

International Conference on Computational Science, ICCS 2011

Inverse Design of Transonic Airfoils Using Variable-Resolution Modeling and Pressure Distribution Alignment

Leifur Leifsson^{a*}, Slawomir Koziel^a, Stanislav Ogurtsov^a

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

Abstract

The paper presents a computationally efficient and robust methodology for the inverse design of transonic airfoils. The approach replaces the direct optimization of an accurate, but computationally expensive, high-fidelity airfoil model by an iterative re-optimization of a corrected low-fidelity model. 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 shape-preserving response prediction technique is utilized to align the pressure distribution of the low-fidelity model with that of the high-fidelity model. This alignment process is particularly suitable since a target pressure distribution is specified in the inverse design problem. The method is applied to constrained inverse airfoil design in inviscid transonic flow. The results show that the proposed method is able to match the target pressure distributions closely and requiring over 90 percent lower computational cost than when using only the high-fidelity model.

Keywords: Inverse airfoil design; transonic flow; pressure distribution; surrogate models; variable-fidelity optimization; design optimization.

1. Introduction

Aerodynamic shape optimization (ASO) involves the design of aerodynamic components such as aircraft wings and turbine blades [1, 2]. The state-of-the-art ASO design methods employ high-fidelity computational fluid dynamic (CFD) simulations as a part of efficient numerical optimization algorithms [3-6]. The benefit of using high-fidelity CFD simulations is a more accurate representation of the flow physics, which, usually, leads to more realistic and attainable designs. However, this presents numerous challenges. High-fidelity CFD simulations are computationally expensive and design optimization normally requires a large number of simulations, leading to a very 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.

In general, ASO design methods can be classified as being either direct or inverse [7]. The direct ASO 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

* Corresponding author. Tel.: +354-599-6379 ; Fax: +354-599-6201; *E-mail address:* leifurth@ru.is

direct ASO methods, for both incompressible and compressible flows, are gradient-based [1, 8, 9], and many also use continuous adjoint methods [10], or discrete adjoint methods [11]. More recently, the introduction of surrogate-based optimization (SBO) [3, 4] 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 [5, 6, 12–14].

Inverse ASO design methods, on the other hand, require the definition of specific flow characteristics a priori [7]. 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 [15]. However, the task of creating pressure distributions that meet the required aerodynamic characteristics is not trivial and the designer has to rely on experience (and/or available data), or employ special optimization methods [16, 17]. 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 [18].

A computationally efficient design optimization methodology for the direct ASO of transonic airfoils has been recently introduced [14]. The method exploits a physics-based low-fidelity model (the transonic small-disturbance equation, TSDE) to construct a surrogate of a high-fidelity model based on the compressible Euler equations [14]. The low-fidelity model is corrected by aligning its airfoil surface pressure distribution with the corresponding distribution of the high-fidelity model. The alignment is carried out using a shape-preserving response prediction (SPRP) technique [19].

In this work, we adopt the optimization methodology [14] to handle the inverse airfoil design problem. In additions to this, the methodology is enhanced by equipping it with the trust-region approach [20] and replacing the TSDE low-fidelity model by the Euler-equation-based one using coarse discretization and relaxed convergence criteria. This improves the generalization capability of the surrogate and, therefore, the overall robustness of the optimization algorithm. The numerical demonstrations of our approach on transonic airfoil design show that target pressure distributions can be matched closely at a low computational cost corresponding to a few evaluations of the high-fidelity model.

2. Problem Formulation

An airfoil is a streamlined aerodynamic surface such as the one shown in Fig. 1(a). The length of the airfoil is called the chord and is denoted by c . The thickness, denoted by t , varies along the chord line. The curvature, called the camber, described by the mean camber line, varies also along the chord line. The leading-edge (LE) is normally rounded and the trailing-edge (TE) is either closed (sharp) or open.

The primary function of the airfoil is to generate a lift force $\mathbf{l}$ (a force component perpendicular to the free-stream) at a range of operating conditions (Mach number M_∞ , Reynolds number, angle of attack α). Normally, the drag force $\mathbf{d}$ (a force component parallel to the free-stream) is to be minimized. For inviscid transonic flow, the lift and drag forces are determined by the pressure distribution acting on the airfoil surface, which is obtained by solving the governing equations of the fluid flow past the airfoil. The pressure distribution is represented by the pressure coefficient, which is defined as

$$C_p \equiv \frac{p - p_\infty}{q_\infty} \quad (1)$$

where p is the static pressure, p_∞ is the free-stream pressure, $q_\infty \equiv (1/2)\rho_\infty V_\infty^2$ is the dynamic pressure, ρ_∞ is the air density, and V_∞ is the free-stream speed. An example pressure distribution on the surface of an airfoil for a flow at a transonic condition is shown in Fig. 1(b).

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 $C_{p,t}$.

