

Typical Velocity Fields and Vortical Structures around a Formula One Car, based on Experimental Investigations using Particle Image Velocimetry

Masaki Nakagawa
Toyota Central R&D Labs., Inc.

Stephan Kallweit and Frank Michaux
Intelligent Laser Applications GmbH

Teppei Hojo
Toyota Motor Corporation

ABSTRACT

This paper presents typical flow structures around a 60%-scale wind-tunnel model of a Formula One (F1) car, using planar particle image velocimetry (PIV). The customized PIV system is permanently installed in a wind tunnel to help aerodynamicists in the development loop. The PIV results enhance the understanding of the mean velocity field in the two-dimensional plane in some important areas of the car, such as the front-wheel wake and the underfloor flow. These real phenomena obtained in the wind tunnel also help maintain the accuracy of simulations using computational fluid dynamics (CFD) by allowing regular checking of the correlation with the real-world counterpart. This paper first surveys recent literature on unique flow structures around the rotating exposed wheel, mostly that on the isolated wheel, and then gives the background to F1 aerodynamics in the late 2000s. It subsequently describes features of the PIV system that is used in the development loop and discusses improvements to the efficiency of the preparation and operation processes. Typical velocity fields behind the rotating front wheel and the underfloor of two F1 car models, one before and one after a regulation change in 2009, are then analyzed. Finally, important vortical structures around the 2009 car are highlighted from the results obtained in the development loop. These results show the initial effect of the regulation change on the flow structures, and how the 2009 car was developed to overcome the initial deficit in downforce.

CITATION: Nakagawa, M., Kallweit, S., Michaux, F., and Hojo, T., "Typical Velocity Fields and Vortical Structures around a Formula One Car, based on Experimental Investigations using Particle Image Velocimetry," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1611.

INTRODUCTION

One of the most challenging tasks in the study of open-wheel Formula One (F1) aerodynamics is to understand actual flow phenomena around the rotating wheel, which is exposed to a high-speed freestream. In particular, the front wheels interact with the wakes of the front wing located upstream, and are sometimes disturbed by turbulence in the wake of the preceding car. In past decades, various studies have conducted experimental and numerical investigations to understand these complex phenomena. This section is a comprehensive survey of the literature of these studies, primarily focusing on wind tunnel testing. It will be shown how quickly research into F1 aerodynamics has increased in complexity to provide solutions that are more practical, owing to the rapid progress of measurement techniques in the 2000s.

Isolated Wheel

The most common investigations are those of the rotating isolated wheel, since the investigations require a relatively simple experimental setup and the single wheel alone involves complex flow phenomena. Since Fackrell and Harvey [1,2] and Fackrell [3] first conducted comprehensive studies of the lift and drag forces computed from measured surface pressure on the rotating isolated full-scale solid wheel in contact with the ground, extensive studies have been conducted both experimentally and numerically, to clarify complex phenomena. In the late 1990s, great efforts were put into developing more accurate pressure measurement techniques; e.g., Hinson [4]. However, an actual flow phenomenon can only be postulated from the surface pressure distribution, leaving open questions about the real flow phenomena. Only classical techniques, such as the use of a pitot probe and hot-wire anemometry, were available to measure the velocity field in the 1990s but these techniques are always subject to directional ambiguity and, importantly, are intrusive. Oil flow and smoke visualization techniques indeed gave useful insights, but only provided quantitative information under limited conditions.

In the 2000s, especially after non-intrusive measurement techniques that employ lasers, such as Laser Doppler anemometry (LDA) and particle image velocimetry (PIV), became available, more accurate and detailed flow structures could be obtained, the level of understanding of flow phenomena around rotating isolated wheels rapidly rose, and the literature became more thorough, as described in the following.

