International Conference on Mechanical, Industrial and Materials Engineering 2015 (ICMIME2015)
11-13 December, 2015, RUET, Rajshahi, Bangladesh.

Paper ID: FM-18

Experimental and Numerical Measurement of Lift and Drag Force of NACA 0015Aerofoil Blade

Md. Kamal Uddin, Md. Zahidul Islam, Md. Rokunuzzaman, Robiul Islam Rubel

Department of Mechanical Engineering, Rajshahi University of Engineering& Technology, Rajshahi-6204, Bangladesh

E-mail: kamaluddin.me10@yahoo.com, jahid10ruet@gmail.com, rzaman.mte@ruet.ac.bd, rubel.ruet10@gmail.com

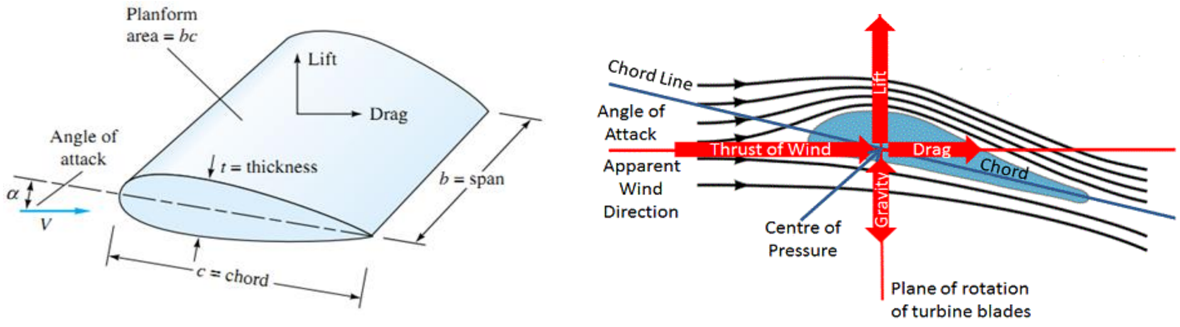
Abstract

A Symmetric aerofoil (NACA 0015) is used in many applications including aircraft vertical stabilizers, submarine fins, rotary and some fixed wings. The ultimate objective of an aerofoil is to obtain the lift necessary to keep an airplane in the air. However, construction of the blade with proper angle of attack implementation has significant effect on lift force. Insufficient lift force might cause failure of airplane flying, especially at high speed. The objective of this paper is to find the aerodynamics characteristics of aerofoil NACA 0015. In this work, numerical and experimental investigation of NACA 0015is studied at different angle of attack (degree) with different velocity of air. The experimental test is conveyed in low speed wind tunnel at the Fluid Mechanics Lab of RUET. The numerical analysis is conducted using ANSYS (combined with CFD and FLUENT FLOW). The use of the CFD technology greatly reduces the overall investment and efforts for aerofoil design. CFD method contributes to visualize the flow pattern inside aerofoil with less time than experimental methods. At the end of investigation, the results of the experimental and numerical data are compared.

Keywords: Aerofoil, CFD, Lift and Drag Force, Angle of Attack.

