Analysis and control of square back Amed body wake : drag reduction

Azeddine KOURTA[1]

1. Univ. Orleans, INSA-CVL, PRISME
8 rue Léonard de Vinci 45072 Orléans, France

Abstract :

This paper presents experimental and numerical simulation results of the near wake of a square back Ahmed body in view of investigating different control strategies in order to reduce the aerodynamic drag. Main objectives are to increase the knowledge of the complex unsteady wake emitted at the model back in order to discuss optimal control parameters to be selected for synthetic jet actuators modeled in the numerical simulation. This work is focused on the recirculation zone that develops in the near wake of the model. Experimental flow characterisation was done. Numerical simulation has been also done to complete the characterisation and to test the effect of a synthetic jet actuator on the development of the wake.

Key words : *turbulent shear flow, Aerodynamics, bluff body,computation, experiment, flow control*

1 Introduction

Despite advances in engine technology and new fuel generation, transport industry and more especially road transport remains the main contributor of energy consumption and pollutant emissions. To improve both of theme one way is to use active flow control to reduce drag. Vehicle wakes are characterized by both large pressure drag and coherent structures [1, 2, 3, 4]. In addition, their three-dimensional features which contribute for large part of performance losses make the physical mechanisms at play even more complex. In this study, we consider the square back Ahmed body [5, 6].

This bluff body is characterized by an abrupt geometry change at the rear of the model that generates the vortices. These vortices interact with each other and lead to the formation of a large recirculation zone. This region is bounded by the shear layers emanating from the edges of the base of the model. Moreover, the boundary layer developing on the ground modifies the lower shear layer. This interaction alters the vortex emission in the wake and thus the pressure distribution on the base of the model responsible for an important part of the pressure drag. The flow of the near wake has a complex dynamic as a bistability [7]. Time dependent data show that two asymmetric topologies coexist in the wake of the model but time averaging leads to a symmetric flow [8, **?**]. This phenomenon is similar to the symmetry breaking found in the wake of backward facing step [10].

The present work explores the wake of the 0.7 scaled square back Ahmed body . The topology of the near wake flow has been characterized first by experiment. Numerical simulations are also done to deeply characterize the Ahmed body wake and to test the effect of a synthetic jet on the dynamic of this wake.

2 Experiment and computation

The model used for this study is a 0.7 scaled square back Ahmed body [11] (l=730.8mm, h=202mm, w=272mm). Tests were run in the subsonic wind tunnel of the PRISME laboratory at the university of Orléans [12]. The square test section is 2 m by 2 m and 5 m long. The model is fixed above a 3 m by 2 m aluminium plate with an ellipsoidal leading edge. The rear part of the plate is equipped with a 3° incline flap in order to suppress the pressure gradient effect in the test section.

The pressure evolution at the back of the body was characterized by using 80 static pressure taps on the rear part of the Ahmed body. The measurement recording lasted 160 s with a 25 Hz sampling frequency. Only a half of the back was instrumented as the pressure distribution on the back of the body is symmetric [13]. The symmetry distribution has been checked with thirteen flush mounted pressure taps at mid-height of the model base.

Spatial characterisation of the wake of the Ahmed body was carried out using a two-component Particule Image Velocimetry (PIV) system. The velocity fluctuations were recorded using one component hot wire probes.

The numerical solver used in this study is elsA software developped by ONERA and CERFACS. It is able to compute internal and external flows. The Navier Stokes equations are solved by using structured mesh and finite volum method. For the case studied here, URANS strategy with turbulent viscoity concept is used. Turbulent models, palart-Allmaras, $k-\epsilon$, $k-\omega$ and $k-\omega$ SST, have been tested. Finaly, this last one has been selected to perform the calculation. The computational domain is parallelepipedic and its dimension is 5.731 m $\times$ 2 m $\times$ 1.05 m.

The square back Ahmed body is placed at 2 m from the inlet section, 3 m from the outlet section, 0.90 m from the lateral surfaces, and wall clearance at 50 mm . The mesh contains 5393952 cells and is divided on 31 blocks (Fig. 1a). The thickness of the first cell si 0.04 mm corresponding to y^+ of 40.H

Concerning the boundary condions, we have walls on the model and at ground (bottom domain boundary) where the slip conditions and wall functions for turbulence are used. For free boundaries, non reflected conditions are used. For more details concerning the numerical method see Lahaye [12].

