Journal of Fluids and Structures 48 (2014) 347–361

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Drag reduction induced by the addition of a multi-cavity at the base of a bluff body

A. Martín-Alcántara [a], E. Sanmiguel-Rojas [b,*], C. Gutiérrez-Montes [a], C. Martínez-Bazán [a]

[a] Área de Mecánica de Fluidos, Departamento de Ingeniería Mecánica y Minera, Universidad de Jaén, Campus de las Lagunillas, 23071 Jaén, Spain

[b] Área de Ingeniería Mecánica, Departamento de Mecánica, Universidad de Córdoba, Campus de Rabanales, 14071 Córdoba, Spain

ARTICLE INFO

Article history:
Received 8 September 2013
Accepted 30 March 2014
Available online 21 April 2014

Keywords:
Multi-cavity
Bluff-body
Drag-reduction
Turbulent-flow
Numerical-simulation

ABSTRACT

We present a three-dimensional numerical study on the drag reduction of a D-shaped body, of chord length L, height H and spanwise width W, with $H/W \ll 1$, aligned with a turbulent incompressible free-stream of velocity U_∞, density ρ and viscosity μ. In particular, an extensive parametric study is performed numerically using the IDDES turbulent model, at a Reynolds number, $\mathrm{Re} = \rho U_\infty H/\mu = 20\,000$, to analyze the effect on the drag coefficient, C_D, of adding a single or a multi-cavity of variable depth, h, at the base of the body. It has been observed that, in the range $0 \le h/H \le 0.3$, the average drag decreases monotonically reaching an asymptotic value in both cases. However, shorter cavity depths are necessary to reach the same reduction in the case of the body with a multi-cavity. On the other hand, the temporal evolution of the drag shows a considerable reduction of the flow randomness with the addition of a multi-cavity, which translates into a much less chaotic and quasi-two dimensional wake.

1. Introduction

The flow around bluff bodies, which is present in multiple natural phenomena and technological processes, is unsteady and highly turbulent in most of the situations of scientific relevance. The wake that forms behind these types of bodies is a source of instabilities and responsible for the appearance of fluctuating forces on the body, which, in the case of vehicles, rules their drag coefficient, among other effects. At this point, car aerodynamics constitutes one of the most important issues, as it is responsible for the flow around them and has a direct impact on the fuel consumption or vehicle stability (e.g. Hucho, 1998). Therefore, an accurate understanding of these kinds of flows is necessary for the design of safer, more efficient and less contaminant vehicles.

In this regard, due to the current economical situation and the market competitiveness, there is a great concern and increasing need to reduce the operating costs and the fuel consumption, in particular, in the goods road transportation industry. This is of special significance in the case of trucks or truck-shaped vehicles, such as buses or those with tractor-trailer rigs, as the need to provide a large cargo volume within the maximum allowable dimensions regulated by laws rules the manufacture of vehicles sacrificing aerodynamics for cargo volume. Consequently, a large pressure difference between

* Corresponding author. Tel.: +34 957 218323.
E-mail address: enrique.sanmiguel@uco.es (E. Sanmiguel-Rojas).

http://dx.doi.org/10.1016/j.jfluidstructs.2014.03.013

the front and rear parts is induced, which is responsible for the appearance of large drag forces, enhancing the fuel consumption in these kinds of vehicles. Ahmed et al. (1984) showed experimentally in a car model that most of the drag is due to pressure drag, being the wake behind the vehicle the flow region which presents the major contribution (see Morel, 1978). Therefore, the implementation of any strategy to modify the wake, improving the flow conditions, will have a direct impact on the drag reduction and, consequently, on fuel consumption (see Fieldler and Fernholz, 1990).