1. Introduction

An aerofoil is defined as the cross section of a body that is placed in an airstream in order to produce a useful aerodynamic force in the most efficient manner possible. The cross sections of wings, propeller blades, windmill blades, compressor and turbine blades in a jet engine, and hydrofoils, aircraft vertical stabilizers, submarine fins, rotary and some fixed wings are examples of aerofoil[1,2]. The basic geometry of an aerofoil is shown in Fig. 1 and Fig. 2. Since an aerofoil is stream lined body, it may be symmetrical or unsymmetrical in shape characterized by its chord length (C), angle of attack (α), and span length (L) [3]. The drag force and lift force significantly depends on its geometrical shape [4]. The proper designing of the aerofoil can minimize the produced drag on the aerofoil [5]. The lift on the aerofoil is due to negative pressure created on the upper part of aerofoil [6]. Low Reynolds number aerofoil aerodynamics is important for both military and civilian applications. The applications include propellers, high-altitude vehicles, sailplanes, light man carrying aircraft, wind turbines, unmanned aerial vehicles (UAVs) and micro air vehicles (MAVs). Flow control over aerofoil is primarily directed at increasing the lift and decreasing the drag produced by the aerofoil [7]. The evaluation of turbulence models for unsteady flows of an oscillating aerofoil has been reported in [8]. In this work, NACA 0015 aerofoil by using five different turbulence models has been considered. It has been found that Spalart-Allmaras turbulence model has a good agreement with experimental results for lift, drag and moment coefficient. The performance of wind turbine using NACA0012 aerofoil using FLUENT programs has been investigated in [9]. Spalart-Allmaras turbulence model for numerical solutions has been used by Lianbing for aerofoil at 3×10^6 Reynolds number, for lift and drag performance and stall angle. NACA 63-415 aerofoil profile has been studied in [10]. Different turbulence model in FLUENT has been used in this work and it is found that SA (Spalart-Allmaras) model is better than others models. Moreover, aerodynamics of aerofoil at low and high angles of attack has been considered. NACA4412 aerofoil profile at 3× 106 Reynolds numbers has been studied in [11]. The transition from laminar flow to turbulence flow by using two different numerical models have been investigated which are k-epsilon and Spalart-Allmaras. Numerical results are compared with experimental results. Two numerical models give similar results at high Reynolds number has also been indicated.



a. Geometry of an aerofoil blade b. Lift and Drag on a turbine blade

Fig. 1. Geometry of an aerofoil blade

A fine mesh is required in the vicinity of the aerofoil to model the flow field precisely. In the vicinity of the aerofoil it is necessary to refine the mesh at the trailing edge in order to confirm accurate modeling, as this would be the point of interest. It is imperative to maintain good aspect ratios of cells through the domain as well as sustaining smooth changes in cell size [12]. Thus quadrilateral elements are used. One of the most practical problems in flight operation is the wing ground interference, or what is called 'collision', during takeoff and landing of aircrafts. Aircraft performance during takeoff and landing is influenced due to the influence in the aerodynamic factors of the wing [13].The aerodynamic characteristics of the wing are changing in the collision phenomena. This paper is intended for finding the aerodynamics characteristics using CFD method and this method has contributed to visualize the flow pattern inside an aerofoil which is faster than experimental methods. The model of NACA 0015 has been chosen for its symmetrical in shape and ease for computation. In this work, Lift and drag force is measured at different velocity by inclined tube manometer. Lift coefficient (C_l), drag coefficient (C_d) and drag polar (C_l/C_d) is also measured.

2. Methodology

The experiment is conducted by an open channel wind tunnel having cross section of .3m×.3m and length 0.4m at 8.5-9.65 m/s wind velocity. The model is first prepared by casting followed by other machining processes to obtain the desired model. The model is placed and tested in the open wind tunnel having a motor operating at 2800 rpm which drives tunnel fan. Lift and drag force are measured from balanced arm and velocity of air is determined from inclined tube manometer after placing the model at an angle of attack (2 degree) with an increment of 2 degree. The chosen model is NACA 0015aerofoil blade. Later on an aerofoil body is designed by ANSYS 14.0 workbench design modeler. Boundary conditions are applied to obtain the distribution of turbulence and pressure.