3 Caracterisation of non-perturbed flow

The main flow is characterized by classical torus structures as has been observed experimentally by Grandemange et al. [15], Ahmed et al. [11], and numerically by Krajnovic et al. [14] and by Rouméas et al. [5]. This zone is the localisation of losses of total pressure. This can be observed in the mediane planes.Two structures are then observed in the longitudinal mediane plane with the same size in agreement with Grandemange et al. [15], Verzicco et al. [16] or Khalighi et al. [13]. Despite the observed agreement between numerical and experimental position of higher vortex, the lower vortex is far from the model base in calculation compared to the experiment (Fig 1b). Another observed difference is the position of stagnation point. In the experiment it is equal to $(x/H, z/H) = (1.45, -0.27)$ and for computation it is at $(x/H, z/H) = (1.61, -0.81)$. Wassen et al. [17] and Rouméas et al. [5] have comparable positions as our numerical simulation. owever, Grandemange et al. [15] obtain $(x/H, z/H) = (1.46, -0.57)$.

4 Synthetic jet effects

This part will focus on the caracterisation of synthetic jet parameters effects (momentum coefficient C_μ, frequency of actuation St_{Act} and jet orientation θ). The synthetic jet is introduced as boundary condition with siunusoidal velocity :

$$\overrightarrow{U}(x,y,z,t) = U_{max} \cdot sin(\omega \cdot t) \begin{pmatrix} cos(\theta) \\ 0 \\ sin(\Theta) \end{pmatrix} \quad (1)$$

This jet velocity is imposed on the slot located on the base along the horizontal edges. The size of this slot is 272 mm length and 2 mm width. The control parameters used are $U_{max} = \{0.5U_\infty, 0.63U_\infty, 0.75U_\infty, 0.88U_\infty, U_\infty\}$ corresponding to : $C_\mu = \{0.007, 0.011, 0.014, 0.019\}$ where :

$$C_\mu = \frac{2 \cdot h \cdot U_{max}^2}{H \cdot U_\infty^2} \quad (2)$$

The non-dimensional frequecy is obtained with the height of the base (H) and (U_∞). Four frequencies have been used $St_{Act} = \{0.10, 0.17, 0.40, 0.80\}$ corresponding to 0.5, 0.85, 2 and 4 times the natural frequency of the flow. Three jet veocity angles have been also used $0°$, $+45°$ and $-45°$. The momentum coefficient effect is plottted on (Fig 2, left side). In the range studied, the distance between the torus vortex and the base decreases. So the drag increases. The frequency effect is plotted on (Fig 2, right side). The distance of the torus vortex to the base decreases first and increase in the last case. So a drag increases and reach a maximum and decreases of the last case. For the jet velocity angle effect, the case with zero angle leads to best results in the range of velocities and frequencies studied.

Conclusions :

Exeprimental and numerical studies have been done to characterize and to modify the near wake of square back Ahmed body. The comparaison of exeperimental and numerical results have been done on the base flow. Numerical parameter studies have been done to test low synthetic jet velocity and frequency with different angles. In the ranges studied no improvement in term of drag reduction has been obtained.

Références

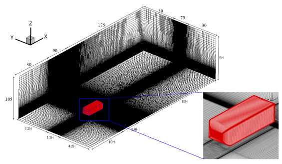
[1] A. Spohn, P. Gilliéron, *Flow separation generated by a simplified geometry of an automotive vehicle.* IUTAM Symposium : unsteady separated flows, (2002).

[2] E. Guilmineau, *Computational study of flow around a simplified car body.* J Wind Eng. Aerodyn., 96, 2008.

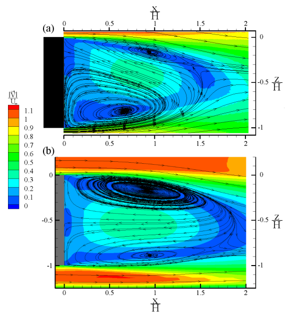
[3] E. Bideaux, P. Bobillier, E. Fournier, P. Gilliéron, M. Hajem, J.-Y. Champagne, P. Gilotte, A. Kourta, *Drag reduction by pulsed jets on strongly unstructured wake : towards the square-back control.* Int. J. Aerodynamics, 1, 2011.

[4] J. MacNally, F. Alvi, N. Mazellier, A. Kourta, *Active flow control on an Ahmed body – An experimental study.* In 53th Aerospace Sceinces Meeting and Exhibit, Kissimmee, Forida, 2015.

[5] M. Rouméas, P. Gilliéron, A. Kourta, *Analysis and control of the near-wake flow over a square-back geometry*, Computers & Fluids 38 (2009) 60–70.