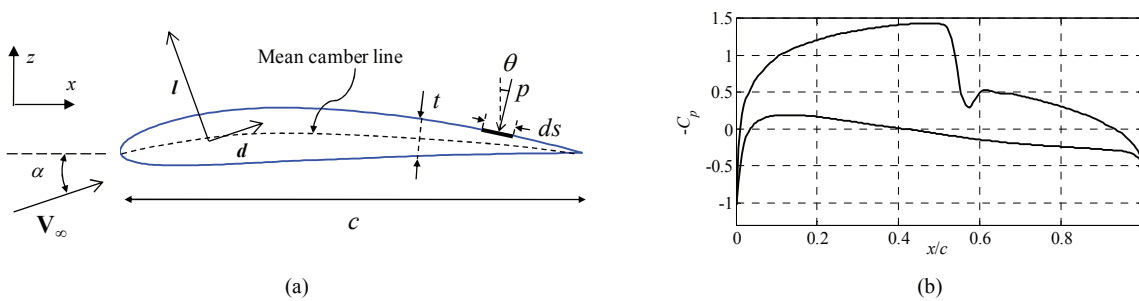


Fig 1. (a) Shown is the NACA 2412 airfoil section in the x - z plane with the free-stream of velocity V_∞ at an angle of attack α . Section lift is denoted by l and section drag is denoted by d . The airfoil thickness distribution is t and the airfoil chord length is c . The pressure p acts normal to a surface panel element of length ds at an angle θ relative to the x -axis; (b) the pressure distribution C_p on the surface of NACA 2412 airfoil at Mach $M_\infty = 0.7$ and angle of attack $\alpha = 1$ deg obtained by the numerical simulation using the Euler equations (see Sec. 3 for further details).

In general, airfoil shape optimization can be formulated as a nonlinear minimization problem. For a given set of operating conditions, solve

$$\min_{\mathbf{x}} f(\mathbf{x}) \quad \text{s.t.} \quad g_j(\mathbf{x}) \leq 0, \quad j = 1, \dots, M; \quad h_k(\mathbf{x}) = 0, \quad k = 1, \dots, N, \quad \text{and} \quad l \leq \mathbf{x} \leq u \quad (2)$$

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 l and u are the design variables lower and upper bounds, respectively.

In inverse airfoil design, the objective function can be the L^2 norm of the difference between the airfoil pressure distribution and the target pressure distribution, or $f(\mathbf{x}) = \frac{1}{2} \int [C_p(\mathbf{x}) - C_{p,t}]^2 ds$. 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 $g(\mathbf{x}) = A_{\min} - A(\mathbf{x}) \leq 0$, where $A(\mathbf{x})$ is the cross-sectional area of the airfoil and $A_{\min}$ is the minimum cross-sectional area.

3. Computational Fluid Dynamic Modeling

A single CFD simulation is, in general, composed of four steps; the geometry generation, 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 each step of the CFD modeling process, for both the high-fidelity and low-fidelity models.

3.1. Airfoil Parameterization

In this paper, for the sake of simplicity, we use the NACA airfoil shapes. 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 Doenhoff [21]. Three example NACA four-digit airfoils are shown in Fig. 2.

3.2. High-Fidelity CFD Model

Transonic flow past airfoils is characterized by regions of locally subsonic ($\text{Mach} < 0.8$) and supersonic ($\text{Mach} > 1.2$) flow that occurs over a body which is moving at Mach numbers near unity. Assuming an inviscid, adiabatic flow with no body forces, the Euler equations are the most accurate description of the fluid flow. The Euler

equations are a set of coupled, non-linear partial differential equations that represent the conservation of mass, momentum and energy, written in Cartesian coordinates as [22]

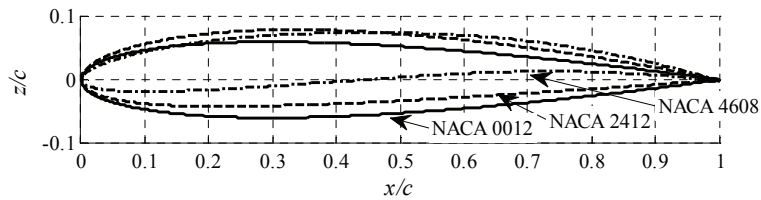


Fig 2. Shown are three different NACA four-digit airfoil sections. NACA 0012 ($m = 0, p = 0, t/c = 0.12$) is shown by a solid line (-). NACA 2412 ($m = 0.02, p = 0.4, t/c = 0.12$) is shown by a dash line (--). NACA 4608 ($m = 0.04, p = 0.6, t/c = 0.08$) is shown by a dash-dot line (-.-).

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} = 0 \quad (3)$$

where $\mathbf{U}$, $\mathbf{E}$ and $\mathbf{F}$ are vectors given by

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E_t \end{bmatrix}, \quad \mathbf{E} = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (E_t + p)u \end{bmatrix}, \quad \text{and} \quad \mathbf{F} = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (E_t + p)v \end{bmatrix} \quad (4)$$

in which ρ is air density, u and v are the velocity components in the x and y directions, respectively, p is static pressure, and $E_t = \rho(e + V^2/2)$ is the total energy per unit volume, e is the internal energy per unit mass, $V^2/2$ is the kinetic energy per unit mass. These equations hold, in the absence of separation and other strong viscous effects, for any shape of the body, thick or thin, and at any angle of attack. However, these four scalar equations contain five unknowns, namely (ρ, p, e, u, v) . An equation of state is needed to close the system of equations. Assuming a perfect gas, the equation of state is

$$p = \rho RT \quad (5)$$

where R is the gas constant. In the current work, the flow is assumed to steady, so the first term of Eq. (6) vanishes.

The computational meshes used in this study are all of structured curvilinear body-fitted C-topology and are generated using the computer code ICEM CFD [23]. 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. A fine mesh was developed with a total of 320 points in the z -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. 3.

The numerical fluid flow simulations are performed using the computer code FLUENT [24]. 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 Sec. 3.4) are monitored for convergence. The criteria used in this work for the high-fidelity model is a maximum residual of 10^{-6} , or a maximum number of iterations of 1000.

