REVIEW OF PASSIVE DRAG REDUCTION TECHNIQUES FOR BLUFF ROAD VEHICLES

ALAMAN ALTAF, ASHRAF A. OMAR AND WAQAR ASRAR

Departmentof Mechanical Engineering, Faculty of Engineering,
International Islamic University Malaysia,
PO Box 10, 50728, Kuala Lumpur, Malaysia.

aao@iium.edu.my

ABSTRACT: This paper presents a review of the techniques used to reduce aerodynamic drag over bluff bodies such as cylinders, spheres, 2D bodies with blunt backs and their application to commercial road vehicles. The recent research carried out on the drag reduction is presented and categorised. A new classification of the techniques is introduced and major contributions under them are shown in this paper. Moreover, it can be concluded that there is not much work done with realistic 3D bluff bodies, especially using passive methods.

ABSTRAK: Kertas kerja ini membentangkan kaji selidik semula teknik yang digunakan untuk mengurangkan seret aerodinamik ke atas jasad tubir seperti silinder, sfera, jasad 2D dengan belakang tumpul dan aplikasinya terhadap kenderaan jalan raya komersial. Pengurangan seretan dibentangkan dan dikategorikan dengan kajian terkini. Klasifikasi teknik terkini diperkenalkan dan sumbangan utamanya diperbentangkan. Secara kesimpulannya terdapat banyak tugasan yang tidak yang dapat dijalankan dengan menggunakan jasad tubir 3D sebenar, terutamanya dengan penggunaan kaedah pasif.

KEYWORDS: *drag reduction; bluff body; passive drag reduction; active drag reduction*

1. INTRODUCTION

Drag reduction is of paramount importance in the transportation field. A major part of the fuel in commercial (aerial/ground/marine) vehicles is spent on overcoming aerodynamic drag - one of the main contributors to overall drag. With the recent recession, increase in global warming and rapid depletion of fossil fuels, researchers are now looking into means to minimize the fuel consumption and reduce emissions; such as to move towards greener technologies. Heavy vehicles due to their large frontal area and bluff shapes are aerodynamically inefficient and take up to 65% of fuel to overcome drag. As mentioned by Hsu and Davis [1], it is estimated that with a drag reduction of about 40%, 10,000 USD/year/vehicle can be saved. However redesigning these vehicles altogether is an impractical solution especially in developing countries which are already burdened with their poor economy.

Following Newton's 3rd law of motion, when a solid body moves through a fluid, the fluid exerts an equal and opposite reaction force on the body; a component of which tries to resist the motion. This force called drag is expressed in the following equation:

$$D = \frac{1}{2}\rho V_{\infty}^{2} C_D A \tag{1}$$

where D is on the drag, ρ is the density of the fluid, V_∞ is the free stream velocity, C_D is the non-dimensional drag coefficient and A is the reference area.

The commonly used reference areas are: frontal area (in case of stubby, bluff bodies e.g. buildings, trucks), planform area (in case of thin, flat shapes e.g. wings) and wetted area (e.g. ships, barges). The total drag on a body is the summation of skin friction drag and pressure drag. From the equation above it is seen that drag basically depends on flow conditions and the geometry of the body. For streamlined bodies, changing the flow conditions (Reynolds number, Re) can effectively reduce drag to some extent. However in case of bluff bodies with sharp corners, change in Re has no effect on drag, thus the need to modify or manage the flow around a body arises in order to reduce drag [2].

Drag reduction techniques are mainly divided into two categories: *passive*, which involves no energy expenditure and *active*, which requires energy and involves a control system. Different techniques offered so far include: blowing, suction, base cavities, ventilated cavities, vortex generators, afterbody modification, splitter plates, boat tailing, injecting additives/droplets or fibers, large eddy breakup devices. These techniques basically try to modify the flow around the body, prevent separation, reduce wake, mass transfer into the boundary layer.

