Journal of Computational Science 7 (2015) 1–12

Contents lists available at ScienceDirect

Journal of Computational Science

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

Simulation-driven design of low-speed wind tunnel contraction

Leifur Leifsson[a,b,*], Slawomir Koziel[a]

[a] Engineering Optimization & Modeling Center, Reykjavik University, Menntavegur 1, 101 Reykjavik, Iceland
[b] Iowa State University, Ames, IA 50011, USA

ARTICLE INFO

Article history:
Received 9 June 2014
Received in revised form
15 December 2014
Accepted 29 December 2014
Available online 8 January 2015

Keywords:
Simulation-driven design
Numerical optimization
Surrogate-based modeling
Low-speed wind tunnel
Experimental verification

ABSTRACT

A low-speed wind tunnel is developed for conducting research on the flow past micro air vehicles. The tunnel is of open suction type and is composed of a square inlet with a honeycomb and turbulence screens, settling chamber, contraction, experimental section housing, diffuser, and axial fan. In this paper, we describe the details of the design optimization procedure of the contraction, which is key to getting a high quality flow in the experimental section. A high-fidelity computational fluid dynamic (CFD) flow solver is used to capture the nonlinear flow physics. Due to the high computational expense of the CFD simulations, surrogate-based optimization (SBO) is used to accelerate the design process. The SBO approach replaces direct optimization of the high-fidelity (accurate but computationally expensive) model by iterative optimization of a properly corrected low-fidelity model. Here, we exploit variable–fidelity CFD simulations, as well as a simple multiplicative response correction technique to construct the surrogate model of the wind tunnel contraction, allowing us to optimize its shape at a low computational cost. To our knowledge, it is the first application of variable–fidelity surrogate modeling to wind tunnel contraction design. The optimum nozzle design is verified using a high-fidelity CFD simulation, as well as by experimental measurements of the fabricated wind tunnel. Experimental validation confirms the correctness of the numerical optimization procedures utilized to design the contraction.

1. Introduction

A low-speed wind tunnel, intended for fluid dynamics research, has been developed at the authors' institution. The utilization of the facility will involve research on micro air vehicles (MAVs), both with fixed- and flapping-wings. The work conducted in the tunnel will involve analysis of various aspects of MAVs, such as the wing shape and flapping parameters, and control effectiveness. High flow quality is essential for the research as the facility is expected to provide capability for the characterization of the aerodynamic performance of MAVs at low Reynolds numbers ($Re < 100{,}000$). The required flow speed range is from 0 to at least 20 m/s.

The wind tunnel is of an open suction type, as shown in Fig. 1, and is composed of a square inlet with a honeycomb and turbulence screens, settling chamber, contraction, experimental section housing, diffuser, and axial fan. The experimental section is located in a large enclosure which houses the experimental equipment (the MAV) and the instrumentation (such as a force balance, high-resolution video cameras, or a particle image velocimetry (PIV) system).

The contraction accelerates and aligns the flow into the test section. The contraction is, therefore, a critical component of the wind tunnel for the provision of high quality test flow in the working section [1]. The size and shape of the contraction controls the quality (such as flow angularity and uniformity, as well as the turbulence intensity levels) of the flow at the outlet [2]. The contraction should be short to minimize boundary layer growth and the flow leaving the contraction should be uniform and steady [3,4]. Moreover, flow separation, due to streamline curvature in the contraction, needs to be avoided at any cost [5].

As the effects of the shape on the flow are highly nonlinear, it is necessary to use computational fluid dynamics (CFD) to make design decisions about key features of the contraction, such as the length, contraction ratio, and, in particular, the wall shape. Examples of such work can be found in Mathew [3] and Mathew et al. [4], Watmuff [6], Su [7], and Doolan and Morgans [8]. The first three are similar as they all employ potential flow analysis and perform parametric studies to find an optimized design. The last one uses numerical optimization techniques.

Watmuff [6] designed an axisymmetric low-speed wind tunnel contraction using potential flow analysis. The wall shape was

* Corresponding author at: Iowa State University, Ames, IA 50011, USA. Tel.: +1 515 294 6549.
E-mail address: leifur@iastate.edu (L. Leifsson).

http://dx.doi.org/10.1016/j.jocs.2014.12.004
1877-7503/

Nomenclature

B	Bézier curve
f	High-fidelity model
H	Contraction inlet height or an objective function
N	Number of mesh points along an axis
n	Order of a Bézier curve
P	Control point of a Bézier curve
s	Surrogate model
V	Mean cross-sectional flow speed
v	Local flow speed
x	Design variable vector
x, y, z	Cartesian axes

Greek letters

α	Local flow angle
β	Penalty factor
δ	Trust-region radius
Δ	Variation
ρ	Gain ratio
∞	Free-stream

Subscripts

c	Coarse model
d	Diagonal
f	High-fidelity model
h	Horizontal
s	Surrogate model
max	Maximum
x, y, z	Cartesian axes

