Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 714–721

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6–8 September 2017, Lecce, Italy

A Discrete Adjoint Formulation for Inviscid Flow Nozzle Optimization

Giovanni Caramia[a,*], Andrea Dadone[a]

[a]*Dipartimento di Meccanica, Matematica e Management, Sezione Macchine ed Energetica, Politecnico di Bari, 70125 Bari, Italy*

Abstract

A discrete adjoint formulation with an ad hoc flow solver has been recently developed and tested on transonic inviscid flow optimization problems. In the present paper the formulation is extended to compressible as well as incompressible flow solvers. First, the adjoint equations are coupled with an accurate in-house flow solver to test the approach on some inverse design problems involving two- and three-dimensional transonic and subsonic flows. Then, the previous design test cases are re-computed coupling the extended adjoint formulation with commercial and open source flow solvers, without noticing any relevant difference in the optimization convergence histories. Finally, incompressible design test cases are successfully computed by means of commercial solver for incompressible flows. The extended compressible adjoint formulation appears to have a wide application, insofar as it allows to perform accurate and efficient design optimization using different flow conditions, different flow solvers and even a solver for incompressible flows.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

Keywords: Shape optimization; Adjoint equations; Nozzle; Inviscid flow.

---

1. Introduction

As discussed in Ref. [1], fluid dynamic design procedures generally require the calculation of many computationally expensive flow analyzes. Most approaches are based upon a sequence of global iterations, each one of them requiring a computation of the converged flow field, a solution for the sensitivity derivatives and an appropriate update of the design parameters. The high computational cost of this serial approach comes principally from the repeated solution of flow equations. In contrast, Ref.[1] has introduced a progressive optimization strategy, whereby the optimization process is based on partially converged flow solutions with the aim to converge the flow solution while converging the design problems. This strategy has been applied to the computation of some two-dimensional flow op-

---

[*] Corresponding author. Tel.: +39-080-596-3795 ; fax: +39-080-596-3411.
E-mail address: giovanni.caramia@poliba.it

1876-6102
Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.304

timization problems and has been compared to serial convergence strategy, showing a reduction of the computational effort by a factor of 50.

Ref. [2], examines issues related to developing robust sensitivity derivatives using an adjoint formulation based on an approximate flow solver, particularly effective for situations when the objective function is noisy or non-smooth. The efficiency of this smoothing procedure and of the progressive optimization strategy has been shown in Ref. [3], which considered the extension of the methodology to three-dimensional inviscid flow problems. The extension to viscous airfoil design problems has been presented in Ref. [4], where adjoint equations based on an inviscid flow formulation have been employed to compute inverse and direct test cases on laminar as well as turbulent transonic airfoils.

The smoothing procedure (Ref. [2]) and the use of partially converged solutions of flow and adjoint equations imply that the optimization process is based on approximate values of the gradient of the objective function. Another example of approximate design sensitivity has been discussed by Matsuzawa and Hafez (Ref. [5]), where adjoint equations based on an inviscid flow formulation have been used to determine approximate gradients of the objective function for airfoil inverse design in laminar flow conditions.

The progressive optimization strategy of Ref. [3] used only an ad hoc flow solver, which was coupled with the suggested adjoint equations to compute transonic flow design problems. The aim of the present paper is to develop a general use discrete adjoint method to be combined with different flow solvers for the inverse design optimization of two- and three-dimensional inviscid flow problems. Some preliminary results have been presented in Ref. [6]; here the optimization tests are extended to different flow solvers to enhance the wide applicability of the methodology.

First, we will present the formulation of Ref. [3], together with the suggested modifications, introducing the objective function, the adjoint formulation and the progressive optimization strategy. The adjoint formulation will be coupled with a flow solver developed in-house, a commercial flow solver and an open source flow solver. The suggested method will be applied to the inverse design of two- and three-dimensional nozzles (Ref. [7, 8, 9]) in transonic and subsonic flow conditions. Finally, the compressible adjoint formulation will be coupled with a solver for incompressible flow and applied to the design of the same two- and three-dimensional nozzles in incompressible flow conditions.

2. Formulation

The inverse design of nozzles consists of finding the geometric shape whose computed pressure distribution along the wall, $p_i(\boldsymbol{\xi})$, matches a target pressure distribution, $\hat{p}_i$ in discrete form. The corresponding discrete objective function, $I(\boldsymbol{\xi})$, may be defined as:

$$I(\boldsymbol{\xi}) = \frac{1}{2\,N_c} \sum_{i=1}^{N_c} [p_i(\boldsymbol{\xi}) - \hat{p}_i]^2, \tag{1}$$

where $\boldsymbol{\xi}$ represents the design parameters, while N_c indicates the number of linear (in 2D cases) or surface (in 3D cases) intervals used to discretize the wall surface.

All the flow computations are performed using three different basic flow solvers. The first one has been developed in-house; it is based on the finite-volume, flux-difference-splitting method of Roe (Ref. [10]), in order to accurately capture shock waves. The second flow solver is provided by the commercial software ANSYS FLUENT (version 14.5); it is based on a flux-vector splitting scheme, called Advection Upstream Splitting Method (AUSM), which was first introduced by Liou and Steffen in Ref. [11]. The third flow solver is provided by the release 2.2.2 of the open source software OpenFOAM (Open Field Operation and Manipulation), an open source library designed for development of multi-dimensional modeling codes. In particular, we use the pressure-velocity coupled solver for compressible transient transonic/subsonic flows, called SonicFoam (Ref. [12]). For the incompressible flow tests we use the SIMPLE solver for incompressible flows, provided by ANSYS FLUENT. SIMPLE (Ref. [13]) is the acronym of Semi-Implicit Method for Pressure-Linked Equations.

The computational flow field is divided into an $N \times M$ finite-volume mesh for two-dimensional flow applications while an $N \times M \times K$ finite-volume mesh is employed for three-dimensional flows, where N is the number of intervals