A. Zanotti, S. Melone, R. Nilifard, A. D'Andrea
Experimental-Numerical Investigation of a Pitching Airfoil in Deep Dynamic Stall
Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, Vol. 228, N. 4, 2014, p. 557-566
doi:10.1177/0954410013475954

EXPERIMENTAL-NUMERICAL INVESTIGATION OF A PITCHING AIRFOIL IN DEEP DYNAMIC STALL

A.Zanotti[1], S.Melone[2], R.Nilifard[1], A.D'Andrea[2]

1. Politecnico di Milano, Dipartimento di Ingegneria Aerospaziale
Campus Bovisa, Via La Masa 34, 20156 Milano, Italy
2. AgustaWestland, Aerodynamics Department
via G.Agusta 520, Cascina Costa di Samarate (VA), Italy

Abstract:

The results of a comprehensive experimental campaign are compared to Computational Fluid Dynamics (CFD) simulations results to assess the modeling capabilities for a NACA 23012 pitching airfoil in deep dynamic stall regime. The experimental campaign involved fast unsteady pressure measurements and Particle Image Velocimetry. Two-dimensional simulations were carried out with EDGE, developed by FOI. The investigated test case consists in a sinusoidal pitching motion with a 10° amplitude and a reduced frequency of 0.1 around a mean angle of attack of 10°. The behaviour of the experimental lift and pitching moment coefficients is in close agreement with the two-dimensional simulations results, also during the downstroke motion where the flow field is characterised by severe unsteadiness conditions. A three-dimensional numerical model was built to evaluate the relevance of three-dimensional effects on the experiments. Three-dimensional simulations were carried out using the commercial code FLUENT. During upstroke motion, three-dimensional simulations results are in better agreement with the experiments, in particular in terms of the lift coefficient curve slope and of the pitching moment coefficient peak. The flow fields evaluated by PIV surveys show strong vortical structures moving on the airfoil upper surface during the downstroke motion that are captured only by the three-dimensional model; then, the flow fields comparison demonstrates the importance of three-dimensional effects for a deep dynamic stall condition.

Keywords: Dynamic stall, Oscillating airfoil, Computational Fluid Dynamics, Unsteady aerodynamics.

1 INTRODUCTION

The dynamic stall phenomenon has become in the recent years one of the more investigated topics in rotorcraft aerodynamics and aeroelasticity fields due to the strong demand for faster helicopters. In fact, several research activities both in experimental and numerical fields are currently focused on the design and develpment of rotor blades equipped with active [1, 2] and passive [3] devices to mitigate the detrimental effects on the helicopter performance produced by the dynamic stall on the retreating side of the rotor [4, 5, 6]. Consequently, the investigation of the fine details involved in this phenomenon has become the object of several experimental and numerical activities [7].

In particular, the prediction of the dynamic stall occurrence and its evolution is a very challenging activity due to the strong unsteadiness that characterises this phenomenon. Indeed, the research about dynamic stall phenomenon is currently focused on the evaluation of reliable numerical tools for the simulation of deep dynamic stall regime featuring the rapid formation, migration and shedding of strong vortices [8]. This dynamic stall regime is more interesting to be investigated, as it represents the most severe condition for helicopter performance. The numerical models require a thorough validation by means of comparison with experimental results, as done for instance by Klein et al. [9]. Recently, a relevant research topic in this field was the investigation of the importance of three-dimensional effects on dynamic stall experiments introduced by the intrinsically three-dimensional nature of phenomenon, by the use of a finite span model and by the presence of the wind tunnel side walls [9, 10, 11].

Due to the common interest about the development and test of new flow control devices integrated on real helicopter blade sections, a cooperation between Politecnico di Milano and AgustaWestland started on this topic. Wind tunnel tests were carried out at the Aerodynamics Laboratory of Politecnico di Milano. An experimental rig designed to test full-scale pitching blade sections at high forward flight speed was used for these tests. The experimental campaign was conducted on a NACA 23012 pitching airfoil in deep dynamic stall condition [4, 12] and it involved the use of different measurement techniques including unsteady pressure measurements, obtained from miniature pressure transducers installed on the midspan airfoil contour, and Particle Image Velocimetry (PIV). The use of the two techniques enabled to completely characterise the time-dependent flow field and to achieve a detailed insight of the different stages of the blade dynamic stall process.

