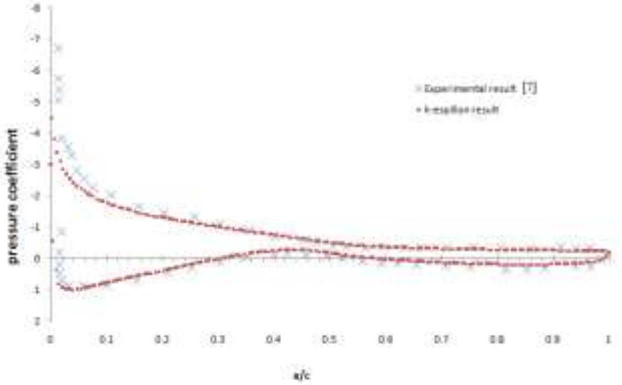

Fig.5 Cp Distribution Over An Airfoil At Angle Of Attack 14.24° For K-Epsilon Standard Turbulence Model.

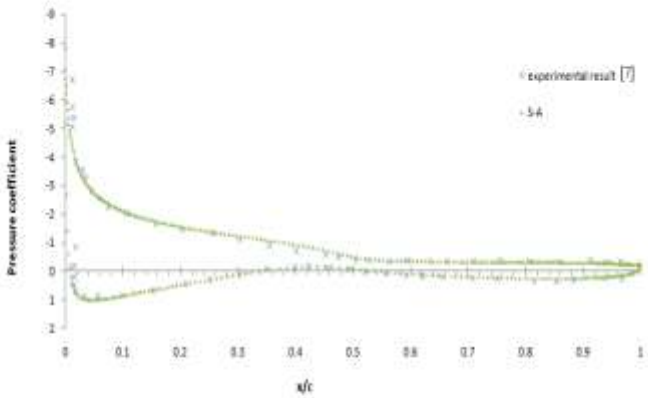

Fig.6 Cp Distributions over an airfoil at angle of attack 14.24° for S-A turbulence model

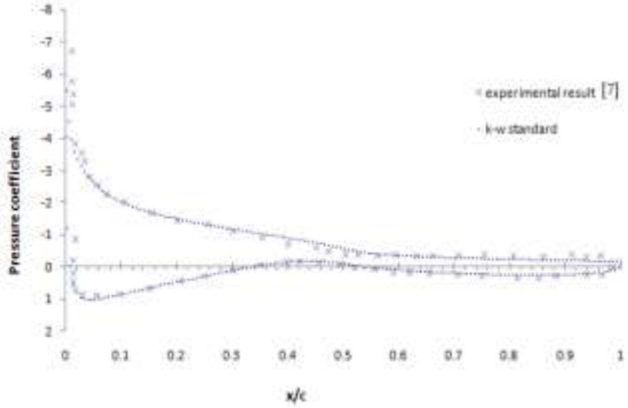

Fig.7 Cp Distributions over an airfoil at angle of attack 14.24° for k-ω standard turbulence model

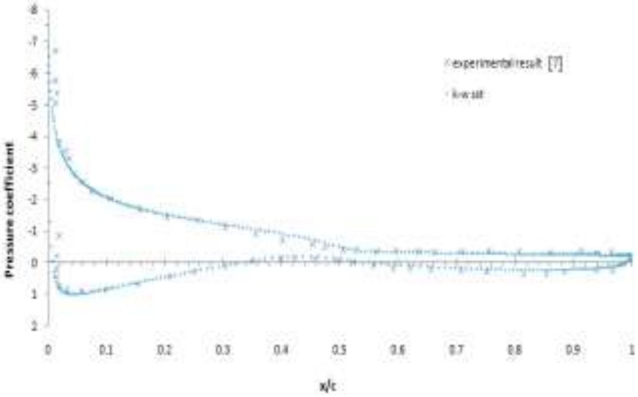

Fig.8 Cp Distribution over an airfoil at angle of attack 14.24° for k-ω sst turbulence model