Over the past two decades with the increasing developments in science and technology, using active flow control methods has become increasingly feasible. With very efficient micro and nano scale sensors and actuators, having a feedback control system has become easier and simpler. But such systems are not economically cost effective. And in the times where the concerns for global warming, depleting natural resources and greener technologies is rising every day, it is logical that the future control strategies should focus on passive techniques. The merits of these approaches are that they require no energy expenditure, no input from the user and are cheaper than active techniques. Many attempts have been made to summarize the work done in the field of drag reduction, but very little (if not none) is done to review only passive drag reduction methods.

2. LITERATURE REVIEW

This section provides the review on the work done in passive drag reduction. The main approaches using passive methods to reduce drag have been broadly categorized into three groups as shown in Fig.1 and are discussed below.

2.1 Retrofits

Muhammed-Kassim and Fillippone [3] conducted a numerical study of fuel saving potential of various drag reducing retrofits. They found that vehicle parameters alone do not affect total drag; operational parameters have a large effect as well. It was found that using aerodynamics devices is optimal when these vehicles travel at high speeds on long haul driving cycles as weight does not have any direct effect on drag. Driving through urban areas utilizes most fuel for acceleration and deceleration. These factors are important because every part of a truck contributes, positively or negatively, to the total drag of the truck. Commonly used retrofits are given in the following subsections.

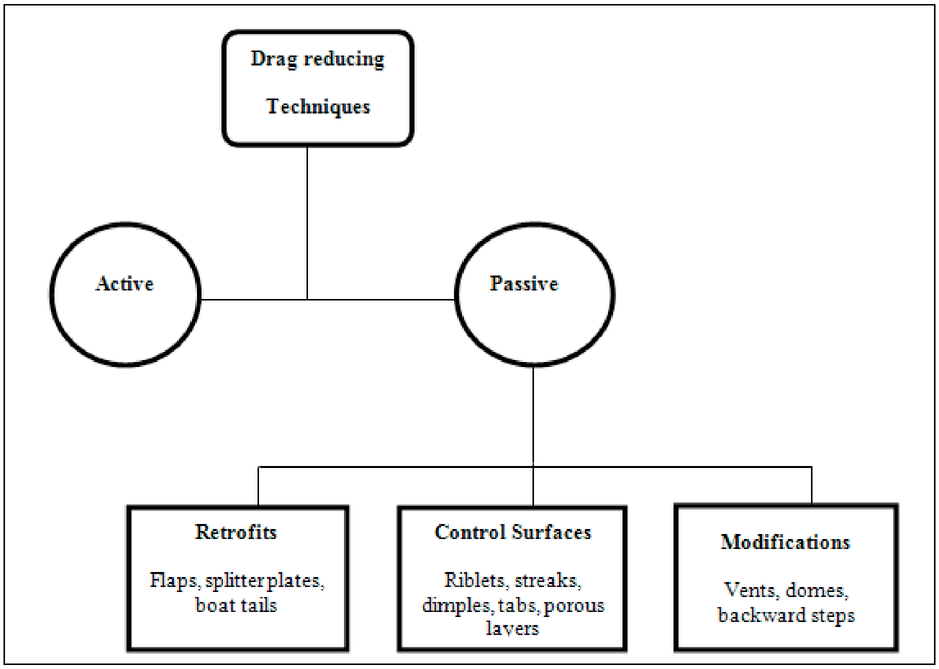

Fig.1: Classification of passive drag reducing techniques.

