

The Aerodynamic Characteristics of a Race Car Wing Operating in a Wake

Michael R. Wilson, Robert G. Dominy and Adam Straker

University of Durham

Copyright © 2008 SAE International

ABSTRACT

It is well documented that the aerodynamic performance of an open wheel race car is degraded when closely following another car. The greatest performance loss is usually experienced by the front-mounted wing leading to reduced aerodynamic downforce, handling imbalance and a reduced overtaking capability. Although previous wind tunnel studies have been performed to investigate this race situation the model scales have generally been compromised in order to achieve representative separation between the two test vehicles within the confines of the wind tunnel working section and particularly within the limited length of the moving ground plane. This study addresses that issue by using a very short, bluff body to create an accurate representation of the wake flow from the leading car in order to provide additional, effective test length ahead of the instrumented model. Data and analysis are presented that identify and quantify the key three-dimensional characteristics of the wake structure of a Formula One Racing car. It is shown that the wake structure is dominated by a trailing vortex pair coupled with turbulence intensities in excess of 40% and a velocity deficit that locally exceeds 50%. The effect of that wake structure upon a following vehicle and, in particular, its front wing is described and the relative significance of each of the primary characteristics of the wake is investigated independently. As the wake develops the effect of loss redistribution and reduced wake intensity suggest the wake influence reduces rapidly with increasing inter-vehicle spacing whilst the vortex structure is more persistent. As a consequence a following car experiences considerable local incidence variation and from the available evidence it could be argued that attempts to frame racing regulations to encourage overtaking should focus primarily on the reduction of the strength of the trailing vortex pair.

INTRODUCTION

It is well known that when drafting, the aerodynamic interaction between the two cars can be very significant. For example, Romberg et al [1] demonstrated drag

reductions of up to 37% for NASCAR stock cars when following another vehicle. For low downforce configurations substantial drag reduction enhances the potential for overtaking but for high downforce designs the associated loss of downforce usually has the opposite effect and in some instances changes in pitching moment can be so severe as to lead to the danger of overturning (Howell [2], Dominy [3]). For open-wheeled cars the effects of drafting are less extreme from the perspective of overturning but there is a dramatic effect upon the cornering and overtaking capability of the downstream car. In an attempt to quantify the aerodynamic characteristics of overtaking Dominy [4] used two similar quarter scale Grand Prix race cars, one placed in front of the other, to investigate the effects of drafting on the performance of the downstream car. He discovered that when separated longitudinally by one car length the downforce on the following car may be reduced by as much as 36 per cent and the drag reduced by 23 per cent relative to the forces that would be generated in 'clean air'. Arguably of even greater significance, was the observed rearward change of the center of pressure that arose as a consequence of the relatively greater loss of downforce that is experienced at the front of the car than at the rear and which results in severe understeer. In that situation the associated reduction of cornering speed, lower entry speed onto the following straight, lower entry speed into the following corner and reduced stability under braking all combine to make overtaking between two cars of similar performance almost impossible.

The studies of both Romberg [1] and Dominy [4] also highlight one of the difficulties associated with investigations of this kind which is the very limited inter-vehicle spacing that can be achieved in conventional wind tunnels especially when a moving ground plane is used. As a consequence, Romberg's study [1] was limited to spacings of less than two vehicle lengths whilst Dominy was able to achieve axial separations of only one vehicle length despite mounting the upstream model on a fixed ground ahead of the wind tunnel's moving ground plane. Blackhurst [5] investigated the effects of wake airflow on the performance of a 1/3rd scale, single-element Formula 1 racing car front wing. His model was tested at different free stream turbulence intensities with

that turbulence being generated by the wakes from circular cylinders. As expected, the measured downforce fell, presumably as a consequence of the reduced axial velocity in the wake but his work also showed that the lower (suction) surface of the aerofoil was affected more than the upper surface implying that the high level of free stream turbulence has an effect that is independent of the free stream velocity deficit. Soso [6] performed a similar investigation using a more sophisticated model in which a single-element wing in ground effect was tested downstream of a bluff body that incorporated a diffuser and rear wing. Like Blackhurst [5], the tests included variations in the height and angle of attack of the wing, whilst data were presented from force and pressure measurements, flow visualization and flowfield surveys. It was found that when fully immersed in the wake from the bluff body the wing experienced a reduction in downforce, relative to the 'clean air' case, and that the proportion of the downforce that was lost was sensitive to the wing's ground clearance. Despite these investigations there has been little advance in our knowledge of the relative influence of the turbulence (and turbulence length scale), velocity deficit and vorticity that are present in the wake, nor in our understanding of the flow structures that must be reproduced in the wind tunnel and in CFD simulations in order to determine the aerodynamic performance of a complete car during drafting and overtaking maneuvers.

