International Journal of Integrated Engineering, Vol. 10 No. 1 (2018) p. 134-137
© Penerbit UTHM
DOI: https://doi.org/10.30880/ijie.2018.10.01.019

A Comparative Study of Turbulence Models on Aerodynamics Characteristics of a NACA0012 Airfoil

Azmahani Sadikin[1], Nurul Akma Mohd Yunus[1], Saiful Anuar Abd Hamid[1], Al Emran Ismail[1], Salihatun Salleh[1], Sufizar Ahmad[1], Mohd Nasrull Abdol Rahman[1], Shahruddin Mahzan[1], Sallehuddin Shah Ayop[2]

[1]Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Pahat, Johor, MALAYSIA

[2]Faculty of Civil and Environmental Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA

Received 28 June 2011; accepted 5 August 2011, available online 24 August 2011

Abstract: This paper presented a computational fluid dynamics (CFD) simulation of air flow past a 2D model NACA0012 airfoil at high Reynolds number ($Re = 3.0 \times 10^6$) at various angles of attack (-10 to 15). The simulations were undertaken to inform on how the fluid flowed around the airfoil by solving the steady state governing equations of continuity and momentum conservation that are combined with one of three turbulence models Spalart-Allmaras, Realizable k-ε and k-ω shear stress transport (SST). It is observed that the Realizable k- ε eliminates the small separation bubble on the upper surface of the airfoil and delaying separation flow. Also, for the lift coefficient, C_L and drag coefficient, C_D investigated in this paper, the predicted data have good agreement with other published data.

Keywords: NACA0012 airfoil, CFD, Spalart-Allmaras, Realizable k- ε, k-ω SST, lift coefficient, drag coefficient

1. Introduction

Aerodynamic is the study of forces and the resulting motion of objects through air such as the example of aerodynamics is flying aircraft, flapping of flags on poles, smoke dispersion from chimney and speeding race cars. The forces are lift force and drag force. Lift most commonly associated with the wing of a fixed-wing aircraft, although lift is also generated by propellers, kites, helicopter rotors, rudders, sails and keels on sailboats, hydrofoils, wings on auto racing cars, wind turbines and other streamlined objects. When an aircraft is flying straight and level (cruise) most of the lift opposes gravity. However, when an aircraft is climbing, descending, or banking in a turn the lift is tilted with respect to the vertical.

An airfoil is a streamlined shape that is capable of generating significantly more lift than drag. Non-streamlined objects such as bluff bodies and flat plates may also generate lift when moving relative to the fluid, but will have a higher drag coefficient, dominated by pressure drag.

The form drag for a 3D wing also includes the induced drag, which is generated at wing tips when high-pressure air from the lower wing surface is driven by a favorable pressure gradient (high to low) around to the low-pressure air on the upper surface, producing wing-tip vortices.

On an airfoil, the resultants of the forces are usually resolved into two forces and one moment. The component of the net force acting normal to the incoming flow stream is known as the lift force and the component of the net force acting parallel to the incoming flow stream is known as the drag force.

Many researchers have numerically and experimentally investigated the aerodynamic performance of airfoil. The dynamics of laminar separation bubble over an airfoil near stall conditions using large eddy simulation of flow around NACA-0012 airfoil at an angle of attack for a Reynolds number and Mach number was investigated by [1]. They found that that computed Strouhal number of the oscillation was in good agreement with the experimental data where a self-sustained low-frequency flow oscillation was observed. The investigation of the unsteady flow past a NACA0015 aerofoil for moderate Reynolds numbers at high angles of attack by solving the full 2-D Navier–Stokes equations with and without the presence of free-stream turbulence (FST) was carried out by [2].

Their investigation focuses on the *by-pass* mode of transition usually encountered in turbomachinery and wind engineering where the flow field around a bluff-

---

Corresponding author: azmah@uthm.edu.my body can experience very high levels of FST. They found out that 5% level of FST is considered and proposed new model for FST based on a moving-average time-series and using it for long-time computation of the Navier–Stokes equations. The experimental investigation on aerodynamics of a NACA2415 aerofoil by varying angle of attack from −12° to 20° at low Reynolds number flight regimes (0.5×10^5 to 3×10^5) was conducted by [3]. They measured the pressure distributions over the aerofoil using a system including a pitot-static tube, a scanivalve unit and a pressure transducer. They also obtained the time-dependant lift and drag forces and pitch moment of the aerofoil by using an external three-component load-cell system. Other than that, they also measured the velocity at different points over the aerofoil using a hot-wire anemometer, and used oil flow visualization method to photograph the surface flow patterns. They found that the angle of attack increased, the separation and the transition points moved towards the leading edge at all Reynolds numbers.