Knowles et al. [5] first presented the velocity field using non-intrusive three-dimensional (3D) velocity measurements around a 40%-scale isolated Champ Car solid wheel, using a long-range LDA system. The measurement was conducted in transverse sections at several downstream positions in the near wake of the wheel. They claimed that the vortex structure in the near wake is different from theoretical models previously proposed by Cogotti [6] and Merker and Berneburg [7], owing to the existence of a support sting and the “hub flow”. Saddington et al. [8] subsequently conducted similar measurements around a 50%-scale F1 isolated grooved solid wheel, where they proposed a revised model of the “trailing vortex system” in the near wake.

Mears et al. [9, 10] conducted the first PIV measurement around an isolated 40%-scale non-deformable pneumatic tire/wheel assembly in conjunction with a conventional pressure measurement on the wheel surface, using a radio telemetry system. In that work, they confirmed the existence of the “jetting phenomenon” at the rear of the contact patch from a flow field investigation for the first time, validating the theoretical prediction made by Fackrell [3]. Mears and Dominy [11] later simulated this phenomenon employing computational fluid dynamics (CFD) to show that it can also be predicted numerically, if the computational grids are fine enough to resolve the viscous sublayer in the boundary layer on the wheel surface.

In recent years, many researchers have begun to use a more practical resting rig. Studies have begun to represent the shapes of the contact patch and sidewall more realistically, and rims have detailed interior geometry, such as spokes, a brake disk, and calipers, in conjunction with a brake scoop as used by the F1 teams.

Sprot et al. [12] used a complete assembly of a 50%-scale F1 “solid” wheel to investigate the “through-hub flow pattern” and its effect on the wheel drag, and downstream wheel-wake structure, employing PIV and CFD. They showed that the open and closed configuration of the brake duct inlet scoop affects the wheel drag and the flow field in the wake significantly. They pointed out that the brake duct scoop must therefore be designed so that its “spillage positively interacts with the downstream component of the car”. They also investigated the effect of the exit position of the outboard rim cover, which was used by many F1 teams until 2009, on the wheel drag.

Sprot et al. [13] used a similar assembly but with a “deformable wheel” to measure the sidewall profile of the rotating wheel. They showed that “the sidewall shape is highly sensitive to the aerodynamic performance of the wheel”, highlighting the importance of having deformable wheels with a proper axle height for the wind

tunnel test. The same assembly was used by Emmanoulides [14] to conduct heat transfer analysis using a hot film sensor on the brake disk inside the wheel assembly.

Issakhanian et al. [15] conducted PIV around a 60%-scale F1 wheel assembly with a deformable grooved tire, provided by an F1 team. The test was conducted under the same conditions as experienced by F1 teams to give similar shapes of the contact patch and sidewall. The purpose of their study was to obtain validation data for subsequent CFD work around stationary and rotating wheels conducted by Axerico-Cilies et al. [16,17], and Axerico-Cilies and Iaccarino [18], respectively. The latter study investigated turbulence models by making detailed comparisons with PIV data, to find the best turbulence closure model in unsteady Reynolds-averaged Navier-Stokes (URANS) equations, without employing large eddy simulation (LES), which has a high computational cost although it gives the closest match of flow structures with PIV data.

Flow Interactions between Front Wing and Wheels

The interaction of the front-wing wake and the rotating front wheel has been studied by many researchers in parallel with the above work. These investigations provided insights that are linked to a regulation change in 2009, which involved an increased span of the front wing.

Keller et al. [19] conducted the first study of the interaction of the front wing and rotating isolated wheel both experimentally and numerically. They set up a simplified front corner assembly with a rotating wheel and half of the two-element front wing with different end-plate designs, using a 40%-scale model. They discovered that the wheel drag can be “significantly reduced” by the design of the front-wing endplate, which is attributed to the “strong vortex shed from the bottom edge of the front-wing endplate”.

Diasinos and Gatto [20] conducted velocity measurements using 3D LDA. They used a generic 1:7.5 front corner assembly, which consisted of a rotating cylinder with an inverted single-element airfoil placed in front with a flat plate attached as an endplate. They conducted a thorough survey of the velocity fields in transverse sections at several downstream positions in the wheel wake. They showed that the position of the “primary vortex”, which is generated at the wing tip and the top of the endplate, is strongly affected by the angle of attack and span of the front wing.

