Progress in Aerospace Sciences 47 (2011) 589–596

Contents lists available at SciVerse ScienceDirect

Progress in Aerospace Sciences

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

From geometry to CFD grids—An automated approach for conceptual design

Maximilian Tomac *, David Eller

Department of Aeronautical and Vehicle Engineering, Royal Institute of Technology (KTH), Stockholm 10044, Sweden

ARTICLE INFO

Available online 19 October 2011

Keywords:
Aircraft design
Aerodynamics
Mesh generation
Automated analysis
Computational fluid dynamics
Simulation

ABSTRACT

The CEASIOM software developed in the EU-funded collaborative research project SimSAC generates stability and control data for preliminary aircraft design using different methods of varying fidelity. In order to obtain the aerodynamic derivatives by CFD, the aircraft geometry must be defined, computational meshes generated, and numerical parameters set for the flow solvers. An approach to automation of the process is discussed, involving geometry generation and mesh generation for inviscid as well as RANS flow models.

Contents

1. Introduction

The CEASIOM software system developed in the EU-funded collaborative research project SimSAC generates stability and control data for conceptual and preliminary aircraft design using a choice of numerical methods of varying fidelity. The system aims at automation of the computation of tables of forces and moments for the rigid or elastic aircraft with control surfaces. In order to obtain this data, the aircraft geometry must be defined, computational meshes for aerodynamic and structural analyses need to be created and finally, the solver parameter settings must be adapted to reliably perform multiple solutions from which the tables are compiled. This paper focuses on the generation of CFD meshes from the parameters that define the geometry. The process is illustrated by examples used in SimSAC, from left to right in Fig. 1, the TCR-C15 TransCruiser design study and wind-tunnel model, the EADS Ranger 2000 jet, and the SACCON UCAV wind-tunnel model. The TCR is a preliminary design of a 200 ton MTOW 200 passenger airliner with M 0.97 design cruise speed and wind-tunnel model scaled 1:40. The EADS Ranger 2000 is a tandem seat light jet trainer which flew, but was not produced.

* Corresponding author.
E-mail address: maxtomac@kth.se (M. Tomac).

0376-0421/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.paerosci.2011.08.005

Nomenclature

Symbols

M	Mach number

Greek letters

α	angle of attack

Abbreviations

ACBuilder	aircraft builder
AMB	aerodynamic model builder
CEASIOM	computerized environment for aircraft synthesis and integrated optimization methods
CFD	computational fluid dynamics
RANS	Reynolds averaged Navier–Stokes
FCS	flight control system
MTOW	maximum take-off weight
NeoCASS	next generation conceptual aero-structural sizing suite
SACCON	stability and control configuration
SDSA	simulation and dynamic stability analysis
SIMSAC	simulating aircraft stability and control characteristics
S&C	stability and control
TCR	transonic cruiser
VLM	vortex lattice method
UCAV	unmanned combat air vehicle
WB	weights and balances

SACCON is the DLR-F17 scale 1:8 low speed wind tunnel model with a wing planform with significant negative twist at the outer kink.

2. Paths to meshable models

It is clear that the conceptual design should be passed to the following design phases in a format which enables easy application of the relevant analysis methods: further down the line, the aircraft lives in a general purpose CAD system, such as CATIA.[1] The whole geometry modeling process could be performed in the CAD system, with mesh generation performed by general, commercial mesh generators. This approach is pursued by several groups, such as the Linköping group [1]. Also, major commercial efforts to speed up the CFD analysis cycle have produced significant progress towards automatic, reliable, physics-aware mesh generation. For external aerodynamics, commercial products are offered for several solver technologies, e.g., ANSYS ICEM-CFD for unstructured tetra–hexa-prism grids,[2] and NUMECA HEXPRESS for multi-block hexa meshes.[3]

While this approach is highly attractive in terms of functionality, it has certain disadvantages. Due to the complex nature of both CAD systems and high-fidelity flow solvers, combined with the associated substantial licensing and training cost, the number of aerospace aerodynamics experts who also are experienced users of both CAD and mesh generation software is rather limited. Furthermore, unless the CAD model is created appropriately, the geometry description exported to the mesh generation software may need "repair" and/or manual simplification efforts, thereby severely encumbering automation.

Another approach is to build a *customized* geometry modeler for aircraft design, as exemplified by the NASA VSP [2] project, which supports export of meshable surface models and/or meshes in standard formats such as IGES [3], STEP [4], and CGNS.[4] CEASIOM is in this category and provides several paths from the set of geometric parameters which describe a CEASIOM aircraft to a computational mesh. The CADac path [5] uses the vendor-neutral geometric modeling API Capri [6] from CADNexus to build a meshable model in the CAD system and the sumo path, Fig. 5, employs the custom surface modeler and -mesher sumo[5] [7], and the TetGen [8] tetrahedral mesh generator. The effort on the CADac path has decreased over time because of the complexities of developing for several different CAD systems, even if CAPRI does hide much of the idiosyncrasies. The "inhouse" sumo path has become our main approach for automatic Euler flow model meshing and for the experiments with automated meshing for RANS simulations.

3. The CEASIOM multi-fidelity framework

CFD computations to estimate aerodynamic forces and moments early in the design stage can provide a head start on the controls design. For this strategy to succeed, so that changes in the aircraft configuration can be assessed at acceptable costs, the simulation methods must be:

(1) fast,
(2) reasonably accurate, and
(3) easy to use.

These three requirements can be addressed by adaptive fidelity CFD. Low order methods are used in the low speed linear region, and higher order solvers in the high speed and non-linear region. Another aspect that needs to be taken into account is the capabilities of the methods to represent the actual geometry, illustrated here by the Ranger 2000, Fig. 1(b).

Fig. 2 shows an overview of the CEASIOM framework. For further details see Refs. [9,10]. The framework integrates discipline-specific tools with main focus on aircraft conceptual design, and we are concerned here only with the left part of the system, from geometry to computational mesh. The round components are in-house developments and the rectangular modules are third party commercial or "free" products.

The sumo path is outlined by a dashed polygon, and the CADac path by a dotted polygon. As mentioned above, the focus of this paper is on streamlining the determination of aerodynamic data. Instead of adopting a fast potential-flow model enhanced with highly refined empiricism, CEASIOM uses adaptive-fidelity CFD. The CFD module models range from DatCom, via vortex lattice modeling to CFD in inviscid Euler or full RANS mode. The choice of flow model depends on how much realism is needed to capture the flow physics at hand. The simplest geometry could be a collection of global shape parameters with a level of accuracy suitable for vortex lattice (VLM) aerodynamics, Fig. 3(a). The next higher level would take into account the actual surface of the aircraft, with some details

[1] http://www.3ds.com/products/catia.
[2] http://www.ansys.com/products/icemcfd.asp.
[3] http://www.numeca.com/index.php?id=16.
[4] http://cgns.sourceforge.net/.
[5] http://www.larosterna.com/sumo.html.