Computers & Fluids 100 (2014) 155–164

Contents lists available at ScienceDirect

Computers & Fluids

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

Static Computational Fluid Dynamics simulations around a specialised delta wing

Christopher Pevitt, Firoz Alam *

School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Melbourne, Australia

ARTICLE INFO

Article history:
Received 6 October 2011
Received in revised form 4 April 2014
Accepted 24 April 2014
Available online 4 May 2014

Keywords:
Computational Fluid Dynamics
Delta wing
TAU
Stability and control
Pitching moment

ABSTRACT

The primary aim of this paper is to determine a suitable and reliable model for the full static angle of attack range in Computational Fluid Dynamics (CFD) applications while determining associated model dependencies. This would allow CFD to be utilised as a more reliable tool in the development of aircraft, reducing dependency on wind tunnel investigations, with a consequent reduction in development costs. The model used in this study is based on a specialised delta wing configuration. The study has been undertaken by incorporating simulation parameters such as mesh resolution, discretisation schemes, turbulence and transition models, time step sizes and the order of the time integration operator. The modelling has been carried out using specialised meshing software, the flow simulation software (TAU) developed by the German Aerospace Agency (DLR), and the graphical interface Tecplot. Findings indicate that current CFD capabilities to model the flight envelope of a configuration are near-sufficient. The findings also show the difficulties in utilising one CFD model to represent the entire angle of attack range and the effect of model dependencies.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

There is no doubt that the determination of the aerodynamic characteristics of a new combat aircraft with highly swept delta wings is complex and time-consuming, because a lengthy iterative process combining semi-empirical, lower-order modelling, wind tunnel, and flight-test is required [1]. Since the 1960s, the period of all major fighter plane developments, the nonlinear aerodynamic and/or fluid–structure interaction issues were not well known until physical flight tests were undertaken, even when utilising the best available predictive tools [2–6]. Cummings and Schütte [1] provided some examples of such constraints found in the development of combat aircraft such as the F-15, the F/A-18, F/A-18A, the AV-8B, the F/A-18C, the B-2 Bomber, and the AV-8B. Furthermore, the F-15, F/A-18A, and AV-8B exhibited significant aero-elastic flutter, the F/A-18C experienced tail buffet at high angles of attack due to leading-edge extension vortex breakdown, and the B-2 Bomber experienced a residual pitch oscillation [1–9].

With recent advances in the aerospace industry, the demand and commonality of Unmanned Aerial Vehicles (UAVs) have increased significantly. Among UAVs, Unmanned Combat Aerial Vehicles (UCAVs) are dominant. These UCAVs often lead to configuration with nonlinear aerodynamic behaviour, dominated by vortical flow across the upper surfaces due to their highly swept wing planform design. Despite the characteristics of flow phenomena associated with highly swept delta wings having been a subject of research for a long time, the flow behaviour around such geometry is still not fully understood. The financial costs for the development of aircraft with highly swept delta wings could have been significantly reduced if their static and dynamic flow characteristics had been identified in the design phase. Under such circumstances, as Cummings and Schütte [1] have said, a high-fidelity tool capable of predicting with confidence and/or identifying aircraft components vulnerable to handling quality instabilities prior to flight testing would be of great interest.

Among all available tools (flight and wind-tunnel testing, semi-empirical lower-order modelling, and Computational Fluid Dynamics), physical flight tests provide better results. However, this is difficult to use in the early stages of aircraft development and this method is also expensive and time-consuming [1]. The second most accurate method is wind tunnel measurement, but this needs correct scaling and poses difficulties in investigating unsteady dynamic behaviour. The third method is semi-empirical lower-order modelling, which provides less accurate results compared to flight and wind tunnel measurements, due to its limitation in reliably predicting unsteady nonlinear aerodynamic behaviour [1]. Therefore Computational Fluid Dynamics (CFD)

* Corresponding author. Tel.: +61 3 99256103; fax: +61 3 99256108.
E-mail address: firoz.alam@rmit.edu.au (F. Alam).

http://dx.doi.org/10.1016/j.compfluid.2014.04.025
0045-7930/© 2014 Elsevier Ltd. All rights reserved.

Nomenclature

AoA	angle of attack
C_D	coefficient of drag
CFD	Computational Fluid Dynamics
C_L	coefficient of lift
C_{my}	coefficient of pitching moment
DLR	German Aerospace Centre
DNW-NWB	German Dutch Wind Tunnel-Low Speed Wind Tunnel Braunschweig
TAU	DLR flow simulation software
UCAV	Unmanned Combat Aerial Vehicle
NASA	National Aeronautics and Space Administration

modelling remains a cheaper and relatively easier alternative. With the current capabilities of computers and CFD codes, CFD modelling can provide a reasonable compromise between flight and wind-tunnel testing and semi-empirical lower-order modelling [1]. To accurately and reliably predict the stability and control characteristics of an aircraft with highly swept delta wings prior to costly wind-tunnel and flight tests, CFD modelling should be attempted with predictive modelling of lower complexity. At present, several research projects (e.g., Computational Methods for Stability and Control (COMSAC) and Simulation of Aircraft Stability and Control Characteristics for Use in Conceptual Design (SimSAC)) have been undertaken to utilise all available tools, including CFD modelling, for the determination of various flight characteristics. The recently formed NATO Research and Technology Organization (RTO) Task Group AVT-161 is researching the enhancement of the ability of computational methods to predict better the static and dynamic stability characteristics of air and marine vehicles [1,40,41].