Two-dimensional simulations were carried out by Politecnico di Milano using EDGE [14], a compressible Navier-Stokes solver developed at FOI, the Swedish Defense Research Agency. A three-dimensional numerical model was built by AgustaWestland to investigate the occurrence of three-dimensional effects on the experiments. Three-dimensional simulations were carried out using FLUENT [15] by ANSYS; the usage of a CFD commercial tool widely used within the industrial community, as FLUENT is, and its application for challenging dynamic stall assessments, represents an interesting element of the present work, as nowadays a lot of works were carried out on dynamic stall CFD modeling using dedicated solvers but very few or none were carried out using an industrial tool.

Figure 1: Layout of the experimental rig for pitching airfoils.

The comparison of the experimental results with two- and three-dimensional simulations ones enabled to assess the modeling capabilities facing with such peculiar flow conditions and to expose the relevance of three-dimensional effects on the deep dynamic stall phenomenon.

In section 2, the experimental set up is briefly outlined. Section 3, presents the description of the two- and three-dimensional CFD models used in this activity. Section 4 reports the comparison of the simulations against the experiments results. Final considerations and comments are given in section 5.

2 EXPERIMENTAL SET UP

The experimental activity was conducted at Politecnico di Milano in the low-speed closed-return wind tunnel of the Aerodynamics Laboratory. The wind tunnel has a rectangular test section with 1.5 m height and 1 m width. The maximum wind velocity is 55 m/s and the turbulence level is less than 0.1%.

The dynamic stall rig layout is presented in Fig. 1. The rig is composed of a blade section model supported by a motorized strut that can move it in pitch around its quarter chord. The NACA 23012 blade section model, with $c = 0.3$ and aspect ratio equal to 3.1, is mounted horizontally in the wind tunnel test section and it is pivoted about the quarter-chord position on two external tubular steel shafts positioned on two self-aligning bearings. The NACA 23012 airfoil was selected since, being a typical helicopter blade airfoil, it was already employed by other researchers in experimental activities about the dynamic stall on pitching blade sections [12, 13]. The aluminium machined blade section model presents an interchangeable midspan section for the different measurements techniques employed: one for PIV flow surveys and another for unsteady pressure measurement equipped with 21 pressure ports positioned along the midspan chord line. During the tests, end plates (with 350 mm chord and 120 mm height) were mounted at the model tips to reduce the interference of the wind tunnel walls boundary layer and to reduce the extremity effects. Nevertheless, wind tunnel tests of airfoils are unavoidably affected by side walls interference and in particular the flow easily becomes quite three-dimensional in stall conditions [12] also if the model spans all the test section width (without gap between walls and model tips) [16]; in fact, the non-slip condition on the walls highly perturbs the flow. A wall boundary layer suction system was considered quite complex to be implemented together with an oscillating airfoil system; indeed, the known reference experimental activities about dynamic stall on pitching airfoils were carried out without suction (see for instance [12, 18, 19, 21]).

The driving mechanism is composed of a brushless servomotor and a 12:1 gear drive shaft. Two encoders are directly mounted on the external shaft on the other side of the model with respect to the motor: a 2048 imp/rev absolute digital encoder is used for feedback control and a 4096 imp/rev incremental analog encoder is used to get the instantaneous position of the model. The model pitching motion is controlled by a proportional and derivative action using an interface software implemented in Labview.

2.1 Unsteady pressure measurements set up

The time histories of the lift and pitching moment during a pitching cycle were evaluated by the integration of the phase averaged pressures collected over 30 complete pitching cycles. The phase average of the pressure signals was computed using a bin of 0.1° angle of attack amplitude. The pressures were measured by means of 21 Kulite fast-response pressure transducers located inside the model interchangeable midspan section. The discretization error was estimated to be less than 2%. The positions of the pressure taps around the model midspan section are listed in Table 1 following a closed loop from the upper to the lower surface of the airfoil, starting from the leading edge.

Tap Number	1	2	3	4	5	6	7
Location x/c	0	0.01	0.044	0.096	0.164	0.28	0.358
Tap Number	8	9	10	11	12	13	14
Location x/c	0.453	0.618	0.76	0.9	0.9	0.767	0.628
Tap Number	15	16	17	18	19	20	21
Location x/c	0.459	0.373	0.285	0.185	0.118	0.06	0.02

Table 1: Pressure taps location on the NACA 23012 model midspan section [17].

