Aerospace Science and Technology 39 (2014) 513–522

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Inverse airfoil design using variable-resolution models and shape-preserving response prediction

Leifur Leifsson *, Slawomir Koziel

Engineering Optimization & Modeling Center, School of Science and Engineering, Reykjavik University, Menntavegur 1, 101 Reykjavik, Iceland

ARTICLE INFO

Article history:
Received 10 October 2013
Received in revised form 27 May 2014
Accepted 31 May 2014
Available online 3 July 2014

Keywords:
Inverse airfoil design
Target pressure distribution
Surrogate models
Variable-resolution modeling
Response surface modeling

ABSTRACT

The paper presents a computationally efficient surrogate-based optimization algorithm for the inverse design of transonic airfoils. Our approach replaces the direct optimization of an accurate, but computationally expensive, high-fidelity airfoil model by an iterative re-optimization of two different surrogate models. Initially, for a few design iterations, a corrected physics-based low-fidelity model is employed, which is subsequently replaced by a response surface approximation model. The low-fidelity model is based on the same governing fluid flow equations as the high-fidelity one, but uses coarser mesh resolution and relaxed convergence criteria. The shape-preserving response prediction (SPRP) technique is utilized to predict the high-fidelity model response, here, the airfoil pressure distribution. In this prediction process, SPRP employs the actual changes of the low-fidelity model response (or the surface approximation model) due to the design variable adjustments. The SPRP algorithm is embedded into the trust region framework to ensure good convergence properties of the optimization procedure. Our algorithm is applied to constrained inverse airfoil design in inviscid transonic flow. A comparison with the basic version of the optimization algorithm, exploiting only a physics-based low-fidelity model, is carried out. While the performance of both versions is similar with respect to their ability to match the target pressure distribution, the improved algorithm offers substantial design cost savings, from 25 to 72 percent, depending on the test case.

1. Introduction

Aerodynamic shape optimization (ASO) involves the design of aerodynamic components such as aircraft wings and turbine blades [6,24]. The state-of-the-art ASO design methods employ high-fidelity computational fluid dynamics (CFD) simulations as a part of efficient numerical optimization algorithms [2,10,27,28]. The accurate CFD simulations typically lead to more realistic and attainable designs. The downside is that the high-fidelity CFD analysis is computationally expensive and design optimization normally requires a large number of simulations, which leads to a time consuming design process. Furthermore, often a large number of design variables are required in order to define, in sufficient detail, the geometry of the component being studied, adding to the dimensionality of the design problem.

Airfoil design methods can be broadly categorized into direct and inverse methods [8]. The direct methods involve changing the geometric shape to maximize a given performance criterion subject to one or several constraints at a given operating condition. The most common objective functions include lift maximization, drag minimization, and lift-to-drag ratio maximization. Often, only an initial shape is prescribed along with the desired constraint values, and nothing is assumed regarding the properties of the fluid flow. Some of the most successful direct ASO methods, for both incompressible and compressible flows, are gradient-based [11,13,24], and many also use continuous adjoint methods [15], or discrete adjoint methods [26]. More recently, the introduction of surrogate-based optimization (SBO) [10,27] methods to ASO have been successful in reducing the overall computational cost, as well as handling noisy objective functions. Examples of such work can be found in [2,3,5,22,28].

Inverse design methods, on the other hand, require the definition of specific flow characteristics a priori [8]. Typically, either a target pressure or a velocity distribution is specified, with the former being more popular. The aerodynamic shape is then modified or designed to achieve these characteristics. Inverse design is, usually, more computationally efficient than direct optimization, because the changes in the geometry can be related to the required change in performance, and, thus, requiring few flow solutions to obtain the final profile [21]. However, the task of creating pressure distributions that meet the required aerodynamic characteristics

* Corresponding author.
E-mail addresses: leifurth@ru.is (L. Leifsson), koziel@ru.is (S. Koziel).

http://dx.doi.org/10.1016/j.ast.2014.05.013
1270-9638/

is not trivial and the designer has to rely on experience (and/or available data), or employ special optimization methods [14,20]. Furthermore, inverse design methods only provide the shape that meets the specific characteristics, which may or may not be optimal. Moreover, one cannot guarantee that an arbitrarily prescribed pressure distribution will yield a realistic airfoil profile [25].

A computationally efficient design optimization methodology for inverse design of transonic airfoils was recently introduced in [23]. The approach replaces the direct optimization of an accurate, but computationally expensive, high-fidelity airfoil model by an iterative re-optimization of a surrogate model, which is constructed during each design iteration using a physics-based low-fidelity model and the shape-preserving response prediction (SPRP) technique [16]. The low-fidelity model is based on the same governing fluid flow equations as the high-fidelity one, but uses coarser discretization and relaxed convergence criteria. The SPRP is utilized to predict the high-fidelity model response (in this case, the airfoil pressure distribution) using the low-fidelity model response changes described by properly defined set of characteristic points.

In this work, we substantially enhance the optimization methodology introduced in [23]. More specifically, the low-fidelity CFD model is replaced—after a few design iterations—by its (local) response surface approximation. This allows us to reduce the overall design cost and obtain faster convergence when compared to the original version of the algorithm. Our approach is demonstrated using several transonic airfoil design cases.