2.1.1 Flaps/Deflectors

Lee and Ko [4] studied flow field behind a perforated Gurney-type flaps and concluded that perforated flaps are better than solid ones to reduce drag and wake width and unsteadiness. However study was done on flaps alone, without any actual 3D body. Fouree et al. [5] in their experimental study using deflector on a generic car model found that drag reduction of up to 9% was obtained depending upon the deflector angle (Fig. 2). Beaudoin and Aider [6] did the experimental study on a 3-D bluff body called Ahmed body, a commonly used 3D bluff body for benchmarking purposes, using flaps at all the edges on the two rear surfaces and found that the most efficient configuration was the two flaps on side edges of rear slant. Depending on various configurations, the drag could be reduced by 25% and lift by 107%. Ha et al. [7] carried out experimental and computational study of drag reducing capability of a rear downward flap on a pickup truck. They found that the C_D was reduced with the increasing flap length. They deduced that the flap displaced the flow attachment enabling more downwash, hence reducing the reverse flow in the wake. With the increase in downward angle, there was an increase in drag reduction. They proposed that this happens because the cabin back pressure increases with the increase in downward angle, which reduces the drag co-efficient. However the performance doesn't stay constant with increasing flap length and downward angle. It was suggested that the rear downward flap needs to be designed which would have an optimum downward angle. While studying the use of back-flaps on a full-sized truck, Muhammed-Kassim and Fillippone [3] found a reduction of 4-5% of total drag. This is one of the few studies conducted on a full-sized truck; however, it did not address the shape or design of the flap, or optimization of the dimensions for maximum drag reduction.

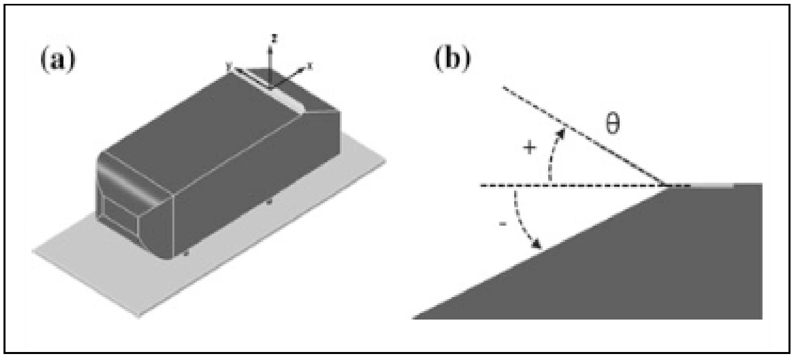

Fig. 2: Description of the Deflector: Overview (a) and definition of deflection angle θ (b). Figure taken from Fouree et al. [5].

2.1.2 Splitter Plates

Gilieron and Kourta [8] did wind tunnel testing on Ahmed body using splitter plates at the front/rear at different skew angles. Drag reduction of 12% was achieved with splitter plates placed downstream and 27%-47% with plates positioned upstream. Drag reduction was achieved by eliminating longitudinal vortices, reducing wake cross section and minimizing pressure loss. The effect of vortex interaction when plates are present at both front and rear was not studied. Park et al. [9] investigated the total drag reducing efficiency of outer layer vertical plates and achieved drag reduction of 9.6% without considering the increased wetted surface area. The height and spacing between the plates were optimized and found to be most efficient when the plates were nearest and shortest. The height of the plates was found to be the most prominent parameter. Plates with height more than the boundary layer thickness caused drag to increase rather decrease. However, many aspects such as loading on blades, vibration and downstream effect need to be studied before this can be applied to real engineering problems.

2.1.3 Boat Tails

Lancer et al. [10] studied the effectiveness of boat tailing on a tractor/trailer at NASA Research Centre wind tunnel. It was concluded that the modified flow due to the boat tails increased the pressure over the base area, hence reducing the aerodynamic drag. Khalinghi et al. [11] extended the work done by Lancer and his team by doing the computational and experimental study on bodies with and without the drag reducing device. The computational fluid dynamics (CFD) simulations were done using both steady and unsteady Reynolds-Averaged Navier-Stokes (RANS) approach. Hot wire anemometry and Particle Imagine Velocimetry (PIV) were used to understand the wake flow better. It was found that the drag coefficient reduced by 6% for steady and 18% for unsteady CFD simulations; as compared to 20% experimentally measured value. They concluded that unsteady RANS is more valid and useful in studying wake flows.