The pressure coefficient (CP) is defined as**:**

$$C_p = \frac{P - P_\infty}{\frac{1}{2}\rho V_\infty^2}$$

Where, the static pressure of air is p∞, the local static pressure acting on the blade surface is p and the dynamic pressure of the free stream is . The pressure coefficient (cp) along the length of airfoil at 14.24° angle of attack for different turbulence model (standard k–ε, spalart-allmaras, k–ω and k–ω SST) is shown in fig.5, fig.6, fig.7 and fig.8. Similarly, at 20.15° angle of attack for different turbulence model pressure distribution is shown in fig.9, fig.10, fig.11 and fig.12 respectively. This cp curve represents changes in pressure at each position on of airfoil. As cp is dimensionless quantity, it has no unit. At 14.24° angle of attack, pressure coefficient over the airfoil surface for all four turbulence models show good agreement. Considering same angle of attack, the experimental results from literature shows that at approximately 45% of the chord length from suction side, the separation will occur. Whenever the pressure coefficient get flattens, it means that the boundary layer separation takes place, while the computational calculations predict the boundary layer separation slightly later. The fig.5 shows that for k–ε model, the separation point is at approximately 60% of chord length. Similarly, for spalart–allmaras, standard k–ω, k–ω sst model separation point is at approximately 55%, 60% and 50%of chord length as shown in fig.6, fig.7 and fig.8.

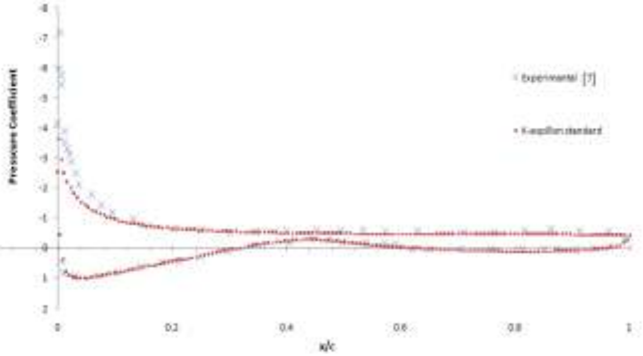

Fig.9 Cp Distribution over an airfoil at angle of attack 20.15° for k-epsilon standard turbulence model

At 20.15° angle of attack, there is early flow separation as compared to 14.24° (The flow is separated from upper most suction side). This is observed from pressure coefficient. The k–ω sst model shows separation at 25% of the chord length. But in case of k–ε, spalart–allmaras and standard k–ω the separation will occurs slightly later i.e. At 32%, 28% and 40% of chord length respectively. Hence the computational pressure distribution shows good experimental result for k–ω sst turbulence model as compare to other turbulence model.

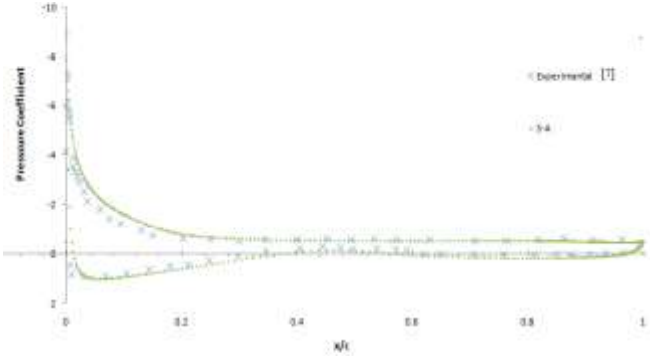

Fig.10 Cp Distribution over an airfoil at angle of attack 20.15° for S-A turbulence model