Van den Berg and Zhang [21] set up a 50%-scale F1 two-element front wing in front of two rotating isolated solid wheels in a wind tunnel. They made various measurements, such as surface pressure, to calculate the aerodynamic loads, oil flow, PIV, and CFD, while varying the ride height. They found that there is a critical ride height, for which wheel drag switches between high and low values. They attributed the critical height to the position of the top vortex generated by the front wing endplate. Heyder-Bruckner [22] extended that study further by conducting an extensive flow field survey using PIV and CFD to support previous conclusions in more detail.

Car Wake

In the late 2000s, many discussions were held to promote overtaking maneuvers and thus increase the entertainment value of F1; e.g., the centerline downwash generating (CDG) wing was proposed as a candidate wing of a new rule for 2008. The drag reduction system (DRS) was officially legalized by the Federation International de l'Automobile (FIA) in 2011 for this purpose. A few studies investigated the flow field in the complete car wake of an F1-car model, although no PIV work has been published in the open literature.

Wilson et al. [23] studied the wake structure behind a 1/6-scale generic F1 car model on a rolling road, using a five-hole pressure probe. According to the results for the wake structure, they invented a “bluff body wake generator” to reproduce a similar wake pattern with a vortex pair in the downstream at a half-car distance. By situating this wake generator in front, they studied the aerodynamic response of the front wing of a following car model, suggesting “the reduction of the strength of the trailing vortex pair” to promote overtaking.

Watts and Watkins [24] conducted a detailed velocity measurement behind a 30%-scale model in a wind tunnel without a rolling road, using a four-hole pressure probe to investigate the effect of the DRS on the wake structure. They drew a similar conclusion, highlighting the dominance of the “counter rotating vortex pair” in the wake, and stating that the “flap must be trimmed to the streamline” to minimize the strength of the vortex pair, making DRS most effective.

Newbon et al [25] recently conducted thorough investigations into the effect of the car wake on the following car using CFD. To find the most sensitive flow features in the wake of the preceding car, they varied fluid dynamic parameters of the inlet plane, which was sampled from the wake of the single-car computation. They found that such an “upwash flow” in the wake actually seemed beneficial in terms of maintaining the overall downforce level as it moved the wake away from the following car, although it is also a main cause of an appreciable rearward-shift in the aerodynamic balance.

Full-Car Model

Knowles et al [26] conducted a wheel wake measurement in the presence of a car body on the rolling road for the first time. They used the same front wheel assembly as used by Saddington [8], but with the installation of a complete 50%-scale car model provided by an F1 team. They conducted 3D LDA measurements on two transversal sections in the near-wake region of the wheel, and compared the measurements with those made behind the isolated wheel. The results showed that the flow separation on the wheels occurs much earlier than that in the isolated case, and the pattern of the trailing vortices seen behind the isolated wheel disappeared. These are the very first results where actual wheel wake structures around a complete car-model were obtained quantitatively in an experiment.

As seen above, a trend of recent research activities has been the rapid shift to more practical applications rather than fundamental research. Apparently, the technical demands of many researchers are growing because of the rapid advance of measurement and simulation technologies. Many findings reported in the relevant literature are

beneficial to F1 teams in terms of making measurements that are more precise in a wind tunnel, raising the quality of wind tunnel testing in the recent development of F1 aerodynamics. In fact, many F1 teams are interested in these techniques. For example, Ogawa et al. [27] reported a PIV measurement made around a 50%-scale F1 model and used for development. They measured the separation point around the front wheel to validate their CFD results.

However, to the best of the authors' knowledge, there has been no PIV study of actual aerodynamic phenomena in various areas around a complete F1 car. Therefore, the aim of this paper is to first present such results obtained from detailed experimental investigations of an F1 car model in a wind tunnel, using PIV to show how PIV can be used in the daily development loop, and what can be learned from these investigations [28].

