ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE) Vol. 5, Issue 4, April 2019

DESIGN AND ANALYSIS OF SECUNT OGIVE AND ELLIPTIC NOSE SECTION OF THE AIRCRAFTS USING CFD

[1]M.AZHAGU SIVA,M.SATHISH KUMAR,M.VIGNESH , [2] E.SHENBAGA MUTHURAMAN
,[2] Assistant Professor
[1] Department of Aeronautical Engineering,[2] Department of Aeronautical Engineering
PSN College of Engineering & Technology, Tamil Nadu.

ABSTRACT

Physics based simulation is widely seen as a way of increasing the information about aircraft designs earlier in their definition, thus helping with the avoidance of unanticipated problems as the design is refined. This paper reports on an effort to assess the automated use of computational fluid dynamics level aerodynamics for the development of tables for flight dynamics analysis at the conceptual stage. These tables are then used to calculate handling qualities measures. The methodological questions addressed are a)geometry and mesh treatment for automated analysis from a high level conceptual aircraft description and b) sampling and data fusion to allow the timely calculation of large data tables. The test case used to illustrate the approaches is based on a refined design passenger jet wind tunnel model. This model is reduced to a conceptual description, and the ability of this geometry to allow calculations relevant to the final design to be drawn is then examined. Data tables are then generated and handling qualities calculated.

Introduction

A prerequisite for the realistic prediction of the flight dynamic behavior of an aircraft is the availability of complete and accurate aerodynamic data. Traditionally, wind-tunnel measurements are used to fill look-up tables of aerodynamic forces and moments related to the flight state. Wind tunnel models only become available late in the design cycle and most data at the conceptual design stage relies on handbook methods or linear fluid mechanics assumptions [20, 25, 29]. These methods provide low cost reliable data only for conventional aircraft in aerodynamically benign regions of the flight envelope. However, current trends in aircraft design towards novel shapes, augmented stability and expanded flight envelopes require a more accurate description of the flight-dynamic behavior of the aircraft. This provides motivation to move towards Computational Fluid Dynamics(CFD) simulations based on the state-of-the-art computer-aided concept designs since these, in principle have no limitations related to geometry. At the highest practical level, simulation based on the Reynolds Averaged Nervier-Stokes(RANS) equations have the potential to predict the full range of regimes

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
Your ult-MATE Research Paper !!!
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE) Vol. 5, Issue 4, April 2019

