Aerospace Science and Technology 20 (2012) 38–51

Contents lists available at SciVerse ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

CFD analysis of the flow around the X-31 aircraft at high angle of attack

O.J. Boelens [1]

Department of Flight Physics and Loads, Aerospace Vehicles Division, National Aerospace Laboratory NLR, P.O. Box 90502, 1006 BM, Amsterdam, The Netherlands

ARTICLE INFO

Article history:
Available online 29 March 2012

Keywords:
CFD
X-31
Vortical flow

ABSTRACT

Accurate and cost-effective Computational Fluid Dynamics (CFD) methods play an increasingly important role, for example in the support of fighter aircraft operations. Prior to deployment of such CFD methods they should be well validated and evaluated against state-of-the-art wind tunnel and/or flight test data. The wind tunnel experiments on the detailed X-31 model performed in the DNW Low-Speed-Wind-Tunnel Braunschweig (DNW-NWB) provide an excellent data set for validation and evaluation purposes. This data set has been investigated in the framework of NATO RTO task group AVT-161 'Assessment of Stability and Control Prediction Methods for NATO Air & Sea Vehicles'. The National Aerospace Laboratory NLR participated in this task group using its in-house developed flow simulation system ENFLOW, which includes both grid generation tools and a flow solver. The focus of the present paper is on the question to what extend leading edge details, flap gaps, need to be taken into account for the X-31 wind tunnel model to properly simulate the flow around this configuration. To investigate this question, three leading edge configurations have been considered, i.e. one with all leading edge flap gaps, one with only the longitudinal flap gaps and one with no leading edge flap gaps. Results obtained for selected test conditions measured during test run VN01004 ($M = 0.18$ and $Re_{m.a.c.} = 2.07 \times 10^6$) of the wind tunnel experiments will be discussed. Properly modeling geometrical details of the wind tunnel model at the leading edge is essential in obtaining the vortical flow phenomena observed in the wind tunnel. Analysis of the pitching moment coefficient demonstrates how in case of not resolving geometrical details a seemingly correct behavior is obtained without, however, resolving the underlying flow physics correctly.

1. Introduction

Cost-effective Computational Fluid Dynamics (CFD) methods with sufficient accuracy are complementary to experimental methods and play an increasingly important role in simulating manoeuvre conditions. Examples are conditions that cannot be simulated in a wind tunnel or are too dangerous to be performed in flight tests. Stability and control characteristics can be obtained for such conditions using CFD. Prior to the usage of the computed stability and control characteristics the CFD methods should be well validated and evaluated against state-of-the-art wind tunnel and/or flight test data.

The proper selection of the level of geometrical detail to be used in a CFD simulation has a large impact on the cost-effectiveness. Including more geometrical detail gives rise to a more complex and thus more expensive grid generation task. Incorporating more geometrical detail will also result in grids with more cells. Since the computing time and hence the computational cost depend directly on the number of grid cells, incorporating more geometrical detail will result in more costly simulations. Omitting these geometrical details may however result in inaccurate or even wrong results. It is well known that geometrical details of the wing leading edge geometry, i.e. the leading edge snag, have triggered a bifurcation in the flow in the first version of the updated F/A-18, Refs. [6] and [5]. Failure of predicting this bifurcation phenomenon during the design stage resulted in a severe penalty in performance, and gave rise to substantial redesign costs.

Another aspect impacting cost-effectiveness is the proper selection of the physical modeling used. Employing a too simple physical modeling may result in not resolving all physics and hence inaccurate or wrong results. Using a too complex physical model will lead to excessive computational times and thus excessive computational costs. The present paper highlights these aspects for the complex flow over the challenging X-31 wind tunnel model geometry.

The wind tunnel experiments on the detailed X-31 model, see Fig. 1, performed in the framework of the DLR project SikMa-"Simulation of Complex Maneuvers" [7,14,13] provide an excellent data set for validation and evaluation purposes. During the course of this project several wind tunnel entries have been executed in the DNW Low-Speed-Wind-Tunnel Braunschweig (DNW-NWB, 3.25 m × 2.80 m). These entries were specifically aimed at generating high quality data which could be used in a later stage for

---

E-mail address: boelens@nlr.nl.

[1] R&D Engineer, Applied Computational Fluid Dynamics.