3.3. Low-Fidelity CFD Model

The low-fidelity CFD model is constructed by using the Euler equations, but with a coarser computational mesh and relaxed convergence criteria. The parameters of the mesh and the number of solver iterations were obtained by performing a parametric study using the NACA 2412 airfoil section at $M_\infty = 0.75$ and $\alpha = 1$ deg.

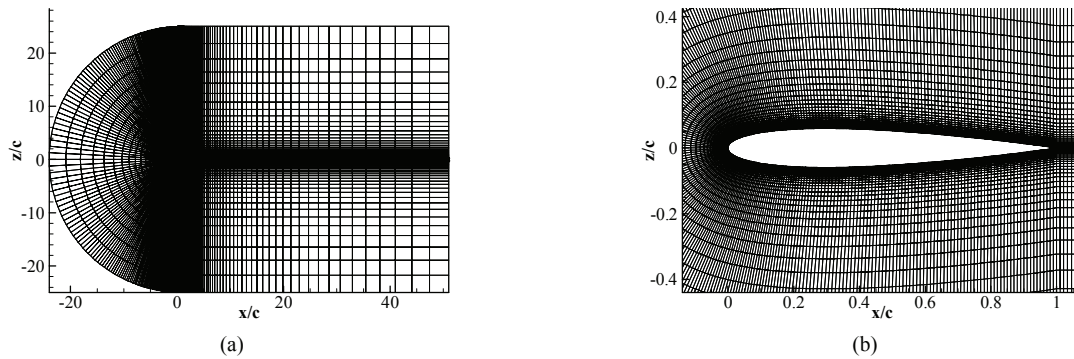


Fig. 3. (a) An example computational mesh with a structured C-topology for a NACA 0012 airfoil, (b) a view of the mesh close to the airfoil surface.

Initially the fine mesh is solved to full convergence. The solver needed 216 iterations to reach a converged solution based on the residuals of the governing equations. However, the lift and drag coefficient values reached a converged value after approximately 50 iterations. Therefore, the number of iterations limit was set to 100 hundred iterations in the subsequent steps.

The mesh points were reduced in two steps. First, the number of mesh points in the z -direction and the number of mesh points behind the airfoil were reduced by (approximately) half. This procedure was repeated and in each step the pressure distribution was plotted. It was observed that the shock location moved rearward. After five subsequent steps, the pressure distribution became highly distorted near the leading-edge. The previous mesh was then retained and the number of mesh points on the airfoil surface was reduced incrementally. Again, the pressure distribution was plotted for each step. It was observed that the shock strength reduced in each step as the mesh got coarser and coarser on the airfoil surface. After three steps the procedure was halted. The resulting mesh has 48 points in the z -direction, 115 points on the airfoil surface, and 20 points in the wake behind the airfoil, with a total of 8295 thousand cells. The overall evaluation time is reduced to about 34 s, which is approximately 13.5 times faster than the high-fidelity model using the fine mesh and traditional convergence criteria.

The overall evaluation time of the high-fidelity model in this parametric study is 471 s with a total of 216 iterations. In many cases the solver does not fully converge with respect to the residuals and goes on up to 1000 iterations. Then the overall evaluation time goes up to 2500 s, and the low-fidelity model is approximately 73 times faster. For the sake of simplicity, we will use a fixed value of 50 in the numerical computations presented later in the paper. Note that the evaluation times reported here includes the time required for connecting to twice to the license server, once for the grid generator, ICM CFD [23], and once for the flow solver, FLUENT [24].

3.4. Aerodynamic Forces

The non-dimensional force coefficients parallel to the x - and z -axes, C_x and C_z , respectively, are calculated by integrating the pressure distribution C_p over the surface of the airfoil as [22]

$$C_x = \oint C_p \sin \theta \, ds, \quad C_z = -\oint C_p \cos \theta \, ds \quad (6)$$

where ds is the surface panel element of length and θ is the angle the panel makes relative to the x -axis. The lift coefficient C_l and the wave drag coefficient C_{dw} are calculated as

$$C_l = -C_x \sin \alpha + C_z \cos \alpha, \quad C_{dw} = C_x \cos \alpha + C_z \sin \alpha. \quad (7)$$

4. Surrogate Modeling

In order to use the low-fidelity model as a reliable prediction tool in the optimization process, it has to be corrected to become a reliable representation of the high-fidelity model. The corrected low-fidelity model is called a

surrogate. The surrogate model can replace the high-fidelity one in the optimization process and thus reduce the overall optimization cost.

Here, we adopt a shape-preserving response prediction (SPRP) methodology introduced in [19] in the context of microwave engineering, and recently applied for direct airfoil design [14]. SPRP is easy to implement, unlike space mapping it does not need any auxiliary transformations or extractable parameters [25]. Also, it does not require high-fidelity model derivative information.

By formulation, SPRP works directly with the model outputs that can be described by certain number of design-variable-dependent characteristic features [19]. In the case of a pressure distribution, these features may include the location and strength of the shock, the pressure at the leading- and trailing-edges, and many others. In direct airfoil design, the pressure distribution is an intermediate simulation result, with the figures of interest, such as lift or wave drag, being derived from it. In inverse design, the pressure distribution is the main object of interest, which makes SPRP well suited for this kind of problem.