of interest to the designer. The current state-of-art in the use of CFD for aircraft design is the generation of data for aerodynamic loads [4]. A number of problems need to be addressed for the routine use of CFD for conceptual design, including the cost of generation of large data tables [13] and the automated handling of geometry [5]. In order to support an automated CFD aircraft design, three procedures need to be considered: geometry definition/mesh generation, the flow simulation and the exploitation of the engineering data from the flow solver output for some design objective. Most dedicated aircraft conceptual design packages construct a simple 3D aircraft model by geometrical lofting techniques. However, these tools do not allow construction of a computational mesh for analysis methods without extensive re-formatting and Computer Aided Design (CAD) repair [17]. In contrast, a disadvantage of a CAD-based geometry model is that generated spline surfaces do not correspond to the parameters that a designer uses to describe the conceptual aircraft geometry (such as wing sweep or thickness) [28]. Some alternatives to CAD are aircraft geometry tools such as Boeing's proprietary tool, General Geometry Generator[8], NASA's Rapid Aircraft Modeler [23, 11], KTH (Royal Institute of Technology)'s CADac [5], Stanford's AEROSURF[1] and KTH's Surface Modeler † . Boeing's GGG software, written in the Python language, has a library of lofting codes to create a parameterized geometry model for conceptual aircraft design. NASA's RAM generates a geometry model and a surface mesh from aircraft parameters such as wing aspect ratio, taper ratio, span and angles of twist, sweep, dihedral, etc. The CADac and AEROSURF software are CAPRI-based applications that link the CAD package and the aircraft design software that requests the variation in the geometry. CAPRI (Computational Analysis Programming Interface) [14] offers a coupling to any supported CAD package by using API to access the geometry and topological data [1]. In this paper, the SUMO code is used for geometry definition/mesh generation. Further information is given below. Historically, many aircraft projects experience problems associated with flight handling qualities, an aircraft attribute that addresses the ability to initiate and subsequently maintain a manoeuvre based on pilot opinion [24]. In the current paper we focus on automating the computational generation of aerodynamic data and its impact on Handling Qualities(HQ) assessment at the conceptual design stage. Three geometry related issues must be considered when applying CFD to conceptual design for handling qualities assessment: The first is, can meshes be generated automatically and calculations run fast enough (to be consistent with use by a designer operating on a short timescale, rather than a CFD specialist on a longer scale)? The second is related to the impact of geometry on the flow predictions. Some aerodynamic phenomena are very sensitive to geometry. As an example, the drag can be significantly increased by wing-fuselage junction separations, and attention to blending to avoid these is a detailed design task which would not be done at the conceptual stage. The level of geometry at the conceptual stage would be likely to promote junction separations that would be predicted by a RANS simulation. In this sense high fidelity simulations on low fidelity geometries may provide misleading information about the underlying properties of the design. The third issue is concerned with the linking of different analysis codes and whether the aerodynamic calculations can be done rapidly enough to be practical. Considering the first issue, the current state of the art in mesh generation does not routinely allow automatic generation of meshes for RANS simulations around full aircraft configurations [3, 18], although progress is being made in this direction [19] . It is however possible to generate unstructured meshes for Euler simulations automatically. As a step in the direction of using RANS for the automated investigation of handling qualities, it therefore seems practical to develop methods based on the Euler equations. To investigate the issue of geometry fidelity, the following approach is taken. We start with the DLR F12 wind tunnel model [22]. This is a refined design of a development model for a large passenger jet, featuring an advanced aerofoil section, a fuselage-wing junction blending, twist and dihedral of the wings, and a realistic fuselage. This geometry has been simplified consistent with conceptual aircraft design. A number of investigations relating to the influence of geometry and aerodynamic model level are then carried out to see what can be learned from the simplified geometries, and how representative the lessons are of the final refined design. The prerequisites of assessing handling qualities are the estimation of mass, centre of gravity, moments of inertia and aerodynamic coefficients for each point in the configuration/flight design space[21]. Such a database could require on the order of 30,000 solutions [27]. At the present time, it is impossible to apply CFD for this number of simulations in a time consistent with design methodology. Fortunately, methods are available that can reduce the computational cost [13]. The paper continues with a description of the geometry handling and the prediction tools. The test case, geometry definition and mesh generation are then detailed. A validation of the aerodynamic tools is made against wind tunnel data. Then, a design study, going from a conceptual geometry description to flight handling quality values in an automated fashion is demonstrated.

Computational Tools

Geometry For a computer-aided analysis and optimization

the geometry of the initial concept must be described. The Computerized Environment for Aircraft Synthesis and Integrated Optimization Methods (CEASIOM) [16], the design code used in the current paper, incorporates a parameterized description of

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE) Vol. 5, Issue 4, April 2019

