

The use of a Bluff Body Wake Generator for Wind Tunnel Studies of NASCAR Drafting Aerodynamics

Roger G. Dominy
University of Durham, UK

Geoff LeGood
G.L Aerodynamics

Copyright © 2008 SAE International

ABSTRACT

It is well documented that the aerodynamic downforce that is developed by a NASCAR race car is degraded when closely following another car. This may lead to handling imbalance and a reduced overtaking capability. Although wind tunnel studies have been performed to investigate this race situation those investigations have frequently been compromised in order to accommodate two or more test vehicles or models within the confines of the wind tunnel's working section. 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 maximize the test length available to the test model. Data and analysis are presented that identify and quantify the key three-dimensional characteristics of the wake structure of a NASCAR race car and 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 60%. It is clearly demonstrated that a simple, bluff body wake generator is an effective tool that allows detailed drafting studies to be performed in existing facilities and without the need to reduce the scale of the test models.

INTRODUCTION

In comparison to the amount of testing and development which is undertaken by the aerodynamics groups within racing teams and race car constructors there has been comparatively little data published in the public domain.

In recent years and perhaps with the increased popularity and more open style of regulations, the aerodynamics of cars racing in the top NASCAR series (Winston Cup/Nextel Cup/Sprint Cup etc) have become the subject of published work, some of which appears to have been attributed to the change in regulations to the Car of Tomorrow concept introduced in 2007.

The basic aerodynamic characteristics of NASCAR-type race cars were shown by Duncan [1], Laise and Bayless [2] and Leuschen and Cooper [3]. These were also

discussed in detail by Nelson, Rousch, Eaker and Wallis [4] during the development of roof mounted flaps which were devised to reduce lifting effects during a spin. Landman and Britcher [5,6,7] also characterized the aerodynamics of NASCAR race cars in their papers describing the development of the Langley Wind Tunnel as a facility suitable for testing race cars. In all of these papers the wind tunnel testing was conducted using a force and moment measurement approach which is comparable to the approach used for other forms of passenger car and race car aerodynamic testing.

Also in a similar fashion to published work on road cars, CFD analyses have been performed on NASCAR geometries to show the flow regimes around the race cars. Duncan and Golsch [8] showed the flow around a 2003 Pontiac NASCAR racer. They compared the flow regimes of fixed ground configurations to track simulations with moving ground and rotating wheels and also the time dependent flow regimes. Singh [9] also used CFD to show the flow regimes around the NASCAR Car of Tomorrow body style in "conventional" fixed ground wind tunnel testing mode and moving ground mode with rotating wheels. Differences in the shape and magnitude of the wake downstream of the base were identified.

In all of the above work the tests were conducted on a single car facing a clean approach flow. In practice, however, the "real world" operating environment of a NASCAR-type race car can be quite different to the conventional wind tunnel simulation.

At some tracks the cars run very close to the outer retaining wall which was shown by Wallis and Quinlan [10] to have a significant influence on drag and lift characteristics. Their work was conducted using 3/8 scale models but without relative movement between the model, the ground and the wall. However, it did illustrate the varying aerodynamic characteristics that a car might be subject to during racing conditions.

Perhaps the biggest difference between the work described above and typical racing conditions is that the nature of the tracks (mainly ovals) and the racing

regulations ensure that cars are often running at high speed in close proximity.

Hoerner [11] and Dalton [12] showed that the aerodynamic drag of both bluff and streamlined bodies was modified when two similar objects were placed in close proximity in a tandem and side-by-side arrangement. The significance of this to NASCAR racing was reported by Romberg, Chianese and Lajoie [13] who conducted tests using 3/8 scale models to investigate changes in aerodynamic characteristics in both drafting and passing situations. The changes in drag and lift were shown to be experienced by both lead and trailing cars in drafting situations – effects which are used strategically by drivers during racing. Two cars running in tandem gave drag reduction benefits to both lead and trailing cars allowing them to go faster than a single car circulating alone. However the lift balance of the cars also changes with the most significant degradation being to the lead car. In passing maneuvers the characteristics of both cars also change. Both of these effects can clearly be observed during racing with cars slipstreaming each other, “bump-drafting” and maneuvering so as to unsettle the car ahead when wishing to overtake.

The complexities of wind tunnel testing in simulated race car conditions with multiple vehicles or models seems to have precluded published data since the work by Romberg et al. It would appear that much aerodynamic development of NASCAR race-cars is still conducted in the single vehicle in clean air mode. It is possible that in the future CFD might help in analyzing the multiple vehicle conditions as recently indicated by Read [14] although for most users the computing requirements to achieve this are beyond practical limits today.