There are three main flow control methodologies (Fieldler and Fernholz, 1990): passive, active open-loop and active closed-loop control. However, the use of passive control systems is more attractive for reasons of cost, as they are energy free and, often, easier to implement. Different mechanisms and devices have been studied since the second part of the last century. Mair (1965) conducted one of the first investigations on the effects of rear mounted plates on the drag of bluff bodies, although only the axisymmetric case with circular plates was considered, while Bearman (1965) described the stabilizing effect of a splitter plate on the wake flow. Subsequently, Tanner (1972) examined the effect of modifying the shape of trailing edge of the body on the drag reduction. The inclusion of wavy trailing edges or vortex disruptors was also proposed for drag reduction by Tombazis and Bearman (1997) and Park et al. (2006). Bruneau and Mortazavi (2008) studied the drag reduction by introducing porous slices on different parts of a vehicle. They found a regularization of the flow and a reduction of the drag coefficient that ranged from 20% to 37%. The replacement of the sharp edge at the roof-rear slant connection of a Ahmed body (see Ahmed et al., 1984) by a rounded connection has also been analyzed by Thacker et al. (2012), who found a drag decrease of 10%. Finally, the inclusion of a cavity at the rear part of the body has been proven to reduce the drag coefficient in bluff bodies (Do et al., 2010; Sanmiguel-Rojas et al., 2011). Specifically, varying different geometrical parameters, Sanmiguel-Rojas et al. (2011) analyzed the effect of the addition of a cylindrical base cavity on the laminar flow around an axisymmetric bluff body at low Reynolds numbers, observing an improvement on the stability of the wake and a drag reduction of 1%. This small reduction was the consequence of an increase in the viscous drag coefficient in their range of Reynolds numbers. However, the pressure coefficient was reduced between 10% and 15%, expecting the reduction to be larger at higher Reynolds numbers (Morel, 1979), for which the viscous contribution is nearly negligible. Additionally, Do et al. (2010) assessed the effect of different base cavity shapes and sizes on the flow around a NACA0015.

The present study is focused on the two-dimensional bluff body proposed by Pastoor et al. (2008). This body, similar to the one used by Park et al. (2006), presents some of the main flow phenomena affecting a real load-truck, with a typical drag coefficient around 0.9. In their study, Pastoor et al. (2008) examined, both experimentally and theoretically, the flow around this D-shaped bluff body and applied different drag reduction techniques by means of an active closed-loop strategy, finding a 15% in drag reduction.

In the current work, we study numerically the effect of placing a multi-cavity at the body base, as passive flow control strategy, on the flow structure and drag reduction in the aforementioned body. Section 2 is devoted to the definition of the numerical method. In Section 3, the main features of the flow without and with the implementation of different cavity configurations, as well as the drag reduction achieved, are described in detail. Finally, Section 4 is devoted to conclusions.

2. Numerical methods

The three-dimensional turbulent flow of an incompressible free-stream of velocity U_∞, density ρ and viscosity μ around the D-shaped body was simulated using the IDDES (Improved Delayed Detached Simulations) model. This numerical approach is a hybrid RANS-LES model developed to simulate massively separated flows at high Reynolds numbers, which works permitting the activation of a RANS model in the attached wall boundary layer and a LES model in the separated flow regions, giving an intricate but well-balanced and powerful numerical approach to simulate complex turbulent flows, such as separated flows over bluff bodies (see Shur et al., 2008). Thus, the IDDES model allows us to resolve the large turbulent scales in the wake region while, near the wall, a RANS model is used having, therefore, the potential of saving attractive amounts of computing resources compared with a pure LES model.

The flow is described in a Cartesian coordinate system, (x, y, z), with the origin placed at the center of the body base, see Fig. 1(a). The characteristic scales of length, velocity, time and pressure are given by H, U_∞, H/U_∞ and ρU_∞^2, respectively, so that the Reynolds number is defined as $\mathrm{Re} = \rho U_\infty H/\mu$. The computational domain, Fig. 1(a), reproduces exactly the same experimental setup used by Pastoor et al. (2008), i.e. a D-shaped bluff body of length $L=262$ mm, height $H=72$ mm and width $W=550$ mm, with a rounded head of radius $R=25$ mm, inside a rectangular cross-section wind tunnel (WT for short) of overall dimensions: $L_{WT}=2500$ mm, $H_{WT}=555$ mm and $W_{WT}=550$ mm. The body was placed at the half-height of the domain and at a length of $10.25H$ downstream from the inlet. The computational domain is enclosed by four boundaries: the inlet Σ_i, the outlet Σ_o, four wind tunnel walls Σ_w and the body surface Σ_b, see Fig. 1(a). Furthermore, Fig. 1(b) shows the front view of the D-shaped body and details of the structured mesh used in the streamwise direction. Finally, Fig. 1(c)–(e) shows the three afterbody configurations under study, i.e. the original D-shaped bluff body, a body with a single-cavity added at the base and a body with a multi-cavity placed at its base, respectively.

The simulations were performed imposing the following boundary conditions: at the inlet Σ_i, a uniform fluid stream of velocity $(U_\infty, 0, 0)$ was fixed, with a turbulent intensity of 0.5% generated by the vortex method (Mathey et al., 2006), which reproduces the experimental inlet conditions reported by Pastoor et al. (2008); at the outlet Σ_o, outflow boundary conditions were applied to all field variables; finally no-slip boundary conditions were imposed at the walls Σ_w and Σ_b. Regarding the wall treatment, we made sure that the average y^+ was of order one, since a Spalding's law (Spalding, 1961) based on unified wall functions was applied. The major advantage of using such unified wall functions is that, the first grid