The pressure transducers signals were acquired by a National Instrument compact data acquisition system equipped with six 24 bit A/D simultaneous bridge modules with 4 channels each. The pressure data from the transducers were acquired with a sampling rate of 50 kHz; the high sampling rate was chosen to capture the fine detail of the dynamic stall phenomenon that is characterised by severe unsteadiness conditions. The model angular position was measured simultaneously to the pressure transducers signals by means of the incremental encoder mounted directly on the model tubular shaft. The simultaneous acquisition of the encoder signals was used for the evaluation of the phase averaged airloads on a pitching cycle.

2.2 PIV set up

A double shutter CCD camera with a 12 bit, 1280 × 1024 pixel array equipped with a 55 mm lens was used to acquire the image pairs. The PIV flow survey covers the entire airfoil upper surface as the measurement field was composed of four 103 × 82 mm measurement windows, spanning the airfoil chord. The CCD camera was mounted on a traversing system guided by stepper motors to move the measurement window along the airfoil chord direction. The observation field was composed of smaller measurement windows to obtain image pairs with better resolution.

A Nd:Yag double pulsed laser, with 200 mJ output energy and wavelength 532 nm, was positioned on the roof of the test section. The laser was mounted on a single axis traversing system to move the light sheet along the wind tunnel flow direction. This solution allowed to use a laser sheet with smaller width and a higher energy level centered on each measurement window. A particle generator with Laskin nozzles was used for the flow insemination. The tracer particles, consisting in small oil droplets with diameter in the range of 1-2 μm, were injected just after the wind tunnel fan to fill the wind tunnel volume with homogeneous density. In order to filter out the effect of turbulent oscillations, the velocity flow fields were phase-averaged over 40 image pairs.

Additional details on the experimental rig and on the layout of the measurement techniques set up can be found in Zanotti and Gibertini [17].

3 CFD models

3.1 Two-dimensional model

The CFD solver used at Politecnico di Milano for the two-dimensional simulations was the EDGE code [14]. The code is capable of solving flows with different regimes from inviscid to fully turbulent using various turbulence models. RANS (Reynolds-Averaged Navier-Stokes), DES (Detached-Eddy Simulation) and LES (Large Eddy simulation) models in 2/3 dimensions are implemented. The RANS equations, considered in the present work, are solved using a node-centered finite volume scheme. The control volumes are formed by dual grid approach obtained from the control surfaces at the edges. Inviscid fluxes are discretized using central scheme with artificial scalar dissipation. A blend of second and fourth order differences is chosen as artificial dissipation which corresponds to a blend of first and third order differences for the fluxes. Time-accurate solutions are obtained using an implicit dual time-stepping technique with explicit sub-iterations and a three-point backward differencing formula. The flow equations in the pseudo time are integrated explicitly using a multi-stage Runge-Kutta scheme with convergence acceleration by agglomeration multigrid and implicit residual smoothing.

A 2D C-grid topology was used with a structured grid made of 318 980 quadrilateral elements (see Fig. 2)

Figure 2: Two dimensional grid for the NACA 23012 airfoil. The grid is made of 318 980 quadrilateral elements.

Figure 3: (a)Three dimensional simulation domain; (b)Two-dimensional grid at model midplane for the three dimensional simulations. The three-dimensional grid is made of 3 700 000 hexaedral elements.

with 190 nodes distributed along the direction normal to the airfoil surface. The dimensionless wall distance y^+ of the first grid point in the normal direction is less than one, which corresponds to a distance of about 10^{-5} c. The farfield boundary is located at 20 c away from the airfoil.

The URANS simulations were carried out using the Hellsten $k - \omega$ turbulence model recalibrated fully consistently with the EARSM (Explicit Algebraic Reynolds Stress Model) assumption by Wallin & Johansson to define the relation between Reynolds stresses and strain rate tensor [22, 23]. The dynamic viscosity was kept constant because the temperature differences are small within the flow field.

In all computations, a sinusoidal pitching motion was imposed on the whole grid which moves as a rigid body. In 2D simulations, 204 physical time steps per period were used, with a maximum number of 250 inner iterations in the pseudo-time. The numerical results were extracted after four full cycles to ensure a periodic state.