Similar concerns apply to open-wheeled cars. In an attempt to quantify the aerodynamic characteristics of overtaking Dominy [15] 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. The studies of both Romberg [13] and Dominy [15] 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 [13] was limited to spacings of less than two vehicle lengths whilst Dominy [15] 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. Various solutions have been proposed including the use of smaller models and the development of a new generation of lengthened wind tunnel working sections. The former has obvious limitations in terms of measurement resolution and Reynolds number effects whilst the latter can lead to flow instability, unacceptable longitudinal pressure gradients and, of course, considerable financial cost. Wilson et al [16] addressed 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 minimize the loss of test section length arising from the use of an upstream model. It was shown that for a Formula 1 car 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 were described and the relative significance of each of the primary characteristics of the wake (turbulence, velocity deficit and flow upwash/downwash) was discussed. Such was the success of the approach for an open wheeled race car that further studies were undertaken to evaluate the approach for use with NASCAR wind tunnel studies. Initially the wake structure of the complete car was evaluated using CFD which allowed the entire flowfield to be evaluated for a variety of configurations. Some further CFD studies were performed to establish whether the major wake flow structures could be recreated using a bluff body simulation before validating the approach by a series of wind tunnel studies on the bluff body. It is the results from that process that are presented here.

CFD STUDY

In order to establish the wake characteristics of a NASCAR race car a series of preliminary CFD simulations were performed using the Cosmos FloWorks software. The model was based upon a 'Car of Yesterday' with its distinctive asymmetry (e.g.[8]) and the car's wake characteristics were investigated with and without a rear spoiler. The model is shown in figure 1.

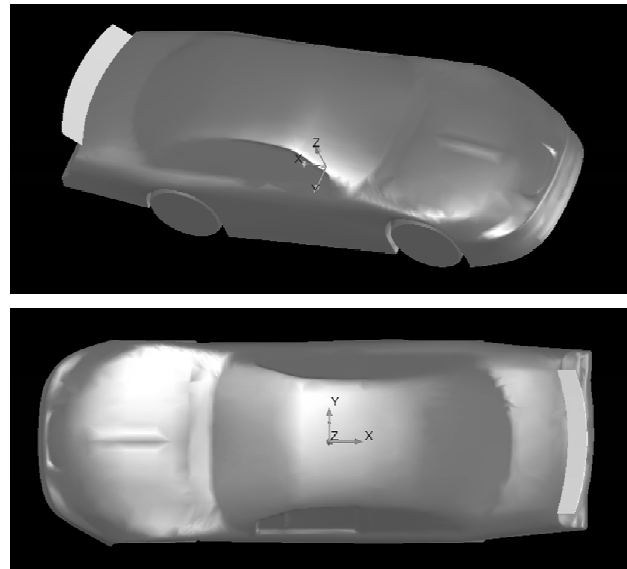


Figure 1: The Asymmetric CFD Model

The calculations were performed at a Reynolds number of 7.6×10^6 based on vehicle length using a structured mesh of approximately 1 million cells. The structure and development of the wake is seen in Figure 2 in which the

cross section of the wake is presented at three different axial planes as defined in Table 1.

Traverse Plane	x / L
1	0.095
2	0.285
3	0.475
x: distance downstream from the rear of the vehicle L: Overall vehicle length	

