BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July, 20–24 2008

TIME-AVERAGED PHENOMENOLOGICAL INVESTIGATION OF A WAKE BEHIND A BLUFF BODY

G.M.R. Van Raemdonck*⋆* and M.J.L. van Tooren*†*

⋆Design Integration and Operations of Aircraft and Rotorcraft
TU Delft, Faculty of Aerospace Engineering, Kluyverweg 1, 2628HS Delft, The Netherlands
e-mail: `g.m.r.vanraemdonck@tudelft.nl`

†Design Integration and Operations of Aircraft and Rotorcraft
TU Delft, Faculty of Aerospace Engineering, Kluyverweg 1, 2628HS Delft, The Netherlands
e-mail: `m.j.l.vantooren@tudelft.nl`

Keywords: Time-Averaged, Bluff Bodies, Experimental investigation

Introduction The main aerodynamic features which are characteristic for a blunt based bluff body are the large region of separated and recirculating flow behind the body, a large pressure drag and highly unstable wake. The abrupt geometrical change at the base of the bluff body leads to flow separation and the formation of a dead air region. It is the low base pressure within the wake that is mainly responsible for the pressure drag of bluff bodies.

The flow contained within the region of the wake is dominated by the formation of large and small vortices combined with locally large amplitude velocity fluctuations in the wake. These fluctuations frequently occurs associated with the shedding of vortices and are most apparent in a two dimensional flow. For two dimensional bodies the near wake is dominated by the periodic and alternate shedding of vortices known as the Von Karman vortex sheet. The most well researched body to show this phenomenon is that of the circular cylinder and a summary of the knowledge that has been accumulated over the years is given by Roshko [1].

For flows around axisymmetric and fully three dimensional bodies, however, wake periodicity is found to be a much less prominent feature. The flow around these types of bodies is extremely complex because vorticity shed from the body has components in all three directions. Another aspect which adds to the complexity of these flows when considering road vehicles is the effect that is induced by the presence of the ground. Although there have been several studies which have been conducted on this topic (Bearman [2] and Hucho [3]), the majority of them only focused on time averaged motion of the flow and not on unsteady flow behavior. The work of Oertel [4] at the beginning of 1990 proved, experiments and corresponding numerical simulations, the existence of unstable regions in the Von Karman vortex street.

The unsteady wake of a blunt based road vehicle was investigated by Duell and George [5]. In their work they hypothesized that the wake is dominated by the pumping effect of the wake behind the bluff body. Duell and George observed that a large ring-type vortex is present within the the recirculation region between the bluff body base and the free stagnation point of the wake. They also suggested that this free stagnation point is in fact a quasi-periodically fluctuating point that shifts as vortices are shed from the shear layer to which Duel and George are referring to as the pumping effect.

Other experimental research concerning the bubble pumping theory are from Khaligini [6], Bayraktar [7] and the thesis of Balkanyi [8]. With the aid of the Large Eddy Simulation (LES) together with the increasing computer power, Krajnovic and Davidson ([9], [10], [11], [12] and [13]) were able to improve the insight in the different flow mechanisms around bluff bodies. Their studies with LES [10] have shown that the instantaneous flow is very different from the time-averaged one, not only in the wake but also along the entire body.

There is suggested by Basford [14] and Hoerner [15], that a thin boundary layer at the side of a bluff body is responsible for more negative pressure in the wake than a thick boundary layer. Or in other words a nice rounded front (that is responsible for the thin boundary layer and a lower drag coefficient) has as consequence a lower pressures at the back of trailer, while a front with sharp corners and thus thicker boundary layers along the body results in higher pressure coefficients at the back.

Balkanyi [8, 16] gives the following interpretation. The abrupt geometrical change at the base of the bluff body leads to flow separation and the formation of a dead air region which results in the formation of a free shear layer. Through this shear layer, mass is entrained from the dead air region, which results in low pressure at the base of the model and an upstream flow at the center of the dead air region. Entrainment also causes the shear layer to grow and eventually merge with the shear layers formed on the other sides of the model. The same principle of mass entrainment and return between the shear region and the recirculation region is stated by Maull [17] for two dimensional bluff bodies.