BACKGROUND TO F1 AERODYNAMICS

The main focus of this paper is a discussion of actual flow fields around a recent F1 car, according to PIV results. In particular, the flow phenomena are compared before and after a change in FIA technical regulations [29] made in 2009. This chapter first reviews the background of F1 aerodynamics from 1980s, followed by details of the regulation change in 2009.

Overview of Aerodynamics of a Racing Car

After the importance of aerodynamics to the performance of an open wheel racing car was first highlighted (e.g., Katz [30, 31]), the theoretical background of the principle downforce generating devices on the racing car became well-established by the mid1990s. The history of these earlier works was summarized by Katz [32].

Agathangelou and Gascoyne [33] presented a general overview of the F1 aerodynamics based on the actual data of the F1 car at that time. They also highlighted the importance of emerging CFD in terms of advancing the theoretical understanding of complicated flow phenomena around the car. They concluded the report by stating “empirical and numerical methods must coexist ... to allow cross validation of investigative tools and progression of the subject”.

Computer technology has since evolved quickly; commercial CFD codes have become increasingly powerful, which has improved understanding of the complex flow phenomena around a full car at a more detailed level. Consequently, the actual flow phenomena around a recent F1 car have become more complex, and development has become more rapid than in the 1990s. However, the development process in F1 remains the same; both empirical and numerical approaches are needed as CFD simulation still requires validation with measurements made in a wind tunnel, although the degrees to which each approach is employed depend on the resources available to each team.

The major factors that determine the performance of an F1 car are broken down to the chassis, the aerodynamics, the engine, and the tires. Figure 1 summarizes the effect of each of these factors on lap time improvement analysis presented by Ara [34], published in 2009 season. It is seen that improving aerodynamic efficiency (the

downforce to drag ratio) by 0.01, which is a typical gain after a week of wind-tunnel development in F1, is comparable to a weight reduction of 10 kg in terms of the lap time gain. Considering that the car is usually made underweight, as the minimum weight is fixed by a regulation, improving the aerodynamic efficiency is apparently the most effective and possibly the only means to improve car performance during the season. For this reason, research and development of the aerodynamics have been conducted tirelessly throughout the racing season, although the current F1 regulation (introduced for the 2014 season) heavily restricts the amount of inseason wind tunnel testing the teams are allowed to perform. Even the smallest gains are implemented immediately so that the car can be best equipped for the upcoming race events, which take place every couple of weeks during a championship.

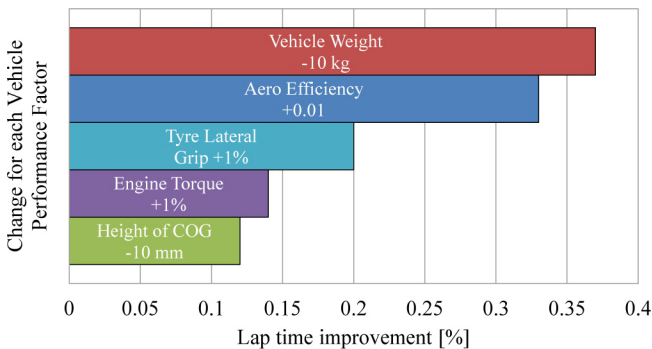


Figure 1. Contributions of nominal performance factors to the improvement of the lap time on average circuits based on the analysis of the 2009 season [34].

Effect of the Regulation Change

The aim of aerodynamic development is to improve key performance indicator (KPI), such as aerodynamic efficiency and stability, by designing a shape of the aerodynamic devices within the geometry specified by the FIA technical regulation (a.k.a., the regulation box), which is subject to change at any time. In fact, there were major regulation changes twice in the first decade of the 2000s, concerning aerodynamics. There are several reasons for changing the rules, such as reducing the cornering speed and promoting overtaking maneuvers. Nevertheless, these changes have always involved reductions in downforce. For example, the total downforce was reduced by 20% initially following the rule change made in 2005, and the subsequent rule change made in 2009 had an even greater impact, reducing the total downforce by 50%. Larsson [35] summarized the initial effect of the latter, according to CFD investigations. However, the recent technology of F1 allows aerodynamicists to quickly recover the loss and finally to exceed the original downforce by the end of the season. The rate of development has increased so dramatically that changes to the regulation now tend to happen more frequently. Figure 2 compares F1 cars before and after the regulation change made in 2009. The appearance of the cars changed appreciably.