Table 1: Axial location of wake cut planes

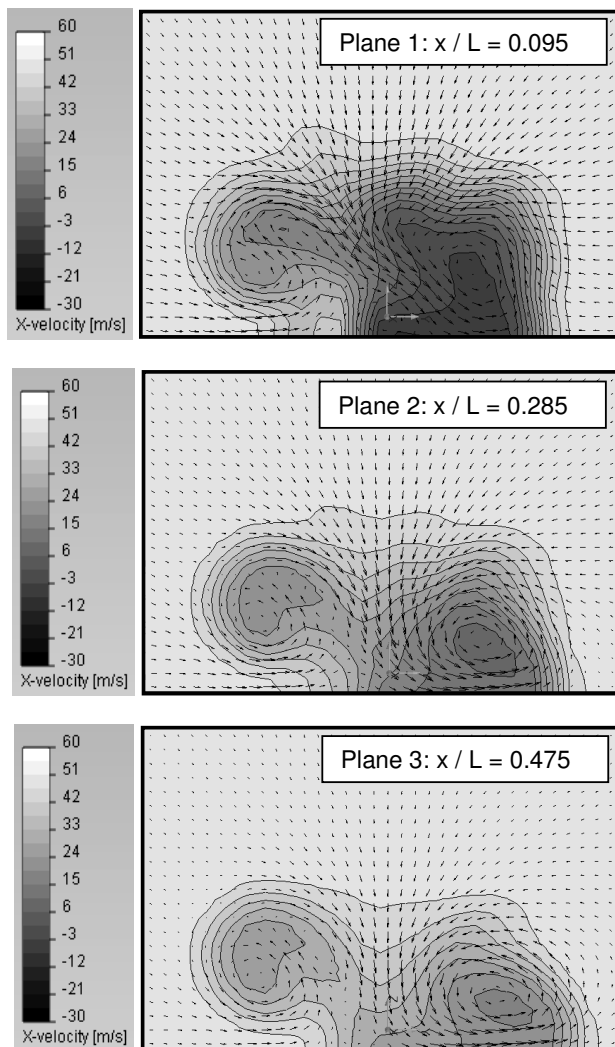


Figure 2: Axial velocity contours and velocity vectors without a rear spoiler (CFD)

The initial investigation was performed without the rear spoiler in order to determine the car's fundamental wake flow structures. The most obvious feature of the wake is its asymmetry arising from the significant asymmetry of the car itself. The classical wake structure for a notchback car is dominated by a pair of trailing vortices

associated with the flow over the roof and C-pillar and these are evident here but in a severely distorted form. The right hand side of the car produces a strong trailing vortex that draws in flow from a much weaker, counter-rotating vortex that is generated at the left hand side of the car. The result is an almost diagonal flow from top left to bottom right. The air that feeds the system is drawn from the lower left hand corner of the wake resulting in a third vortex, located slightly below and to the left of the left hand side C-Pillar vortex, which rotates in the opposite direction from its near neighbor. The wake structure develops rapidly and by Plane 2 the three vortex system has developed into a pair of dominant vortices, both of the same sign with only the remnant of the counter vortex still visible. The mixing of the flow in the wake results in reduced velocity gradients as the flow continues to develop downstream but at plane 3 the wake structure remains much as observed at plane 2.

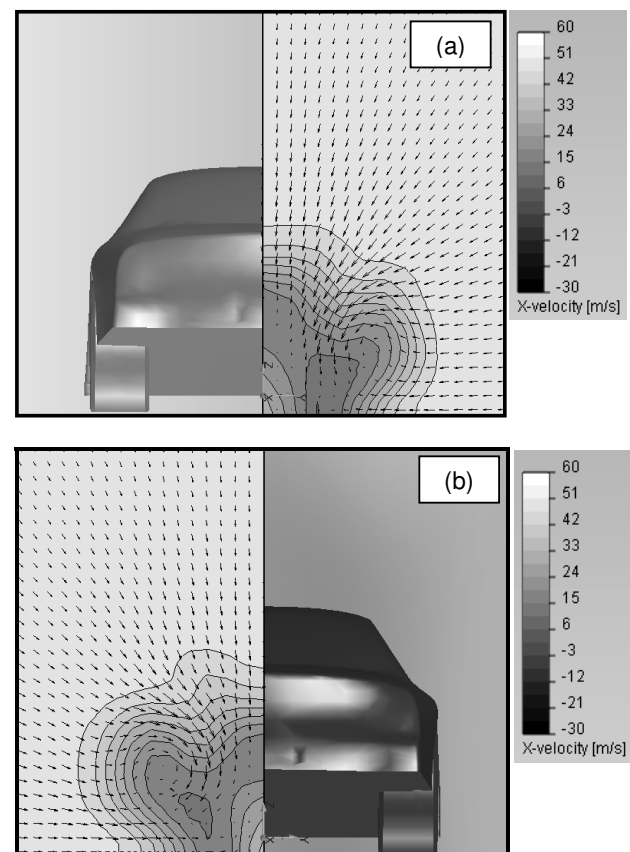


Figure 3: Axial velocity contours and velocity vectors without a rear spoiler (CFD) with a symmetry plane

The influence of the vehicle's asymmetric geometry is emphasized by comparison between the asymmetric data (figure 2) and simulations performed on each side of the car independently using a plane of symmetry as the centerline boundary condition. The simulated, symmetrical wake structure at Plane 2 is shown in figures 3(a) and 3(b). As expected the right hand trailing vortex is a little stronger (figure 3(a)) but the wake structures are essentially mirror images of each other