Superscripts

i	Design iteration

parameterized by polynomials with a single design parameter. The design criterion was to minimize the adverse pressure gradient near the inlet while not significantly increasing the adverse pressure gradient near the exit. Su [7] considers rectangular wind tunnel contractions with incompressible potential flow analysis. A comparative parametric study is conducted based on seven geometric parameters and five criteria (including pressure extrema, flow non-uniformity, and crossflow features). Mathew [3] designed a low-speed wind tunnel for aeroacoustic measurements. He performs a parametric study of the geometry parameters (contraction length, contraction ratio, outlet aspect ratio, and a single parameter governing the wall shape (described by polynomials)) to minimize the non-uniformities in the axial flow and the magnitudes of the other flow components, as well as checking for separated flow. A three-dimensional potential flow analysis and Stratford's criterion for separation for a two-dimensional boundary layer are employed in the parametric studies. The optimized design is validated by solving the steady, turbulent, three-dimensional, Reynolds-averaged Navier–Stokes equations using Menter's SST k–ω turbulence model.

Doolan and Morgans [8] employ a potential flow solver, but they couple it with a laminar viscous solver. They move away from performing parametric studies and integrate their computational model into an automated contraction design software tool which uses numerical optimization techniques to search for the optimized design. Three different optimization techniques are used to optimize a contraction shape. Sequential quadratic programming [9], a gradient-based method, solved the problem efficiently, but not robustly. DIRECT [10], a gradient-free method, provided robust global optimization at the expense of a large number of function evaluations. Efficient global optimization [11], a surrogate-based method, was robust and always gave acceptable results, but its efficiency depended on the initial random sampling required to construct the functional surrogate model (Kriging [12] was used in this case). Further work on contraction physics and design can be found in Refs. [30–33].

All of the aforementioned work use traditional fluid flow solvers (coupled potential and viscous fluid flow solvers), which are computationally cheap, as well as conventional optimization techniques. The likely reason for not using three-dimensional high-fidelity CFD models during design is that the simulations are computationally expensive and numerical optimization requires a large number of function calls, resulting in a prohibitively large design time.

The objective of this work is to design the shape of the contraction so that the flow in the experimental section is of high quality. It is necessary to use high-fidelity CFD simulations to capture accurately the nonlinear flow physics throughout the optimization process. The coupled potential and viscous flow solvers, as used in previous work, will not capture these effects with sufficient accuracy. As a matter of fact, high-fidelity simulations are essential. As demonstrated in Section 5, the contraction design obtained based on low-fidelity CFD analyses does not ensure the high-quality flow.

In this work, we describe the design of the wind tunnel contraction using high-fidelity CFD simulation and surrogate-based optimization (SBO). SBO replaces direct optimization of the high-fidelity (accurate but computationally expensive) model by iterative optimization of a properly corrected low-fidelity model. This reduces the number of evaluations of the high-fidelity model and, consequently, accelerates the optimization process. In particular, we adopt the variable–fidelity approach with the high-fidelity model based on the steady Reynolds-averaged Navier–Stokes (RANS) equations, and the low-fidelity model based on the same equations, but with coarse discretization and relaxed convergence criteria. We use a simple response correction technique to create

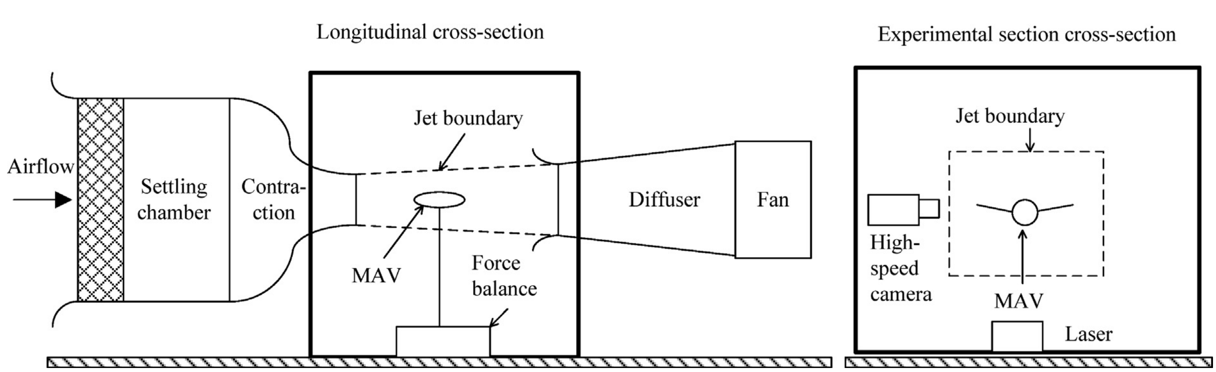

Fig. 1. A schematic of the wind-tunnel from the side, as well as a cross-sectional view of the test section.