The determination of static flow characteristics in combat aircraft development is an essential part of the development cycle in flight physics. When reviewing an unstable aircraft, knowledge of the flight characteristics is critical for design of the flight control systems, as many future unmanned aircraft with highly swept delta wing configurations exhibit aerodynamic stability and control issues in various regions of the flight envelope. Although several research papers have been reported in the public domain on the delta wing configuration and its effects in CFD simulations using static and dynamic modelling, a significant disagreement in angle of attack (AoA) over the linear range was reported [5–7,10–14,18]. These deviations become even more noticeable and serious over the nonlinear range, where the leading edge vortex-breakdown begins to develop. Hence the objectives of this paper are to be able to model the flow phenomena around a highly swept delta wing configuration and be capable of understanding and visualising the key characteristics of the flow. These simulations are to be performed using the TAU flow simulation system, which was developed by the German Aerospace Center (DLR) [1]. These objectives include modelling a range of parameters and determining how each can influence the flow accuracy and characteristics around a specialised delta wing configuration under a range of AoA.

In this study, individual parameters would be assessed. These include dependencies on configurations (with and without sting), mesh resolutions, discretisation schemes, turbulence and transition models, time step sizes and order of the time integration operations. The results will then be compared to the characteristics of both in-phase and out-of phase contributions to the aerodynamic forces and moments.

2. Delta wing configurations

As technology and demands on modern aircraft are advancing, the desire for additional speed and manoeuvrability capabilities is becoming more imperative. These factors should be considered in the design and developmental phase of modern aircraft. The configurations associated with both supersonic and subsonic aircraft vary greatly. The delta wing is not new technology: its initial concept was developed in 1867 [15]. It is one of the most efficient ways to achieve the desired high speed capabilities of a wing. Delta wings are a common feature of aircraft tailored for supersonic flight. The majority of modern aircraft have some aspects of swept wings to gain the beneficial effect of preventing the high speed shock effect [16]. There is a large number of delta wing types, including (a) Standard, Ogival, (b) Compound, (c) Cropped, (d) Tailless, (e) Cranked Arrow, and (f) Diamond/Lambda configurations [17]. These delta wing configurations are shown in Figs. 1 and 2.

A Lambda-type delta wing configuration with a 53° swept angle has been selected for this study. In addition to the delta configuration, the sweep angle of the wings characterises slender and non-slender delta wings [19]. A non-slender delta wing is defined as having a sweep angle equal to or less than 55° [16]. These are known as low sweep angle wings. Therefore the delta wing used in this study is a low sweep angle and non-slender Lambda delta wing. This wing possesses combined rounded and sharp leading edge geometry, and the Lambda wing model is a specifically designed UCAV delta wing configuration. It has been specifically designed in order to develop key aerodynamic characteristics such as flow separation and the development of vortices [20]. The exact configuration is not shown here, but a close representation of the configuration is shown in Figs. 1g and 2, and more details of such a configuration can be found in Cummings and Schütte [1]. As mentioned above, the model has a 53° swept leading edge, with the capability of interchanging a sharp or rounded leading edge. In this study, the rounded leading edge is considered. The rounded leading edge configuration is created with a sharp inboard leading edge, which transitions into a medium round leading edge on the outer panels of the wing. The outer panel has a parallel leading and trailing edge with a washout twist of 5° [21].

The model consists of three main sections: the fuselage, the wing section and wing tips. It is made of light weight reinforced plastics, with an overall mass of less than 10 kg [22]. The purpose of the extra-light model is to reduce the dynamic inertial loads [23]. This allows for a more accurate and sensitive balance that leads to better force and moment resolution. The model contains more than 200 pressure taps on its upper and lower sides to obtain the dynamic measurement of unsteady pressures. The model was designed to gather both static and dynamic pressures. For the scope of this study with the delta wing configuration, experimental data was made available to allow for detailed CFD simulations to be undertaken and compared.

3. The DLR TAU-Code

The CFD modelling was undertaken using the DLR TAU-Code package developed by the DLR Institute of Aerodynamics and Flow Technology. The DLR TAU-Code has been developed to undertake complex CFD simulations. The Solver is based on compressible three-dimensional, steady, and unsteady Reynolds Averaged Navier–Stokes equations [24]. This has been achieved by using

Download English Version:

https://daneshyari.com/en/article/761960

Download Persian Version:

https://daneshyari.com/article/761960

Daneshyari.com