There is a much broader body of research relating to the operation of race-car wings in ground effect under low free stream turbulence conditions. Ranzenbach and Barlow [7,8] investigated essentially two-dimensional, single element aerofoils above a stationary ground plane with complementary CFD simulations that allowed the study to be extended to include the relative motion between the wing and the ground. They clearly demonstrated the widely accepted relationship between reduced ground clearance and increased downforce that continues until a limiting ground clearance is reached which corresponds to the onset of stall. Further work such as that by Knowles et al [9], Jasinski and Selig [10], Kieffer et al [11], Martinez [12], and Zeriha and Zhang [13] have extended the breadth of the research to more complex wing geometries but little additional insight has been added in terms of the complex flow physics and none of those aerofoil in ground effect studies considered the influence of free stream turbulence. Any such information that does exist is confined to proprietary studies performed by the Motorsport industry. Research into the effect of high free stream turbulence upon any kind of aerofoil is sparse and the most complete work to date is that of Swalwell [14] who investigated the effect of turbulence upon aerofoil performance in the context of wind turbines. The results from that study showed clearly that low turbulence, wind tunnel-derived data generally under predicts lift close to the stall point and that in 'real' conditions stall is delayed relative to the idealized wind tunnel situation.

The objectives of the study presented in this paper were threefold: firstly to characterize the wake structure of a generic F1 car, secondly to recreate that wake using a

very short bluff body in order to increase the effective inter-car spacing in the wind tunnel and thirdly to begin to investigate the effect of the various aspects of the wake structure on the following vehicle. In this study it is only the aerodynamic response of the front wing of the following car that has been investigated and not the complete car since that is the component that is most significantly affected [e.g. 4].

EXPERIMENTAL STUDY

In order to establish the wake characteristics of a current generation Formula One car a representative vehicle wake was generated using a 1/6 scale model. The model (figure 1) was designed to meet the 2006 FIA F1 regulations [15] and included all generic appendages that were expected to have a significant influence upon the wake structure.

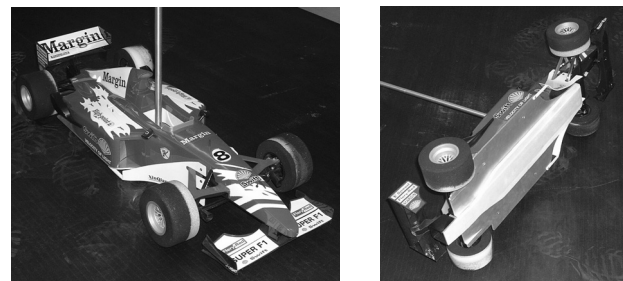


Fig 1: Wind Tunnel model (complete vehicle) – upper and lower surfaces

The testing was performed in the 1m automotive wind tunnel at the Durham University School of Engineering [16] which incorporates a moving ground plane in a $\frac{3}{4}$ open jet working section. The model was run at a Reynolds number of 6×10^5 (based on car length); high enough to generate representative wake structures although arguably not sufficient to give confidence in the Reynolds number independence of individual components such as the wings. The model was mounted from above and its rotating wheels were attached to the model's own suspension system and driven by rolling contact with the moving ground plane.

The Wake studies were performed using the wind tunnel's three-axis probe traversing system. A calibrated 5-hole pressure probe was adopted for the majority of the measurements because of its versatility for the acquisition of pressure field and three-dimensional velocity field data [17]. The probe was calibrated for pitch and yaw angles within the range $\pm 45^\circ$ and its transfer function correction was determined [18] to allow the investigation of unsteady wake structures at frequencies up to 500 Hz. These data were supplemented by hot wire measurements to obtain higher frequency velocity data to a cut-off frequency of 20 kHz. To maximize the spatial resolution of the traverse data that could be achieved within the available testing period the wake was assumed to be symmetrical about the centerline and consequently only half the car's wake was surveyed. This method was deemed appropriate since testing was

performed only at zero yaw although it is acknowledged that in some instances periodic, lateral structures have been observed in vehicle wakes that can lead to asymmetry [e.g. 19]. It should be noted that asymmetric flow structures have not, to date, been reported in the literature relating to race car wakes and for open wheel race cars the driving flow structures for such phenomena are expected to be weak. The wake was measured at various downstream distances and the data presented here refer to traverse planes located one quarter and one half a car's length downstream from the rearmost point on the car (i.e. $x/L=0.25$, $x/L=0.5$ respectively where x is the distance downstream and L is the overall car length).

