Numerical Investigation of Dynamic Stall for Non-Stationary Two-Dimensional Blade Airfoils

G.S.T.A. Bangga¹*, H. Sasongko²

1,2Department of mechanical engineering
Field of study energy conversion
Institut Teknologi Sepuluh Nopember (ITS), Surabaya, Indonesia
*Corresponding author : probability.schrodinger@gmail.com

Abstract

In the analysis of the helicopter blades, turbo-machinery compressor blades, wind turbines and other streamlined structures operating at high angle of incidence, the necessity and importance of including a non-stationary effect of the local two-dimensional flow, the so-called dynamic stall becomes increasingly important. It could affect performance and also reduce fatigue life of many modern lightweight blades through the induced vibrations. The event of the dynamic stall occurred on the suction side of airfoils that are shown by dynamic lift exceeds maximum static lift at the static stall angle. Numerical methods using two turbulence URANS model, namely standard-kω and SST-kω, is used in the analysis to investigate the dynamics induced by the present model on a two-dimensional airfoil and also to explain physics of these phenomenon. It is demonstrated that the SST-kω performs better than standard-kω to predict hysteresis lift coefficient. The onset of travelling vortex phenomenon could be well captured. The intended applications of this paper lie in the field of unsteady rotary-wing aerodynamics and its characteristics on the induced vibration.

Keywords: Dynamic stall, Non-stationary two dimensional airfoils, CFD, External flows

Introduction

The term dynamic stall usually refers to an unsteady flow characterized by the formation, convection and shedding of a vortex on the suction side of an airfoil [1]. The event of the dynamic stall on the aerodynamic bodies involve complex separated flow and even in the transition zone. The occurrence of dynamic stall is caused by the high aeroelasticity on the airfoil at high angle of attack. Unlike the static stall, the stall on the dynamic conditions can be delayed on airfoil with angle of attack increases rapidly and even beyond the critical static stall angle of attack significantly. However, when dynamic stall occurs, the aerodynamic loads generally larger than the static stall and lead to increased stress acting on the airfoil significantly and potentially harmful the airfoil structure itself [2]. Dynamic stall events initiated by the leading edge separation, leading edge vortex developed and moves downstream along the airfoil surface. Vortex reaches the trailing edge and detach followed by a trailing edge vortex formation and indicate the onset of stall. The detachment of the trailing edge vortex occurs as the increase in angle of attack followed by a leading edge vortex breakdown. These events resulted in a drastic decrease in lift coefficient and the flow reattachment will occur at fairly low angle of attack again. Figure 1 shows the event of stall at the dynamic conditions by CFD calculation on the NACA 0015 [3]. Fig. 1(a) illustrates the onset of leading edge separation with the entire boundary layer starting to detach. Fig. 1(b) shows the build-up of the leading edge vortex, which in Fig. 1(c) detaches and moves downstream, while a trailing edge vortex starts building up. Finally, Fig. 1(d) shows the detachment of the trailing edge vortex and breakdown of the leading edge travelling vortex. Both experimental data and CFD calculations indicate that the flow changes, caused by the leading edge separation vortex, generate an increased suction contribution, leading to an increased lift even after flow separation has occurred [3]. This effect may be seen in figure 2 as the dynamic curve that continues to increase above the static stall angle.
Since the first analysis on aircraft aeroelasticity, unsteady aerodynamic airfoil studied experimentally and theoretically. Dynamic stall, until the 1950’s only studied experimentally, in the late 1970’s a new mathematical model of dynamic stall was introduced by Friedmann and since then the modeling of dynamic stall developed [6]. Mathematical modeling of dynamic stall lot to do ranging from simple to complex, including the ONERA [7], Boeing [8], Johnson [9], Ye [10], Ris [6] and also Leishman-Beddoes model [11, 12]. Meyer, et al. [13] modeled the dynamic stall based on the DLR models by utilizing the state-space representation. This involves modeling of the instationary effects using solution of Theodorsen equation in the time domain called Wagner function and Küssner function. These two functions describe the dynamic stall hysteresis curve in the linear zone. Based on the verification, this modified numerical ODE method produces dynamic stall plot corresponding to the occurring physical phenomena. However, the mathematical formulation of the model is difficult to trace DLR physically because of the mathematical formalism that is used not accompanied by a translation of the differential analysis. Dynamic stall on wind turbine models are developed by JW Larsen, et al. [3] with a backbone models based on the static lift curve. Modeling of the static lift is described by two parameters, at fully attached flow and lift depending on the degree of attachment. This mathematical modeling used the data obtained from the inviscid static stall lift and is corrected using a viscous separation factor on a flat plate approach based on Kirchhoff’s potential flow theory. Based on data obtained static stall, the effects of instationary dynamic stall will be involved by using the solution of convolution integral on the impulse response function. CFD analysis for the dynamic stall at low Reynolds number was developed by Wang [14] to simulate the NACA 0012 oscillating airfoil. This study used two turbulence modeling, using Standard-\(k\omega\) and SST-\(k\omega\) and revealed that the used of SST-\(k\omega\) give more accurate result better Standard-\(k\omega\)models. Therefore, the objective of this paper is to assess the ability of the Standard-\(k\omega\) model and the SST-\(k\omega\) model to correctly simulate dynamic stall on the two dimensional Boeing-Vertol V23010-1.58 rotorcraft airfoil which is found in the wind turbines and make a contribution towards a better understanding of the flow physics of dynamic stall in order to assist in the design optimizations of wind turbines intended for the built and urban environment in the future.