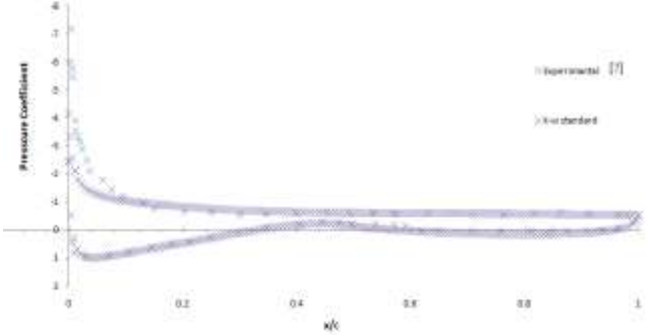

Fig.11 Cp Distribution over an airfoil at angle of attack 20.15° for k-ω standard turbulence model

The velocity contour is shown in Fig.13, Fig.14, Fig.15 and Fig.16 for four different turbulence models (i.e. standard k–ε, Spalart–Allmaras, k–ω and k–ω SST). This model shows in variation in velocity distribution over an airfoil. The dark blue color shows zero velocity. At 14.240, Angle of attack the stagnation point is shifted to lower side of an airfoil surface from suction side. The point is shifted at lower side of an airfoil as angle of attack is 14.24°.

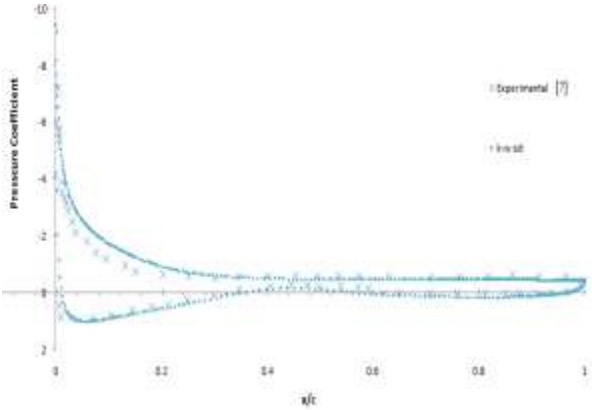

Fig.12 CP Distribution Over An Airfoil At Angle Of Attack 20.15° For K-Ω Sst Turbulence Model

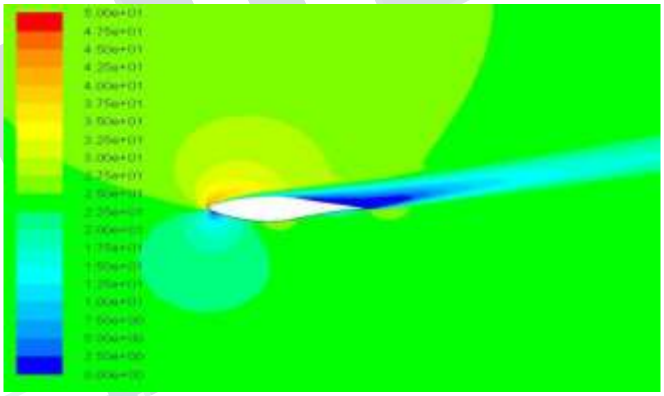

Fig.13 Contour of Velocity Magnitude for K-Epsilon Standard Turbulence Model At Angle Of Attack14.24°

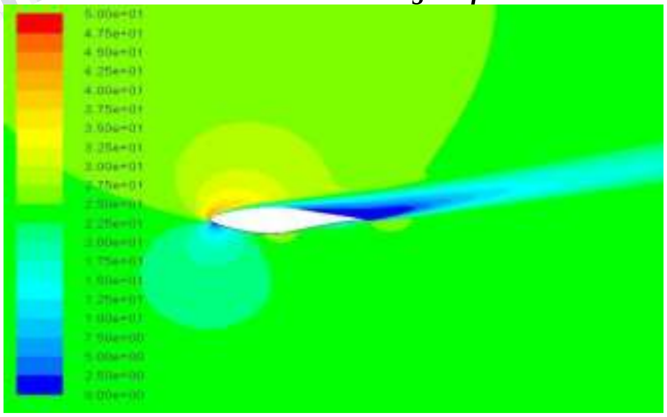

Fig.14 Contour of Velocity Magnitude for S-A Turbulence Model at Angle Of attack14.24°