We will denote the vector of design variables as $\mathbf{x}$. The pressure distribution for the high- and low-fidelity models will be denoted as $C_{p,f}$ and $C_{p,c}$, respectively. The surrogate model is constructed assuming that the change of $C_{p,f}$ due to the adjustment of the design variables $\mathbf{x}$ can be predicted using the actual changes of $C_{p,c}$. The change of $C_{p,c}$ is described by the translation vectors corresponding to certain (finite) number of its characteristic points. These translation vectors are subsequently used to predict the change of $C_{p,f}$, whereas the actual $C_{p,f}$ at the current design, $C_{p,f}(\mathbf{x}^{(i)})$, is treated as a reference.

Figure 4(a) shows the pressure distribution $C_{p,c}$ of the low-fidelity model at $\mathbf{x}^{(i)} = [0.02 \ 0.4 \ 0.12]^T$ (NACA 2412 airfoil) for $M_\infty = 0.7$ and $\alpha = 1$ deg, as well as $C_{p,c}$ at $\mathbf{x} = [0.025 \ 0.56 \ 0.122]^T$; $\mathbf{x}^{(i)}$ will denote a current design (at the i th iteration of the optimization algorithm; the initial design will be denoted as $\mathbf{x}^{(0)}$ accordingly). Circles denote characteristic points of $C_{p,c}(\mathbf{x}^{(i)})$, here, representing, among others, x/c equal to 0 and 1 (leading and trailing airfoil edges, respectively), the maxima of $C_{p,c}$ for the lower and upper airfoil surfaces, as well as the local minimum of $C_{p,c}$ for the upper surface. The last two points are useful to locate the pressure shock. Squares denote corresponding characteristic points for $C_{p,c}(\mathbf{x})$, while small line segments represent the translation vectors that determine the “shift” of the characteristic points of $C_{p,c}$ when changing the design variables from $\mathbf{x}^{(i)}$ to $\mathbf{x}$.

In order to obtain a reliable prediction, the number of characteristic points has to be larger than illustrated in Fig. 4(a). Additional points are inserted in between initial points either uniformly with respect to x/c (for those parts of the pressure distribution that are almost flat) or based on the relative pressure value with respect to corresponding initial points (for those parts of the pressure distribution that are “steep”). Figure 4(b) shows the full set of characteristic points (initial points are distinguished using larger markers).

The pressure distribution of the high-fidelity model at the given design, here, $\mathbf{x}$, can be predicted using the translation vectors applied to the corresponding characteristic points of the pressure distribution of the high-fidelity model at $\mathbf{x}^{(i)}$, $C_{p,f}(\mathbf{x}^{(i)})$. This is illustrated in Figure 5(a) where only initial characteristic points and translation vectors are shown for clarity. Figure 5(b) shows the predicted pressure distribution of the high-fidelity model at $\mathbf{x}$ as well as the actual $C_{p,f}(\mathbf{x})$. The agreement between both curves is very good.

Rigorous formulation of the SPRP technique can be found in [19]. We omit the details here for the sake of brevity. It should be mentioned that the SPRP model assumes that the high- and low-fidelity model pressure distributions have corresponding sets of characteristic points. This is usually the case for the practical ranges of design variables because the overall shape of the distributions is similar for both models. In case of a lack of correspondence, original definitions of characteristic points are replaced by their closest counterparts. The typical example would be non-existence of the local minimum of the pressure distribution for the upper surface for the high- and/or low-fidelity model at certain designs. In this case, the original point (local minimum) is replaced by the points characterized by the largest curvature.

5. Optimization Methodology

5.1. Objective Function

In inverse design, the primary objective is the alignment between the pressure distribution of the airfoil being designed and the prescribed target. The alignment can be measured using a norm.

Due to unavoidable misalignment between the pressure distributions of the high-fidelity model and its SPRP surrogate, it is not convenient to use some of the constraints (e.g., a maximum allowable drag) directly. This is because the design that is feasible for the surrogate model, may not be feasible for the high-fidelity model. In

particular, the design obtained as a result of optimizing the surrogate model $C_{p,s}^{(i)}$, i.e., $\mathbf{x}^{(i+1)}$, will be feasible for $C_{p,s}^{(i)}$. However, if $\mathbf{x}^{(i+1)}$ is not feasible for the high-fidelity model, it will not be feasible for $C_{p,s}^{(i+1)}$ because we have $C_{p,s}^{(i+1)}(\mathbf{x}^{(i+1)}) = C_{p,f}(\mathbf{x}^{(i+1)})$ by the definition of the surrogate model. In order to alleviate this problem, we shall use the penalty function approach to handle the drag constraint. Of course, this does not apply to constraints that depend exclusively on the airfoil geometry, such as the minimum airfoil cross-sectional area.

In this work, all the constraints are handled through penalty functions so that the objective function is defined as follows

$$H(C_p(\mathbf{x})) = F^2 + \beta [\Delta C_{dw,s}(C_p(\mathbf{x}))]^2 + \delta [\Delta C_{L,s}(C_p(\mathbf{x}))]^2 + \gamma [\Delta A(\mathbf{x})]^2 \quad (8)$$

with $F = \|C_p(\mathbf{x}) - C_{p,t}\|$, where $C_{p,t}$ is the target pressure distribution, $\Delta C_{dw,s} = 0$ if $C_{dw,s} \leq C_{dw,s,max}$ and $\Delta C_{dw,s} = C_{dw,s} - C_{dw,s,max}$ otherwise, $\Delta C_{L,s} = 0$ if $C_{L,s} \geq C_{L,s,min}$ and $\Delta C_{L,s} = C_{L,s} - C_{L,s,min}$ otherwise, and $\Delta A = 0$ if $A \geq A_{min}$ and $\Delta A = A - A_{min}$ otherwise. In our numerical experiments we use $\beta = \delta = \gamma = 1000$. Here, the pressure distribution for the surrogate model is $C_p = C_{p,s}$, and for the high-fidelity model $C_p = C_{p,f}$. Also, $C_{L,s}$ and $C_{dw,s}$ denote the lift and wave drag coefficients (both being functions of the pressure distribution).