the geometry, named in this paper the XML-aircraft. Its basic parameters for describing lifting surfaces are: the wing area (퐴); leading edge sweep (ΛLE) and quarter-chord sweep (ΛC/4); aspect ratio (퐴푅); taper ratio; dihedral (Γ), the mean thickness to chord ratio (푡/푐푟) and the aerofoil section. The definition covers cranked lifting surfaces and any number of LE and TE moving devices. The fuselage consists of a three-segment body. The centre body is assumed to have a constant cross-section symmetric about the x-z plane. An ovoid cross section is described by a distortion coefficient (0 < 휆푓 < 1), where 휆푓 = 0.5 shows a circular cross section [15]. The XML-aircraft also includes definitions of wing-fuselage fairing, ventral fin, engines, fuel tanks, baggage hold and cabin dimensions. In summary, the XML-aircraft model is defined by around 100 parameters. The aerodynamic module in CEASIOM converts the XML-aircraft description into the native geometry format for the aerodynamic prediction tools. Of particular interest is the approach used to automatic mesh generation for the Euler solver. The surface modeling package, SUMO, produces a surface model, and its triangulation. The model can be passed to an extended CAD system or mesh generator as a standard CAD interface file, and the surface mesh direct to a tetrahedral volume mesh generator. The parametrization can be extended to the model in the external CAD system through the CADac/CAPRI tools [5]. SUMO ‡ is a rapid geometry modeling tool for parametrically-defined aircraft configurations. The code, written in C++ , has a library of geometric primitives based on B-spline curves and surfaces to create a parameterized watertight surface model of the complete XML-aircraft. The automatic mesh generation tool in SUMO provides an unstructured surface mesh. The mesh control parameters are estimated from the model geometrical features, such as radii of curvature and the presence of sharp edges. From the surface mesh, unstructured volume meshes can be automatically generated using the tetrahedral mesh generator TetGen.

CFD Methods Edge

[10] is a parallelized CFD package developed by Swedish Defence Research Agency, FOI. The code can be applied to 2D/3D viscous(RANS) or inviscid(Euler), compressible flow problems on unstructured grids with arbitrary elements and is used in Euler mode in CEASIOM. Also, Edge allows both steady state and time accurate calculations. The space discretisation exploits a node-centred finite-volume technique using an edge based data structure. The computational elements are a set of non-overlapping cells formed as the dual of the primary tetrahedral mesh. Explicit Runge-Kutta time stepping integrates the discrete equations in time. Accelerated convergence to steady state is promoted using agglomeration multigrid and implicit residual smoothing. A Matlab interface allows Edge calculations to be prepared and run from CEASIOM. This call

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
Your ult-MATE Research Paper !!!
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE) Vol. 5, Issue 4, April 2019

runs the preprocessing routines, launches the calculation and processes the flow solution for the forces and moments. CEASIOM also provides the possibility to exploit external CFD codes. The example code used in the current work is the Parallel Multiblock Code (PMB)[2]. The Euler and RANS equations are discredited on curvilinear multi-block body conforming grids using a cell-centered finite volume method which converts the partial differential equations into a set of ordinary differential equations. The convective terms are discretised using Osher's upwind method. Monotone Upwind Scheme for Conservation Laws (MUSCL) variable extrapolation is used to provide second-order accuracy with the Van Albada limiter to prevent spurious oscillations around shock waves. The spatial residual is modified by adding a second order discretisation of the real time derivative to obtain a modified steady state problem for the flow solution at the next real time step, which is solved through pseudo time. This pseudo time problem is solved using an unfactored implicit method, based on an approximate linearisation of the residual. The linear system is solved in unfactored form using a Krylov subspace method with Block Incomplete Upper Lower (BILU) preconditioning. The preconditioner is decoupled between blocks to allow a high efficiency on parallel computers with little detriment to the convergence of the linear solver. For the Jacobian matrix of the CFD residual function, approximations are made which reduce the size and improve the conditioning of the linear system without compromising the stability of the time marching. Given a block structured mesh, CEASIOM can prepare input files and launch calculations using PMB.