2.1 Design Criteria

In this paper, the NACA 0015, the well documented airfoil from the 4-digit series of NACA airfoils, is utilized. The NACA 0015 airfoil is symmetrical; the 00 indicates that it has no chamber. The 15 indicates that the airfoil has a 15% thickness to chord length ratio. It is 15% as thick as it is long. The parameters of the NACA 0015 aerofoil blade are the following:

Chord length of the aerofoil, C= 0.06 m

Maximum chamber, m = first digit × %C = $0\times\frac{1}{100}\times 0.06 = 0$

Distance from leading edge to maximum wing thickness, p = second digit× 10% C = $0\times\frac{10}{100}\times 0.06 = 0$

Maximum wing thickness, t = last two digit × % C = $15\times\frac{1}{100}\times 0.06 = 0.009$m

2.2 Reynolds number

The Reynolds number is dimensionless number which is defined by the eq. 1.

$$Re_L = \frac{\rho V L}{\mu} \quad (1)$$

Where, density of air ρ = 1.23 kg/m^3, kinematic viscosity, μ = 1.973kg.m^{-1} s^{-1}, span length, L = 26cm

2.3 Lift force

Lift on a body is defined as the force on the body in a direction normal to the flow direction. Lift will only be presented if the fluid incorporates a circulatory flow about the body such as that which exists about a spinning cylinder. The velocity above the body is increased and so the static pressure is reduced. The velocity beneath is slowed down giving an increase in static pressure. Consequently, there is a normal force upwards called the lift force. Lift and drag force data are usually expressed in dimensionless terms by using lift coefficient and drag coefficient. Lift force is a component of total force F perpendicular to the stream of $Fcos\alpha$. The lift coefficient (C_l) is defined mathematical by eq. 2.

$$C_l = \frac{2F_L}{\rho V^2 A} \tag{2}$$

Where, F_L = lift produced, ρ = density of air, V = velocity of the air and A= (C×L) = area of the aerofoil.

2.4 Drag force

The drag on a body in an oncoming flow is defined as the force on the body in a direction parallel flow direction. For a windmill to operate efficiently the lift force should be high and drag force should be low. For small angles of attack, lift force is high and drag force is low. If the angles of attack (α) increases beyond a certain value, the lift force decreases and the drag forces increases. Therefore, the angle of attack plays a vital role. Lift and drag is presented in Fig. 1(b). There is also a component of total force F in the direction of the stream, which is $Fsin\alpha$. The drag coefficient (C_d) is defined as mathematical by eq. 3.

$$C_d = \frac{2F_D}{\rho V^2 A} \tag{3}$$

Where, F_D = Drag produced, ρ = density of air, V = velocity of the air, and A = (C×L) is the area of the body or aerofoil.

2.5 Inputs and Boundary condition

We consider the problem of flow around the aerofoil at 0º - 8º angle of attack. For this, we take some initial inputs and impose some boundary conditions for our problem which is shown in the table 1.

Table 1. Inputs and Boundary condition for CFD analysis

Inputs	Boundary conditions
Velocity of flow	8.5-9.65 m/s
Operating temperature	30º C
Operating pressure	1 atm
Model	Transition (sst)
Density of fluid	1.23 Kg/m^3
Kinematic viscosity	1.973 kg.m^{-1} s^{-1}
Reynolds number	Vary with air velocity
Length	0.06 m
AOA	0-8 degree respectively
Fluid	Air as an ideal