2.1.4 Fences

Modi et al. [12] did wind tunnel and full scale road tests to see the effect of ‘fences’ (boundary layer control method) on drag reduction. Wind tunnel testing done on a 1/6th scale model at subcritical Re showed drag reduction of 31.4% while the road tests showed the drag reduction of only 16.6%. The discrepancies in results were attributed to the much idealized circumstances in wind tunnel as compared to the roads; the wind tunnel model was not exactly same as the full scale model; the scaling of fence and their orientation for full scale trucks were not accurate; and that the number of fences used on the wind tunnel model and the full scale trucks were not same. Also it was found that the effectiveness of the 'fences' reduced at speeds lower than 30 km/h. The optimization in terms of location of fences was left as future work.

2.2 Control Surfaces

Bruneau et al. [13–15] presented a new passive flow control method using a porous layer. In their paper they explained the solid-porous-fluid physical models used;the setup of the boundary conditions for such flow; integration of Darcy equations with Navier Stokes equations and a penalization method applied. Numerical simulations were done to understand the passive flow control capability of porous layer at high Reynolds number and it was found to reduce vortex induced vibrations. Based on the location of porous layer on an Ahmed body, it was found that drag coefficient could be reduced by 40%. However the drag reduction was not very significant in case of Ahmed body with a rear window. Bruneau et al. [16] found that using a porous layer on the roof of the body modifies the shear forces reducing the drag by 22%. The porous layer induces the Kelvin Helmotz instabilities which change the size of vortices on the top of the body hence reducing the drag. However their study was performed only on 2D body and concluded that the more detail understanding of vortex dynamics is required to be able to modify it substantially.

2.2.1 Riblets

Chen et al. [17] did a numerical study to see the effect of bionic riblets on drag reduction of revolution body. It was found that the drag could be reduced by almost 9%. The basic reason of drag reduction was found to be reduction in skin friction drag due to control and correction of boundary layer. Sirovich and Karlson [18] showed that by having 'vee' shaped protrusions, drag can be reduced by almost 10%. However, studying various protrusion heights, 'vee' angles and their effects was left as future work. Parker and Sayers[19]found that symmetric V groove riblets on a flat plate perform better at reducing drag than unsymmetrical U groves. These grooves suppress the streamwise vortices, turbulent mixing and hence turbulent shear stress. Gruneberger and Hage [20, 21] found that triangular riblets with trapezoidal grooves aligned parallel to flow reduce drag upto 7.6% by effectively increasing wetted area.

2.2.2 Vents and Strips

Falchi et al. [22] studied the effect of passive ventilation on flow control. It was found that 7-8% drag reduction was achieved for clean vented and 20% drag reduction with strips. However no direct pressure measurements were done, so the results could be under/overestimated. Suryanarayana and Prabhu [23] found that drag can be reduced by 65% in a vented sphere.

2.2.3 Streaks

Pujals et al. [24, 25] studied the suppression of separation behind an Ahmed body by large scale coherent streaks forced on the roof. The streaks were formed using small cylinders upstream slanted rear end. Drag reduction of about 10% was obtained. However, the minimum streak amplitude necessary for separation was not found.

2.2.4 Dimples

Mode [26] did a numerical study of dimples on a surface of a flat plate to understand the physics involved and effect of the dimples on drag reduction. It was found that the presence of the dimples cause the flow to accelerate, producing a favorable pressure gradient which prevents the BL separation and hence reduces the wake.

2.2.5 Tabs

Park et al. [27] proposed a new passive control method to reduce form drag over 2-D blunt bodies. They found that by attaching small tabs on the trailing edge of a body, flow characteristics in wake changes, leading to the drag reduction. Both experimental and computational studies were done to comprehend the efficiency in reducing drag. Parametric studies were done by varying spacing, height and width of tabs at three different Reynolds numbers. It was found that the tabs reduce the strength of the vortices in the wake, leading to increase in base pressure (decreased drag). Norman and McKeon [28] studied effect on a stud (tab) on an otherwise smooth sphere using PIV (Fig. 3). It was found that in subcritical regime, with an isolated roughness added, the flow separation was delayed further downstream. The stud seems to increase the Reynolds stresses which leads to decrease in the shear layer instabilities and hence the narrower wake. However in supercritical regime, it seems that the stud causes premature separation and hence an increase in drag. The researchers also studied the effect of the size of stud (tab). Though only cylindrical tabs were used, the effect of other geometries was left as future work.

