

Aerodynamic Structure and Development of Formula 1 Racing Car Wakes

Matthew Watts and Simon Watkins
RMIT Univ.

ABSTRACT

For the modern Formula 1 racing car, the degradation in aerodynamic performance when following another car is well documented. The problem can be broken into two parts; firstly the wake flow generated by these vehicles and the subsequent interaction a following car has with this field. Previous research [1, 2 & 3] has focused upon investigating the later without completely characterizing the former. This paper seeks to address this deficiency with initial data from a newly commissioned 30% scale Formula One wind tunnel model built to the 2011 technical regulations. Experimentation was carried out in the Industrial Wind-Tunnel (IWT) at RMIT University. In the absence of a rolling road an elevated ground plane was implemented; the results obtained show good agreement with the limited published material available. Using a high frequency response, four-hole pressure probe the aft body flow was investigated at multiple downstream locations. Time-averaged velocity, turbulence and secondary flow vectors were plotted. These data were supplemented through flow visualization techniques to better understand the salient aerodynamic features. The influence of geometric changes to the aerodynamic devices upon the rear of the car (i.e. drag reduction system currently used within Formula 1) was also investigated.

CITATION: Watts, M. and Watkins, S., "Aerodynamic Structure and Development of Formula 1 Racing Car Wakes," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(3):2014, doi:10.4271/2014-01-0600.

INTRODUCTION

Classically the aerodynamic development of vehicles has taken place in a smooth flow environment, be it in the wind tunnel, Computational Fluid Dynamics (CFD) or controlled real world testing. For vehicles operating in isolation this methodology has yielded favorable results, however, when flow conditions change from this ideal state the aerodynamic impact can be great. This is of particular importance for racing cars and especially Formula 1 where vehicle performance is increasingly aerodynamically dependant. The design of these vehicles is such that this performance comes at the cost of highly adverse wake conditions. The combination of these shortcomings is a vehicle which cannot operate in close downstream proximity to a vehicle of similar design; overtaking a leading car in racing conditions proves unduly difficult without a disproportionate performance advantage.

Despite this being a well known problem, few studies investigating wake structure of Formula 1 cars exists in the public domain. Most research has instead focused upon the performance changes a trailing car or lifting surface would experience during drafting, without complete characterization of the conditions behind a leading car. For example, Dominy [1] investigated the aerodynamic changes experienced by a 1989 Formula 1 car when following in the wake of an identical model. Here two quarter-scale models were positioned

longitudinally with the trailing car on a moving ground while the leading car was located on the test section floor. Longitudinal spacing was limited to one car length owing to experimental limitations however lateral movement was investigated. Downforce and drag measurements for the trailing car showed a drop of 36 and 23 percent respectively over the free-stream case. The centre of pressure was also seen to move rearwards with the conclusion that the front wing is most affected during drafting. Despite these insights, the flow field generated by the leading car was not interrogated. Soso [2] performed a similar investigation but used a much simplified model in order to de-couple this behavior. An instrumented single element wing capable of force and pressure measurements was used in place of an entire vehicle, while wake generation was achieved with a bluff body featuring a diffuser and single element rear wing with end plates. In addition to force and pressure data, flow field surveys were performed at a fixed position of three characteristic lengths downstream of the wake-generating model. Stream-wise and vertical velocity components were measured at several lateral planes encompassing the width of the wing model. The most adverse conditions were found at the centerline; stream-wise velocity deficits of 20 percent were seen while vertical velocity increased some 15 percent. Turbulence intensity followed a similar trend with a value of 11

percent at the centerline that reduced to 7 percent at the lateral limit. Unfortunately wake development was not investigated nor vorticity.