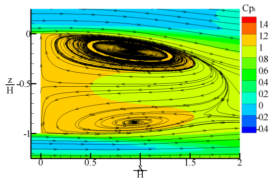


(a) Computational domain and Mesh

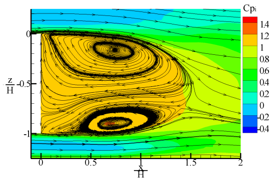


(b) Comparison of the main flow in the longitudianl mediane plane between experiment (a) and computation (b)

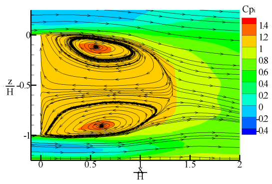
FIGURE 1 – mesh and comparison experiment-computation



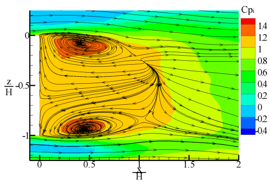
(a) Base flow



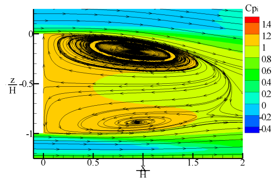
(b) $C_\mu = .005\ St_{Act} = .17$



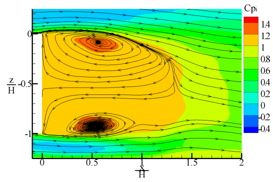
(c) $C_\mu = .011\ St_{Act} = .17$



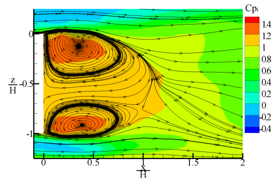
(d) $C_\mu = .019\ St_{Act} = .17$



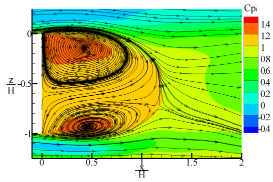
(e) Base flow



(f) $C_\mu = .019\ St_{Act} = .1$



(g) $C_\mu = .019$ a$St_{Act} = .4$



(h) $C_\mu = .019\ St_{Act} = .8$

FIGURE 2 – Jet Amplitude and Frequency effects : Cp_i and streamlines

[6] S. Krajnovic, L. Davidson, *Flow Around a Simplified Car, Part 2 : Understanding the Flow*, Journal of Fluids Engineering 127-5 (2005) 919.

[7] M. Grandemange, M. Gohlke, O. Cadot, *Reflectional symmetry breaking of the separated flow over three-dimensional bluff bodies*, Physical Review E 86 (2012), 035302.

[8] M. Grandemange, M. Gohlke, V. Parezanovic, O. Cadot, *On experimental sensitivity analysis of the turbulent wake from an axisymmetric blunt trailing edge*, Physics of Fluids 24 (2012), 035106.

[9] R. Volpe, P. Devinant, A. Kourta, *Experimental characterization of the unsteady natural wake of the full-scale square back Ahmed body : flow bi-stability and spectral analysis*, Exp Fluids 56 (2015), 99.

[10] B. Herry, L. Keirsbulck, L. Labraga, J.B.Paquet, *Flow Bistability Downstream of Three-Dimensional Double Backward Facing Steps at Zero-Degree Sideslip*, Journal of Fluids Engineering 133 (2011), 054501.

[11] S.R. Ahmed, R. Ramm, G. Falting, *Some Salient Features Of The Time-Averaged Ground Vehicle Wake* SAE technical paper series 840300 (1984).

[12] A. Lahaye, *Caractérisation de l'écoulement autour d'un corps de Ahmed à culot droit*, PhD (Thèse de Doctorat), Orléans University ,France, 2014.

[13] B. Khalighi, S. Zhang, C. Koromilas, *Experimental and computational study of unsteady wake flow behind a bluff body with a drag reduction device*, SAE paper (2001), 1-15.

[14] S. Krajnovic, L. Davidson, *Numerical Study of the Flow Around a Bus-Shaped Body*, ournal of Fluids Engineering 125 (2003), 500.

[15] M. Grandemange, A. Mary, M. Gohlke, O. Cadot, *Effect on drag of the flow orientation at the base separation of a simplified blunt road vehicle*, Experiments in Fluids 54 (2013), 1-10.

[16] R. Verzicco, M. Fatica, G.Iaccarino, P. Moin, B. Khaligh, *Large eddy simulation of a road vehicle with drag-reduction devices*, AIAA Journal 40 (2002), 2447-2455.

[17] E. Wassen, S. Eichinger, F. Thiele, *Simulation of active drag reduction for a square-back vehicle*, Active Flow Control II (2010), 241-255