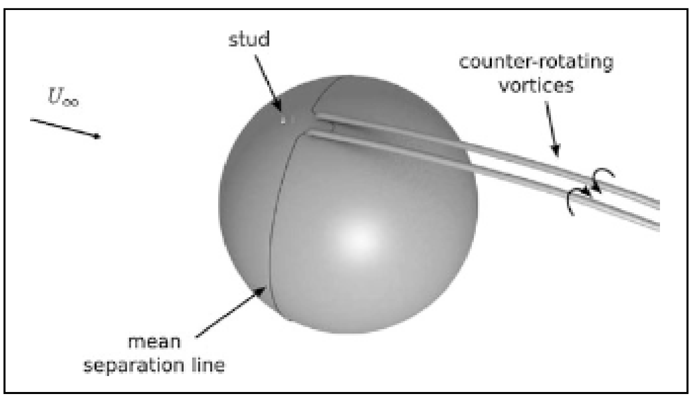

Fig. 3: The stationary stud locally delays separation on the subcritical regime, likely leading to the production of weak counter-rotating vortices.
Figure taken from Norman and McKeon [28].

2.3 Modifying / Optimizing Existing Designs

Park et al. [29] investigated flow over the backward facing step. It was found that the 51% reduction of reattachment length was achieved using rectangular tabs located at the edge of the step. The tabs caused the span wise modulation of streamwise velocity profile. Hsu and Davis [1]found that by adding humps on the top, bottom and/or sides of trailer along with boat-tail flaps (as shown in Fig. 4) can reduce drag by about 50.9%. But the method isn't feasible for real automobiles. Jing et al. [30] did a comparative study of commercial truck with a dome on top and deflector at the rear, using CFD and wind tunnel testing. They concluded that drag could be reduced by about 10%.

Strachan et al. [31] studied wake flow of Ahmed body with overhead supporting struts and both with and without moving ground. The results obtained were compared to two previous studies done. They found that the overhead supporting struts reduced the strength of the trailing counter rotating vortices, which in turn lead to the separation of flow earlier than the critical back angle. It was also concluded that the moving ground had little to no effect on the overall results. They also found that there were small vortices developing near the ground (behind the model) which were not reported by any previous researches; most probably due to the interference of the struts used to mount the model. However they could not provide any accurate measurement of the distance the vortices propagate.

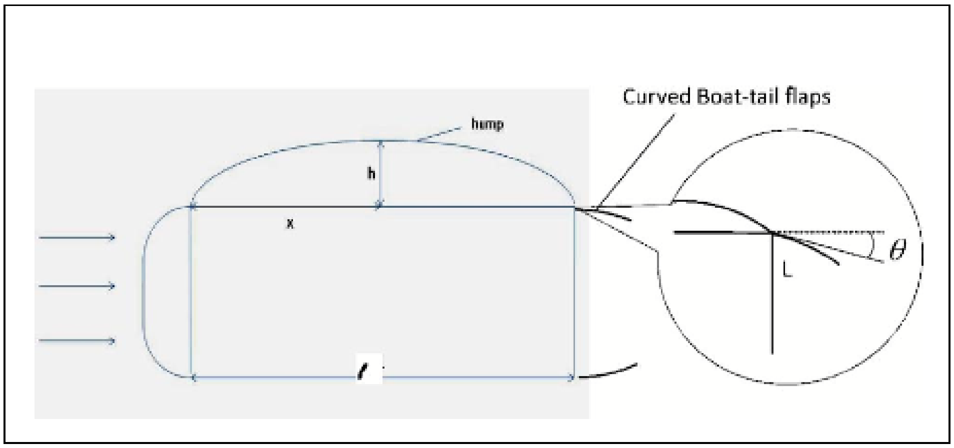

Fig. 4: Hump and curved boat-tail flaps geometry. Figure taken from Hsu and Davis [1].