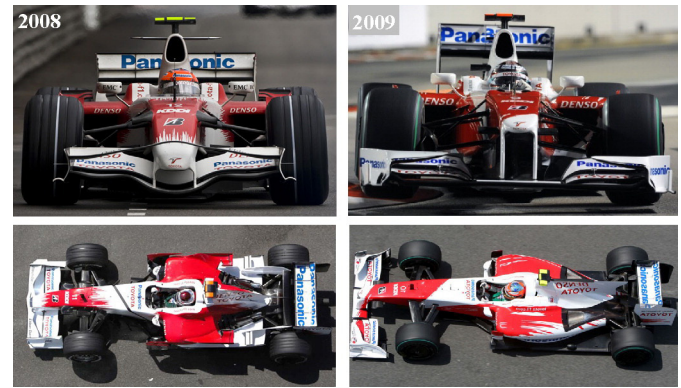


Figure 2. Pictures of F1 machines before (in 2008, left) and after (in 2009, right) the technical regulation change made in 2009.

The following explains these changes in more detail by highlighting important development areas of F1 cars. Although F1 teams are free to develop their cars according to the F1 regulations, the designs of different teams are similar enough to consider a generic F1 car for each year in the following discussion.

The front wing of F1 cars produced about 25% of the total downforce, as reported by Agathangelou and Gascoyne [33], which is the same level as achieved nowadays even though the level of the total downforce and efficiency are much higher than 1990s. While the front wing efficiently produces a large downforce thanks to the ground effect, the outboard area, which features multiple-element flaps and endplates, also plays an important role. The outboard area controls the flows over the front wheels and other downstream components of the car [19, 20, 21]. The trailing vortices, produced by the top and bottom of the endplates and vertical fences underneath the main plane, called strakes, are important structures, as shown in Figure 3. There is an appreciable difference in these flows before and after the regulation change, since it changed the span of the front wing. Consequently the development strategy had to be completely altered: In 2008, the main focus was on the control of the endplate vortices to have a favorable interaction with the wake of the front wheel. In 2009, as the span of the wing was increased to cover the complete width of the car, the endplate vortices travel outboard. Please note that the front wing span was reduced again by 150 mm in the 2014 regulation, and the authors believe that vortices now travel both inboard and outboard of the front wheels, which could be the reason why the outboard area of the front wing of the most recent F1 car seems to add more complexity to the geometry.

Managing the front-wheel wake has been a constant challenge in F1 aerodynamic development since the 1980s, as the rotating wheels create the largest disturbances acting on the car and can thus reduce the effects of downstream aerodynamic devices. To minimize this potentially detrimental effect, the wheel wake must be well managed using aerodynamic devices on the chassis side. This was done using devices called turning vanes (TV) and bargeboards (BB) situated between the front wheel and the sidepod for the 2008 car as in Figure 3. These devices produce strong downwash flow to feed energized air under the floor, as well as expelling the front-wheel wake outboard so that the underfloor flow can be kept undisturbed. Meanwhile, the front-wheel wake of the 2009 car had to be managed differently, since

no such devices existed on the 2009 car, owing to the restricted area in the regulation box. It turns out that the vortices generated by the inboard tips of the front wing flaps, so-called Y250 vortices, became prominent and became a major target of the development; details will be discussed later.

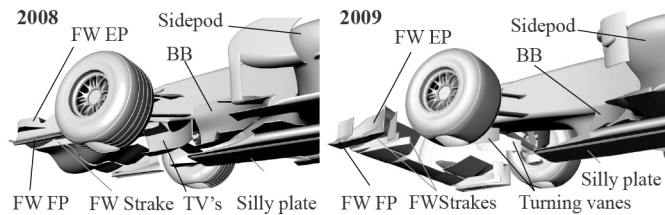


Figure 3. Major aerodynamic devices on the front half of the machines before (left) and after (right) the regulation change in 2009.