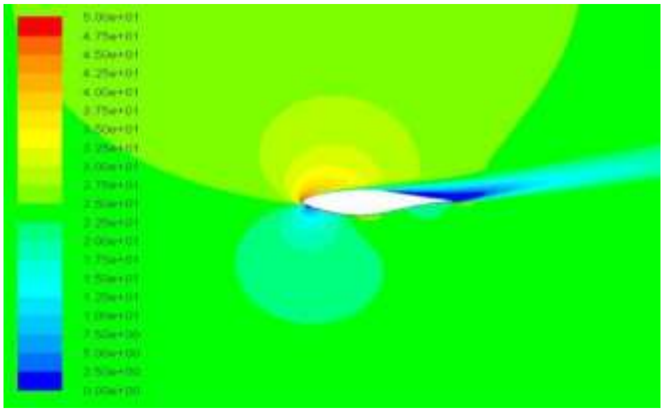

Fig.15 Contour Of Velocity Magnitude For K-Ω Standard Turbulence Model At Angle Of Attack14.24°

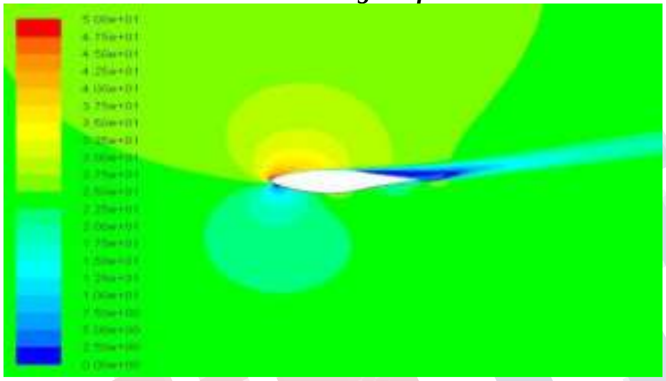

Fig.16 Contour Of Velocity Magnitude For K-Ω Sst Turbulence Model At Angle Of Attack14.24°

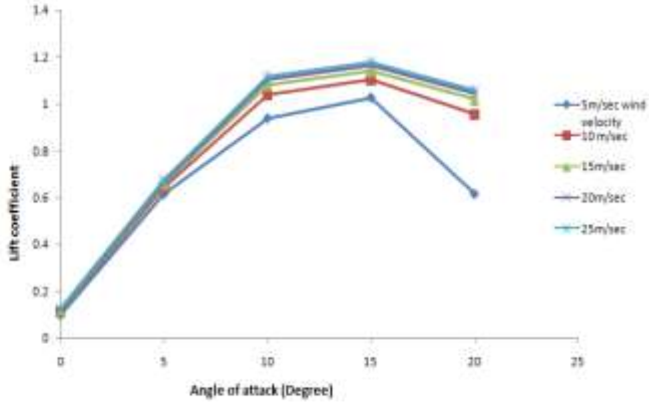

Fig.17 coefficient of lift at various angle of attack wind velocities to determine the most convenient turbulence model table1 could be referring.

The study is based on flow control. There expected a large separation areas, so that there is significant influence of turbulence model on numerical solution [8]. Table1. Effect of turbulent model at 16° angle of attack and 24.34 wind velocity

Turbulence Model	C_l	C_d	C_l/C_d
k–ε model	0.9498	0.1436	6.6142
S-A	1.2457	0.0892	13.9652
k–ω Standard	1.297	0.1113	11.6531
k–ω SST	1.1638	0.0747	15.5796

So it is observed that k–ω sst is best for an airfoil. By selecting k–ω sst turbulent model some results are as shown in fig.17 and fig.18. The fig.17 shows effect of wind velocity on lift coefficient. It seems that with increase in wind velocity lift coefficient increases. Fig 18 shows effect of wind velocity on drag coefficient. This shows higher the wind velocity lower will be drag force acting on it. . The reynolds number defined as:

$$Re = \frac{\rho V_{\infty} l}{\mu}$$

Where is v_{oo} the free stream flow velocity i.e. Wind velocity, μ is the dynamic viscosity of air and is the chord length of an airfoil i.e. (1m).