3. CONCLUSION

From the literature provided in the previous section it is seen that passive drag reduction is a fast growing field. A lot of work has been done on this topic and researchers all over the world continue adding to the knowledge. In this review, existing passive flow control methods were classified into three categories: retrofits, control surfaces and design modifications (shown in Fig. 1) and the work done under each category was presented and discussed.

As mentioned earlier, control methods are either active or passive. Due to the simplicity, cost effectiveness and easier implementations, passive techniques are gaining popularity. Though as compared to the active flow management, this field still seems to be in infancy. Much of the work done previously in this field is on 1D and 2D bodies. However it is only when a reliability of a method is proven in 1D or 2D, that it can be developed to more complex geometries. The flow around a 3D body is much more complex and requires much deeper understanding. Retrofits, amongst the three categories presented, is widely used and studied on 3D bodies. Splitter plates, flaps and deflectors are already employed on various commercial vehicles. Following the same trend it won't be long before other methods would be proven to be useful on 3D bodies. More studies should be done to apprehend the flow around 3D bodies and to obtain an effective drag reduction.

REFERENCES

[1] Hsu, F.-H., Davis, R. L. “Drag reduction of tractor-trailers using optimized add-on devices.” *Journal. Fluids Engineering*, *132*(8), 084504. doi:10.1115/1.4001587

[2] Hucho, W.-H. Ed. “Aerodynamics of Road Vehicles”, 4th ed. *Society of Automotive Engineers.*

[3] Mohamed-Kassim, Z., Filippone, A”Fuel savings on a heavy vehicle via aerodynamic drag reduction.” *Transportation Research Part D* Vol. 15. pp 275-285. (2010).

[4] Lee, T., Ko, L. S. “PIV investigation of flowfield behind perforated Gurney-type flaps.” *Experiments in fluids* 46(6) 1005-19, 2009.

[5] Fourrié, G., Keirsbulck, L., Labraga, L., Gilliéron, P. “Bluff-body drag reduction using a deflector.” *Experiments in. Fluids,* (2011), 50 (2): 385-395.

[6] Beaudoin, J.-F., Aider, J.-L. “Drag and lift reduction of a 3D bluff body using vortex generators, “*Experiments in Fluids*, (201) 48, 771-789.

[7] Ha, J., Jeong, S., Obayashi, S. “Drag reduction of a pickup truck by a rear downward flap.” *International. Journal. Automotive. Technology*, (2011) 12.3:369-74.

[8] Gilliéron, P., Kourta, A. “Aerodynamic drag reduction by vertical splitter plates.” *Experiments in Fluids, 48(1), 1–16. doi :10.1007/s00348-009-0705-7*, .

[9] Park, H., An, N. H., Hutchins, N., Choi, K.-S., Chun, H. H., Lee, I. “Experimental investigation on the drag reducing efficiency of the outer-layer vertical blades“*Journal. Mar. Science. Technology*, *16*(4), 390–401. doi:10.1007/s00773-011-0135-0

[10] Lancer, W. R., Ross, J., Kaufman, A. E. “*Aerodynamic Performance of a Drag Reduction Device on a Full-Scale Tractor/Trailer.”* Aerospace Technology Conference and Exposition SAE Technical Paper, 1991.

[11] Khalingi, B., Zhang, S., Koromilas, C., Balkanyi, S. R., Bernal, L. P., Iaccarino, G., Moin, P. “Experimental and Computational Study of Unsteady Wake Flow behind a Bluff Body with Drag Reducing Device.” SAE Trans., 110.6(2001):1209-222.

[12] Modi, V. J., Hill, S. S., Yokomizo, T“Drag reduction of trucks through boundary-layer control“J. Wind Eng. Ind. Aerodyn., 54.55(1995):583-94.

[13] Bruneau, C H., Mortazavi, I., Gilliéron, P.”Flow Regularisation and Drag Reduction around Blunt Bodies Using Porous Devices.” *IUTAM Symposium on Flow Control and MEMS* Springer Netherlands. 2008.

