European Journal of Mechanics B/Fluids 55 (2016) 380–393

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

journal homepage: www.elsevier.com/locate/ejmflu

Numerical study of the turbulent transonic interaction and transition location effect involving optimisation around a supercritical aerofoil

Damien Szubert [a,*], Ioannis Asproulias [a], Fernando Grossi [a], Régis Duvigneau [b], Yannick Hoarau [c], Marianna Braza [a]

[a] *Institut de Mécanique des Fluides de Toulouse, UMR N° 5502 CNRS-INPT-UPS, Allée du Prof. Camille Soula, F-31400 Toulouse, France*

[b] *INRIA Sophia Antipolis - Méditerranée, ACUMES Team, France*

[c] *Laboratoire ICUBE, UMR N° 7357, Strasbourg, France*

ARTICLE INFO

Article history:
Available online 22 October 2015

ABSTRACT

The present article analyses the turbulent flow around a supercritical aerofoil at high Reynolds number and in the transonic regime, involving shock-wave/boundary-layer interaction (SWBLI) and buffet, by means of numerical simulation and turbulence modelling. Emphasis is put on the transition position influence on the SWBLI and optimisation of this position in order to provide a maximum lift/drag ratio. A non-classical optimisation approach based on Kriging method, coupled with the URANS modelling, has been applied on steady and unsteady flow regimes. Therefore, the present study contributes to the so-called 'laminar-wing design' with the aim of reducing the drag coefficient by providing an optimum laminar region upstream of the SWBLI.

1. Introduction

The present study has been carried out in the context of the European research program TFAST, "Transition location effect on shock-wave/boundary-layer interaction", project No. 265455. One of the main objectives of this research is to provide optimal laminarity in the boundary layer upstream of the shock-wave/boundary-layer interaction (SWBLI), in order to reduce the skin friction comparing to the fully turbulent case and therefore reduce drag, in the context of greening aircraft transport (a major objective of the *Horizon 2020* European programme). Due to increased aerodynamic loads and aero-engine components nowadays, supersonic flow velocities are more frequent, generating shock waves that interact with boundary layers. Laminar shock-wave/boundary-layer interaction can rapidly cause flow separation, which is highly detrimental to aircraft performance and poses a threat to safety. This situation can be improved by imposing the laminar–turbulent transition upstream of the interaction, but this should be carefully done in order to keep the aerodynamic efficiency high (lift/drag ratio).

In the context of the European research program TFAST, several ways of controlling the position of the transition are carried out. To this end, a supercritical laminar wing, the so-called V2C, has been designed by Dassault Aviation. This profile allows the boundary layer to remain laminar up to the shock foot, even in the environment of transonic wind tunnels of the laboratories involved in the project, and up to the angle of attack of 7.0°. Experimental results for the present configuration are not yet available in the present research project. Regarding the related literature, the transonic buffet has been studied experimentally in detail since the 70s on circular-arc aerofoils [1,2], and most recently on supercritical aerofoils [3]. In this latest study, a fixed transition tripping was applied at 7% of the chord. The physics governing the transonic buffet is complex and several theories have been proposed, like the effect of the feedback mechanism of waves propagating from the trailing edge, or the onset of a global instability [3–5]. Comparison of numerical results by Deck [6], Grossi et al. [7] and Szubert et al. [5] with the experimental results by Jacquin et al. [8] concerning the transonic buffet around supercritical wings with fixed transition showed the predictive capability of recent CFD methods and a physical analysis of the interaction between buffet and trailing-edge instabilities. The SWBLI involving transonic buffet and laminar wing design currently highly interests the aeronautical industries (Cleansky European project, "Advanced, high aspect ratio transonic laminar wing" [9]). Laminar wing design in transonic regimes has been studied in respect of transition control by means of Discrete Roughness Elements (DRE's) [10]. Navier–Stokes simulations of transonic buffet as well as of the shock–vortex

* Corresponding author.
E-mail address: damien.szubert@imft.fr (D. Szubert).