To date the most complete study is that carried out by Wilson [3]. The methodology used was a further refinement to that explored by both Dominy [1] and Soso [2]. Testing was again performed in a moving ground wind tunnel with a single element wing downstream of a bluff body. However Wilson addressed the gaps in knowledge of previous research by ensuring the flow generated by this body conformed to that experienced by a real racing car during drafting. To achieve this, flow field studies were performed on a 1/6 scale 2006 Formula 1 model, full flow properties were documented at two transverse planes located at 1/4 and 1/2 car lengths respectively. A compacted wake generator was then developed to match these data; comparisons between the flows generated by each model were deemed sufficiently similar for the research goals. Although thorough in approach a significant limitation was the Reynolds number achieved during testing which was limited to 6×10^5 based on car length. Wilson's work [3] also highlighted a problem common to wake mapping experiments and one experienced by other researchers [1, 2, 4 & 5]; wind tunnels with short working sections limit the inter-vehicle spacing's that can be achieved. This problem is only compounded for large scale models and facilities featuring a moving ground, in most case longitudinal spacing's less than one vehicle length are all that is practically achievable. However in real world racing scenarios Formula 1 vehicles are influenced by wake flows at far greater distances.

To achieve increased inter-vehicle spacing's it is proposed that wake characterization studies be performed in a conventional wind tunnel with a long test section. The findings from this work could then be applied to generate characteristic wakes in a moving ground facility. This would allow detailed analysis of the wakes influence on downstream vehicles while maintaining correct boundary conditions. Furthermore, recent changes to the technical regulation binding Formula 1 have allowed the introduction of movable aerodynamic devices with the aim to increase overtaking frequency [6]. The idea behind this system is to increase the speed of a trailing vehicle by allowing the driver to reduce the rear wing flap incidence and thus affect a passing maneuver. As the rear wing is a key component to wake generation, changes in its geometry undoubtedly affect downstream flow. Thus the goals of this research are firstly to reproduce Wilson [3] and Soso's [2] results using a simplified experimental method; including a stationary floor and model with static wheels. Second, to catalogue the wake development at longitudinal spacing's greater $\frac{1}{2}$ car lengths downstream and lastly quantify the influence of rear wing flap angle upon wake structure.

EXPERIMENTAL STUDY

In order to generate the required wake a suitable wind tunnel model was required. A review of current reference forms revealed no publically available design which would be

appropriate [7] and thus a model was designed and built specifically for this experiment. The result was a 30 percent scale model conforming to the 2011 technical regulations [6] which featured all relevant appendages envisioned to influence the wake structure (Figure 1). Included in this model were also some more advanced features such as adjustable wing sections, cooling and exhaust flows, adjustable suspension and provisions for wheel rotation. In this research, the model configuration was set to the most basic parameters with fixed wheels and only rear wing flap angle varied. Adjustment to the rear wing was directly modeled on the Drag Reduction System (DRS) currently in use within Formula One; the flap pivots at the trailing edge and thus disrupts the Fowler condition at certain settings (Figure 2). Three flap settings are investigated in this research; the first being the design condition, the second being the DRS setting which for this model places the flap at -15° angle-of-attack (AoA) and the third where the flap is trimmed to the free-stream at -47° AoA.