Not much is known about this subject and the behavior of three dimensional shear layers and a straightforward explanation that answers all the questions is not yet available. For two dimensional profiles the wake and its size is a measure for the total drag which includes the friction and the pressure drag of that profile. But, is this still the case for bluff bodies with a blunt base? The goal of this research is to define a relation between models with different boundary layer thicknesses, the total drag of the corresponding body, the pressures at the back surface and the size of its wake. This relation should provide more insight in the flow behavior around and in the wake of bluff bodies.

Methodology The relations between what is happening at the front and the back of the bluff body, the pressure drag, the size of the wake compared with the total drag, together with the thickness of the boundary layer are not understood well. Therefore a phenomenological investigation in two different phases of the flow, of which the first phase is presented here, is being executed with a bluff body, which represents a tractor-trailer combination. Four different roughness's are added to the front of the body which induces different boundary layers thicknesses. Performing time-averaged force measurements and pressure measurements, boundary layers thickness and planar PIV (Particle Image Velocimetry) measurements will contribute to define a relation and to a better understanding of the behavior of the flow past the bluff body.

In a second phase time-dependent pressure at the surfaces and velocity field measurements (spectral analysis) together with more advanced PIV (stereoscopic and high speed) will be executed. Extensive three dimensional numerical simulations will help with the preparation of these wind tunnel tests.

Wind tunnel and model set-up The experiments were executed in the Low Turbulence Tunnel at the Delft University of Technology, The Netherlands. This closed circuit windtunnel has a octagonal test section with a cross sectional area of $2.07m^2$ (width of 1.8m; height of 1.25m) and a maximum speed of 120m/s. The used wind tunnel test section is calibrated with a pitot tube that measures the dynamic pressure.

The windtunnel is restricted by not having a moving belt which is desirable to investigate the ground effect of, for instance, race cars. Ground simulations and its study have been the subject of many research projects [18, 19, 20, 21, 22]. The major issue is that the approaching floor boundary layer is too thick. According to Cooper [23, 24] one can conclude that a fixed-floor with a thinned boundary layer is sufficient for current automotive and commercial vehicle applications.

To test the vehicle the model is suspended on a parallel floor which has an offset of 250mm with respect to the upper (horizontal) wind tunnel wall and has the same width as the test section. On the rounded front edge of the plate develops a new thinner boundary layer in comparison with the boundary layer on the wind tunnel wall.

The Reynolds number for a European full-scaled truck, based on the square root of the frontal area of $A = 10.34m^2$ and a driving velocity of 25m/s, becomes 5.4×10^6 [23]. The scaled wind tunnel model has a Reynolds number of 0.84×10^6 which is high enough for bluff bodies, [25]. No Reynolds effects were deducted around the testing velocity. Also no corrections methods were applied on the dynamic pressure.

In order to initiate good boundary layer development, without the interruptions of several vehicle parts, towards the back of the body a generalized representation of a tractor-trailer combination, according to European dimensions, is being used, fig. 1. The GETS (Generalized European Transport System) model is build-up with aluminum plates in a modular way and is equipped with more than 170 pressure orifices. The main part of the orifices are situated at the back surfaces of the GETS model.

Figure 1: GETS model

Measuring techniques The six-component mechanical balance system measures the resulting forces acting on the truck model. Only the drag coefficient C_T of the vehicle will be discussed. The pressure on the different surfaces is measured, to calculate a corresponding pressure coefficient C_p, with the aid of Esterline pressure scanners [26]. A single probe of DANTEC mounted on a mechanical transverse system is used to define the boundary layer thicknesses through measuring the velocity towards the model surface. Hard- and software of Davis is used to perform the PIV measurements.