Figure 1. Flow visualization of a CFD calculation performed on a NACA-0015 wing section during dynamic stall conditions. (a) Leading edge separation starts. (b) Vortex build-up at the leading edge. (c) Detachment of leading edge vortex and build-up of trailing edge vortex. (d) Detachment of trailing edge vortex and breakdown of leading edge vortex [4].
Figure 2. Lift coefficient under static and dynamic stall situations:
– –, static lift; —, dynamic lift [5]

Figure 3. Diagram of the sub-grid structure: (a) domain classification and (b) meshing near airfoil.

Figure 4. Domain of the analysis.

Numerical Method

Turbulence simulation methods are generally divided into three main forms, namely the Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Unsteady Reynolds-averaged Navier-Stokes (URANS) [14]. DNS as the most advanced computational methods in solving both space and time scales require high computing technology so that the use of this method would require a high computational cost when applied to the case in this study. The use of LES will also require high computing technology to be applied to the phenomenon of dynamic stall and 3D simulation should be performed due to the nature of the eddy in 3D. URANS is a suitable method of dynamic stall approach with low computational cost and acceptable accuracy, therefore URANS would be used in this present study. Although the dynamic stall flow studied here is inherently a 3D phenomenon,
measurements have been taken to ensure a 2D flow in the mid-span plane, where the experimental data were obtained, in the case investigated. Thus, in the present simulations, 2D geometrical configurations are employed to model the experimental investigations and a 2D incompressible unsteady CFD solver, based on the finite volume method in the commercial software package Fluent, is employed to solve the full URANS governing equations. Due to the incompressibility of the flow studied, the pressure-based solver, which employs an algorithm which belongs to the so-called “projection method” and is traditionally implemented to solve low-speed incompressible flows, is chosen [14]. All the governing equations for the solution variables, which are decoupled from each other, are solved sequentially and the SIMPLE algorithm is applied as the pressure-velocity coupling algorithm. With respect to the discretization of the convection terms in the transport equations for the velocity and the turbulence quantities, second-order upwind schemes are utilised [14]. All the governing equations for the solution variables, which are decoupled from each other, are solved sequentially and the SIMPLE algorithm is applied as the pressure-velocity coupling algorithm. With respect to the discretization of the convection terms in the transport equations for the velocity and the turbulence quantities, second-order upwind schemes are utilised [14].

