Hydrodynamic Shape Optimization of Axisymmetric Bodies Using Multi-fidelity Modeling

Leifur Leifsson, Slawomir Koziel, and Stanislav Ogurtsov

Engineering Optimization & Modeling Center, School of Science and Engineering, Reykjavik University, Menntavegur 1, 101 Reykjavik, Iceland
{leifurth,koziel,stanislav}@ru.is

Abstract. Hydrodynamic shape optimization of axisymmetric bodies is presented. A surrogate-based optimization algorithm is described that exploits 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 the same governing equations as the high-fidelity one, but exploits coarser discretization and relaxed convergence criteria. A multiplicative response correction is applied to the low-fidelity CFD model output to yield an accurate and reliable surrogate model. The approach is implemented for both direct and inverse design. In the direct design approach the optimal hull shape is found by minimizing the drag, whereas in the inverse approach a target pressure distribution is matched. Results show that optimized designs are obtained at substantially lower computational cost (over 94%) when compared to the direct high-fidelity model optimization.

Keywords: Shape Optimization, Surrogate-based Optimization, Multi-fidelity Modeling, Axisymmetric Body, CFD, Direct Design, Inverse Design.

1 Introduction

Autonomous underwater vehicles (AUVs) are becoming increasingly important in various marine applications, such as oceanography, pipeline inspection, and mine counter measures [1]. Endurance (speed and range) is one of the more important attribute of AUVs [2]. Vehicle drag reduction and/or an increase in the propulsion system efficiency will translate to a longer range for a given speed (or the same distance in a reduced time). A careful hydrodynamic design of the AUVs, including the hull shape, the protrutions, and the propulsion system, is therefore essential.

The fluid flow around an underwater vehicle with appendages is characterized by flow features such as thick boundary layers, vortices and turbulent wakes generated due to the hull and the appendages [3]. These flow features can have adverse effects on, for example, the performance of the propulsion system and the control planes. Moreover, the drag depends highly on the vehicle shape, as well as on the aforementioned flow features. Consequently, it is important to account for these effects during the design of the AUVs.

The prediction of the flow past the full three-dimensional configuration of the AUVs requires the use of computational fluid dynamics (CFD). Numerous

N. Pina et al. (Eds.): *Simulation & Modeling Methodologies, Technologies & Appl.*, AISC 197, pp. 209–223.
DOI: 10.1007/978-3-642-34336-0_14

applications of CFD methods to the flow past AUVs and other underwater vehicles are in the literature, see, e.g., [4-6]. The purpose of these investigations is to predict properties such as added masses, pressure and friction distributions, drag, normal force and moment coefficients, wake field, and stability derivatives. Comparison with experimental measurements show that CFD is reliable and can yield accurate results.

Numerous studies on underwater vehicle design and optimization have been reported which focus on the clean hull only, i.e., the appendages and the propulsion system are neglected and the flow is taken to be past an axisymmetric body at a zero angle of attack. Examples of such numerical studies can be found in [7-13]. Allen et al. [2], on the other hand, report an experimental investigation of propulsion system enhancements and drag reduction of an AUV.

The hydrodynamic design optimization of AUVs in full configuration, taking into account the appendages and the propulsion system, is still an open problem. One of the main challenges involved is the high computational cost of a CFD simulation. A single CFD simulation of the three-dimensional flow past an AUV can take a few hours up to several days, depending on the computational power, the grid density, and the flow conditions. Therefore, the direct optimization, as shown in Fig. 1(a), can be impractial, especially using conventional gradient-based methods.

An important research area in the field of aerodynamic optimization is focused on employing the surrogate-based optimization (SBO) techniques [14,15]. One of the major objectives is to reduce the number of high-fidelity model evaluations, and thereby making the optimization process more efficient. In SBO, the accurate, but computationally expensive, high-fidelity CFD simulations are replaced—in the optimization process—by a cheap surrogate model (Fig. 1(b)). SBO has been successfully applied to the aerodynamic design optimization of various aerospace components, such as airfoils [16], aircraft wings [17], and turbine blades [18].

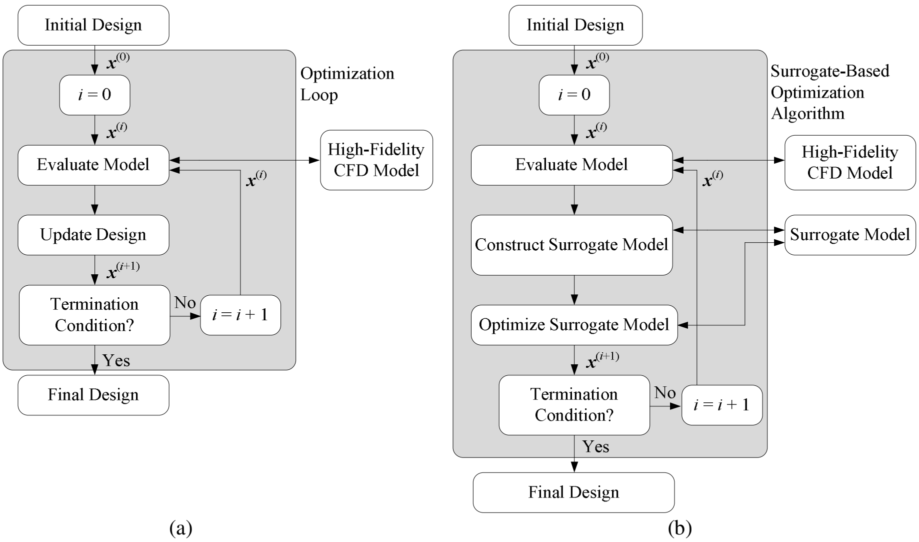

Fig. 1. (a) Direct optimization, (b) surrogate-based optimization. Here, $\boldsymbol{x}^{(i)}$ denotes the design variable vector at iteration i.