Before the actual measurements the boundary layer at the front round edge of the model is checked, by using the microphone and oil visualization, if separation is occurring and if the boundary layer becomes turbulent. In order to prevent separation bubbles zig-zag tape is used at the front edges of the model to trigger the corresponding boundary layer

Experimental results Currently one is working on the results obtained during the wind tunnel test in order to make a good comparison between the different configurations. The objective is to compare all five different configurations (and thus five different boundary layer thicknesses) of the GETS model: a clean configuration without zigzag-tape and four configurations with four different zigzag-tape thicknesses. Also comparison between the drag and the pressure coefficient around the model will be made. Processing the PIV data gives more insight in the steady wake structure and its length.

Below a first indication is given of the drag measurements results. Fig. 2 shows the time-averaged drag coefficient (at zero yaw angle) of the different configurations. The clean configuration (without triggering boundary layer) has the highest drag coefficient which can be expected due to the flow separation at the front edge. The configuration with the 0.65mm zigzagtape indicated no separation bubbles, which results in the lowest C_T value. An increasing zigzag-tape thickness goes together with an increasing drag coefficient. The boundary layer thicknesses measurements should clarify this out.

The pressure coefficient at the back surface $C_{P,back}$, also plotted in fig. 2, has the same trend as the total drag coefficient. It is remarkable, however, that at the lowest drag coefficient, the most negative pressure coefficient (highest suction at the back surface) is registered. A profound drag breakdown and analysis of the different drag contributions will contribute to the understanding of this behavior.

The right picture, fig. 3, indicates the absolute difference and the difference in terms of percentage of the drag coefficient considering the clean configuration as reference value.

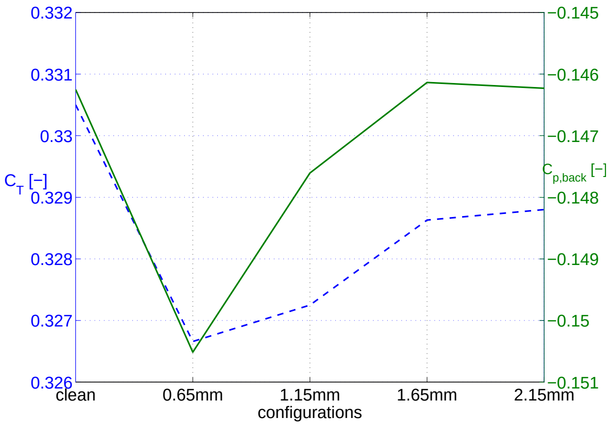

Figure 2: Drag coefficients

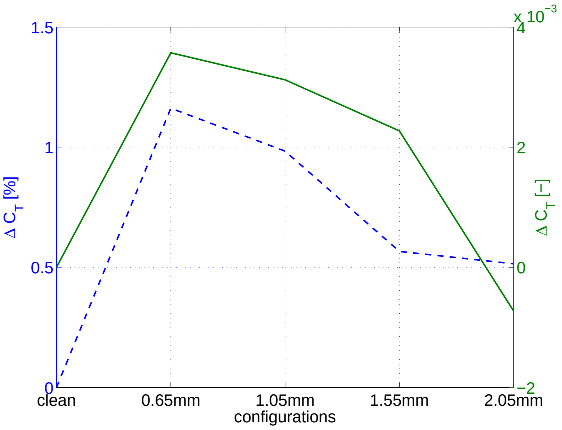

Figure 3: Overview drag coefficient differences

REFERENCES

[1] Roshko, A., *Perspectives on bluff body aerodynamics*, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 49, 1993, pp. 79.

[2] Bearman, P., *Some observations on road vehicles wakes*. SAE Pape, Vol. 840301, 1984.

[3] Hucho, W.-H., *Aerodynamics of road vehicles*, fourth edition, Hardbound, 1998.

[4] H.jr., O., *Wakes behind blunt bodies.* Anual Review of Fluid Mechanics, Vol. 22, 1990, pp. 539–564.