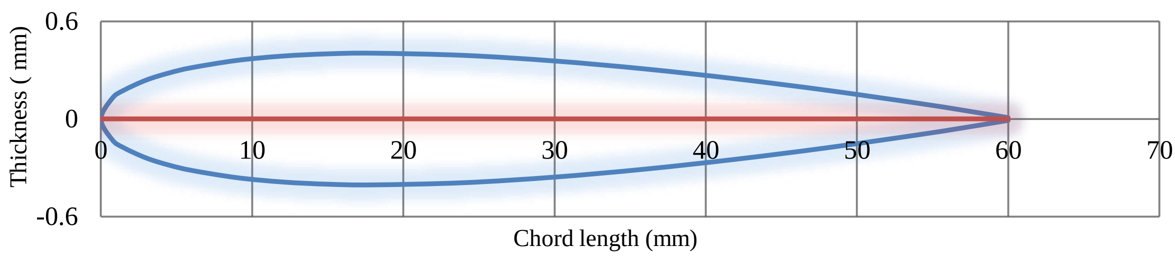


Fig. 2. NACA 0015 aerofoil profile

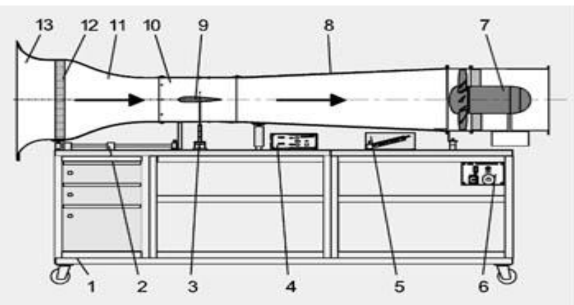


Fig. 3. Diagram of an open type wind tunnel

Fig. 4. Experimental setup

a. Aerofoil blade b. Balance arm c. Protector

Fig. 5. Photography of experimental work

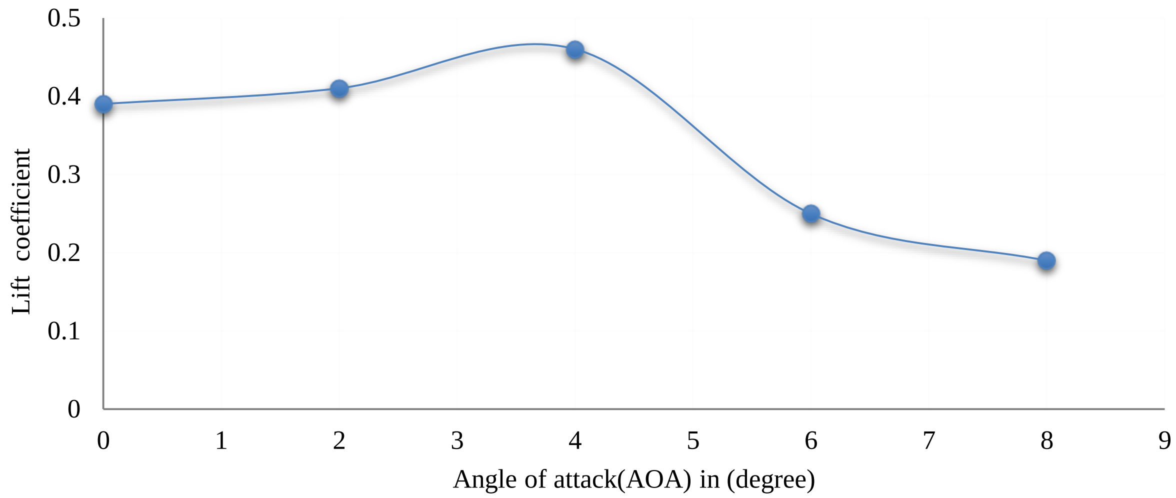


Fig. 6. Lift coefficient VS angle of attack at different Reynolds's number for NACA 0015

Figure 3 is the diagram of an open type wind tunnel with the following components numbered by (1) Base, (2) Moving carrier, (3) Balance Arm, (4) Speed Controller, (5) Inclined tube manometer, (6,7) Drive section (Motor, fan), (8) Diffuser, (9) Model, (10) Test Section, (11,13) Contraction Cone. The experimental setup is shown in Fig. 4. The different components of the setup are shown in Fig. 5.

3. Results

The experimental data are tabulated in Table 2. The lift coefficient and drag coefficient both depend on angle of attack. The experimental results obtained from our model NACA 0015 are plotted on graph. The Fig. 6 shows that lift coefficient increases with increasing angle of attack and after a certain angle of attack it decreases and this angle is called stall angle. Lift coefficient C_l is maximum (0.46) at 4 degree of angle of attack. The stall angle is caused due to transition from laminar to turbulence flow. It is clear from Fig. 7 that, the value of drag coefficient is increased as angle of attack is increased. From Fig. 8 it is noticed that C_l/ C_d is gradually decreased as the value of AOA is increased. The value of lift coefficient is maximum (0.46) at Reynolds number 1.46×10^5 as shown in Fig. 9. According to Fig. 10 the variation of drag coefficient slightly increases and gives upward line with the increase of Reynolds number of the air for each AOA.

