Journal of Fluids and Structures 74 (2017) 158–177

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Drag reduction of slender blunt-based bodies using optimized rear cavities

M. Lorite-Díez, J.I. Jiménez-González *, C. Gutiérrez-Montes, C. Martínez-Bazán

Departamento de Ingeniería Mecánica y Minera, Universidad de Jaén, Campus de las Lagunillas, 23071, Jaén, Spain

ARTICLE INFO

Article history:
Received 19 November 2016
Received in revised form 4 May 2017
Accepted 13 July 2017

Keywords:
Drag reduction
Drag sensitivity
Wake instability
Shape optimization
Rear cavities

ABSTRACT

We have investigated the use of the adjoint sensitivity formulation to design efficient passive control strategies aiming at reducing the drag coefficient of a slender blunt-based body with a straight rear cavity. In particular, two control techniques consisting in wake modifications generated by placing small control cylinders in the near wake and by geometry variations of the cavity respectively, have been evaluated numerically. Thus, we have computed the turbulent flow sensitivity of the drag coefficient to localized forcing for a two-dimensional body with a straight cavity at $Re = \rho U_\infty H/\mu = 2000$, where U_∞ is the free-stream velocity, ρ and μ the fluid density and viscosity respectively and H the body height, showing that the highest values of sensitivity are obtained near the cavity edges. The effect of placing a pair of control cylinders around the most sensitive locations has been studied, obtaining a largest drag reduction of only 0.6%. Alternatively, a most efficient control strategy based on shape optimization has been thoroughly investigated. The drag shape sensitivity on the body surface (Othmer, 2014), computed using the former linear adjoint formulation, has been used in combination with a free-form deformation algorithm (Han et al., 2011), to guide the local structure deformations of the cavity, providing progressive drag reductions until the optimal, curved, shape is achieved. To deeply analyze the physical mechanisms behind the drag reduction provided by the optimal cavity, we have also performed more realistic three-dimensional numerical simulations using an IDDES model at two different Reynolds numbers, $Re = 2000$ and 20 000. The results corroborate the sensitivity analysis, obtaining a total drag reduction of 25.6% at $Re = 2000$ and 43.9% at $Re = 20\,000$, with respect to the original body without cavity, and 21.7% at $Re = 2000$ and 29.6% at $Re = 20\,000$ with respect to the body with a straight cavity. These reductions are mainly achieved by the inwards deflection of the flow upon detachment and a flow deceleration at the trailing edge due to an adverse pressure gradient introduced by the curved shape of the optimal cavity walls. Both combined effects reduce the size of the recirculation bubble formed behind the body, increasing the base pressure, and consequently, decreasing the drag. Furthermore, the addition of the optimized base cavity reduces the amplitude of the velocity fluctuations behind the body and stabilizes the wake, which becomes less chaotic and more two-dimensional.

* Corresponding author.
E-mail addresses: mldiez@ujaen.es (M. Lorite-Díez), jignacio@ujaen.es (J.I. Jiménez-González), cgmontes@ujaen.es (C. Gutiérrez-Montes), cmbazan@ujaen.es (C. Martínez-Bazán).

http://dx.doi.org/10.1016/j.jfluidstructs.2017.07.011
0889-9746/

1. Introduction

The flow around heavy vehicles, such as trucks or buses, is highly three-dimensional, chaotic and turbulent. This flow is mainly characterized by a massive flow separation, which induces the development of a wake behind the body and the periodic shedding of vortices, resulting into the appearance of major fluctuating aerodynamic forces in the structure. These forces have a great impact on the vehicle stability and, more importantly, on the fuel consumption, the latter being governed by the drag force. In this regard, it is estimated that 20% of the whole energy in a heavy vehicle is devoted to overcome the aerodynamic losses (Choi et al., 2014). Thus, these kinds of flows have been extensively studied in the last decades (Ahmed et al., 1984; Han et al., 1996; Pastoor et al., 2008) with the aim at better understanding the present features and the governing physical mechanisms as well as to develop new, more efficient flow-control methods or to improve the existing ones in terms of drag reduction and flow induced loads.