The rear wing of F1 cars generally produced about 25% to 30% of the total drag, as reported by Agathangelou and Gascoyne [33]. Over the course of F1 history, many attempts have been made to reduce undesired drag when the car is traveling down straights, and the DRS was officially legalized in 2010 to promote overtaking. The rear wing has also been subjected to major rule changes; in 2009, for example, the rear wing had to be narrower and taller than that in 2008, as shown in Figure 4. The 2009 rear wing is more isolated in freestream, resulting in a higher downforce generation and stronger interaction with the low beam and diffuser.

The diffuser is the most efficient downforce-generating device, and its dimensions were frequently altered following the regulation changes of 2005 and 2009, restricting the design flexibility. In 2009 in particular, a double-deck diffuser received much attention and was standard in F1 until it was banned in 2011. The diffuser is usually designed to have the maximum allowable expansion such that it generates large negative pressure under the expanded channel in order to collect as much of available mass flow as possible. Several vertical fences are used to prevent stalling, which produces strong longitudinal vortices, also inducing negative pressure. The upwash from the rear end of the car was much steeper in 2009, owing to the combined effect of the taller rear wing and the double-deck diffuser that had a steeper expansion. The outboard shape of the diffuser is also important in preventing disturbances of the rear-wheel wakes from entering inboard.

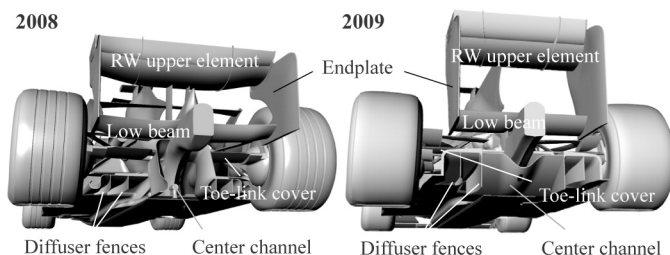


Figure 4. Major aerodynamic devices on the rear half of machines before (left) and after (right) the regulation change made in 2009.

EXPERIMENTAL APPARATUS

Wind tunnel Facility

The wind tunnels used in this study are owned by Toyota Motorsport GmbH (TMG) in Cologne, Germany. TMG used to be a base factory for the Panasonic Toyota Racing Team, until they withdrew from the F1 championship at the end of 2009. The facility has twin wind tunnels, both of which were operated 24 hours a day, seven days a week at peak time, until the agreement of the Formula One Teams' Association (FOTA, disbanded in 2014) to limit the "wind-on" hours became effective at the beginning of 2009 season.

Test Section

The schematics of the test section are illustrated in Figure 5. Each wind tunnel is a closed-loop (Göttingen) type with a nozzle exit that is 4.1 m wide and 3.7 m tall. The 15-m-long test section can be used with either a slotted-wall or semi-open configuration. The full-belt rolling-road system is equipped with a proper boundary layer treatment in front of it. Aerodynamic loads of the scale model are measured using an external balance on the roof and under-belt load cells just below the wheels. Figure 6 shows that a scale model can be suspended by a strut equipped with a vertical traverse system inside a carbon fairing. One of the wind tunnels is capable of testing a full-scale car, where the downforce can be measured via the under-belt load cells below the wheels, while the drag forces are measured via the load cells mounted inside the support towers of the side struts. The other tunnel is capable of conducting exhaust blowing tests, which were essential for the development of the exhaust blown diffuser in 2010. The detailed specifications are given in Table 1. The maximum wind speed achieved in conjunction with the rolling road is 70 m/s. All the measurements reported in this paper were conducted at a wind speed of 50 m/s, corresponding to a Reynolds number of 5.9×10^6 for the length of a 60%-scale 2009 car model.