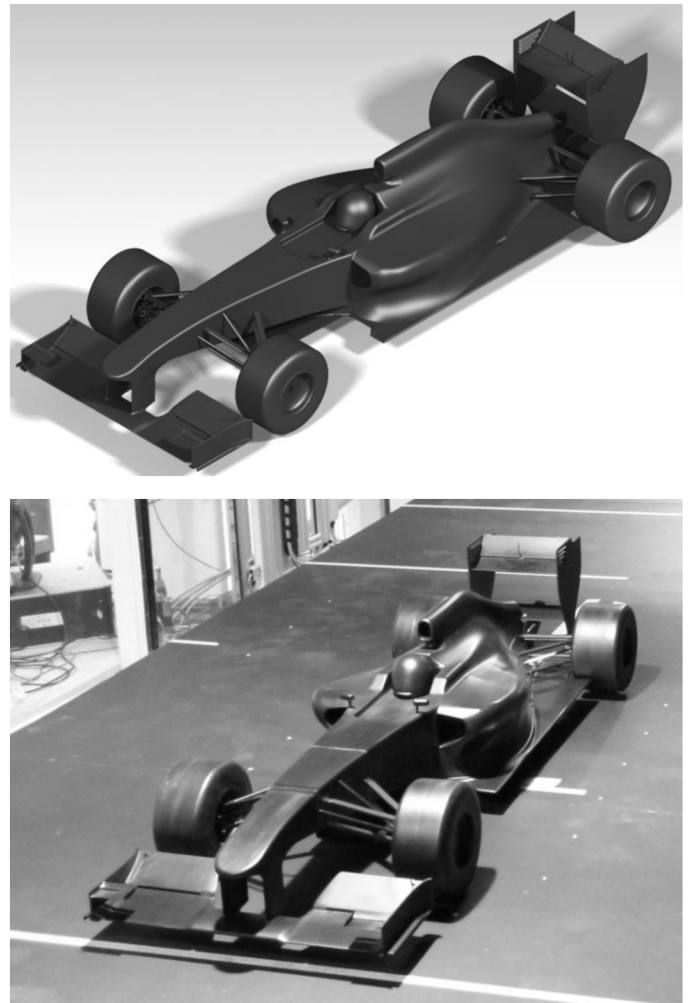


Figure 1. Top; computer render of Formula 1 model, bottom; actual model during experimentation.

Testing was performed in the 3 by 2 meter Industrial Wind Tunnel (IWT) at RMIT University. Featuring a 9 meter long test section, this facility provided adequate down stream space for flow mapping studies however the fixed floor present resulted

in considerable boundary layer growth. To minimize this effect, an elevated ground plane was manufactured that featured a super ellipsoidal leading edge with a cross-section conforming to an optimal low speed design [8] and a 2 degree nose down AoA. These measures resulted in a slight positive pressure gradient that promoted boundary layer attachment and minimized thickness. Longitudinal boundary layer development is shown in Figure 3, 95% thickness grows from 9.5 mm at the model nose to a maximum of 35 mm at the rear measurement plane. It is recommended for fixed ground wind tunnel testing that the displacement thickness (δ^*) be no more than 10% of the model ground clearance [9]. As can be seen this condition is met for only a small fraction of the model. However this is not an absolute requirement for valid data; using the same wind tunnel Paggiarella [10] accurately reproduced results from Ahmed [11] with less clearance than that achieved here. Boundary layer influence although present, was thus expected to be minimal.

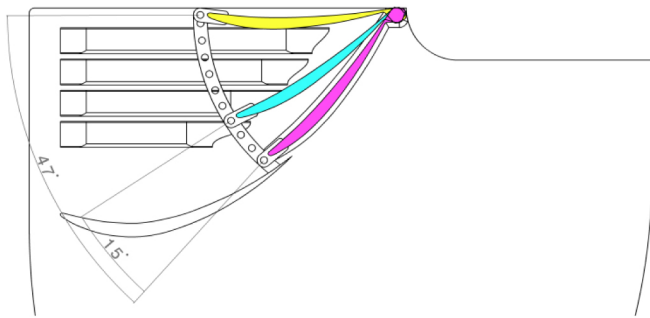


Figure 2. Side sectional view of rear wing showing angle settings.

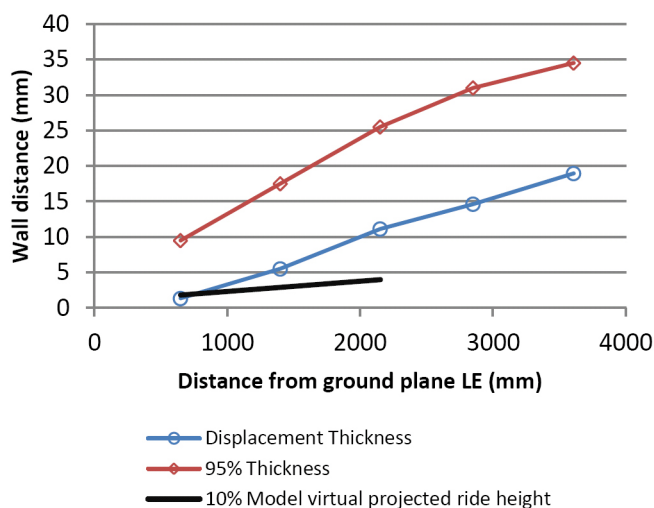
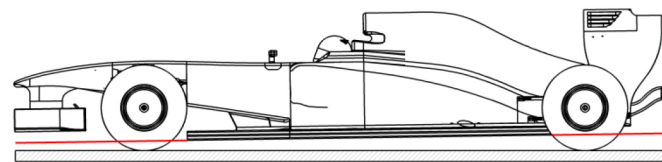


