C. R. Mecanique 342 (2014) 363–375

Contents lists available at ScienceDirect

Comptes Rendus Mecanique

www.sciencedirect.com

Separation flow control

Optimization of jet parameters to control the flow on a ramp

Emmanuel Guilmineau [a,*], Régis Duvigneau [b], Jérémie Labroquère [b]

[a] LHEEA, CNRS UMR 6598, École centrale de Nantes, 1, rue de la Noë, BP 92101, 44321 Nantes cedex 3, France
[b] INRIA Sophia Antipolis–Méditerrannée, OPALE Project Team, 2004, route des Lucioles, 06902 Sophia-Antipolis, France

ARTICLE INFO

Article history:
Received 30 August 2013
Accepted 3 December 2013
Available online 11 June 2014

Keywords:
Flow control
Optimization
Ramp

ABSTRACT

This study deals with the use of optimization algorithms to determine efficient parameters of flow control devices. To improve the performance of systems characterized by detached flows and vortex shedding, the use of flow control devices such as oscillatory jets are intensively studied. However, the determination of efficient control parameters is still a bottleneck for industrial problems. Therefore, we propose to couple a global optimization algorithm with an unsteady flow simulation to derive efficient flow control rules. We consider as a test case a backward-facing step with a slope of 25°, including a synthetic jet actuator. The aim is to reduce the time-averaged recirculation length behind the step by optimizing the jet blowing/suction amplitude and frequency.

1. Introduction

Flow-control technology has been widely researched to improve the aerodynamic performance of transportation systems such as aircrafts and cars. The common objectives in many cases are to prevent massive flow separations. Actuator devices, such as synthetic jets or vortex generators, have proved their ability to modify the flow dynamics. However, the determination of efficient flow control parameters, in term of location frequency, amplitude, …, is tedious and highly problem dependent [1,2]. To overcome this issue, the numerical simulation of controlled flows is often considered to determine optimal control parameters, and the use of an automated optimization procedure is more and more observed [3,4]. Several studies have shown that the simulation of controlled flows is a difficult task, since result may be dependent on the turbulence close used [5,6] but also on numerical errors [6].

Therefore, this paper presents numerical results in the context of the optimization of flow control parameters for the flow on a ramp.

2. Flow solver

The ISIS–CFD flow solver, developed by "École centrale de Nantes" and CNRS (France), solves the incompressible unsteady Reynolds-averaged Navier–Stokes equations. This solver is based on the finite volume method to build a spatial discretization for the transport equations.

The incompressible unsteady Reynolds-averaged Navier–Stokes equations can be written (using the generalized form of Gauss' theorem) as:

* Corresponding author.
E-mail address: Emmanuel.Guilmineau@ec-nantes.fr (E. Guilmineau).

http://dx.doi.org/10.1016/j.crme.2013.12.009

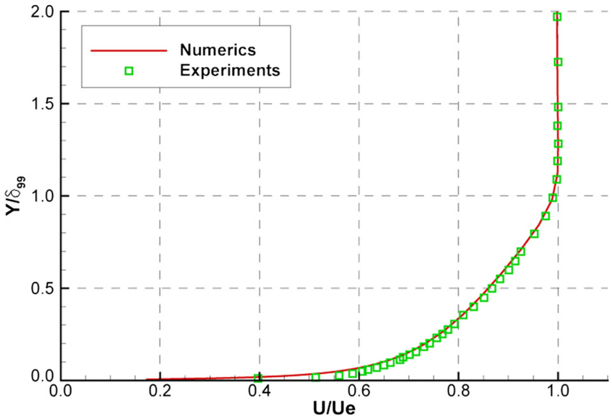

Fig. 1. Efficient global optimization loop.

$$\frac{\partial}{\partial t}\int_V \rho \mathrm{d}V + \int_S \rho(\vec{U} - \vec{U}_d)\cdot\vec{n}\,\mathrm{d}S = 0 \tag{1a}$$

$$\frac{\partial}{\partial t}\int_V \rho U_i\,\mathrm{d}V + \int_S \rho U_i(U_j - U_{dj})\cdot n_j\,\mathrm{d}S = \int_S (\tau_{ij}n_j - pn_i)\,\mathrm{d}S \tag{1b}$$

where V is the domain of interest, or control volume, bounded by a closed surface S moving at a velocity $\vec{U}_d$ with a unit outward normal vector $\vec{n}$ where n_j is the jth component. $\vec{U}$ and p are respectively the velocity and pressure fields. τ_{ij} are the components of the Reynolds stress tensor.

All flow variables are stored at the geometric center of arbitrarily shaped cells. Volume and surface integrals are evaluated with second-order accurate approximations. The face-based method is generalized to two-dimensional or three-dimensional unstructured meshes for which non-overlapping control volumes are bounded by an arbitrary number of constitutive faces, which means that cells can be polyhedral. A centered scheme is used for the diffusion terms, whereas for the convective fluxes, the Gamma Differencing Scheme (GDS) [7] is used for this study. Through a Normalized Variable Diagram (NVD) analysis [8], this scheme enforces local monotonicity and convection boundedness criterion. For more details, see Queutey and Visonneau [9].

The velocity field is obtained from the momentum conservation equations, and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation. The pressure equation is obtained in the spirit of Rhie and Chow [10]. Momentum and pressure equations are solved in a segregated manner as in the SIMPLE coupling procedure [11].

A second-order backward difference scheme is used for time discretization. All spatial terms appearing in Eqs. (1a) and (1b) are treated in a fully implicit manner. In this paper, the geometry is fixed. Therefore, the velocity $\vec{U}_d$ in Eqs. (1a) and (1b) is null.

In the case of turbulent flows, additional transport equations for modeled variables are discretized and solved using the same principles. Various turbulence closures are implemented. The method features sophisticated turbulence models: apart from the classical two-equation k–ε and k–ω models, the anisotropic two-equation Explicit Algebraic Reynolds Stress Model (EARSM), as well as Reynolds Stress Transport Models, are available, see [12] and [13]. Recently, a Detached Eddy Simulation (DES) approach has been introduced, see [14].

We consider actuations based on synthetic jets. The actuation is implemented as boundary conditions. The velocity is imposed on the jet boundary and is defined as:

$$\mathbf{U} = U_j A(X)\sin(\omega t)\mathbf{d}_\mathrm{j} \tag{2}$$

with $A(x)$ a unit profile function, U_j the amplitude, ω the angular frequency with $\omega = 2\pi f_j$ where f_j is the frequency of the synthetic jet, and $\mathbf{d}_\mathrm{j}$ the direction of the jet. In this study, $A(X)$ is a sine squared function and $\mathbf{d}_\mathrm{j}$ is perpendicular to the boundary.

3. Optimizer

A surrogate-based optimizer, included in the FAMOSA optimization toolbox developed at INRIA by the Opale Project Team, is used for this study. It is based on the construction of a kriging model to describe the variation of the objective function value, for example the recirculation length or the drag coefficient, with respect to control parameters. Kriging models, also called Gaussian process models, belong to response surface models, which allow one to predict a function value f at a given point x, on the basis of a set of known function values $\boldsymbol{F_N} = \{f_1, f_2, \cdots, f_N\}$ at some points $\boldsymbol{X_N} =$