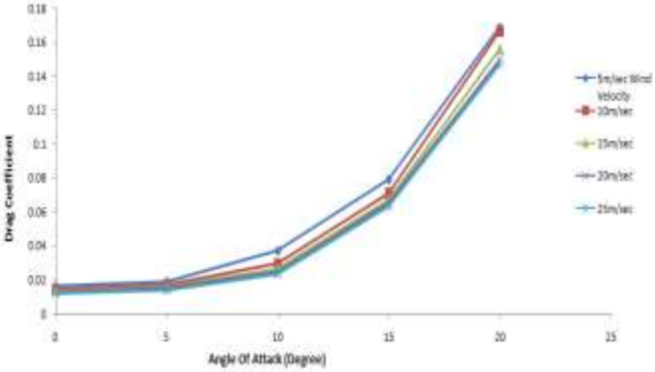

Fig.18 Coefficient of drag at various angle of attack and wind velocities

IV. CONCLUSION

There is variation in computational result as compared to experimental result. On the basis of pressure coefficient distribution over an airfoil, it is conclude that k–ω SST is best turbulence model for aerodynamic analysis of an airfoil on the basics of Reynolds number or wind velocity, because it gives relatively good result as compared with experimental literature result [7]. It is also observed that wind velocity is directly proportional to the lift coefficient i.e. increase in wind velocity increase in lift coefficient. But wind velocity is inversely proportional drag force. To increase the aerodynamic performance of blade, higher will be the Reynolds number, as the Reynolds number is depends on wind velocity.

REFERENCES

[1] M.SerdarGença, UnverKaynakb, HüseyinYapicic, “Performance of transition model for predicting low Re aerofoil flows without/ with single and simultaneous blowing and suction”, European Journal of Mechanics B/Fluids 30 (2011) 218–235.

[2] . Bhushan S. Patil, Hitesh R. Thakare, “Computational Fluid Dynamics Analysis Of Wind Turbine Blade at various angle of attack and different Reynolds number, International conference on computational heat and mass transfer 2015, Procedia Engineering 127 (2015) 1363 – 1369.

[3] . Kianoosh Yousefi, Reza Saleh and Peyman Zahedi, “Numerical study of blowing and suction slot geometry optimization on NACA 0012 airfoil”, Journal of Mechanical Science and Technology 28 (4) (2014) 1297~1310.

[4] . Tarun B Patel, Sandip T Patel, Divyesh T Patel, Maulik Bhensdadiya,” An Analysis of Lift and Drag Forces of NACA Airfoils Using Python”, International Journal of Application or Innovation in Engineering & Management (IJAIEM) Volume 4, Issue 4, April 2015 ISSN 2319 – 4847.

[5] . Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, ” CFD Analysis of an Aerofoil”, International Journal of Engineering Research ISSN: 2319-6890) (online) , 2347 – 5013 (print).

[6] . P. B. Makwana1, J. J. Makadiya, ” Numerical Simulation of Flow Over Airfoil and Different Techniques to Reduce Flow Separation Along with Basic CFD Model: A Review Study”, International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 3 Issue 4, April – 2014.

[7] . Riyadh Belamadi, Adbelouaheb Djemili, Adrian ilinca, Ramzi Mdouki, “Aerodynamic analysis of slotted airfoils for application to wind turbine blade”, J. Wind Eng. Ind. Aerodyn. 151(2016) 79-99.

[8] . David Wafula Wekesa, Cong Wang, Yingjie Wei and Joseph N. Kamau, “Wind Resource Assessment and Numerical Simulation for Wind Turbine Airfoils”, 15th International Workshop on Research and Education in Mechatronics (REM), Elgouna, Egypt, September 9-11, 2014.

[9] . Ansys Fluent Theory Guide Ansys, November 2013.