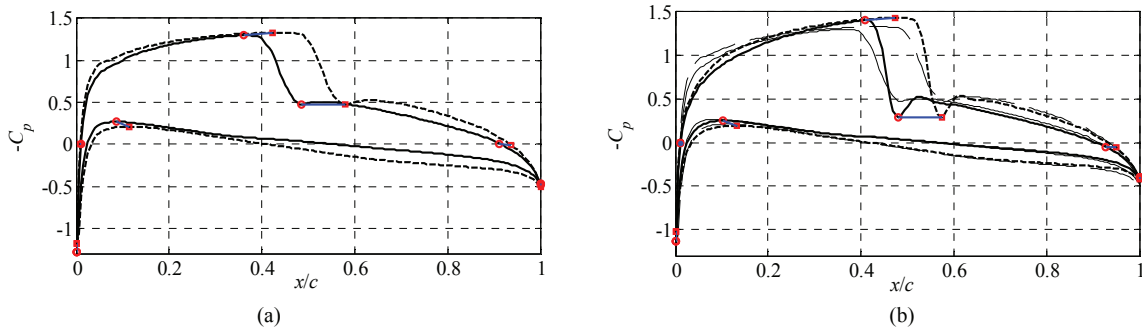


Fig. 4. (a) Example low-fidelity model pressure distribution at the design $\mathbf{x}^{(i)}$, $C_{p,c}(\mathbf{x}^{(i)})$ (solid line), the low-fidelity model pressure distribution at other design $\mathbf{x}$, $C_{p,c}(\mathbf{x})$ (dotted line), characteristic points of $C_{p,c}(\mathbf{x}^{(i)})$ (circles) and $C_{p,c}(\mathbf{x})$ (squares), and the translation vectors (short lines); (b) low-fidelity model pressure distributions, initial characteristic points (large markers) and translation vectors from Fig. 4(a) as well as additional points (small markers) inserted in between the initial points either uniformly with respect to x/c (for those parts of the pressure distribution that are almost flat) or based on the relative pressure value with respect to corresponding initial points (for those parts of the pressure distribution that are “steep”).

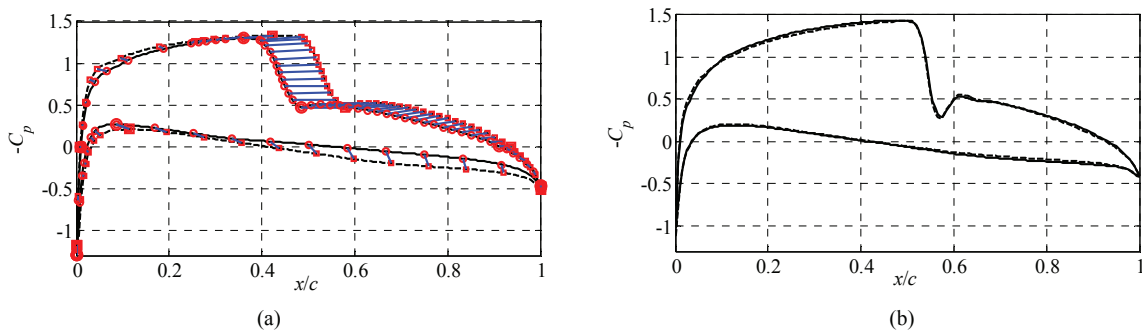


Fig. 5. (a) High-fidelity model pressure distribution at $\mathbf{x}^{(i)}$, $C_{p,f}(\mathbf{x}^{(i)})$ (solid line) and the predicted high-fidelity model C_p at $\mathbf{x}$ (dotted line) obtained using SPRP based on characteristic points of Fig. 4(b); characteristic points of $C_{p,f}(\mathbf{x}^{(i)})$ (circles) and the translation vectors (short lines) were used to find the characteristic points (squares) of the predicted high-fidelity model pressure distribution (only initial points are shown for clarity); low-fidelity model distributions $C_{p,c}(\mathbf{x}^{(i)})$ and $C_{p,c}(\mathbf{x})$ are plotted using thin solid and dotted line, respectively; (b) high-fidelity model pressure distribution at $\mathbf{x}$, $C_{p,f}(\mathbf{x})$ (solid line), and the predicted high-fidelity model pressure distribution at $\mathbf{x}$ obtained using SPRP (dotted line).

5.2. Optimization Algorithm

The optimization algorithm exploits the SPRP-based surrogate model and a trust-region convergence safeguard [20]. It can be summarized as follows:

1. Set $i = 0$; Select λ (initial trust region radius); Evaluate $C_{p,f}(\mathbf{x}^{(0)})$;
2. Set up SPRP model;
3. Obtain $\mathbf{x}^{(i+1)} = \operatorname{argmin}\{l \leq \mathbf{x} \leq u, \|\mathbf{x} - \mathbf{x}^{(i)}\| \leq \lambda : H(C_{p,s}^{(i)}(\mathbf{x}))\}$;
4. Evaluate high-fidelity model to get $C_{p,f}(\mathbf{x}^{(i)})$;
5. If $H(C_{p,f}^{(i)}(\mathbf{x}^{(i+1)})) < H(C_{p,f}^{(i)}(\mathbf{x}^{(i)}))$ accept $\mathbf{x}^{(i+1)}$; Otherwise $\mathbf{x}^{(i+1)} = \mathbf{x}^{(i)}$;
6. Update λ ;
7. Set $i = i + 1$;
8. If termination condition is not satisfied, go to 2.