Force and Moment Generation Using Sampling and Data Fusion

The aerodynamic prediction methods are used to generate tables of forces and moments for a set of aircraft states (e.g. aircraft angle of attack, side-slip angle, control deflections and etc.), which spans the flight envelope. This potentially entails a large number of calculations, which will be a particular problem due to the computational cost if CFD is the source of the data. This issue has been addressed by sampling and reconstruction based on Kriging interpolation model and data fusion using Co-Kriging as described in reference [13]. Two scenarios were considered, based on (1) a requirement for tables for a completely new design and (2) for updating tables for an existing design which is being altered. In the first scenario it is assumed that a high fidelity model is required and that this can be generated without user intervention. The emphasis is on a sampling method which will identify nonlinearities in the force and moment tables. Approaches to the sampling based on the Mean Squared Error (MSE) criterion of Kriging and the Expected Improvement Function (EIF) were considered in reference [13]. The second scenario has a designer involved in an interactive session. It is assumed that the aircraft geometry is incremented from an

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE) Vol. 5, Issue 4, April 2019

initial design, perhaps selected from a library, and that a high fidelity model is available for the initial design from the first scenario. Data fusion based on Co-Kriging is then used to update this initial model, based on a small number of calculations at an acceptable cost (which at present rules out RANS). In this scenario it is assumed that the main flow features present for the initial geometry are not changed appreciably by the geometry increments. If this is not the case, for example if the wing sweep angle increases so that vortical flow starts to dominate, then either a new initial geometry needs to be selected, or the interactive session needs to be suspended so that a new high fidelity model can be generated under the first scenario. Using these techniques it was shown that tables which are practically useful could be generated in the order of 100 calculations under the first scenario and 10 calculations under the second scenario.

Mesh Generation

Multiblock structured Euler and RANS grids were available for the WT geometry. The Edge meshes were generated automatically by SUMO for the XML geometry. The grid sizes are shown in table 1, and views of the two types of grid in Fig. 3. Comparison of the predictions of the lift,

Conclusions

The results in this paper demonstrate that the automated calculation of flight handling qualities of a conceptual aircraft design is now possible using physics based aerodynamic simulation data. The steps that allow this are

(a) the automated generation of meshes for Euler calculations around complete aircraft starting from a high level conceptual definition of the geometry;

(b) fast generation of aerodynamic tabular models based on sampling and data fusion and

(c) the coupling of these tools with an analysis code for flight dynamics. Results were presented to benchmark and assess the impact of the geometry definition on the Euler calculations, to compare the predictions of different aerodynamic modelling levels on handling quality predictions, and to show that the expected trends from changing geometry parameters are obtained. Future work will go in two directions. First, the opportunity is now there to exploit simulation for driving control design at the conceptual stage. It is expected that this will enable the consideration of a wider range of potential designs, with the use of active control to remove previously insurmountable obstacles arising from the geometry. Secondly, the extension of the automated analysis to include RANS simulations poses the challenge of automatically generating suitable meshes, and revisiting the impact of geometry roughness when flow separation is possible.

Acknowledgements

Liverpool was supported under funding from the Sixth Framework programme of the European Union for the SimSAC project, and the Engineering and Physical Sciences Research Council and the Ministry of Defence under EP/D504473/1. KTH was supported by the SimSAC project. Multiblock structured Euler and RANS grids were generated by CERFACS for the WT geometry.

References

[1] J. J. Alonso, J. R. Martins, J. J. Reuther, R. Haimes, C. A. Crawford. High-fidelity aerostructural design using a parametric CAD-based model.16th AIAA Computational Fluid Dynamics Conference, Orlando, FL, June 23- 26, 2003, AIAA-2003-3429.

[2] K. J. Badcock, B. E. Richards, M. A. Woodgate. Elements of computational fluid dynamics on block structured grids using implicit solvers. Progress in Aerospace Sciences, 36 (2000), 351–392.

[3] J. T. Baker. Mesh generation: Art or science? Progress in Aerospace Sciences, 41 (2005), 29–63.

[4] K. Becker, J. Vassberg. Numerical aerodynamics in transport aircraft design. Notes on Num. Fluid Mechanics, 100 (2009), 209–220.

[5] A. Berard, A. Rizzi, A. T. Isikveren. CADac: a new geometry construction tool for aerospace vehicle pre-design and conceptual design. 26th Applied Aerodynamics Conference, Honolulu, Hawaii, USA, August 18-21, 2008, AIAA-2008-6219.