3.2 Three-dimensional model

Three-dimensional simulations were carried out at AgustaWestland using FLUENT version 12 [15]. A control-volume-based technique is used in the code to discretize the governing equations that are solved using a segregated solution method with a pressure-velocity coupling scheme. In the present work, a second order scheme was used for spatial discretization while a first order implicit integration was used fot temporal discretization. In order to evaluate the three-dimensional effects on the present experiments, the three-dimensional model reproduced the wind tunnel test section including the 35 mm gap between the side walls and the blade section model tips. The end plates were not considered in the numerical model. The whole domain (depicted in Fig. 3(a)) was split into two sub-domains: a rigid rotating domain enclosing the airfoil and a stationary outer domain. FLUENT "non-conformal" sliding mesh approach allowed to manage the interface between the two domains during rigid rotation. The baseline (2D) mesh for the three-dimensional model was a O-type structured mesh made of 160 000 quadrilateral elements, see Fig. 3(b). The wall normal resolution of the closest internal node from the airfoil surface was such as $y^+ \approx 1$. A structured mesh was used in the whole domain with a total amount of 3.7 millions elements. The free-stream conditions were imposed using pressure-far-field boundary conditions where, based on the Riemann invariants, only the free-stream Mach number and the static conditions

Figure 4: Comparison of experimental and numerical airloads for $\alpha = 10° + 10° \sin(\omega t)$, $k = 0.1$ (Re $= 6 \times 10^5$ and Ma $= 0.09$).

have to be specified. A no-slip adiabatic and a slip wall boundary conditions were imposed respectively along the airfoil surface and on the four test section walls.

For the three-dimensional simulations both fully turbulent $k - \omega$ SST model and free transition SST model were used and the results compared. Simulations were carried out for four cycles up to periodic steady state forces and moments convergence. Time resolution was set in order to have 628 physical time steps per period.

4 Numerical and experimental results

The dynamic stall condition considered in the present activity was a pitching cycle characterised by a mean angle of attack of $\alpha_m = 10°$ with constant oscillation amplitude of $\alpha_a = 10°$ and reduced frequency of $k = 0.1$. This condition corresponds to the deep dynamic stall regime according to the definition of McCroskey [4]. In fact, as it will be described by the present results, the tested pitching cycle is characterised by the vortex shedding phenomenon and therefore, by a large amount of airloads hysteresis, representing the predominant features of this regime.

The pitching condition was tested at relatively low speed (30 m/s), corresponding to a Reynolds number of Re $= 6 \times 10^5$ and a Mach number of Ma $= 0.09$. Such a low testing speed, which is far beyond the test rig capabilities, was chosen because the long run time required by the PIV measurement might have raised fatigue issues on the model strut.

Figure 4 shows the comparison between the experimental and numerical $C_L - \alpha$ and $C_M - \alpha$ curves. The experimental airloads coefficients are not corrected by the wind tunnel effects because well-established correction methods are not available for pitching airfoils tests, in particular for pitching cycles with high oscillation amplitudes. The standard deviation of the airloads coefficients are plotted on the experimental curves, as done in the recent works by Gardner et al. [19, 20]. The evaluation of the airloads standard deviation allows to evaluate the aperiodicity of the flow during the collected pitching cycles. In particular, the small values observed during the upstroke motion suggest the quite periodic behaviour of the flow during this phase of the motion. On the other hand, during the downstroke the flow is characterised by a high degree of unsteadiness due to the rapid formation and shedding of strong vortical structures. The rather chaotic behaviour associated to the flow structures typical of this phase of the motion explains the obtained higher values of the airloads coefficients standard deviation, in agreement with the results by Gardner et al. [19] for a deep dynamic stall test case.

Figure 5: Comparison of experimental and numerical C_L computed for 2D model with various time steps (a) and different grids (b): $\alpha = 10^\circ + 10^\circ \sin(\omega t)$, $k = 0.1$ (Re $= 6 \times 10^5$ and Ma $= 0.09$).

The airloads comparison shows some discrepancies between the 2D and 3D simulations results with respect to the wind tunnel data. During the upstroke motion, where the flow on the airfoil upper surface is fully attached, the lift coefficient curve slope obtained from three-dimensional simulations is closer to the experimental one with respect to the two-dimensional simulations results. For the three-dimensional simulations, the use of a free transition model does not introduce important modification or improvement with respect to the fully turbulent approach. In particular, during the upstroke, at low angles of attack, a local modification of the lift and moment coefficient curves slope can be observed for the free transition model. This local slope modification could be related with a possible laminar bubble, likely to what happen at such a low Reynolds and Mach numbers

During the downstroke motion the flow field features the formation of strong vortices that move on the airfoil upper surface, causing the large variations of the airloads. The two-dimensional model reproduces fairly well the oscillations of the lift and pitching moment curves observed in the experimental data. In particular, the computed airloads oscillations in downstroke present a delay in angle of attack with respect to the experimental curves that is higher for the three-dimensional models. Despite of this angle delay, during downstroke the peak of the pitching moment coefficient is well captured by three-dimensional fully turbulent model when compared to the experimental measurements.