The SPRP surrogate model is updated before each iteration of the optimization algorithm using the high-fidelity model data at the design obtained in the previous iteration. The trust-region parameter λ is updated after each iteration, i.e., decreased if the new design was rejected or the improvement of the high-fidelity model objective function was too small compared to the prediction given by the SPRP surrogate, or increased otherwise. We use classical updating rules (see, e.g., [20, 26]). The algorithm is terminated if $\|\mathbf{x}^{(i+1)} - \mathbf{x}^{(i)}\| < 0.001$ or $\lambda < 0.001$.

6. Verification Examples

The results of inverse design using the proposed algorithm are presented for four test cases. The grid-search algorithm (see, e.g., [27]) is used in the surrogate model optimization. The design variables are the airfoil shape parameters in the NACA four-digit parameterization (described in Sec. 3.1), i.e., $\mathbf{x} = [m \ p \ t]^T$. Note that the chord length is set to 1. The only inequality constraint is the minimum cross-sectional area constraint. There are no equality constraints. The side constraints are $0 \leq m \leq 0.1$, $0.2 \leq p \leq 0.8$, and $0.05 \leq t \leq 0.20$. For comparison, the inverse design test cases are also solved by using only the high-fidelity CFD model and the grid-search algorithm. The test cases were chosen only for verification purposes and they do not represent optimal airfoil designs. The results are presented in Table 1.

In case 1, the target pressure distribution is the one of NACA 3210 at $M_\infty = 0.75$ and $\alpha = 1^\circ$. The initial design is NACA 0012 and the minimum cross-sectional area is $A_{\min} = 0.065$. Fig. 6(a) shows the pressure distributions for the target, initial, and optimized designs. There is clearly a large difference between the initial and target pressure distributions. On the upper surface, the target pressure distribution has a much stronger shock located further downstream than the shock on the initial design. Near the leading-edge on the lower surface of the target pressure distribution, the pressure is reduced sharply behind the stagnation point, but then it rises and recedes again, before tending towards the pressure value at the trailing-edge. For the initial design, on the lower surface, the pressure distribution is much smoother, and does not have this nonlinearity near the leading-edge. However, the proposed algorithm is able to match the target pressure distribution relatively well, with the norm of the difference between the two distributions being $F = 0.0182$, using a total of 6 high-fidelity function calls and 457 surrogate function calls. The total optimization cost, expressed in terms of the equivalent number of high-fidelity function calls, is less than 16 high-fidelity function calls. The largest difference in the pressure distributions of the target and the optimized design is near the leading-edge, where the pressure distribution is highly nonlinear. Inverse design using the high-fidelity CFD model and the grid-search algorithm required 202 function calls. The optimized design is fairly similar to the one obtained using the surrogate-based algorithm.

The target pressure distribution is the same for cases 2, 3, and 4, i.e., $\mathbf{x} = [0.0163 \ 0.4004 \ 0.1118]^T$ at $M_\infty = 0.75$ and $\alpha = 0^\circ$. In case 2 the initial design is NACA 2412 and $A_{\min} = 0.078$. Our method hits the cross-sectional area constraint and matches the target pressure distribution only relatively closely ($F = 0.0243$). For case 3, the constraint value is lowered to $A_{\min} = 0.075$, but other parameters are kept the same. Now, our approach is able to match the target much closer ($F = 0.0025$) with the design cost of about 13 equivalent high-fidelity function calls, whereas using the high-fidelity CFD model and the grid-search algorithm required 152 function calls. Case 4 starts with a different initial design, namely the NACA 0012. Here, the proposed method is able to match the target distribution closely ($F = 0.0023$) as can be seen in Fig. 6(b), using less than 16 equivalent high-fidelity function calls. The comparison method required 201 high-fidelity function calls and is unable to match the target as close as the proposed method ($F = 0.0181$).

Table 1. Numerical results of four inverse design cases studies using the proposed optimization methodology. F is the norm of the difference of pressure distributions for the optimized and the target designs. N_c is the number of low-fidelity model evaluations and N_f is the number of high-fidelity model evaluations. All the numerical values are from the high-fidelity model.

Variable	Case 1 $M_\infty = 0.75, \alpha = 1^\circ, A_{min} = 0.065$				Case 2 $M_\infty = 0.75, \alpha = 0^\circ, A_{min} = 0.078$			
	Initial	Target	Grid-Search [#]	VR-SPRP [§]	Initial	Target	Grid-Search [#]	VR-SPRP [§]
m	0.0000	0.0300	0.0287	0.0293	0.0200	0.0163	0.0153	0.0154
p	0.4000	0.2000	0.2067	0.2057	0.4000	0.4004	0.4089	0.4043
t	0.1200	0.1000	0.1083	0.1022	0.1200	0.1118	0.1159	0.1158
C_l	0.2082	0.8035	0.7786	0.7905	0.4745	0.3832	0.3663	0.3663
C_{dw}	0.0024	0.0410	0.0424	0.0405	0.0115	0.0049	0.0049	0.0050
A	0.0808	0.0675	0.0731	0.0689	0.0050	0.0753	0.0780	0.0780
F	N/A	N/A	0.0492	0.0182	N/A	N/A	0.0249	0.0243
N_c	N/A	N/A	0	457	N/A	N/A	0	330
N_f	N/A	N/A	202	6	N/A	N/A	201	11
Total cost [*]	N/A	N/A	202	< 16	N/A	N/A	201	< 18