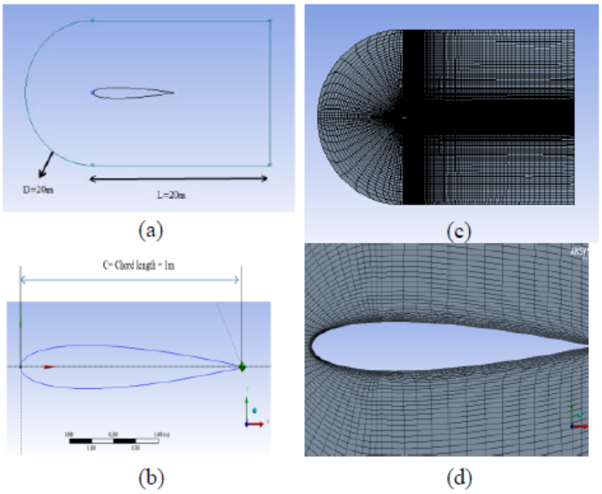
A CFD simulation of an aerodynamic performance of rough wind turbine airfoil and its blunt trailing-edge modification with sensitive roughness height was studied by [4]. They used k-ω SST turbulence model, to calculate the lift and drag coefficients of S834 airfoil with smooth or rough surface. They found that the sensitive roughness height of suction surface is 0.5 mm, and the pressure surface is insensitive to the roughness height. Through the blunt trailing-edge modification, the lift coefficient and the maximum lift-drag ratio obviously increase for rough airfoil, and the sensitivity of airfoil to roughness height is reduced. The effects of Mach number, length, installation angle and installation position of the small plate on the flow separation control to the airfoil (NACA4405) was investigated by [5]. They found out that by setting a small plate at the leading-edge of the airfoil can effectively delay the flow separation phenomenon. They also found out that the Mach numbers inferior to 0.5 can lead to maintain a relatively high lift coefficient even at very large angles of attack. Other researcher also studied the airfoil aerodynamic performance at low Reynolds numbers for lift over drag coefficient [6]. They used the XFOIL code, the Shear Stress Transport turbulence model and a refurbished version of transition model to predict both coefficients. Computational Fluid Dynamics (CFD) is becoming increasingly popular in the design and optimization of devices that depend on aerodynamics.

Thus, it is of great significance to study the aerodynamic performance of NACA0012 airfoil using ANSYS CFD. The NACA airfoil is an airfoil shape for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of the NACA airfoil is described using a series of digits following the word NACA. The parameters in the numerical code can be entered into equations to precisely generate the cross-section of the airfoil and calculate its properties. In this present study, the curves of the lift coefficient, C_L and the drag coefficient, C_D are shown for various angles of attack in a range of -10° to 15°. This study also demonstrate the capability of three turbulence models which are Spalart-Allmaras, Realizable k-epsilon and k-ω Shear Stress Transport (SST) to predict the flow separation around the airfoil.

2. Methodology

In this study, ANSYS CFD version 14.5 is used to simulate high Reynolds number flow ($Re = 3 \text{ x } 10^6$) past two-dimensional airfoil. The flow is assumed incompressible based on airfoil chord length while the angles of attack varied from -10° to 15°. The simulation was conducted in steady state. The airfoil geometry and the mesh are shown in Fig. 1. The C-type mesh topology was chosen because it can minimize the skewness of a near wall mesh as the structured quadrilateral element has the advantages of a higher degree of control and accuracy, a lower memory consumption and a faster convergence rate.

Three mesh configurations of 16 940, 57 040 and 78 408 cells were conducted for the grid independency test. The pressure coefficient versus distance of y-axis were plotted and analyzed. The results show that there is no significant difference between the 57 040 and 78 408 configurations as all lines of both configurations are almost overlapped. These indicate that using finer mesh does not improve the model prediction. Thus, meshing with lower number of mesh cells does not sacrifice the solution accuracy.



(a) (c)

(b) (d)

Fig. 1 The model (a) flow domain (b) airfoil geometry (c) the meshing (d) the mesh refinement at the airfoil surface.

Since the Central Processing Unit (CPU) time increases exponentially with the number of grids, the lower mesh cells, 57 040 were chosen. Less mesh cells reduce CPU time during CFD simulation which permits a significant number of cases to be run. The meshing gave a total of 29 862 nodes and had 57 040 elements, and the near wall of the airfoil is refined using the boundary layer as shown in Fig. 1(d). The boundary conditions for the airfoil are shown in Fig. 2. The airfoil was set to solid surfaces with no slip and the top and bottom lines were set to the symmetrical boundary condition.