A grid and time-step dependence assessment for the two-dimensional model is reported in Fig. 5 to support the suitability of the numerical model. Figure 5(a) presents the lift coefficient curves computed with 68, 136 and 204 time-steps per period, using the high resolution grid (HR 2D grid) made of 318 980 elements described in Sec. 3, compared to the experimental one. The numerical lift coefficient curves are overlapped for the whole upstroke motion, where the flow field is attached to the airfoil surface. In downstroke motion, featuring a significant flow separation and a high degree on unsteadiness, the lift coefficient curves presents some differences, but for $\alpha < 10^\circ$, corresponding to the part of the motion where the flow reattachment occurs, the C_L curves computed with 136 and 204 time-steps still overlap. On the base of this consideration, as supported by Wang et al. [24] in a recent work about a similar test case, the numerical solution can be considered converged with respect to time-step. Moreover, the good agreement between experimental results and the numerical ones obtained using 204 time-steps shows that the resolution in time used for the simulations is enough to represent quite well the dynamics of the stall.

Figure 5(b) presents the results of a grid dependence study for the 2D model, where simulations using 204 time-steps were carried out over the high resolution grid and a lower resolution grid (LR 2D grid with 108 380

Figure 6: Comparison of experimental and numerical flow fields for $\alpha = 10^\circ + 10^\circ \sin(\omega t)$, $k = 0.1$ (Re $= 6 \times 10^5$ and Ma $= 0.09$).

elements). Analogously to what observed in the time-step dependence study, the lift coefficient curves computed with the different grids are overlapped for the whole upstroke motion, showing in this phase of the motion a good convergence of the grid with respect to space. Otherwise in downstroke, the C_L curves still exhibit some discrepancies. Nevertheless, the closer agreement with the experimental results obtained over the finer grid supports the reliability of these numerical results. Similar conclusions about the required space resolution are reported in the work by Wang et al. [24].

Moreover, the grid dependence study on the 2D numerical model suggests that the use of a high resolution in space can improve model capabilities, in particular where the unsteadiness conditions become severe, as it occurs during the downstroke motion. Indeed in the deep dynamic stall regime, downstroke, with its highly separated flow and incipient reattachment region, is characterised not only by the leading edge vortex but also by important secondary vortex sub-structures that need to be resolved. This consideration could explain the better agreement with experimental airloads curves shown in downstroke by the 2D simulations with respect to the 3D simulations; indeed, the use of a finer 3D grid could probably recover the missing features of the present 3D model observed in terms of airloads behaviour during the downstroke. A 3D simulation carried out over a grid with more resolution would clarify this topic but, unfortunately, due to the unsteadiness of the problem, it represents a very demanding task with respect to the computation capabilities available for this activity. However, the present 3D numerical model can be considered as a good compromise between accuracy and computational time, as demonstrated by its capability to capture the overall experimental airloads behaviour. In particular, the proper computation of the experimental pitching moment peak and the aerodynamic damping associated to the C_M hysteresis loop [6] can be considered interesting results, as they represent important parameters to be evaluated for new blade airfoils performance assessment and for the development and sizing phase of new control devices for dynamic stall effects mitigation.

Moreover, in the present activity an important goal of the 3D simulations was the achievement of an insight on the relevance of 3D effects on the flow phenomena. With this aim, the results of the PIV surveys carried out at model midspan were compared to the flow fields obtained by the two- and three-dimensional simulations. In the following figures, the flow fields are shown by means of instantaneous streamlines patterns; the illustrated three-dimensional numerical flow field is evaluated at midspan.

Figures 6 and 7 show the comparison between the experimental and numerical flow field at four angles of attack of interest.

Figure 7: Comparison of experimental and numerical flow fields for $\alpha = 10° + 10° \sin(\omega t)$, $k = 0.1$ (Re $= 6 \times 10^5$ and Ma $= 0.09$, Continued).