Many of the aforementioned investigations considered simplified vehicle models (see e.g. Ahmed et al., 1984; Pastoor et al., 2008), to emulate the flow conditions around real heavy vehicles. As a result, a large amount of approaches and devices have been proposed, which can be grouped into active and passive flow control techniques, the latter being easier to implement. In general, these strategies aim at modifying the near wake, and can be implemented either by means of open (Parkin et al., 2014) or closed (Henning et al., 2007; Pastoor et al., 2008) flow control loops. For instance, regarding active flow control systems, sophisticated and complex devices have been studied, as those based on the use of actuators (Beaudoin et al., 2006), suction/blowing strategies (Sevilla and Martínez-Bazán, 2004; Sanmiguel-Rojas et al., 2009; Bohorquez et al., 2011; Pasquetti and Peres, 2015), or spinning motions (Jiménez-González et al., 2013, 2014). Their implementation in real problems usually requires an important power input, what limits somehow its applicability and efficiency, rendering passive control systems more attractive because of their simplicity. Among other passive strategies, it is worth mentioning the works from Mair (1965) who studied the effect of including plates on the base of the body, Park et al. (2006) who introduced trailing edge modifications in the form of small tabs, those from Sanmiguel-Rojas et al. (2011), Cai and Chng (2009) and Kruiswyk and Dutton (1990) on the use of open base cavities or similarly (Han et al., 1992; Mair, 1978; Choi et al., 2014), on boat-tailed after-bodies. Besides, Martín-Alcántara et al. (2014) proposed a multi-cavity device that produces a maximum drag reduction of 25% at $Re = 20\,000$, obtaining additionally a less chaotic wake topology. Notwithstanding the aforementioned positive features, the performance of any type of rear cavity as a control mechanism is related to their depth, obtaining larger drag reductions as the cavity deepens (Sanmiguel-Rojas et al., 2011). In that sense, functionality requirements and practical geometrical limitations in vehicles usually restrict the depth, subsequently hindering its capability as control mechanism. Consequently, in order to overcome such limitations, it seems interesting to investigate any potential performance improvement of the addition of a base cavity with a fixed depth, for instance, by means of shape modification.

Most of the above investigations base the design of control strategies on the general knowledge of the physical mechanisms governing the flow around bluff bodies. Alternatively, the use of sensitivity analysis applied to flow control may help to precisely identify the regions which contribute the most to wake instability (Marquet et al., 2008; Meliga et al., 2012) or drag and lift generation (Meliga et al., 2014). This identification allows a more specific design and an optimal placement of control elements aiming at suppressing instabilities or reducing the drag, without having to carry out parametric studies or testing different locations empirically. In fact, these sensitivity analyses have been already satisfactorily proven, obtaining accurate predictions when compared with results from experimental work or numerical simulations (see e.g. Meliga et al., 2012; Parezanović and Cadot, 2012). Regarding aerodynamic forces, Meliga et al. (2014) have lately derived a linearized adjoint formulation to analyze the sensitivity of drag to local force perturbations on the flow around a square cylinder, for laminar and turbulent regimes. By simply introducing small control cylinders in those flow regions featuring the highest sensitivity, reductions up to 20% in the main drag were obtained. Consequently, this adjoint approach stands out as a powerful tool to maximize the drag reduction and to design optimized control devices. In that sense, adjoint-based optimization has long been recognized to be efficient for studies with large numbers of design variables, especially in the aerospace sector (Jameson, 1988; Burgreen and Baysal, 1996). In fact, Othmer (2014) has recently considered the use of adjoint shape optimization techniques applied to car aerodynamics, focusing on the reduction of drag force and using the open source computational fluid dynamics (CFD) toolbox OpenFOAM®. Interestingly, the combined use of sensitivity analysis and body topological modifications provides with efficient solutions for control devices or reduction of the aerodynamic forces acting on the body, which are achieved by means of iterative processes that characterize the sequential interactions between flow and structure. Since, in most applications, the geometry modifications are constrained by functionality and other requirements, it might be interesting to consider, as a solution for flow control and forces reduction, the implementation of control devices whose design stems from adjoint-based sensitivity and optimization analysis, what would ensure a high performance.

Following this idea, the present work explores the use of adjoint formulation to assess the sensitivity of drag to flow and structure topological modifications, in the wake behind a blunt-based body with a base cavity, which is a widespread configuration in real wake control applications (see e.g. Choi et al., 2014), in an attempt to evaluate the potential for improvement on the efficiency of rear cavities of fixed depth as passive control strategy. Thus, the study aims at achieving additional drag reduction on the wake behind a two-dimensional bluff body of semi-ellipsoidal nose and straight rear cavity, by analyzing the effect of small control cylinders in the wake (Meliga et al., 2014) and optimal geometry modifications of the cavity structure (Othmer, 2008). The predictions of drag reduction obtained by means of the adjoint formulation analysis and topological modifications, will be compared with results from numerical simulations of the flow around the obtained