In order to simulate the sinusoidal pitching motion of the blade, the dynamic-mesh technique is employed. As seen on the figure 3, the hybrid mesh [15] domain were divided into four sub-domain with, domain 1 contains fine mesh and region 2 to region 4 contains coarser mesh. Corresponding to the pitching motion of the blade in reality, the blade geometry with domain 1, 2, and 3 pitches like a rigid body with the sinusoidal mode, \[ \alpha = \alpha_0 + \alpha_l \cos \omega t \], as the blade with \( \alpha_0 = 14.92^\circ \), \( \alpha_l = 4.85^\circ \), and \( \omega = 0.062 \), whilst the fixed mesh zone is kept stationary and being remeshed using remeshing method. Spring based smoothing is used in the dynamic mesh scheme, zero damping coefficient is used in this analysis and indicates the damping value on the airfoil surface is zero [16]. The grid for airfoil has 84 nodes on the suction side and pressure side. The height of the first row of cells is set at a distance to the wall of \( 10^3 c \) and this corresponds to \( y' \sim 1 \) to predict the phenomenon of laminar sub-layer accurately [14, 16]. The total number of meshes is of the order of \( 10^5 \). Simulation was performed in an open test region, the computational domain consists of two boundaries which are 16c and 12c away from the blade, respectively, see figure 4. The two boundaries have been placed sufficiently far away to eliminate their effect on the flow near the blade. The inlet and outlet boundaries are placed respectively 8c upstream and 45c downstream of the blade, making the uniform freestream velocity boundary condition at the inlet accurate enough and allowing a full development of the wake [14].

Results and Discussion

Numerical simulation of a two-dimensional dynamic stall has been done using two turbulence model, namely standard-\( k\omega \) and SST-\( k\omega \) on the Boeing-Vertol V23010-1.58 rotorcraft airfoil. Experimental works and analytical model by Liiva [17] and Larsen [3] were used to validate numerical works in this paper. Figure 5 shows the comparison between lift coefficient which is produced using numerical simulation with those obtained from experiment and analytical works. In the case of \( \alpha_0=14.92^\circ \), the lift coefficient in figure 5 shows the effect of oscillations of the airfoil at the fully attached and separated flow. This is indicated by the operation area in this case is on the angle \( 10.34^\circ < \alpha < 20.04^\circ \), with the static stall position is \( \alpha \approx 12.5^\circ \) [17]. The results which obtained showed significant differences by minimizing time-step size up to 0.001 s compared with the larger time-step size which is not presented in this paper. Accuracy of numerical simulation has significantly increased compared with the use of larger time-step size. The maximum position of the lift coefficient that indicates the occurrence of dynamic stall is well captured using this time-step size. Both of turbulence model, standard-\( k\omega \) and SST-\( k\omega \), generates lift coefficient value under the existing experimental result and analytical model. Lift coefficient which is produced by mathematical modeling conducted by Larsen [3] did not fluctuate in the stall area, as well as with experimental results by Liiva [17]. This phenomenon is different with numerical result that generates high fluctuation in the high angle of attack, the failure is due to the 3D effect becomes very significant compared to the lower angle of attack [18], therefore, both models fail predict the lift coefficient corresponding with experimental data [14]. In general, the results of numerical simulations on the stall area would significantly different with the experimental results, the results obtained in the simulation tend to show deep stall, similar results were also found in Wang's study [14].
Figure 5. Dynamic lift coefficient between CFD and experimental and analytical works.

Figure 6. Comparison of the lift coefficient on different reduced frequency [21].

Figure 7. Vorticity field in dynamic and static conditions (1/s).
It is clearly seen that the SST-ko turbulence model generates lift coefficient data which is more accurate than standard-ko model. The position of maximum lift coefficient could not be well predicted using standard-ko turbulence model, and this model predict this location at $\alpha \approx 15.5^\circ$ by value 1.43 compared with the experimental results at angle $16.5^\circ$ by value 1.66. SST-ko turbulence model were able to predict the maximum position of the lift coefficient at $\alpha \approx 16.8^\circ$ by value 1.57, and this is much closer to the experimental data compared with the standard-ko. This is probably due to the standard-ko model is more dissipative in terms of eddy energy so that failed to predict the high adverse pressure gradient on the corresponding angle [16, 17]. Lift coefficient value at the highest angle of attack ($\alpha = 20.04^\circ$) could be well predicted using the SST-ko turbulence models, although at the lowest angle ($\alpha = 10.34^\circ$), this value is still not well predictable. This might because of the computational errors in numerically large airfoil section [19]. Unlike the case with the SST-ko model, standard-ko model fails to predict the lift coefficient value at the lowest angle of attack as well as the highest angle. Similar results were also obtained in the study [20], where the use of the SST-ko model has a good performance for low and medium angles of attack.

