International Journal of Heat and Fluid Flow 32 (2011) 192–200

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Numerical simulation of the flow around a simplified vehicle model with active flow control

Siniša Krajnović *, João Fernandes

Division of Fluid Dynamics, Department of Applied Mechanics, Chalmers University of Technology, SE-412 96 Göteborg, Sweden

ARTICLE INFO

Article history:
Received 12 November 2009
Received in revised form 30 March 2010
Accepted 22 June 2010
Available online 15 July 2010

Keywords:
Active flow control
LES
Ahmed body
Drag reduction
Simplified vehicle

ABSTRACT

Large-eddy simulation (LES) was used to study the influence and the resulting flow mechanisms of active flow control applied to a two-dimensional vehicle geometry. The LES results were validated against existing Particle Image Velocimetry (PIV) and force measurement data. This was followed by an exploration of the influence of flow actuation on the near-wake flow and resulting aerodynamic forces. Not only was good agreement found with the previous experimental study, but new knowledge was gained in the form of a complex interaction of the actuation with the coherent flow structures. The resulting time-averaged flow shows a strong influence of the extension of the actuation slots and the lateral solid walls on the near-wake flow structures and thereby on the resulting drag.

1. Introduction

Energy-efficient ground vehicles require careful geometrical styling to decrease drag, primarily due to the formation of the wake region. However, commercial vehicles such as trucks and buses do not permit any great change of the rear part of the vehicle. Instead of changing their geometry, attempts can be made to influence the flow using flow control, which can be either active or passive. Both strategies suffer from the inability to adjust to the flow conditions if the flow control is not a function of the actual flow and thus cannot adapt to the flow conditions.

The development of an efficient strategy for closed-loop active flow control requires a better understanding of open-loop active flow control and the flow mechanisms that are likely to produce an increase in the base pressure of ground vehicles. The goal of the present paper is twofold: to investigate the applicability of the LES technique for the purpose of flow control and to improve our understanding of the flow mechanisms acting in an active flow control process.

The present study employed a two-dimensional bluff body with a lateral shape similar to a so called Ahmed body (Ahmed et al., 1984) used in the experimental study by Pastoor et al. (2008). The interaction of the upper and lower shear layers after the trailing edges of the 2D Ahmed body (Fig. 1) results in von Karman-like instabilities. Such instabilities rapidly produce two large 2D vortices in alternating order. As the vortices are formed very early, the near-wake separation bubble (the dead water) is short, producing a low base pressure and large drag. An increase of the base pressure can be achieved by an elongation of the near-wake region and suppression or delay of the shear layer interaction. To achieve this objective, the present work applies the strategy used in Pastoor et al. (2008) to force symmetric vortex shedding and thereby delay the wake instabilities.

2. Description of the model and numerical set-up

Both the natural and the controlled flows were studied at two Reynolds numbers, $Re = 2 \times 10^3$ and $Re = 2 \times 10^4$. The lower Reynolds number simulations were used for parametric studies of the location of the actuation slots, the actuation amplitude and actuation frequencies, as the low Re allows a large number of wall-resolved LES simulations.

The only difference between the set-up of the low and high Reynolds numbers is that the support rods were excluded in the case of low Reynolds number to make the simulations even less computer intensive. The model has the side shape of an Ahmed body (Krajnović and Davidson, 2005a,b) with no slanted surface at the rear end. It extends from the one to the other lateral wall, forcing the flow above and below it. The geometry of the body is shown in Fig. 1. It has a cross section from the side of a simplified bus with chord length $L = 262$ [mm], height $H = 72$ [mm] and spanwise width $W = 550$ [mm]. The front of the body has a radius of 25

* Corresponding author.
E-mail address: sinisa@chalmers.se (S. Krajnović).
URL: http://www.tfd.chalmers.se/~sinisa (S. Krajnović).

0142-727X/$ - see front matter © 2010 Elsevier Inc. All rights reserved.
doi:10.1016/j.ijheatfluidflow.2010.06.007

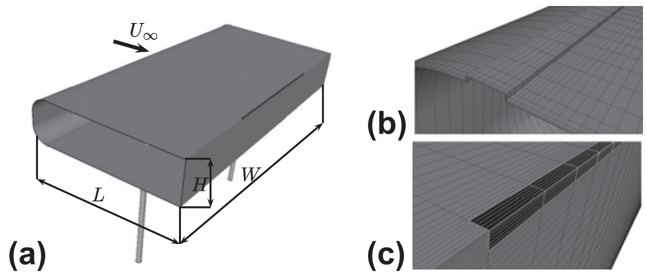

Fig. 1. (a) Geometry of the 2D Ahmed body. (b) Zoom of the trip tape. (c) Zoom of the actuation slots.

[mm]. Two trip tapes of a height of 0.8 [mm] and length of 5 [mm] were placed on the upper and the lower face of the body 30 [mm] downstream of the front.

The position of the body in a computational domain is shown in Fig. 2. The model was centered in the computational domain with a height of 7.7*H*. The distance from the model to the inlet and the outlet was 10.25*H* and 20.83*H*, respectively.