http://dx.doi.org/10.1016/j.euromechflu.2015.09.007
0997-7546/

interaction at moderate Reynolds numbers were reported by Bouhadji and Braza [11], as well as DNS by Bourdet et al. [12]. In the high Reynolds number range, typical of aerodynamic applications, the use of appropriate turbulence modelling is necessary. Concerning transonic buffet, the unsteady shock-wave/boundary-layer interaction represents a major challenge for turbulence models and the low frequencies associated with the shock-wave motion can make the simulations very expensive. Since the first simulations by Seegmiller et al. [2] and Levy Jr. [13] for a circular-arc aerofoil, Unsteady Reynolds-Averaged Navier–Stokes (URANS) computations using eddy-viscosity turbulence models have been largely used to predict the phenomenon over two-dimensional aerofoils. Pure LES simulations, even combined with specific wall-models, are yet quite costly for the high Reynolds number range of real flight configurations. For this reason, hybrid RANS–LES methods have been developed in the last decade and start to be largely used in the industrial context together with adapted, advanced URANS approaches. The hybrid methods combine the robustness and near-wall physics offered by URANS in the near region, as well as LES advantages in capturing the physics of unsteady vortices and instabilities development in the detached flow regions. Among the hybrid methods, the Detached-Eddy Simulation (DES) does not need to impose the interface between the statistical and LES regions. This is provided inherently by the choice of the turbulence length scale to use in the transport equations [14]. In order to avoid approaching the near-wall region by the LES zone, the Detached-Eddy Simulation has been improved in respect of the turbulence length scale, ensuring a quite significant statistical zone around the body, in the context of the Delayed Detached-Eddy Simulation (DDES) [15]. Moreover, improvement of the near wall modelling has been achieved by means of a suitable Wall-Modelled LES (WMLES) in order to allow the flow physics modelling in the very near wall region covering the viscous sublayer by means of finer grids (but more economic than the LES) in the context of the Improved Delayed Detached-Eddy Simulation (IDDES) [16]. Regarding the transonic buffet simulations, Deck [6] has used a successful zonal DES approach, using mostly statistical modelling in the outer regions far from the body. He provided a detailed prediction of the transonic buffet around the supercritical aerofoil OAT15A. Regarding the same configuration, Grossi et al. [7] performed a Delayed Detached-Eddy Simulation in the context of the ATAAC (Advanced Turbulence Simulations for Aerodynamic Application Challenges) European programme. This study succeeded in the prediction of the shock-wave self-sustained motion near the critical angle of incidence for the appearance of buffet, based on experimental results by Jacquin et al. [3,8]. Moreover, Szubert et al. [5] provided a detailed analysis of the buffet dynamics by means of the Organised Eddy Simulation (OES) approach, resolving the organised coherent modes and modelling the random turbulence background and using upscale turbulence modelling through stochastic forcing in order to keep the turbulent–non-turbulent shear-layer interfaces thin. In the present paper, the transonic buffet is applied on the V2C aerofoil within the TFAST program, at 7.0°, the maximum angle of attack allowed by the design, upstream Mach number 0.70 and Reynolds number 3.245×10^6. The fully turbulent case is studied by different URANS and DDES modelling in two and three dimensions respectively. The predictive capabilities of statistical and hybrid turbulence modelling approaches are discussed. A 2D study is first carried out to investigate the main flow characteristics in respect of the angle of attack as well as the influence of the transition location. The transition location effects are also studied in the buffeting regime, by imposing the laminarity at several positions. Based on these results, the main objective of the present article is to put ahead a coupling of the aforementioned CFD methods with a non-classical optimisation approach of the transition location in the steady and unsteady transonic regimes in respect of the drag reduction and lift-to-drag ratio maximisation.

2. Numerical method and turbulence modelling

2.1. Flow configuration

Concerning the design of the V2C wing, it was validated numerically by Dassault on a 0.25 m-chord length (c) profile by means of RANS computations for various angles of attack at freestream Mach numbers of 0.70 and 0.75, yielding chord-based Reynolds numbers of approximately 3.245×10^6 and 3.378×10^6 respectively. The study was performed using a compressible Navier–Stokes code adopting a two-layer $k - \varepsilon$ model, with the transition location being determined from the fully-turbulent flowfield using a three-dimensional compressible boundary-layer code by means of the N-factor amplification with a parabola method. The technique employed for laminarity and an initial design in respect of the transition prediction was based on the e^N method (Ref. [17] for instance). The aerofoil surface was generated in such a way that the N-factor remains small for low-to-moderate turbulence intensity levels, similar to the wind tunnel turbulence levels used for the present test-case for the experimental study currently in progress in the TFAST project. At Mach number 0.70, the flow separated between $\alpha = 6°$ and 7°. The amplification factor N was shown to be smaller than 3 up to the shock wave, thus guaranteeing laminar flow. At Mach number 0.75, the value of N remained smaller than 2 up to $\alpha = 7°$. For this Mach number, there was not buffeting phenomenon, whatever the angle of attack. Moreover, for incidences higher than 1°, the shock induces a separation of the boundary layer up to the trailing edge.

2.2. Numerical method

The simulations of the V2C configuration at upstream Mach number $M = 0.70$ and Reynolds number $\mathrm{Re} = 3.245 \times 10^6$ have been carried out with the Navier–Stokes Multi-Block (NSMB) solver. The NSMB solver is the fruit of a European consortium that included Airbus from the beginning of 90s, as well as main European aeronautics research institutes like KTH, EPFL, IMFT, ICUBE, CERFACS, Karlsruhe Institute of Technology, ETH—Swiss Federal Institute of Technology in Zurich, among others. This consortium is coordinated by CFS Engineering in Lausanne, Switzerland. NSMB solves the compressible Navier–Stokes equations using a finite volume formulation on multi-block structured grids. It includes a variety of efficient high-order numerical schemes and turbulence modelling closures in the context of URANS, LES and hybrid turbulence modelling. NSMB includes efficient fluid–structure coupling for moving and deformable structures. For the present study, the third-order of accuracy Roe upwind scheme [18] associated with the MUSCL flux limiter scheme of van Leer [19] is used for the spatial discretisation of the convective fluxes. A similar upwind scheme (AUSM) was used by Deck [6]. For the diffusion terms, second-order central differencing has been used. The temporal discretisation has been done by means of dual-time stepping and of second order accuracy. A physical time step of 5 μs has been adopted for 2D simulations. For the 3D simulations, the time step has been reduced to 0.1 μs after detailed numerical tests. A typical number of inner iterations of 30 was necessary for the convergence requirements in each time step.

The 2D grid has a C–H topology, and is of size 163,584 cells. The downstream distance of the computational domain is located at a mean distance of 80 chords from the obstacle. A grid refinement study has been carried out, by means of steady-state computations and using local time stepping, for the flow at $M_\infty = 0.70$ and $\alpha = 4.0°$ using the $k - \omega$ SST model [20] and assuming fully-turbulent boundary layer, with two other grids: one 50% coarser, and another 30% finer. Detailed results of this convergence study can be found in [7]. The grid retained for the present study gave