Figure 3. Top; definition of virtual projected ride height shown in red, bottom; longitudinal boundary layer growth along ground plane

Limited force balance measurements were also made during experimentation to assess the flow speed to be used during wake mapping. Here Reynolds number sensitivity formed the primary metric; downforce ($-C_L$) and drag coefficient (C_D) independence was achieved above a Reynolds number of 2.0×10^6 based on car length (Figure 4) and thus all wake mapping experiments were conducted at 2.4×10^6 to ensure data integrity.

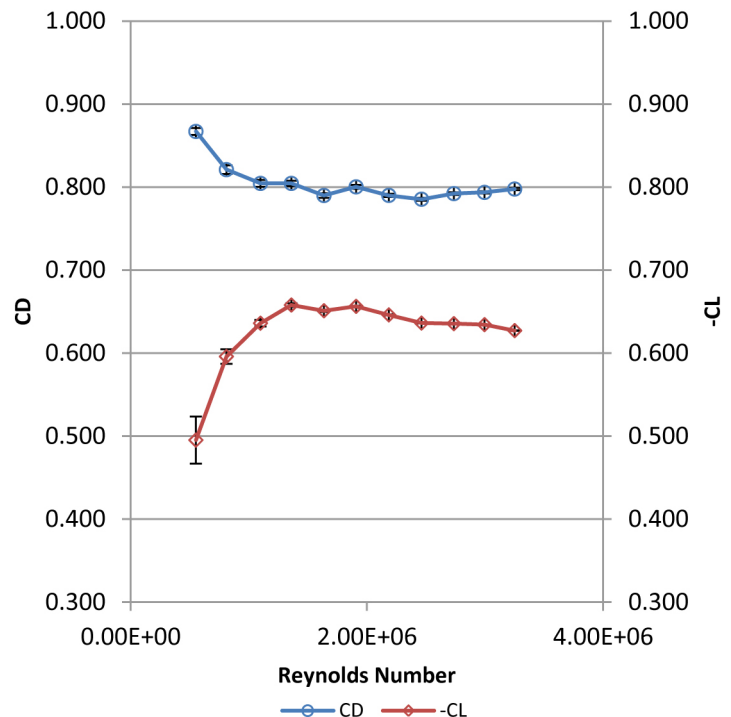


Figure 4. Downforce and drag sensitivity to Reynolds number

Wake studies utilized a calibrated 4-hole pressure probe capable of a dynamic response of up to 2.5 kHz and resolving pitch and yaw angles within a $\pm 45^\circ$ range. The former is achieved through application of the Inverse Transfer Function (ITF) method to allowing phase lag and amplification corrections [12]. A singular probe was used for this research and thus each flow map constitutes a series of non-simultaneous measurements. Time-resolved reconstruction of such data sets using a reference signal is possible; however in practice this method has shown limited success in three-dimensional flows and remains limited to the near field [13]. Given wake maps in this research are far downstream of the model, mixing effects would preclude reconstruction methods. Analysis was therefore limited to the time-averaged domain.

To allow automation of the experimental procedure, the probe was coupled to a numerical controlled 3-axis traverse system. Wake interrogation was performed at two downstream locations corresponding to transverse planes located $\frac{1}{2}$ and 1 car lengths behind the rearmost point of the model. To account for wake expansion the measurement area was increased with longitudinal spacing. To facilitate analysis, wake flow is presented in the form of contour maps of velocity magnitude and intuitive data representation. The color scheme used is of