European Drag Reduction and Flow Control Meeting – EDRFCM 2015
March 23–26, 2015, Cambridge, UK

DRAG REDUCTION ON A SIMPLIFIED 3D BLUFF BODY

C. Sardu, S. Sedda, G. Iuso
Department of Mechanical and Aerospace Engineering
Politecnico di Torino, Corso Duca degli Abruzzi 24, Torino, IT

INTRODUCTION

A complete flow control over road vehicles is difficult to achieve because of different phenomena occurring in flow evolution, i.e. the wheel rotation and ground effect deeply influence the flow features. In spite of this, the problem can be approached following different strategies considering both passive and active flow control techniques. The flow around bluff body road vehicle exhibits a massive separation region and as a consequence the drag is mainly due to pressure losses. The total aerodynamic drag of such bodies can be roughly split into three main contributions. The rolling tire account for up to 25-30% whereas the rear part gives rise to 40-45% of the total drag. The remaining is due to the underbody flow and interferences. The wake structure is greatly influenced by the rear geometry of the body that determines the pressure field in this region. As showed by Ahmed [1] and by Iuso [2] the afterbody geometry plays a key role for the drag contributions and also for drag reduction when passive solutions are adopted. In cases where a square back rear part is imposed by different requirement such as internal space in commercial vehicles, passive or active control techniques can be used to achieve drag reduction. Passive and active vortex generators as those used by Aider et al [3] and piezoelectric vortex generators as proposed by Orazi et al. [4] can be adopted. Also synthetic jets located in the body rear part as done by Park et al. [5] or continuous blowing slots as done by Rouméas et al. [6] have been also used. Since a favorable energy budget is an essential requirement for a real application of active flow control the technique efficiency is of crucial importance. From this point of view very low energy absorption devices or direct drag alleviation are promising possible solutions.
In the present study a simplified car geometry has been chosen as a reference shape and a wake control system has been designed based on continuous or synthetic jets. The results of preliminary CFD analysis focused on the investigation of natural and controlled flow behavior are presented here. Moreover, this study has allowed the design of the physical experiment in the wind tunnel.

EXPERIMENTS DESCRIPTION

The model for the wind tunnel tests is reported in figure 1 a). It has a square-back rear shape typical of commercial vehicles. The flow control actuators are continuous jets injected through four rectangular slots mounted around the perimeter of the rear part as reported in red color in figure 1 b). Furthermore, four curved slots are disposed near the wheel to control the flow separation in this region. The choice of continuous jets stands on their energy saving capability. Similarly to the work of

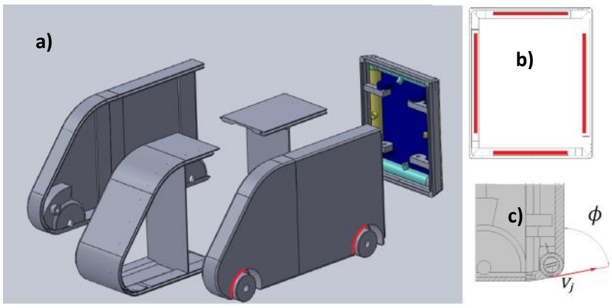

Figure 1: a) Exploded view of modular model for wind tunnel test. b) Jet slots disposition in rear part of the model. c) jet orientation angle.

Orazi et al. [1] where the use of piezoelectric actuator allowed an energy recovery, in this study the impulse of jet has a beneficial effect as a drag alleviation contribution.
The flow control parameters for the present technique are the jet orientation ϕ_i and its strength given through the momentum coefficient c_μ defined as:

$$c_\mu = \frac{h_j V_j^2}{L V_\infty^2} \tag{1}$$

where h_j is the slot width and L is a reference model length. Since the actuators geometry is fixed in the following the velocity ratio V_j/V_∞ is considered to represent the jet strength. Finally it has to be stressed that the control parameters the can be set independently for each jet slot.
To preliminarily evaluate the flow behavior a commercial code (STAR CCM+® by Cd-Adapco) was used for CFD analysis. RANS simulations with k-epsilon turbulence model were used. The results reported are referred to the same slot orientations and the same velocity ratio.

RESULTS

In figure 2 the drag reductions as a function of the velocity ratio V_j/V_∞ and the angle ϕ are reported. The results show that the drag reduction increases almost linearly with the velocity ratio independently from the angle ϕ. This behavior is respected up to $V_j/V_\infty = 1.5$ then a trend reversal occurs. For $V_j/V_\infty = 2$ the control becomes highly sensitive to jet orientation. Setting $\phi = 55°$ and $\phi = 65°$ the flow control leads to a maximum drag reduction of about 16%. On the other hand it becomes less convenient for $\phi = 50°$ and $\phi = 75°$.

Furthermore from the wake analysis it was observed that the presence of the jets can deeply modify the wake structures. The natural flow presents a toroidal vortical motion adherent to the back surface. This vortex allows a partial recovery of pressure in the back surface. The flow control gives rise essentially to two different wake topologies. The first one presents a wake structure similar to natural flow, namely a toroidal vortex attached to the rear part of the body and characterized by low drag. This flow behavior takes place for velocity ratios V_j/V_∞ up to 1.5. The second wake structure is instead governed by stream-wise counter-rotating vortices originating low or high drag according to the jet orientation. Further investigations are in progress to analyze the flow behavior around $V_j/V_\infty = 2$ and ϕ around 65°. Moreover considering also the wheel flow control higher benefits have been observed up to 18%.