Table 2. Experimental data for NACA 0015aerofoil blade

Angle of attack in degree	Lift Coefficient, C_l	Drag coefficient C_d	Reynolds's Number $\times 10^5$	C_l/C_d
0	0.39	1.70	1.31	0.230
2	0.41	2.22	1.39	0.180
4	0.46	2.60	1.46	0.170
6	0.25	2.61	1.48	0.096
8	0.19	2.64	1.49	0.072

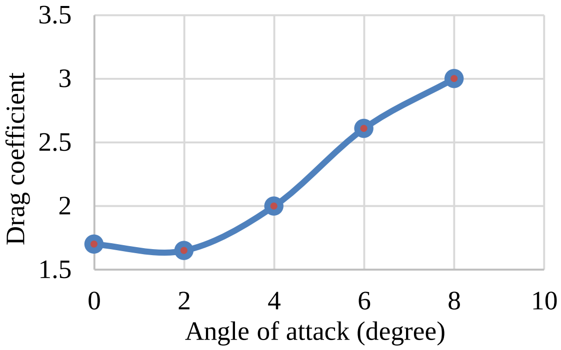


Fig. 7. Drag coefficient vs. angle of attack

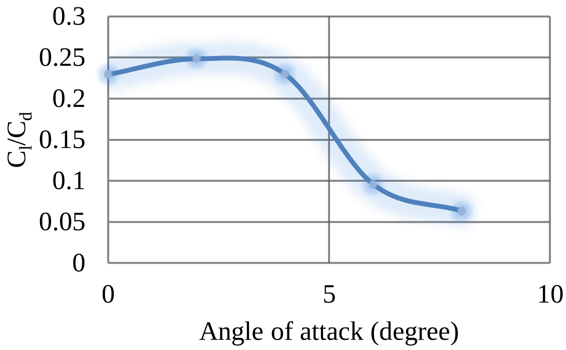


Fig. 8. Drag polar vs. angle of attack

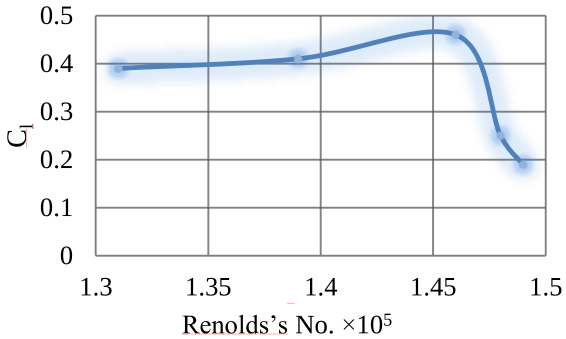


Fig. 9. Lift coefficient vs. Reynolds's number at different angle of attack for NACA 0015

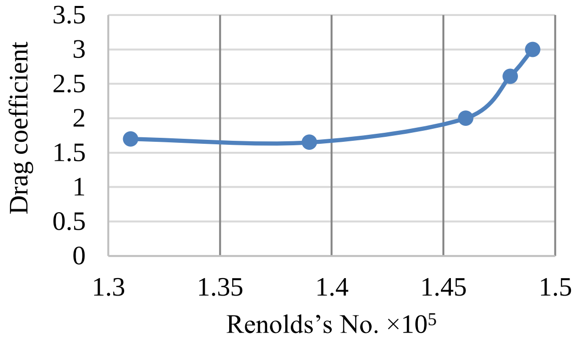
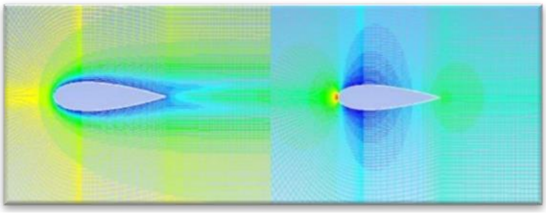
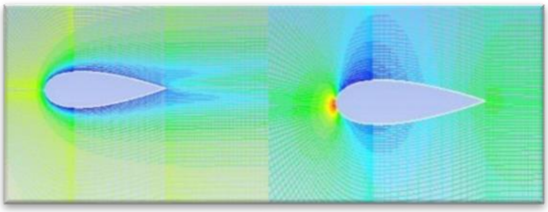