Lift coefficient decreasing at $16.8^\circ < \alpha < 20.04^\circ$ indicates shedding phenomenon on the leading edge vortex, followed by formation of trailing edge vortex. Lift coefficient would increase again and this is the indication of the formation of leading edge vortex and followed by shedding of trailing edge vortex. This is explained by McCroskey [21], phase and value of the dynamic forces is depend on the value of reduced frequency $\bar{\omega}$. As the $\bar{\omega}$ increases, the position of the lift coefficient shifts to the higher angle of attack, or in other way, stall is delayed as illustrated on figure 6. In $\bar{\omega} = 0.05$, shedding of leading edge vortex occurs before the airfoil reach its maximum angle in upstroke phase, in $\bar{\omega} = 0.15$, the formation of secondary vortex is delayed to the downstroke phase, and in $\bar{\omega} = 0.25$, the shedding of leading edge vortex occurs in the downstroke phase. Figure 5 shows that the position of the lift coefficient shifts to the right (On higher angle), followed by increasing of lift coefficient value, by the increase of $\bar{\omega}$ in range $0.05 < \bar{\omega} < 0.15$. By this explanation, it could be seen that this simulation has similar results with $\bar{\omega} = 0.05$, agree with $\bar{\omega} = 0.062$ that being used in present study.

Figure 7 shows the comparison of vorticity patterns on the static and dynamic conditions using SST-ko turbulence model. At $\alpha = 10.34^\circ$, there is no significant difference occurred in both static and dynamic conditions. It could be seen that the pattern of fully attached flow occurs in both the simulation results, generating no different lift coefficient value on the corresponding angle of attack as shown in Figure 5. Along with increasing angle of attack, there was a significant difference in both conditions, as shown in Figure 7. At angle $16.0^\circ$, the position of static stall angle ($\alpha \approx 12.5^\circ$) has been passed, lift coefficient value of static condition has fallen and there has been a massive separation on the suction side of airfoil, while on the dynamic condition, similar things do not happen. With increasing angle of attack, vortex formed on the leading edge, convected along suction side of airfoil, separated, and followed by trailing edge vortex formations shown in Figure 7.e, and this is clearly different from the static condition at the corresponding angle. At an angle $16.80^\circ$, lift coefficient in the dynamic condition achieves its maximum value, the further increase in angle of attack will result in decreasing lift coefficient as shown in Figure 5, characterized by trailing edge vortex formation at $17.0^\circ$.

Development of fluid flow phenomena in dynamic conditions has been discussed in this section. The results of numerical computation is able to capture qualitative data in the form of traveling vortex phenomenon that characterizes the main features of dynamic stall. Predictions which is obtained can provide more detailed information on the development of fluid flow on the phenomenon of dynamic stall.

**Conclusion**

In this study, two URANS turbulence model, namely standard-ko and SST-ko is used to perform numerical simulations of fluid flow on an two-dimensional oscillating airfoil. Both turbulence models underpredicted the results when it is compared to experimental data. Standard-ko looks too dissipative to predict the position of maximum lift coefficient, which causes the phenomenon of dynamic stall to occur. Lower angle in this model. More accurate results is demonstrated by the use of the SST-ko model, where the position of maximum lift coefficient can be predicted better than standard-ko, except for high angle of attack where the 3D effects become more significant. The main characteristics of dynamic stall, such as the dominance of leading edge shedding...
edgevortexandliftcoefficienthysteresiscurvescan be capturedwellusingthis model. Nevertheless, the development of fluid flow phenomena of transition before and after the dynamic stall are less able to be accurately captured. In order toobtainmore detailedinformation, the useof more advancedCFDmethodssuch asLESandDESshould be made toaccommodatethe failure ofURANSin predictingthe flowphenomenainareaswithahighangle of attackdue tothe dominance of3D effects.

References