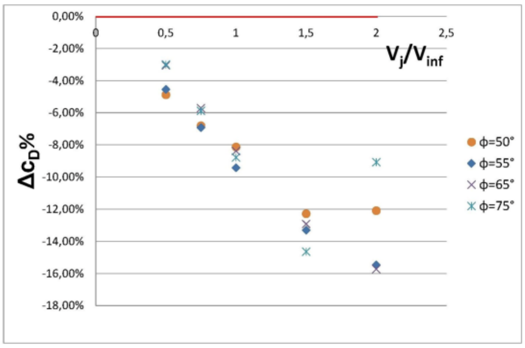

Figure 2: Drag reductions with control of rear part.

An estimation of the energy budget has been done considering the efficiency ζ introduced by Rouméas et al. [6] and defined below:

$$\zeta = \frac{P_{saved}}{P_{spent}} = \frac{\Delta D V_\infty}{K\dfrac{\rho V_j^3 S_j}{2}} \tag{2}$$

The efficiency ζ represents the ratio between the power saved through the drag reduction and the power used to control the flow. It has to be remarked that ζ has to be greater than 1 to have a positive energy budget. The coefficient K represents the total pressure losses and here it is assumed equal to 5 following the work of the previous authors. A dedicated experimental investigation will be performed in order to obtain a more rigorous evaluation.
In figure 3 the efficiency related to the velocity ratio and the jet orientation is reported.
From the graph it is evident that the efficiency decreases as the velocity ratio grows whichever is the jet orientation. Only for $V_j/V_\infty < 1$ the energy budget becomes favorable being $\zeta > 1$. The drag reduction corresponding to these values is $\Delta D/D \leq$ 8%. The control parameters pertaining the most convenient energy budget correspond to ϕ=50° and $V_j/V_\infty = 0.5$ which allow 5% of drag reduction.
The control parameters can be still optimized considering asymmetric jet orientation and jet velocity. In fact in the case of a real road vehicle, a strong flow asymmetry is present between the lower and upper part of the vehicle. As a consequence also the jet injections have to be different in order to control more efficiently the underbody flow and the upper flow.

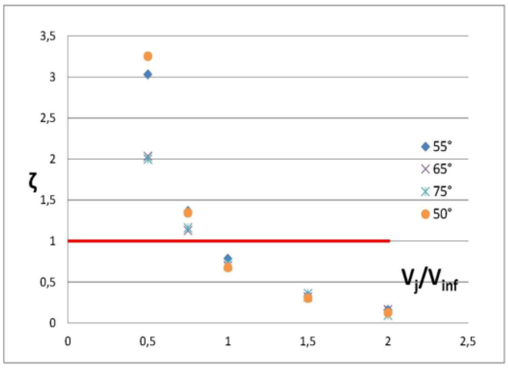

Figure 3: Control efficiency in symmetric configuration

CONCLUSIONS

A preliminary CFD analysis of flow control over a simplified 3D car shape was conducted. Continuous jets injected through four slots are used to control the near wake region. The control is applied in symmetrical configurations in terms of equal jet orientation and velocity ratio. Drag reduction up to 16% is obtained with jet orientation $\phi = 65°$ and velocity ratio equal to 2. It has to be remarked that the wake control showed higher sensitivity to the velocity ratio respect to the effects of the jet orientation. The analysis of the efficiency of the technique showed that a reduced velocity ratio of $V_j/V_\infty \leq 1$ allows positive energy budget to which correspond lower drag reductions. Works for the optimization of the velocity ratio and the jet orientation are in progress as also the preparation of the physical experiment in a wind tunnel.

REFERENCES

[1] S.R. Ahmed, Influence of base slant on the wake structure and drag of road vehicles. *J Fluids Eng 105:429–434, 1983*

[2] G. Iuso, Base pressure control by passive methods. AIAA *18th Fluid Dynamic and Plasmadynamics And Laser Conf., 1985 Cincinnati Ohio*

[3] J.L. Aider, J.F. Beaudoin, J.E. Wesfreid, Drag and Lift reduction of a 3D bluff -body using active vortex generator. *Exp Fluids, 48:771–789, 2010*

[4] M. Orazi, D. Lasagna, G. Iuso, Circular cylinder drag reduction using piezoelectric actuators. *Advanced in Aircraft and Spacecraft Science, Vol.1 027-041, 2014*

[5] H. Park, J-H. Choi, J. Lee, D-H. Lee, K. H. Kim, Experimental study on synthetic jet array for aerodynamic drag of a simplified car. *Journal of Mechanical Science and Technology, 27 3721-373,* 2013

[6] M. Rouméas, P. Gilliéron, A. Kourta, Analysis and control of the near-wake flow over a square back geometry. *Computer & Fluids, 38 60-70, 2*