[5] Duell, E. and George, A., *Experimental study of a Ground vehicle body unsteady near wake.* SAE Paper, Vol. 1999-01-0812, 1999.

[6] Khalighi, B., Zhang, S., Koromilas, C., Balkanyi, S., Bernal, L., Iaccarino, G., and Moin, P., *Experimental and computational study of unsteady wake flow behind a bluff body with a drag reduction device.* SAE Paper, Vol. 2001-01-1042, 2001.

[7] Bayraktar, I., Landman, D., and Baysal, O., *Experimental and computational investigations of ahmed body for ground vehicle aerodynamics.* SAE Paper, Vol. 2001-01-2742, 2001.

[8] Balkanyi, S., *An experimental investigation of drag reduction devices for blunt based road vehicles.* Master's thesis, Delft University of Technology, Faculty of Aerospace Engineering, 2001.

[9] Krajnovic, S. and Davidson, L., Large-eddy simulation of the flow around a ground vehicle body. SAE Paper, Vol. 2001-01-0702, 2001.

[10] Krajnovic, S. and Davidson, L., *Numerical study of the flow around a bus-shaped body.* Journal of Fluid Engineering, Vol. 125, 2003, pp. 500–509.

[11] Krajnovic, S. and Davidson, L., *Large-eddy simulation of the flow around simplified car model.* SAE Paper, Vol. 2004-01-0227, 2004.

[12] Krajnovic, S. and Davidson, L., *Flow around a simplified car, Part1: Large Eddy Simulation.* Journal of Fluid Engineering, Vol. 127, 2005, pp. 907–918.

[13] Krajnovic, S. and Davidson, L., *Flow around a simplified car, Part2: Understanding the flow.* Journal of Fluid Engineering, Vol. 127, 2005, pp. 919–928.

[14] Basford, W., *Apparatus to reduce drag behind bluff bodies*, United States Patent, Vol. 2002/0030384, 2002.

[15] Hoerner, S., *Fluid-dynamic drag*. Hoerner Fluid Dynamics, Vancouver, 1965.

[16] Balkanyi, S., Bernal, L., Khalighi, B., and Sumantran, V., *Dynamics of manipulated bluff body wakes.* AIAA Journal, Vol. 2000-33888, 2000.

[17] Maull, D., *Mechanism of two and three dimensional base drag.* Symposium on Aerodynamic drag mechanisms of bluff bodies and road vehicels proceedings, Plenum, New-York, September 1976, pp. 137-153.

[18] Berndtsson, A., Mason, W. E., and Mercker, E., *The effect of ground plane boundary layer control on automotive testing in a wind tunnel.* SAE Paper, Vol. 880248, 1988.

[19] Hacket, J., Baker, H., Williams, J., and Wallis, S., *On the influence of ground movements and wheel rotation in tests on modern car shapes.* SAE Paper, Vol. 750245, 1975.

[20] Mercker, E. and Knape, H., *Ground simulation with moving belt and tangential blowing for full-scale automotive testing in a wind tunnel.* SAE Paper, Vol. 890367, 1989.

[21] Carr, G., *A comparison of the groun-plane-suction and moving-belt ground-representation techniques.* SAE Paper, Vol. 880249, 1988.

[22] Eckert, W., Singer, N., and Vagt, J.-D., *The Porsche wind tunnel floor-boundary-layer control - A comparison with road data and results form moivng belt.* SAE Paper, Vol. 920346, 1992.

[23] Cooper, K., *Bluff-body aerodynamics as applied to vehicles.* Journal of Wind Engineering and Industrial Aerodynamics, Vol. 49, 1993, pp. 1–21.

[24] Cooper, K., *The wind tunnel testing of heavy duty trucks to reduce fuel consumption.* SAE Paper, Vol. 821285, 1982.

[25] Society of AutomotiveEngineers, *SAE wind tunnel test procedure for truck and buses*. SAE, Vol. J1252, 1981.

[26] Systems., E. P., http://www.pressure-systems.com/esp_scanners.html.