The outlet boundary condition was set to atmospheric pressure and the inlet boundary was set to a velocity inlet of 2.19 m/s. The free stream temperature is 300 K. The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) scheme was selected for the pressure-velocity coupling while for spatial discretization section, the green-gauss node based was set. The second order upwind was used for the momentum, turbulent kinetic energy and turbulent dissipation rate to arrive at the best solution. Turbulences model from the viscous model which were Spalart-Allmaras, k-ε Realizable and k-ω SST were selected.

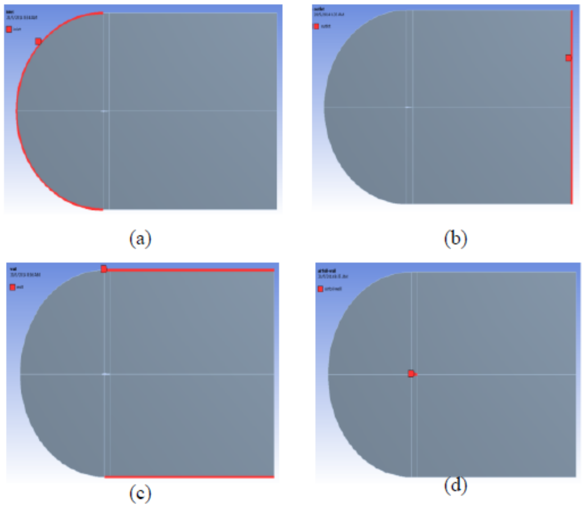


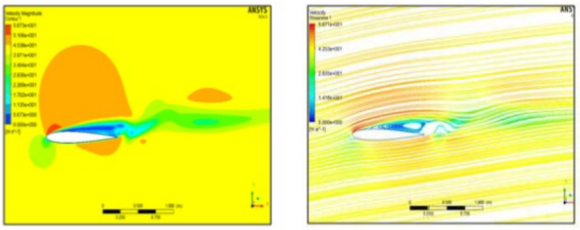
Fig. 2 The boundary conditions (a) inlet (b) outlet (c) symmetry (d) wall

3. Results and Discussion

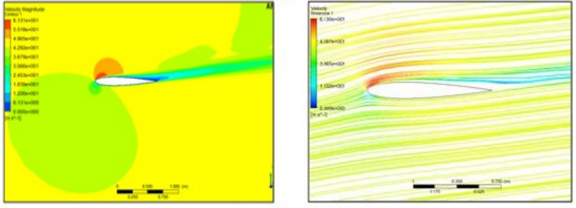
3.1 Flow over airfoil

In the present study, comparisons made on the time-averaged streamlines and velocity contours near trailing edge of the airfoil at 15° angle of attack. Fig. 3 gives respectively the streamlines and velocity magnitude obtained by using different turbulence models. We can observe obviously that the flow fields around the airfoil predicted by using of all three different turbulence models are in agreement with each other. It can be seen that there is a small separation bubble near the trailing edge using Realizable k-ε. It is obvious that the Realizable k-ε turbulence model eliminates the small separation bubble on the upper surface, so that the air can flow smoothly along the upper surface.

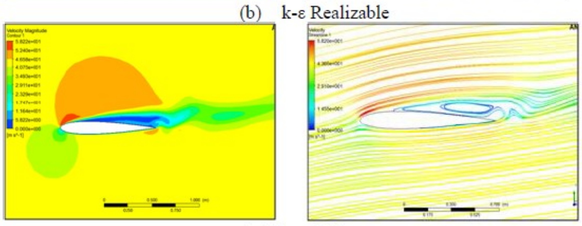
Thus, this turbulence model can increase lift by delaying the flow separation in the trailing edge of the airfoil. Similarly, a similar comparison is made at other turbulence models. The flow has separated earlier from the leading edge for both Spalart-Allmaras and k-ω SST, resulting in a large separation bubble, which greatly reduces the suction on the upper surface of airfoil. There is a wake region downstream of the airfoil, where a counter-rotating vortex pairs exist. The reattached zone corresponds to the layer colored in blue that is attached at the upper surface of the airfoil. This reattachment of the negative vorticity zone can lead the fluid to rotate in a clockwise direction. The results are quantitatively summarized.



