Aerospace Science and Technology 47 (2015) 216–237

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Numerical investigation of three-dimensional effects during dynamic stall

M. Costes *, F. Richez, A. Le Pape, R. Gavériaux

ONERA, The French Aerospace Lab, F-92190 Meudon, France

A R T I C L E I N F O

Article history:
Received 27 February 2015
Received in revised form 15 July 2015
Accepted 21 September 2015
Available online 30 September 2015

A B S T R A C T

The numerical simulation of dynamic stall around a 3D finite-span oscillating wing of constant OA209 airfoil section is compared to experimental results obtained in the ONERA F2 wind-tunnel, assuming fully turbulent flow. A deep dynamic stall case is considered for a reduced frequency typical of helicopter problems. A detailed comparison with the experimental data available (unsteady pressure distribution and velocity field) shows that the main flow features are captured by the numerical simulation, more especially the large spanwise flow component induced by separation.

1. Introduction

Dynamic stall is one of the most difficult aerodynamic problems encountered in helicopter rotors. Its origin lies in the dissymmetry between the advancing and the retreating blade in forward flight. The lateral trim of the helicopter requires decreasing the blade pitch angle on the advancing blade and increasing it on the retreating blade. When the forward speed or the load factor of the helicopter is high enough, the retreating blade is led to operate above its static stall angle, this condition being met by the blades once per revolution. Dynamic stall allows the blade to increase its maximum lift capabilities with respect to static stall, but it also induces large negative pitching moments and consequently strong pitch-link and vibratory loads, thus limiting the flight domain of the helicopter.

The quest for improving rotor performance and for reducing helicopter operation cost may rapidly lead helicopter designers to be confronted with dynamic stall problems, e.g. at high speed or at reduced rotor RPM. It is thus necessary to better understand the complex physics of dynamic stall and to develop numerical simulation capabilities of sufficient fidelity in order to master stall-induced problems in rotorcraft design. Indeed, numerous fundamental problems are raised by dynamic stall, such as flow separation on a smooth surface, laminar-turbulent transition created either by instabilities or downstream a laminar separation bubble, unsteady and turbulent separated flows, etc. At ONERA, a significant research effort has been devoted to dynamic stall studies during the last decade, in cooperation with the German aerospace research centre DLR. The goal of these activities was to develop a better knowledge of dynamic stall phenomena together with the capability to simulate them, in order to be able to develop the techniques to control this configuration, by means of active or passive devices, afterwards [1–5].

In this research, detailed wind-tunnel testing of the OA209 airfoil under dynamic stall conditions was performed in the ONERA low-speed F2 wind-tunnel. These series of tests have allowed measuring the pressure, velocity field, skin-friction around several static and oscillating models for a range of representative Reynolds numbers and reduced frequencies. Most of the tests were performed for 2D flow conditions, the wing covering the full span between the wind-tunnel walls. Some of them also concerned a 3D wing in order to investigate finite-span effects on dynamic stall, for different sweep angles.

Up to now, most of the published work mentioned above concerns 2D airfoil test data that have been used to validate Computational Fluid Dynamics (CFD) methods at dynamic stall. The results obtained so far have shown the importance of transition effects in the Reynolds Averaged Navier–Stokes (RANS) models in order to predict the onset of stall correctly. Nevertheless, when massive flow separation is obtained, too strong dynamic stall vortices are shed over the airfoil, so that the negative pitching moment is significantly overestimated by numerical predictions. As a matter of fact, flow separation is known to generate three-dimensional effects, and it is important to see how these three-dimensional phenomena are predicted by CFD. This is the topic of the present paper.

The computation of 3D dynamic stall is indeed quite uncommon in the literature, especially for high Reynolds number flows. During the last decade, a number of researchers have begun to consider three-dimensional simulations of 2D oscillating airfoils

---

* Corresponding author.
E-mail address: mcostes@onera.fr (M. Costes).

http://dx.doi.org/10.1016/j.ast.2015.09.025
1270-9638/

experiments, considering that the physics of separated flows is inherently three-dimensional. This was more especially the case when hybrid RANS/LES methodologies were first used for this problem (Szydlowski et al. [6], Martinat et al. [7]), but although the occurrence of three-dimensional structures could be noted in their simulations, it is clear that the mesh resolutions used at that time were too reduced, more especially along the spanwise direction in terms of distance or spacing, and the computed flow was mostly two-dimensional. More recently, Wang et al. [8] have compared 2D and 3D URANS simulations using various turbulence and transition models with DDES simulations for a low-Reynolds number case ($\mathrm{Re} = 10^5$), showing no significant gain of the 3D approach, the space and time resolutions used being still too coarse to capture the complex physical phenomena with massive flow separation. Three-dimensional simulations of 2D dynamic stall experiments have also been undertaken with URANS simulations only. Klein et al. [9] computed the complete airfoil setup of their experiment in the DNW-TWG wind-tunnel taking into account the lateral walls of the tunnel. However, although great care was put in getting turbulent boundary layer profiles of the lateral walls, too strong corner flow separation was obtained and the computed 3D solutions did not correlate better with experiment than their 2D counterparts. This difficulty of RANS methods to predict corner separation is a well-known problem [10]. Zanotti et al. [11] compared 2D and 3D dynamic stall simulations of their own experiment without considering the wind-tunnel walls, the 3D simulations being carried out over 1.5 chord spanwise extension and applying an inviscid wall condition over the lateral sides of the 3D mesh. Their 3D results slightly improved the comparison with experiment, showing a reduced over-prediction of the pitching moment peak at stall, and a better correlation of the computed velocity field with PIV. Nevertheless, 3D URANS simulations are known to be dependent on the size of the spanwise discretization [12], and this factor was not addressed in the present work.