[14] Bruneau, C. H., Gilliéron, P., Mortazavi, I. “Passive control around the two dimensional square back ahmed body using porous devices” *Journal Fluids Engineering,* 130.6(2008):6-11.

[15] Bruneau, C. H., Mortazavi, I. “Numerical modelling and passive flow control using porous media. “*Computers and Fluids*, 37.5(2008):488-98.

[16] Bruneau, C. H., Depeyras, D., Mortazavi, I. “Active and Passive Flow Control around Simplified Ground Vehicles.” *Journal of Fluid Mechanics*, 5.1(2012):89-93.

[17] Zhang, C., Wang, J., Shang, Y“Numerical simulation on drag reduction of revolution body through bionic riblet surface. “*Science. China Technology. Sci* 53.11(2010):2954-959

[18] Sirovich, L., Karlsson, S. “Turbulent Drag Reduction by Passive Mechanisms”. Nature, 388(1997):753-55.

[19] Parker K., Sayers, A. T. “The effect of longitudinal microstriations and their profiles on the drag of flat plates”. Proc. Inst. Mech. Eng. Part C *Journal. Mechanical. Engineering. Science,* 213.8(1999):775-85.

[20] Grüneberger R., Hage, W. “Drag characteristics of longitudinal and transverse riblets at low dimensionless spacings.” *Experiments. Fluids*, 50.2(2010):363-73.

[21] Falchi, M., Provenzano, G., Pietrogiacomi, D., Romano, G. P“Experimental and numerical investigation of flow control on bluff bodies by passive ventilation.” *Experiments in Fluids,* 41.1(2006):21-33.

[22] Suryanarayana G. K., Prabhu, A. “Effect of natural ventilation on the boundary layer separation and near-wake vortex shedding characteristics of a sphere.” *Experiments in Fluids*, 29.6(2000):582-91.

[23] Pujals, G., Depardon, S., Cossu, C. Drag reduction of a 3D bluff body using coherent streamwise streaks. *Experiments in Fluids*, 49.5(2010):1085-94.

[24] Pujals, G., Cossu, C., Depardon, S. Forcing large-scale coherent streaks in a zero-pressure-gradient turbulent boundary layer. *Journal.ofTurbulence,* 11(2010):25. doi:10.1080/ 14685248.2010.494607

[25] Mode, J. M. Simulation of the flow over a flat dimpled plate. Arizona State University. MSc thesis. (2010).

[26] Park, H., Lee, D., Jeon, W.-P., Hahn, S., Kim, J., Kim, J., Choi, J., Choi, H. Drag reduction in flow over a two-dimensional bluff body with a blunt trailing edge using a new passive device. *Journal of Fluid Mechanics,* 563(2006):389, doi:10.1017/S0022112006001364.

[27] Norman, A. K., McKeon, B. J. Unsteady force measurements in sphere flow from subcritical to supercritical Reynolds numbers. *Experiments.in Fluids* 51.5(2011):1439-53.

[28] Park, H., Jeon, W.-P. Choi, H. and Yoo, J. Y. Mixing enhancement behind a backward-facing step using tabs. *Physics.of Fluids*, 19.10(2007):103-05.

[29] Jing, C., Yun-zhu, W., Deng-feng, W., Wei-gang, X., Gui-Jin, WResearch on aerodynamic drag reduction of heavy commercial vehicle. 2nd *International Conference on Computer Engineering and Technology,* 5(2010):194.

[30] Strachan, R. K., Knowles, K., Lawson, N. J. The vortex structure behind an Ahmed reference model in the presence of a moving ground plane. *Experiments in Fluids*, 42.5(2007):659-669.

NOMENCLATURE

1D	*One Dimensional*
2D	*Two Dimensional*
3D	*Three Dimensional*
DR	*Drag Reduction*
Re	*Reynolds Number*
PIV	*Particle Image Velocimetry*
CFD	*Computational Fluid Dynamics*
RANS	*Reynolds Averaged Navier Stokes*