Variable	Case 3 $M_\infty = 0.75, \alpha = 0^\circ, A_{min} = 0.075$				Case 4 $M_\infty = 0.75, \alpha = 0^\circ, A_{min} = 0.075$			
	Initial	Target	Grid-Search [#]	VR-SPRP [§]	Initial	Target	Grid-Search [#]	VR-SPRP [§]
m	0.0200	0.0163	0.0162	0.0162	0.0000	0.0163	0.0156	0.0163
p	0.4000	0.4004	0.4007	0.4014	0.5000	0.4004	0.4252	0.3978
t	0.1200	0.1118	0.1122	0.1121	0.1000	0.1118	0.1139	0.1117
C_l	0.4745	0.3832	0.3816	0.3822	0.0005	0.3832	0.3776	0.3835
C_{dw}	0.0115	0.0049	0.0049	0.0048	0.0002	0.0049	0.0045	0.0049
A	0.0808	0.0753	0.0756	0.0755	0.0673	0.0753	0.0767	0.0752
F	N/A	N/A	0.0025	0.0025	N/A	N/A	0.0181	0.0023
N_c	N/A	N/A	0	240	N/A	N/A	0	468
N_f	N/A	N/A	152	8	N/A	N/A	201	6
Total cost [*]	N/A	N/A	152	< 13	N/A	N/A	201	< 16

[#] Design obtained using the high-fidelity model and the grid-search algorithm [27].

[§] Design obtained using the algorithm proposed in Sec. 5; surrogate model optimization performed using the grid-search algorithm [27].

^{*} The total optimization cost is expressed in terms of the equivalent number of high-fidelity model evaluations. The ratio of the high-fidelity model evaluation time to the corrected low-fidelity model evaluation time varies between 13.5 to 73 depending on the design. For the sake of simplicity we use a fixed value of 50 here.

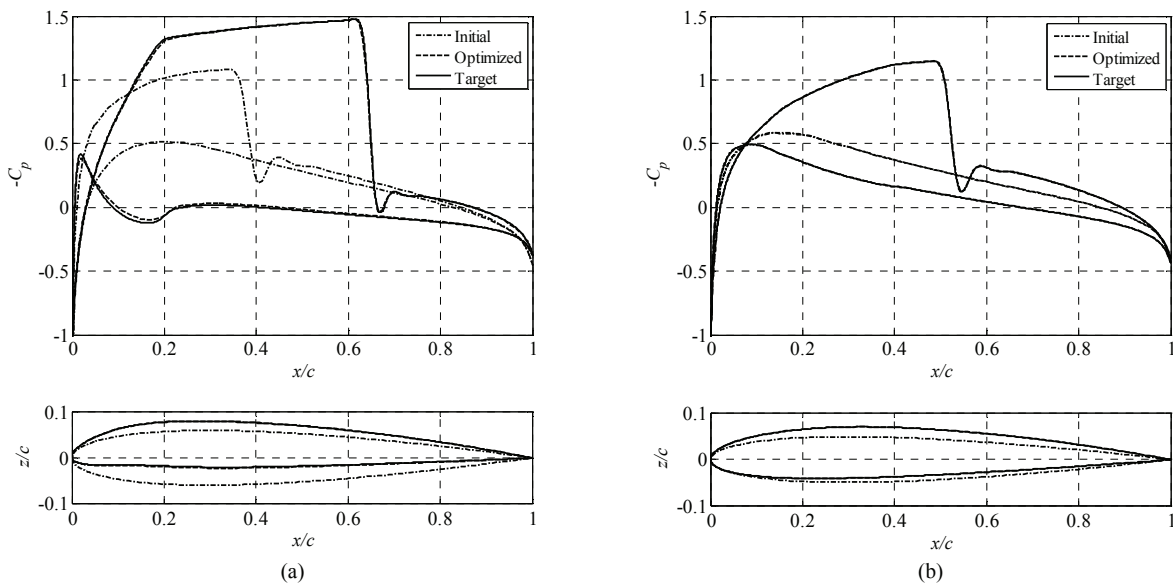


Fig. 6. The pressure distributions and airfoil shapes for the initial, optimized and target designs for (a) case 1; and (b) case 4.

7. Conclusions

A variable-resolution optimization algorithm for the inverse design of transonic airfoils has been presented. The algorithm uses a computationally cheap low-fidelity model to construct a surrogate of an accurate but CPU-intensive

high-fidelity model. The low-fidelity model is based on same governing equations as the high-fidelity model, but exploits coarser discretization and relaxed convergence criteria. The low-fidelity model is corrected by aligning the airfoil surface pressure distribution with the corresponding distribution of the high-fidelity model by means of the shape-preserving response correction prediction technique, which ensures a good generalization capability of the surrogate model with respect to both objectives and constraints. The alignment process is particularly suitable since a target pressure distribution is specified in the inverse design problem. The algorithm performance is enhanced by embedding it in the trust region framework. Our approach is simple to implement and its robustness is demonstrated through several test cases with target pressure distributions obtained using only a few high-fidelity model evaluations.

