Applied Mathematical Modelling 40 (2016) 4543–4559

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

A novel implementation of computational aerodynamic shape optimisation using Modified Cuckoo Search☆

D.S. Naumann*, B. Evans, S. Walton, O. Hassan

Civil & Computational Engineering Centre, School of Engineering, Swansea University, Swansea Wales SA2 8PP, UK

ARTICLE INFO

Article history:
Received 24 March 2015
Revised 15 October 2015
Accepted 10 November 2015
Available online 2 December 2015

Keywords:
Mesh movement
Cuckoo search
Computational fluid dynamics
Aerodynamic shape optimisation
Shape parameterisation

ABSTRACT

This paper outlines a new computational aerodynamic design optimisation algorithm using a novel method of parameterising a computational mesh using 'control nodes'. The shape boundary movement as well as the mesh movement is coupled to the movement of user–defined control nodes via a Delaunay Graph Mapping technique. A Modified Cuckoo Search algorithm is employed for optimisation within the prescribed design space defined by the allowed range of control node displacement. A finite volume compressible Navier–Stokes solver is used for aerodynamic modelling to predict aerodynamic design 'fitness'. The resulting coupled algorithm is applied to a range of test cases in two dimensions including aerofoil lift–drag ratio optimisation intake duct optimisation under subsonic, transonic and supersonic flow conditions. The discrete (mesh–based) optimisation approach presented is demonstrated to be effective in terms of its generalised applicability and intuitiveness.

1. Introduction

During the last 30 years, the aerodynamic design problems faced by the aerospace industry have been revolutionised by computational fluid dynamics (CFD). Particularly unstructured mesh methods [1–3] these days allow the mesh generation on complex three–dimensional geometries within a few hours, that initially required several months using multiblock techniques for quasi–structured meshes [4,5]. Simultaneously, the development of Computer Aided Design (CAD) has had a strong impact on the design cycle of aerodynamic problems [6]. In light of this, CFD and CAD have become integral parts of a typical aerodynamic design cycle apparent in current aerodynamic design projects. The flow chart in Fig. 1 [7] indicates the emphasis now placed on CFD and CAD within the inner and outer design loops.

Despite these advancements, significant challenges remain for the computational modelling community in order to efficiently transfer geometry between CAD and CFD systems and improve the computationally expensive mesh re-generation process and CFD evaluation during optimisation [7,8]. Main challenges include a lack of standardised shape parameterisation approaches and the alignment of CAD geometry definition with the CFD solver geometry definition as well as the lack of consensus regarding most suitable optimisation scheme given the application. Current research tries to overcome some of these problems by linking CAD systems, CFD tools and the mesh generation process. Examples include Isogeometric Analysis [9–11] and NURBS Enhanced Finite

Abbreviations: FDGD, Fast Dynamic Grid Deformation; DG, Delaunay Grid; CS, Cuckoo Search; MCS, Modified Cuckoo Search.

☆ The intellectual property rights in the research data are asserted by the authors and their research partners.

* Corresponding author. Tel.: +44 7805243278.
E-mail addresses: 717761@swansea.ac.uk, dn_arbeitsadresse@arcor.de (D.S. Naumann), b.j.evans@swansea.ac.uk (B. Evans), s.p.walton@swansea.ac.uk (S. Walton), o.hassan@swansea.ac.uk (O. Hassan).

http://dx.doi.org/10.1016/j.apm.2015.11.023

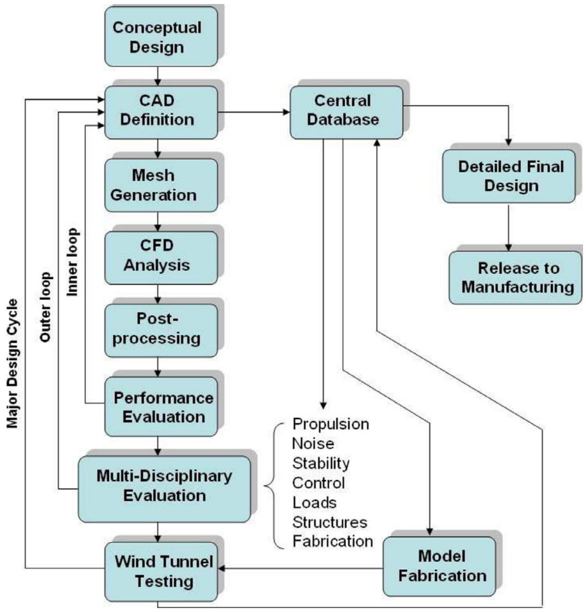

Fig. 1. A typical multi–disciplinary aerospace design cycle.

Elements Methods [12,13], Reduced Order Models (e.g. POD) [14,15] and surrogate models (e.g. Kriging) [16,17]. These approaches have been shown to reduce the computational cost involved in the CFD evaluation and design space exploration; yet, analysis of their impact on the proposed optimisation strategy was deemed to be beyond the scope of this study. However, limited progress has been made to link these approaches to computational aerodynamic shape optimisation algorithms in contexts of generalised applicability. The selected approach usually severely limits the explorable design space (i.e. range of potential shapes) for the optimiser. Considerable effort into research concerning coupling CFD modelling with aerodynamic shape optimisation has only been invested in significantly over the last 10 years [18]. Particularly the use of global optimisation algorithms in this field is only just emerging [19].

This paper presents a novel implementation of computational aerodynamic shape optimisation in which the parameterisation of the geometry and coupling with an optimisation algorithm is unique. The approach makes use of the concept of 'control nodes' in the mesh as the method for both defining the geometry movements and as the design parameters for the optimisation process. The Fast Dynamic Grid Deformation (FDGD) approach [20] has been applied to move the mesh and results in a self-contained algorithm formulated to propagate the effect of the 'control node' displacement throughout the discrete shape boundary and computational mesh. There is no requirement to re–mesh at each stage in the optimisation. Since all knowledge of the geometry is 'stored' in the discrete boundary, there is no requirement to convert the geometry definition stored in the mesh into any other format during the optimisation process. This reduces the problem of translation of CAD-based geometry definitions to CFD meshes.

Aerodynamic designers prefer to use tools that are both intuitive and have wide–ranging applicability. The optimisation and design process requires an effective geometry parameterisation to allow sufficient exploration of a design space. Furthermore, a minimisation of the number of parameters defining the position in a design space is of benefit in order to also minimise computational cost. The approach described in this paper is a 'control node' mesh–based parameterisation. Well–known mesh–based optimisation test cases were performed by Jameson [21,22] using control theory coupled with an adjoint approach [7,22,23] to solve for gradients. Other implementations of parameterisation schemes in the literature include CAD based [24], analytical, basis vector [25], free form deformation (FFD) [26,27], domain element methods [28], and the control grid approach [29]. A thorough review of shape parameterisation techniques is provided by Samareh in [30]. In this paper it will be argued that the 'control node' approach presented here has advantages over these methods in terms of ease of implementation, user intuition and generalised applicability.

2. Methodology

2.1. Geometry shape parameterisation

One of the common practical problems in industrial implementations of CFD–based aerodynamic design is the translation of geometries from CAD systems into computational meshes for simulation. This is often referred to as the bottleneck of the design process [7,8] due to differing tolerance levels required for CAD systems compared with CFD [31,32]. Solutions currently researched, for example Isogeometric Analysis and NURBS Enhanced Finite Elements Methods emphasize the development of a new CFD solver based on the geometry definition of the CAD system [9–13].