As far as the dynamic stall of finite-span wings is concerned, first comparisons between CFD analysis and the experimental data of Piziali [13] were presented by Ekaterinaris [14]. However, the coarse mesh used at that time, the moderate stall configuration considered and the availability of surface pressure data only for comparison gave limited information about the specificities of dynamic stall for a 3D finite-span oscillating wing. The work of Spentzos et al. [15,16] about 10 years later made a significant step forward on this problem, although the number of mesh points used in the computations was still moderate according to nowadays standards. In [15], the capability of the simulation to predict the Ω-shaped vortex resulting from the interaction between the dynamic stall vortex and the wing tip vortex was demonstrated, and a comparison of 2D and 3D dynamic stall showed the milder character of 3D dynamic stall versus its 2D counterpart. In [16], several wing planforms were computed and the three-dimensional structure of the flow field was investigated. The interaction between the dynamic stall vortex and the wing tip vortex was more particularly emphasized, showing strong similarities between all planforms considered and also good qualitative agreement with experiment.

For the last years, numerical research activities in 3D dynamic stall have also concerned rotor applications, more especially in the US with the UH-60 flight tests database which includes a high-loaded case. Sitaraman et al. [17] coupled the TURNS CFD method to UMARC comprehensive analysis, using Chimera overset structured grids to discretize the aerodynamic field, with more than 13 million grid points. The URANS equations were solved using the Spalart–Allmaras one-equation turbulence model with rotation correction, and a loose coupling between TURNS and UMARC. The dynamic stall on the retreating side of the blade was qualitatively well captured, but with a phase shift variable in span and azimuth between computation and experiment. Furthermore, the magnitude of lift and moment stall peaks was generally not well predicted, although the order of magnitude was correct. Potsdam et al. [18] also applied the loose coupling methodology between OVERFLOW-D and CAMRAD II to the same configuration. Chimera overset structured grids of about 26 million points were used, together with the Baldwin–Barth one-equation turbulence model to solve the URANS equations. The results were also in good qualitative agreement with test data, more especially as far as pitching moment is concerned, although similar discrepancies in terms of phase and peak magnitude as those noted above could be found. The predicted location of stall on the rotor disc was in fairly good agreement with flight test data, both showing the progression of dynamic stall occurrence from blade midspan to outboard. In [19], Sankaran et al. applied the Helios computational platform loosely coupled with RCAS comprehensive analysis to the UH-60A dynamic stall case. Unstructured near-body meshes overset with Cartesian background grids are used to solve the URANS equations with the one-equation Spalart–Allmaras turbulence model. Again, the dynamic stall on the retreating blade side is qualitatively captured, the fine adapted mesh with more than 150 million nodes showing a concentration of vorticity in the fourth quadrant of the rotor disc. Finally, Biedron et al. [20] applied the FUN3D unstructured CFD methodology loosely coupled with CAMRAD II comprehensive analysis to the same problem, using again the Spalart–Allmaras turbulence model to close the URANS set of equations. The results obtained were similar to those discussed above, showing a good qualitative prediction of dynamic stall but differences in phase and in peak values of lift and pitching moment. However, refining the grid from more than 17 million nodes to values above 37 million nodes degraded moment stall prediction. Adding the fuselage in the computation did not show any improvement. Finally two other turbulence models were tested on the finest mesh (Menter SST, HRLES) with small effect on the predicted loads, the SST model providing however the best prediction of the second pitching moment stall. In all this published work [17–20], fairly big time steps were used (between 0.25° and 1° of azimuth) with respect to the dynamics of separated flows, which may not be sufficient to pick the details of dynamic stall. Furthermore, this assessment of methodologies relies on blade surface and integrated data (integrated pressure, rotor loads ...) only, giving therefore limited insight into the details of the dynamic stall mechanisms in this complex environment.

The objective of the present work is twofold. First, it aims to compare 3D finite-span oscillating wing computations with experimental data obtained in the ONERA F2 wind-tunnel for an oscillating finite-span wing under dynamic stall conditions. Second, both numerical and experimental data are used to better describe the unsteady separated flow occurring on the upper surface of the wing. Ideally, when dealing with unsteady massive separation and high-Reynolds number flows, advanced simulation techniques such as LES or hybrid RANS/LES are expected to provide the more accurate results possible with current computer technology. However, although a few applications of such kind of methodology have been mentioned earlier, even in a “simple” hybrid ZDES approach the requirements on mesh resolution are very demanding ($\Delta x^+ \leq 200$ in the streamwise direction, $\Delta y^+ \leq 1$ in the wall normal direction and $\Delta z^+ \leq 200$ in the spanwise direction), leading to a number of mesh points greater than 150 millions on the present configuration [21]. The requirements on time discretization are even more stringent since, with a maximum CFL number smaller than 15 for accurately computing the advection of turbulent structures, time steps of about 0.1 μs are required, while the wing oscillates at frequencies of the order of 1 Hz. As a result, 10 million time steps per cycle are necessary, and even if the data is phase-averaged over only five oscillating periods – which is far