In order to determine the influence of the wake upon an isolated aerofoil that is fully immersed in that wake the wake structure was recreated at a larger, 1/3 scale by means of a short, bluff body. The design of that body and the details of the flow structures that were recreated are described in more detail in the results section. The bluff body was mounted immediately upstream of the moving ground plane in order to minimize the effect of its presence upon the effective length of the working section that remained available for the aerofoil study. The aerofoil selected for the investigation was the single element NASA/Langley LS(1)-0417 (GA(W)-1) (figure 2). This was chosen because of the considerable database for this profile that already exists for this profile (e.g. Knowles et al [9], Soso [6]). The chord of the model wing was 250mm, its span was 600mm and the wing was fitted with endplates throughout the test programme. The aerofoil was mounted from a six-component overhead force balance.

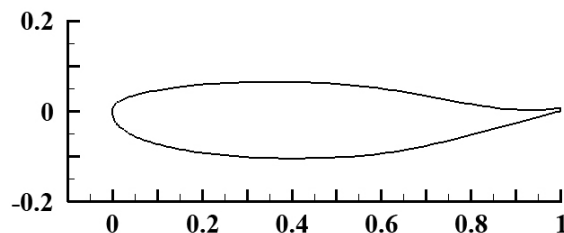


Figure 2: LS(1)-0417 Aerofoil Section
(dimensions normalized against chord)

RESULTS

Wake Survey

Figure 3 ($x/L=0.25$) shows secondary velocity vectors (i.e. the velocity component in a plane perpendicular to the direction of travel) that shape the entire wake. The structure of the wake created by this model of a generic, 2006 F1 car is clearly dominated by the wing tip vortex which is centered at $y=80\text{mm}$, $z=120\text{mm}$ where y is the lateral distance from the centerline and z is the height above the moving ground plane. These vortices are enhanced by the upward flow component exiting from the underbody diffusers but any streamwise vorticity from

the diffuser corners is largely obscured by the dominant vortex pair.

Figure 4 shows contours of stagnation pressure coefficient with the zone of lowest stagnation pressure (highest loss) lying close to the centers of the wing tip vortices but extending across the full span of the wing. In general high loss is associated with a vortex core [e.g. 19] and its lateral extension across the wing, between the vortex cores, results from the migration of low energy fluid from the upper body surfaces under the influence of the wing tip vortices. That low energy fluid is associated not only with the body surface boundary layers but also with other sources of energy loss such as the wake from the front wing and from the front tires. It should be noted that under normal operating conditions the leading edge of the front wing of the following car would lie in the range 150mm to 270mm (25mm to 45mm at model scale) which lies well below this zone of particularly extreme stagnation pressure deficit. Of more relevance to the following car is the low stagnation pressure that is observed at $y=50\text{mm}$, $z=30\text{mm}$. This too is a result of accumulated low energy fluid drawn primarily from the wheel wakes and also from the underbody diffuser boundary layer. In this case it lies close to the ground where it might be expected to have a significant impact upon the following car.

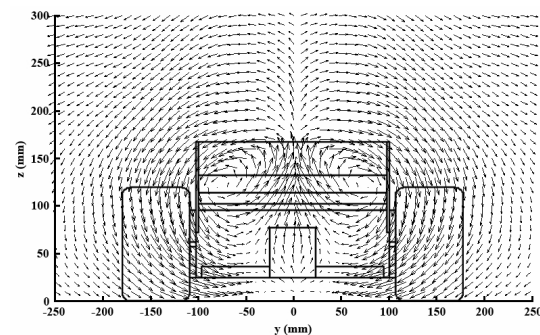


Figure 3: Secondary Velocity Vectors at $x/L = 0.25$

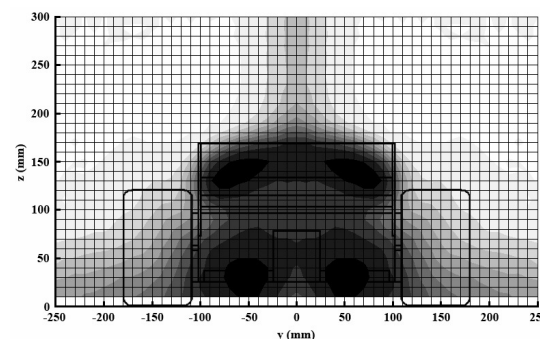


Figure 4: Contours of Stagnation Pressure Coefficient
at $x/L = 0.25$

The corresponding axial velocity contours are shown in figure 5. Not surprisingly these closely resemble the form of the stagnation pressure data since low stagnation pressure in a wake is associated with reduced dynamic