(a) Spalart-Allmaras



(b) k-ε Realizable



(c) k-ω SST

Fig. 3 Velocity contours and streamlines of the airfoil at angle of attack of 15° (a) Spalart-Allmaras (b) k-ε Realizable and (c) k-ω SST

3.2 C_L and C_D at various angles of attack

Fig. 4 and 5 show the graph C_L and C_D versus angle of attack for the all three turbulent models. For lower angles of attack, the C_L - angles of attack curve is nearly linear, and very closely matches the one predicted by other researchers due to attached flow. At higher values of angles of attack, the flow can no longer follow the upper surface of the aerofoil and becomes detached. There is a region above the upper surface, near the trailing edge, where the velocity is low and the flow reverses direction in places in a turbulent motion. This phenomenon is trailing edge separation.

As the angle of attack is increased further, the beginning of the region of separated flow (trailing edge separation) moves towards the leading edge of the aerofoil. At a critical angle of attack, the lift component of the aerodynamic force falls off rapidly and the drag component increases rapidly as shown from C_L - angles of attack and C_D - angles of attack curves. This phenomenon is called stall, and this critical angle of attack is called stall angle. The maximum lift coefficient defines the angle at which the aerofoil will stall. It can be seen that all the predictions are in good agreement with the published data. All three turbulence model are capable to accuratey predict the C_L and C_D and similar finding with [3,6].

4. Conclusion

The CFD simulation for a steady 2D symmetric airfoil NACA0012 at various angles of attack using three different turbulence models was successfully carried out. It is observed that all of three turbulence models agree well with published data. The best turbulence model to simulate the flow past an airfoil is Realizable k-ε. This is due to the fact the delay of flow separation occur when using this model. Therefore, this resulted in the increases of lift and decreases of drag for airfoil through the fluid. Also, the predicted C_L and C_D are found to have a good agreement with the published data.

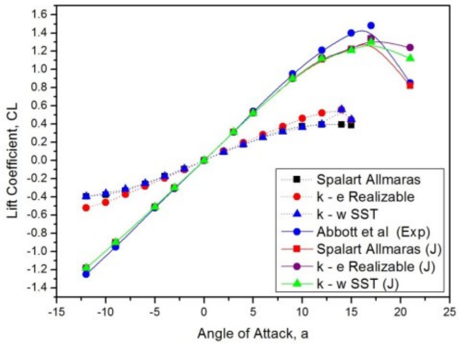


Fig. 4 Curves of C_L versus angles of attacks

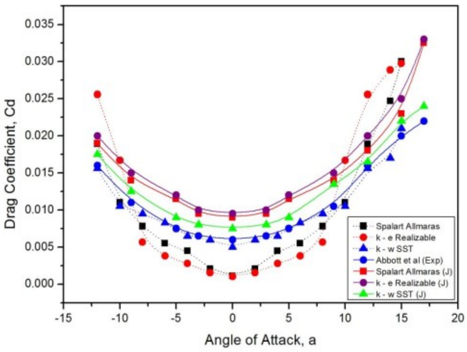


Fig. 5 Curves of C_D versus angles of attacks

References

[1] AlMutair J., ElJack E., AlQadi I., Dynamics of laminar separation bubble over NACA-0012 airfoil near stall conditions, *Aerospace Science and Technology*, Volume 68 (2017), pp 193-203.

[2] Sengupta T.K., Dek.Gupta S., Effect of free-stream turbulence on flow over aerofoil section at high incidence, Journal of Fluids and Structures, Volume 15, Issue 5 (2001) pp 671-690.

[3] Serdar Genç M., Karasu I., Açıkel H. , An experimental study on aerodynamics of NACA2415 aerofoil at low Re numbers, *Experimental Thermal and Fluid Science*, Volume 39 (2012), pp 252–264

[4] Zhang X., Wang G., Zhang M., Liu H., Li W., Numerical study of the aerodynamic performance of blunt trailing-edge airfoil considering the sensitive roughness height, *International Journal of Hydrogen Energy*, Volume 42 Issue 29 (2017), pp 18252-18262.

[5] Zhou Yangwei, Hou Longfeng,Huang Diangui, The effects of Mach number on the flow separation control of airfoil with a small plate near the leading edge, *Computers & Fluids*, Volume 156 (201), pp 274-282.

[6] Morgado J., Vizinho R., Silvestre M.A.R., Páscoa J.C., XFOIL *vs* CFD performance predictions for high lift low Reynolds number airfoils, *Aerospace Science and Technology*, Volume 52 (2016) pp 207-214.