2. Problem formulation

Airfoil shape optimization can be formulated as a constrained nonlinear minimization problem, i.e., for a given set of operating conditions, solve

$$\begin{aligned} \min_{\mathbf{x}} \quad & f(\mathbf{x}) \\ \text{s.t.} \quad & g_j(\mathbf{x}) \le 0, \quad j = 1, \ldots, M \\ & h_k(\mathbf{x}) = 0, \quad k = 1, \ldots, N \\ & \mathbf{l} \le \mathbf{x} \le \mathbf{u} \end{aligned} \tag{1}$$

where $f(\mathbf{x})$ is the objective function, $\mathbf{x}$ is the design variable vector, $g_j(\mathbf{x})$ are the inequality constraints, M is the number of the inequality constraints, $h_k(\mathbf{x})$ are the equality constraints, N is the number of the equality constraints, and $\mathbf{l}$ and $\mathbf{u}$ are the design variables lower and upper bounds, respectively.

In inverse design, the role of the designer is to specify a particular flow feature, which typically is a target pressure distribution, $C_{p.t}$, on the surface of the airfoil. The task is then to find the airfoil shape that can give the target pressure distribution at the desired flow condition. This can be done by minimizing the difference between the pressure distribution of the airfoil C_p and the target distribution $C_{p.t}$.

The objective function can be formulated as the difference between the airfoil pressure distribution and the target pressure distribution, or $f(\mathbf{x}) = 1/2 \int [C_p(\mathbf{x}) - C_{p.t}]^2 \, ds$. A minimum thickness is normally specified so that the optimizer does not reduce the airfoil to a thin plate. The thickness constraint can be written as $g(\mathbf{x}) = A_{min} - A(\mathbf{x}) \le 0$, where $A(\mathbf{x})$ is the cross-sectional area of the airfoil and A_{min} is the minimum cross-sectional area.

In this paper, we use the NACA airfoil shapes to illustrate the use of the proposed methodology. In particular, we use the NACA four-digit airfoil parameterization method, where the airfoil shape is defined by three parameters m (the maximum ordinate of the mean camberline as a fraction of chord), p (the chordwise position of the maximum ordinate) and t/c (the thickness-to-chord ratio). The airfoils are denoted by NACA *mpxx*, where *xx* represents the value of t/c. The shapes are constructed using two polynomials, one for the thickness distribution and the other for the mean camber line. The full details of the NACA four-digit parameterization method are given in Abbott and von Doenhoff [1]. Three example NACA four-digit airfoils are shown in Fig. 1.

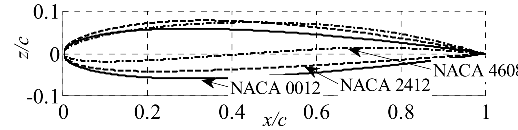

Fig. 1. Shown are three different NACA four-digit airfoil sections; NACA 0012 ($m = 0$, $p = 0$, $t/c = 0.12$) solid line (–), NACA 2412 ($m = 0.02$, $p = 0.4$, $t/c = 0.12$) dashed line (--), NACA 4608 ($m = 0.04$, $p = 0.6$, $t/c = 0.08$) dash–dot line (-·-).

3. CFD modeling

A single CFD simulation is, in general, composed of four steps; the geometry generation (described here in Section 2), meshing of the solution domain, numerical solution of the governing fluid flow equations, and post-processing of the flow results, which involves, in the case of numerical optimization, calculating the objectives and constraints. In this section we present the high- and low-fidelity CFD models.

3.1. High-fidelity CFD model

The flow is assumed to be steady, inviscid, and adiabatic with no body forces. The Euler equations are taken to be the governing fluid flow equations (see e.g., Tannehill et al. [29]). The computational meshes used in this study are all of structured curvilinear body-fitted C-topology. The solution domain boundaries are placed at 24 chord lengths in front of the airfoil, 50 chord lengths behind it, and 25 chord lengths above and below it. The meshes are generated with the computer code ICEM CFD [12]. A fine mesh was developed with a total of 320 points in the vertical direction, 180 points on the airfoil surface and 160 points in the wake behind the airfoil, with a total of 106 thousand cells. An example computational mesh is shown in Fig. 2.

The numerical fluid flow simulations are performed using the computer code FLUENT [9]. Asymptotic convergence to a steady state solution is obtained for each case. The iterative convergence of each solution is examined by monitoring the overall residual, which is the sum (over all the cells in the computational domain) of the L^2 norm of all the governing equations solved in each cell. In addition to this, the lift and drag forces (defined in Section 3.3) are monitored for convergence. The solver convergence criteria is to reduce the maximum residual by six orders of magnitude, or a maximum number of iterations of 1000, whichever comes first.

3.2. Low-fidelity CFD model

In order for SBO to have an impact on the overall efficiency of the design process, the simulation time of the surrogate model needs to be substantially lower than that of the high-fidelity model. Here, we construct the low-fidelity CFD model by using the high-fidelity CFD model (as described in Section 3.1), but with a coarser computational mesh and relaxed convergence criteria.

We performed a parametric study on a computational mesh of a typical airfoil section by reducing the number of mesh points and reducing the number of required solver iterations. The NACA 2412 was selected for this study. The free-stream Mach number is taken to be $M_\infty = 0.75$ and the angle of attack is set to $\alpha = 1$ deg. A fine mesh was developed for this case with a total of 320 points in the vertical direction, 180 points on the airfoil surface and 160 points in the wake behind the airfoil, with a total of 106 thousand cells.