At $\alpha = 18°$ in upstroke, corresponding to a post-stall condition in the steady case, the flow field on the airfoil upper surface is fully attached, as shown in Fig. 6(a). As it is well known, the flow separation is delayed by a reduction of the adverse pressure gradient produced by a kinematic induced camber effect due to the positive rapid pitching rate [5]. For this condition both the 2D and 3D numerical models reproduce fairly well the experimental flow field at midspan (see Figures 6 (c),(e) and (g)).

For the present test case, the fow separation does not occur during almost all the upstroke motion. In fact, the flow separation starts only at the end of the upstroke motion. The PIV flow field at $\alpha = 20°$ (Fig. 6(b)) shows a flow separation region which spans over more than half of the airfoil chord and it is characterised by small vortical structures within the shear layer. The 3D simulations capture the extension of the separation region on the airfoil upper surface better than the 2D model (see Figures 6 (d) and (h)) and in particular, the 3D free transition simulation is able to reproduce better the flow physics exhibited by the PIV survey (see Fig. 6 (f)).

During the downstroke motion the flow on the airfoil upper surface is fully separated and it is characterised by the formation, migration and shedding of strong vortical structures. At $\alpha = 18°$ in downstroke the measured flow field shows a small vortex near the leading edge and a very large vortex at about half of the chord, as it is illustrated in Fig. 7(a). In this case, the experimental flow field is fairly well captured by the 3D fully turbulent simulation (see Fig. 7 (c)), while, the 2D and 3D free transition numerical flow fields show two counter-rotating vortices near the trailing edge region (see Figures 7 (e) and (g)).

Also at $\alpha = 16°$ in downstroke the experimental flow field presents a very good agreement with the 3D fully turbulent numerical flow field. In fact, the simulation reproduces very well the large region of reversed flow on the upper surface, a small vortex located near the leading edge and a larger size vortex detaching the trailing edge, as it can be observed in Figures 7(b) and (d). The large recirculating flow region on the upper surface recalls air from the airfoil lower surface causing the formation of a counter-clockwise vortex located very close to the trailing edge. The 2D and 3D free transition numerical models are not capable to capture the experimental flow field in this condition (see Figure 7(f) and (h)).

A thorough explanation about the different behaviour exhibited in upstroke and in downstroke by the 3D simulations results with the free transition and the fully turbulent models with respect to the experimental flow fields is difficult to be argued. During the wind tunnel tests transition strips were not used on the model to enforce a turbulent boundary layer, therefore, a fair matching between the experimental flow fields and the

Figure 8: 3D fully turbulent simulation results for $\alpha = 10° + 10° \sin(\omega t)$, $k = 0.1$, (Re $= 6 \times 10^5$ and Ma $=$ 0.09): pressure coefficient distribution on the model surface and on midspan plane.

ones obtained with a laminar-turbulent transition model (as the SST free transition model) can be reasonably expected. Indeed, at the top of the upstroke motion, which features a small region of separation, the free transition model captures better the experimental flow field. Nevertheless, in downstroke motion, where the flow fields feature strong vortical structures traveling on the airfoil upper surface, the SST free transition model is not able to accurately reproduce the evolution of flow separation, probably due to the fact that this semi-empirical model has been accurately tuned for flow conditions where no large separation regions are present.

The comparison between the simulations and experimental results shows that strong three-dimensional effects influence the flow field over the pitching airfoil tested in the wind tunnel for a deep dynamic stall regime condition. The analysis of the 3D fully turbulent simulations results allows to point out the phase of the pitching cycle where these three-dimensional effects are important.

Figure 8 present the pressure coefficient distribution on the model surface and on the midspan plane for the four angles of attack considered in the flow field comparison; only half airfoil model is depicted, from the tip up to the midspan plane. At $\alpha = 18°$ during upstroke clearly no vortex structures are visible in the midspan plane and spanwise pressure distribution is quite uniform along the whole model span. Only at model tip a modification in the pressure spanwise distribution is visible due to the effects of the finite model span and the wind tunnel side walls. At $\alpha = 20°$ the vortical structures start to alter the spanwise pressure distribution. It can be understood that dynamic stall starts mainly from the model midspan section where a local slight modification in pressure distribution can be seen, while all over the model the pressure distribution seems to be quite uniform except, again, at the tip region. At $\alpha = 18°$ and $\alpha = 16°$ during downstroke, the strength of vortex structure observed at midspan plane is such that a complex three-dimensional structure is shed downstream causing a strong alteration of pressure distribution on the whole model surface up to the tip. It seems that the finite span model and the wind tunnel walls effects tend to prevent the propagation and formation of the vortex structure towards the outer part of the model where, roughly, the same pressure distribution can be observed with no evidence of leading edge vortical structures.