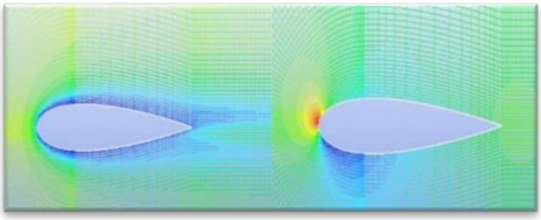
Fig. 10. Drag coefficient vs. Reynolds number at different angle of attack for NACA 0015



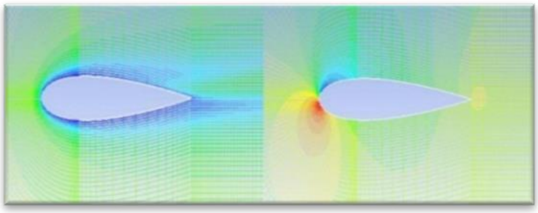
a. At 0 degree angle



b. At 2 degree angle



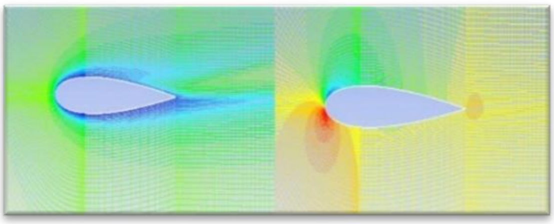
c. At 4 degree angle



d. At 6 degree angle

e. At 8 degree angle

f. Total pressure volume rendering



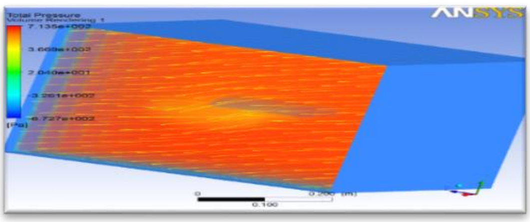


Fig. 11. Distribution of turbulence and distribution of pressure and total pressure volume rendering around NACA 0015 obtained by simulation (ANSYS)

It is clear from Fig. 11(a)-11(e) that pressure developed increases with increasing angle of attack. Thus negative pressure at angle of attack 8 degree is greater than pressure created at angle attack 0,2,4,6 degree. Figure 11(f) shows the total pressure volume rendering after simulation with ANSYS.

4. Conclusions

After preparing a NACA 0015 aerofoil blade, experimental and numerical measurement of lift and drag force has been performed. The experiment is compensated for NASA 0015 by an open type wind tunnel. Fluent flow program is used for numerical analysis to show distribution of turbulence, distribution of pressure and total pressure volume rendering around NACA 0015 aerofoil blade. The maximum value of lift coefficient is found as 0.46. The stall angle is found as 4 degree. Both lift and drag coefficient increases as angle of attack is increased. The drag coefficient is gradually decreased as Reynolds's number increases. However, with the increase of Reynolds's number lift coefficient increases slightly and after a certain point it decrease. The optimum value of lift coefficient is found as 0.46 at Reynolds's number of 1.4×10^5.There is a large negative pressure created on the aerofoil which accounts for most of the lift. The results of the experiments help in designing wind turbine and aircraft wings. Moreover, the proper blade angle can be selected based on the experimental results.