1270-9638/$ – see front matter © 2012 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.ast.2012.03.003

Nomenclature

AVT	Applied Vehicle Technology (one of the seven panels within RTO)
CFD	Computational Fluid Dynamics
C_p	pressure coefficient $(= (p - p_\infty)/(1/2\rho_\infty u_\infty^2))$
DLR	Deutsches Zentrum für Luft- und Raumfahrt, German Aerospace Center
DNW	German–Dutch Wind tunnels
EARSM	Explicit Algebraic Reynolds Stress Model
FAS	Full Approximation Storage
FMG	Full Multi-Grid
k	turbulent kinetic energy
L_{ref}	reference length
M	Mach number
$m.a.c.$	mean aerodynamic chord
NATO	North Atlantic Treaty Organization
NLR	Nationaal Lucht- en Ruimtevaartlaboratorium, Netherlands National Aerospace Laboratory
p	pressure
p_∞	free-stream pressure
RANS	Reynolds-Averaged Navier–Stokes
Re	Reynolds number
RTO	Research and Technology Organization – scientific arm of NATO
TNT	Turbulent Non-Turbulent
u	velocity
u_∞	free-stream velocity
X-LES	Extra-Large Eddy Simulation
x	distance along the model body axis, positive aft
y	distance along the span, positive outward
y^+	Re-like term for flat plate turbulent boundary layer
α	angle of attack, °
β	side-slip angle, °
ρ	density
ρ_∞	free-stream density
ω	specific turbulent dissipation rate

Fig. 1. X-31 model in rear sting configuration in DNW Low-Speed-Wind-Tunnel Braunschweig (DNW-NWB) (photo by courtesy of Deutsches Zentrum für Luft- und Raumfahrt, DLR).

the validation of CFD methods. Both steady-state measurements as well as simulations of complex manoeuvres employing a test rig with 6 degrees of freedom have been performed.

This data set has been provided by Deutsches Zentrum für Luft- und Raumfahrt DLR to the partners participating in NATO RTO task group AVT-161 'Assessment of Stability and Control Prediction Methods for NATO Air & Sea Vehicles' [8,12]. The objectives of this task group are defined as follows:

(i) to assess the state-of-the-art in computational fluid dynamics methods for the prediction of static and dynamic stability and control characteristics of military vehicles in the air and sea domains, and
(ii) to identify shortcomings of current methods and identify areas requiring further development.

The National Aerospace Laboratory NLR participates in this task group using its in-house developed flow simulation system ENFLOW [2], which includes both grid generation tools and a flow solver.

This article concentrates on the question to what extend leading edge details, flap gaps, need to be taken into account for the X-31 wind tunnel model to properly simulate the flow around this configuration. The layout of the article is as follows. The wind tunnel experiments will be discussed in Section 2. The focus will be on a test run which shows a sudden drop in the pitching moment coefficient between angles of attack α of 17° and 23°, whereas both the lift and drag coefficient increase monotonously for those angles. The grid generation will be discussed in Section 3. Section 4 will discuss the important features of the flow solver ENSOLV, which is part of the flow simulation system ENFLOW. Section 5 will discuss the results obtained at NLR for the above mentioned test run. A section with conclusions (Section 6) completes the paper.

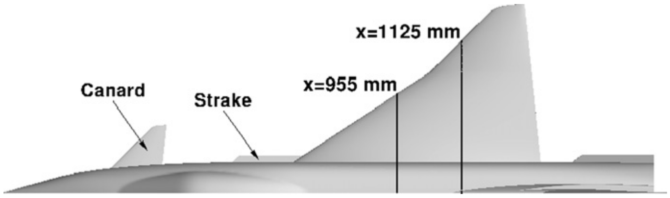

Fig. 2. Pressure sensor locations: $x = 955$ mm (60% chord) and $x = 1125$ mm (70% chord).

2. Wind tunnel experiments

The model used during the wind tunnel experiments [7,14,13] is based on the X-31 experimental high angle-of-attack aircraft configuration, see Fig. 1. The model is equipped with control devices driven by remotely controlled internal servo-engines to deflect the canard, the two leading edge flaps and the trailing edge flap. Sensors are installed to measure the aerodynamic forces and moments on the model, as well as sectional surface pressure distribution at $x = 955$ mm (60% chord length) and $x = 1125$ mm (70% chord), see Fig. 2. The experiments also included steady-state measurements using Pressure Sensitive Paint (PSP), which provides detailed information on the surface pressure distribution on the whole wing. Measurements were performed for a large range of angles of attack and side-slip angles. During the measurements also the effect of different canard settings as well as leading and trailing edge flap settings has been investigated.

Test run VN01004 [7] has been selected from the wind tunnel data set as reference for the sectional surface pressure distributions and the aerodynamic force and moment data. This test run constitutes an α-sweep for angles of attack α ranging from −6° to 55°.