5 Conclusions

Two-dimensional and three-dimensional CFD models were developed for the NACA 23012 pitching airfoil in deep dynamic stall regime. The results obtained from simulations were compared to wind tunnel measurements of the airloads and to PIV flow surveys, to validate the capabilities of the numerical models when facing with such a demanding aerodynamic phenomenon. Moreover, the three-dimensional numerical model was used to point out the relevance of three-dimensional effects on the deep dynamic stall experiments.

For the considered deep dynamic stall condition the two-dimensional model reproduces quite well the measured airloads hysteresis and the rapid oscillations of the airloads occurring during the downstroke motion. However, the two-dimensional model does not reproduce the detailed physics and vortex dynamics that characterise the flow on the airfoil upper surface during the downstroke, as shown by the comparison with the PIV surveys at the model midspan. The results obtained using the three-dimensional models show a similar overall airloads behaviour, in particular, with a better agreement with the experiments during the upstroke motion but with some missing features in downstroke. An interesting issue shown by the 3D model in view of the development activity of dynamic stall control devices is its capability to compute fairly well the experimental pitching moment peak and the net aerodynamic damping over the pitching cycle.

Therefore, in the present activity the three-dimensional model can be considered a good compromise between accuracy and computational time.

The comparison of the flow field computed by 3D simulations with the PIV surveys results shows that three-dimensional effects characterise the aerodynamics of the pitching airfoil in the deep dynamic stall regime. These three-dimensional effects on the flow field are important in particular in post-stall conditions and they could be due to the intrinsic three-dimensional nature of dynamic stall but, in particular, they depends on the use of a finite-span wind tunnel model and on the wind tunnel side walls. Indeed, the analysis of the pressure coefficient distribution on the model surface computed by the 3D simulation shows that the finite model span and the wind tunnel side walls cause, in particular, a modification of the flow field at the tip, producing during the downstroke the alleviation of the vortical structure propagation typical of this dynamic stall regime.

Moreover, the 3D fully turbulent numerical flow field captures the fine details of the dynamic stall phenomenon observed by the PIV surveys during the downstroke motion and in particular, the formation and shedding of strong vortical structures that move fast on the airfoil upper surface. The free transition approach on the same 3D model fails to accurately reproduce the evolution of flow separation during the downstroke motion.

The quite good level of confidence in the numerical reproduction of wind tunnel experiments assesses the suitability of the numerical models to be used as an important tool for blade airfoils performance assessment as well as for the investigation of deep dynamic stall aerodynamics. In particular, the good reliability of a three-dimensional numerical model built using a CFD commercial tool as FLUENT can be considered another interesting result of the activity, assessing the suitability of an industrial tool facing with such a demanding challenging activity, up to now mainly coped with dedicated solvers.

APPENDIX

Nomenclature

α	angle of attack [deg]
α_m	mean angle of attack [deg]
α_a	pitching oscillation amplitude [deg]
δt	time interval
ω	circular frequency [rad/s]
b	blade section model span [m]
c	blade section model chord [m]
C_L	lift coefficient
C_M	pitching moment coefficient about the airfoil quarter chord
C_P	pressure coefficient
f	oscillation frequency [Hz]
k	reduced frequency $= \pi f c / U_\infty$
LR	Low Resolution
HR	High Resolution
Ma	Mach number
PIV	Particle Image Velocimetry
Re	Reynolds number
U_∞	free-stream velocity [m/s]
y	wall distance
y^+	dimensionless wall distance

References

[1] Singh C, Peake D, Kokkalis A, Khodagolian V, Coton F and Galbraith R. Control of Rotorcraft Retreating Blade Stall Using Air-Jet Vortex Generators. *Journal of Aircraft* 2006; 43: 1169-1176.

[2] Chandrasekhara M, Martin P and Tung C. Compressible Dynamic Stall Control Using a Variable Droop Leading Edge Airfoil. *Journal of Aircraft* 2004; 41: 862-869.