It is worth mentioning here that there is a dependence of the flow separation on the Reynolds number and curvature of the surface at the front of the body (Cooper, 1985). In the flow studied here at $Re \leqslant 2 \times 10^4$ and a radius of the front end of the body of $R = 25$ [mm], the flow is expected to separate (see Cooper, 1985). This means that no tripping of the transition is needed. However, in the experimental study, trip tapes are placed on both the upper and the lower sides of the body just after the curved part. Although they are rather small ($d_1 = 8$ mm and $d_2 = 5$ mm), they were included in the LES in order to remove sources of discrepancies between the experiments and the simulations. Two actuation slots were placed at the rear edges of the body (Fig. 3). The slots in the experimental study were inclined at a 45° angle. However, all the LES used two pairs of slots around the upper and the lower edges, as shown in Fig. 3. The reason for this discrepancy is that a better hexahedral computational grid was obtained with this slot

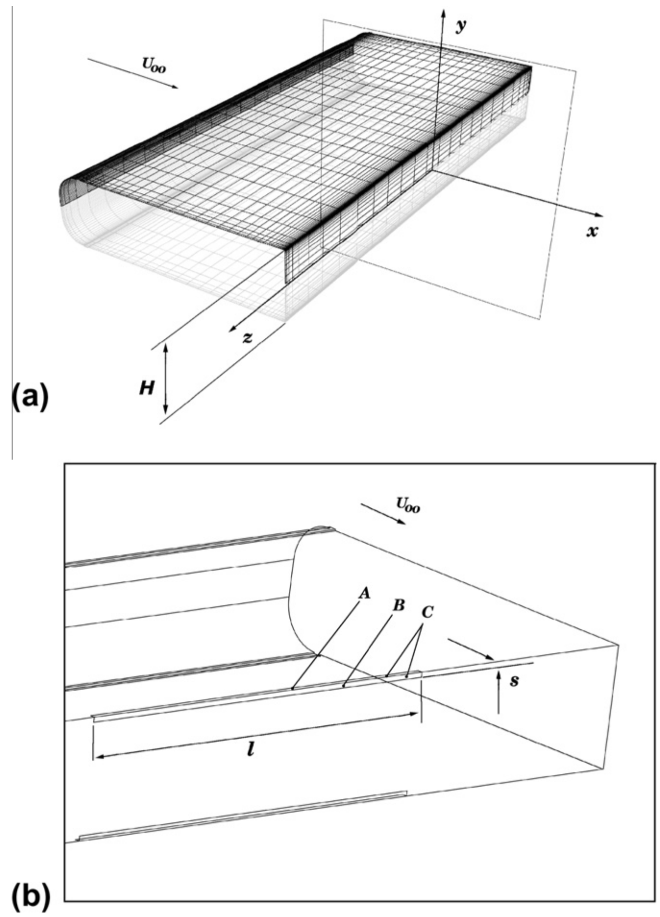

Fig. 3. (a) Geometry of the body with the coordinate system and reference axis. (b) Three slot configurations.

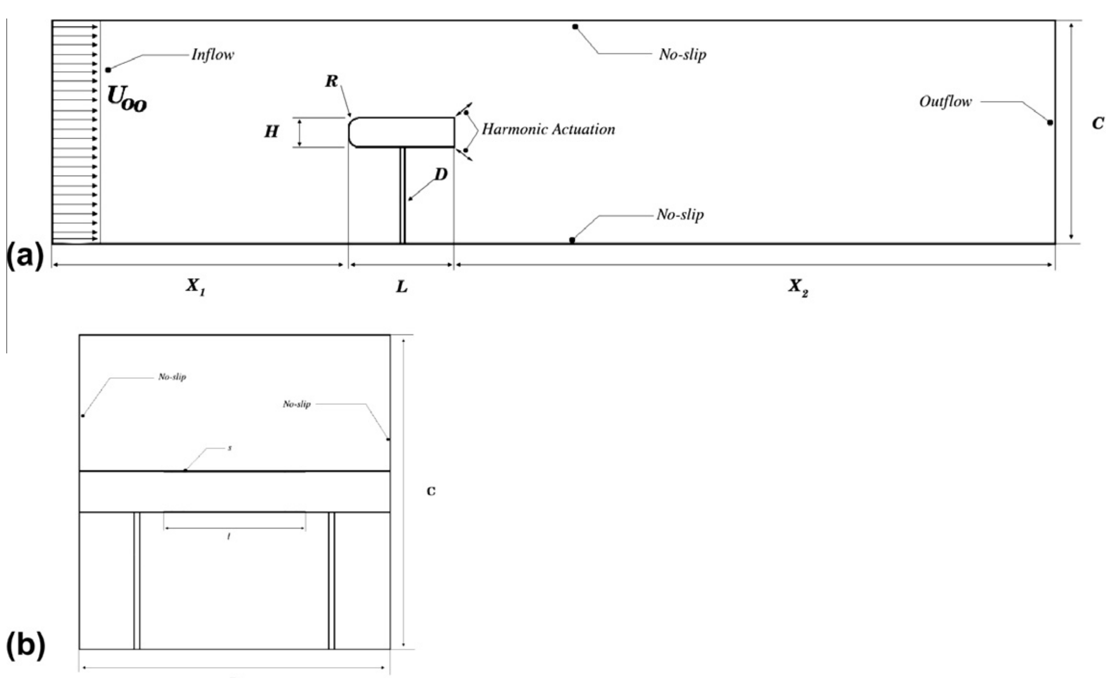

Fig. 2. Geometry of the body placed in the wind tunnel in the case of $Re = 2 \times 10^4$.