References

1. Leoviriyakit, K., Kim, S., and Jameson, A., "Viscous Aerodynamic Shape Optimization of Wings including Planform Variables," *21st Applied Aerodynamics Conference*, Orlando, Florida, June 23-26, 2003.
2. Braembussche, R.A., "Numerical Optimization for Advanced Turbomachinery Design," Optimization and Computational Fluid Dynamics, Thevenin, D. and Janiga, G., editors, Springer, 2008, pp. 147-189.
3. Queipo, N.V., Haftka, R.T., Shyy, W., Goel, T., Vaidyanathan, R., and Tucker, P.K., "Surrogate-Based Analysis and Optimization," *Progress in Aerospace Sciences*, Vol. 41, No. 1, 2005, pp. 1-28.
4. Forrester, A.I.J., and Keane, A.J., "Recent advances in surrogate-based optimization," *Progress in Aerospace Sciences*, Vol. 45, No. 1-3, 2009, pp. 50-79.
5. Alexandrov, N.M., Lewis, R.M., Gumbert, C.R., Green, L.L., and Newman, P.A., "Optimization with Variable-Fidelity Models Applied to Wing Design," *38th Aerospace Sciences Meeting & Exhibit*, Reno, NV, AIAA Paper 2000-0841, Jan. 2000.
6. Robinson, T.D., Eldred, M.S., Willcox, K.E., and Haimes, R., "Surrogate-Based Optimization Using Multifidelity Models with Variable Parameterization and Corrected Space Mapping," *AIAA Journal*, Vol. 46, No. 11, November 2008.
7. Dulikravich, G.S., "Aerodynamic Shape Design and Optimization," *29th AIAA Aerospace Sciences Meeting*, Reno, NV, 1991.
8. Hicks, R.M., and Henne, P.A., "Wing Design by Numerical Optimization," *Journal of Aircraft*, Vol. 15, No. 7, 1978, pp. 407-412.
9. Jameson, A., "Aerodynamic Design via Control Theory," *Journal of Scientific Computing*, Vol. 3, 1988, pp. 233-260.
10. Kim, S., Alonso, J.J., and Jameson, A., "Design Optimization of High-Lift Configurations Using a Viscous Continuous Adjoint Method," AIAA paper 2002-0844, *AIAA 40th Aerospace Sciences Meeting & Exhibit*, Reno, NV, January 2002.
11. Nemec, M., and D.W. Zingg, D.W., "Optimization of high-lift configurations using a Newton-Krylov algorithm," *16th AIAA Computational Fluid Dynamics Conference*, Orlando, Florida, June 23-26, 2003.
12. Booker, A.J., Dennis Jr., J.E., Frank, P.D., Serafini, D.B., Torczon, V., and Trosset, M.W., "A rigorous framework for optimization of expensive functions by surrogates," *Structural Optimization*, Vol. 17, No. 1, 1999, pp. 1-13.
13. Barrett, T.R., Bressloff, N.W., and Keane, A.J., "Airfoil Design and Optimization Using Multi-Fidelity Analysis and Embedded Inverse Design," AIAA Paper 2006-1820, *47th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference*, Newport, Rhode Island, May 1-4, 2006.
14. Leifsson, L., and Koziel, S., "Multi-fidelity design optimization of transonic airfoils using physics-based surrogate modeling and shape-preserving response prediction," *Journal of Computational Science*, Vol. 1, No. 2, 2010, pp.98-106.
15. Lane, K.A., and Marshall, D.D., "Inverse Airfoil Design Utilizing CST Parameterization," *48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, Orlando, Florida, 2010.
16. Labrujere, T.E., and Slooff, J.W., "Computational Methods for the Aerodynamic Design of Aircraft Components," *Annual Review of Fluid Mechanics*, Vol. 25, 1993, pp. 183-214.
17. Kim, H.J., and Rho, O.H., "Aerodynamic Design of Transonic Wings Using the Target Pressure Optimizaiton Approach," *Journal of Aircraft*, Vol. 35, No. 5, 1998, pp. 671-677.
18. Lieback, R.H., "Subsonic Airfoil Design," Applied Computational Aerodynamics, edited by P.A. Henne, Vol 125 of *Progress of Astronautics and Aeronautics*, AIAA, Washington, DC, 1990, pp. 133-165.
19. Koziel, S., "Shape-preserving response prediction for microwave design optimization," *IEEE Trans. Microwave Theory and Tech.*, Vol. 58, 2010, pp. 2829-2837.
20. Conn, A.R., Gould, N.I.M., and Toint, P.L., *Trust Region Methods*, MPS-SIAM Series on Optimization, 2000.
21. Abbott, I.H., and Von Doenhoff, A.E., *Theory of Wing Sections*, Dover Publications, 1959.
22. Anderson, J.D., *Modern Compressible Flow – With Historical Perspective*, 3rd edition, McGraw-Hill, 2003.
23. ICFM CFD, ver. 12.1, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, 2006.
24. FLUENT, ver. 12.1, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317, 2006.
25. Koziel, S., Cheng, Q.S., and Bandler, J.W., "Space mapping," *IEEE Microwave Magazine*, Vol. 9, No. 6, 2008, pp. 105-122.
26. Koziel, S., Bandler, J.W., and Madsen, K., "Space-mapping based interpolation for engineering optimization," *IEEE Trans. Microwave Theory and Tech.*, Vol. 54, No. 6, 2006, pp. 2410-2421.
27. Koziel, S., "Multi-fidelity multi-grid design optimization of planar microwave structures with Sonnet," *International Review of Progress in Applied Computational Electromagnetics*, Tampere, Finland, April 26-29, 2010, pp. 719-724.