[3] Mai H, Dietz G, Geissler W, Richter K, Bosbach J, Richard H and de Groot K. Dynamic stall control by leading edge vortex generators. *Journal of American Helicopter Society* 2008; 56: 26-36.

[4] McCroskey WJ. The Phenomenon of Dynamic Stall. NASA TM 81264, 1981.

[5] Leishman JG. *Principles of helicopter aerodynamics.* Cambridge Aerospace Series, 2000.

[6] Carta FO. An Analysis of the Stall Flutter instability of Helicopter Rotor Blades. *Journal of American Helicopter Society* 1967; 12: 1-8.

[7] Mulleners K and Raffel M. The Onset of Dynamic Stall Revisited, *Experiments in Fluids* 2012; 52: 779-793.

[8] Visbal MR. Numerical Investigation of Deep Dynamic Stall of a Plunging Airfoil. *AIAA Journal* 2011; 49: 2152-2170.

[9] Klein A, Richter K, Gardner AD, Altmikus ARM, Lutz T and Kramer E. Numerical Comparison of Dynamic Stall for 2D-Airfoils and an Airfoil Model in the DNW-TWG, *37th European Rotorcraft Forum*, Gallarate (VA), Italy, 13-15 September, 2011.

[10] Spentzos A, Barakos G, Badcock K, Richards B, Wernert P, Schreck S and Raffel M. Investigation of Three-Dimensional Dynamic Stall Using Computational Fluid Dynamics, *AIAA Journal* 2005, 43: 1023-1033.

[11] Costes M, Richez F, Le Pape A and Gavriaux R. Numerical investigation of three dimensional effects during dynamic stall, *37th European Rotorcraft Forum*, Gallarate (VA), Italy, 13-15 September 2011.

[12] Leishman JG. Dynamic stall experiments on the NACA 23012 aerofoil. *Experiments in Fluids* 1990; 9: 49-58.

[13] Raffel M, Kompenhans J, Stasicki B, Bretthauer B and Meier GEA. Velocity measurement of compressible air flows utilizing a high-speed video camera. *Experiments in Fluids* 1995; 18, 204-206.

[14] Eliasson P. EDGE, a Navier-Stokes solver for unstructured grids. *Proc. to Finite Volumes for Complex Applications III* 2002; 1: 527-534.

[15] ANSYS FLUENT 12.0, Getting Started Guide, ANSYS Inc., 2009.

[16] Nishino T and Shariff K. Numerical Study of Wind-Tunnel Sidewall Effects on Circulation Control Airfoil Flows, *AIAA Journal* 2010; 48: 2123-2132.

[17] Zanotti A and Gibertini G. Experimental investigation of the dynamic stall phenomenon on a NACA 23012 oscillating airfoil, *Proc of IMechE, Part G: Journal of Aerospace Engineering*,in press, doi:10.1177/0954410012454100.

[18] Raffel M, Kompenhans J and Wernert P. Investigation of the unsteady flow velocity field above an airfoil pitching under deep dynamic stall conditions. *Experiments in Fluids* 1995; 19, 103-111.

[19] Gardner AD, Richter K, Mai H, Altmikus ARM, Klein A and Rohardt CH. Experimental Investigation of Dynamic Stall Performance for the EDI-M109 and EDI-M112 Airfoils, *37th European Rotorcraft Forum*, Gallarate (VA), Italy, 13-15 September 2011.

[20] Gardner AD, Richter K, Mai H and Neuhaus D. Experimental Control of Compressible OA209 Dynamic Stall by Air Jets, *38th European Rotorcraft Forum*, Amsterdam, The Netherlands, 4-7 September 2012.

[21] Carr LW, McAlister KW and McCroskey WJ. Dynamic stall Experiments on the NACA 0012 Airfoil, NASA TP-1100, 1978.

[22] Wallin S. Engineering turbulence modeling for CFD with a focus on explicit algebraic Reynolds stress models, Doctoral Thesis Norsteds Tryckeri AB, Stockholm, Sweden, 2003.

[23] Hellsten A. New advanced $k-\omega$ turbulence model for high-lift aerodynamics, *AIAA Journal* 2005; 43: 1857-1869.

[24] Wang S, Ingham D, Ma L, Pourkashanian M and Tao Z. Turbulence modeling of deep dynamic stall at relatively low Reynolds number, *Journal of Fluids and Structures* 2012; 33: 191-209.