5. Acknowledgement

The authors gratefully thanks to Prof. Dr. Mohammad Rofiqul Islam, Department of Mechanical Engineering, RUET, Bangladesh for allowing the use of wind tunnel placed in fluid mechanics lab, support and guidance. Great thanks to Prof. Dr. Rokunuzzaman, Head of the Department of Mechatronics Engineering, RUET, Bangladesh, for his instructive supervision during experimental investigation by the wind tunnel. Also the authors gratefully acknowledge to Rajshahi University of Engineering and Technology owing to allowing use of machine shop lab and necessary help to make NACA 0015aerofoil blade model.

6. References

[1] Md. Rasedul Islam, Md. Amzad Hossain, Md. Nizam Uddin, and Mohammad Mashud, "Experimental Evaluation of Aerodynamics Characteristics of a Baseline Aerofoil", *American Journal of Engineering Research (AJER*), Vol.4, Issue.1, pp. 91-96, 2015

[2] S. Kandwal, and S. Singh, "Computational Fluid Dynamics Study of Fluid Flow and Aerodynamic Forces on an Aerofoil", *IJERT*, Vol.1, Issue.7, 2012

[3] R. K. Bansal, "Fluid Mechanics And Hydraulic Machines", *Laxmi Publication (P) Ltd*, 9th Edition, pp. 686- 687, 2010.

[4] M. Morshed, S. B. Sayeed, S. A. Al Mamun, and G.M. Jahangir Alam, "Investigation of Drag Analysis of Four Different Profiles Tested at Subsonic Wind Tunnel", Journal *of Modern Science and Technology*, Vol.2, No.2, pp. 113-126, 2014

[5] 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, pp. 22-25, 2015

[6] E. H. Lewitt, "Hydraulics and Fluid Mechanics", *Sir Isaac Pitman & Sons Ltd*, 10th Edition, pp.382-383, 1963

[7] Zifeng Yang, Hirofumi Igarashi, Mathew Martin, and Hui Hu, "An Experimental Investigation on Aerodynamic Hysteresis of a Low-Reynolds Number Aerofoil", *American Institute of Aeronautics and Astronautics*, AIAA-2008-0315, 2008

[8] G. R. Srinivasan, J. A. Ekaterinaris and W. J. McCroskey, "Evaluation of Turbulence Model for Unsteady Flows of an Oscillating Aerofoil", *Elsevier Science Ltd.*, Computers & Fluids, Vol.24, No.7, pp. 833-861, 1995

[9] L. B. Li, Y. W. Ma, and L. Liu, "Numerical Simulation on Aerodynamics Performance of Wind Turbine Aerofoil", *IEEE Conference on World Automation Congress (WAC), Puerto Vallarta, Mexico,* ISSN. 2154-4824, pp. 1-4, 2012

[10] F. Villalpanda, M. Reggio, and A. Ilinca, "Assessment of Turbulence Model for Flow Simulation Around a Wind Turbine Aerofoil", *Modeling and Simulation in Engineering, Hindawi Publishing Corp.*, New York, United States Vol.2011, No.06, 2011

[11] H. C. Ravi, N. Madhukeshwara, and S. Kumarappa, "Numerical Investigation of Flow Transition for NACA 4412 Aerofoil Using Computational Fluid Dynamics", *International Journal of Innovative Research in Science Engineering and Technology*, Vol.2, Issue.7, pp. 2778-2785, 2013

[12] Hakim Tarteeb Kadhim, "Computational Modeling of Turbulent Flow around Aerofoil Using Different Turbulence Models" *Al-Qadisiya Journal for Engineering Science,*Vol.7, No.3, pp. 278-289, 2014

[13] H. H. Al-Kayiem and AK Kartigesh, "An Investigation on the Aerodynamic Characteristics of 2-D Airfoil in Ground Collision", *Journal of Engineering Science and Technology,* Vol.6, No.3, pp. 369-381, 2011