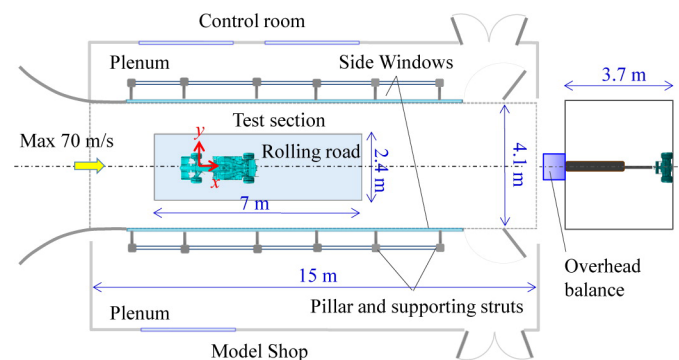


Figure 5. Layout and dimensions of the test section.

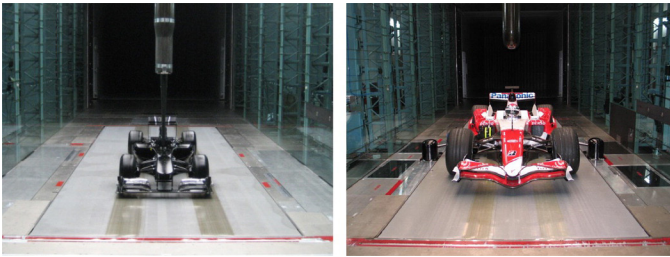


Figure 6. Views of the test section of TMG wind tunnel 1 with the 60%-scale model (left) and the full-scale car (right).

Table 1. Specifications of the TMG wind tunnel.

Operation since	March, 2002
Duct size	11.65 m (H) × 25.0 m (W) × 68.0 m (L)
Duct length	15.0 m (W) × 59.07 m (L) (Total line length 148.14 m)
Contraction ratio	6.7:1
Test section size	3.7 m (H) × 4.1 m (W) × 15 m (L)
Test section type	Closed (slotted for full scale)
Fan dimensions/power	Coaxial fan (diameter 6.3 m)/2.4 MW
Temperature control	22 ± 1°C
Rolling road dimensions	2.4 m (W) × 7.0 m (L) in steel (MTS)
Wind/Rolling road speed	max 70 m/s (nominally 50 – 60 m/s)
Boundary layer treatment	Double suction at scoop, perforated plate, and tangential blowing (WT1) ; $u/V = 0.98$ up to $z = 10$ mm at model nose
Turbulence intensity	Generally between 0.04% and 0.22%
Load cells	Six-component external (Aerotech)

Scale Model

Initially, a 50%-scale model with a wheel supporting system (wheel arms and towers) was used. This model was eventually replaced by a 60% model with no wheel supporting system. We call the former the “wheel-off” system, as the wheel assemblies are detached from the model, and the latter the “wheel-on” system. Switching to the wheel-on system was a particularly important step forward in minimizing the blockage effect as the model size increased to 60%. This change in size was motivated by the 2009 regulation change that resulted in losses in downforce as mentioned earlier. The model was enlarged to better utilize the measurement capacity, which improved the measurement resolution and correlation with the full-scale phenomena as a result, owing to the higher Reynolds number.

Figure 7 shows the 60% wind tunnel model of the 2008 F1 car. The model dimensions are width of 2.78 m, length of 1.08 m, and height of 0.57 m. Usually, a model consists of more than 500 parts built around a steel skeleton that encloses several essential components: a model motion system, mechanical components such as the active suspension control system that applies preloads to the deformable tire supplied by the tire manufacturer, the front wheel steering system for yaw/steer tests, and pressure sensor modules that measure the surface pressure under the floor and radiators. Different materials are used for the parts that reproduce the actual car shape. The areas on which high aerodynamic loads were expected, such as the front and rear wings,

were made from machined aluminum, steel, or carbon. Most of the other parts were made through rapid prototyping with various materials, such as ceramic, resin, and aluminum powder.

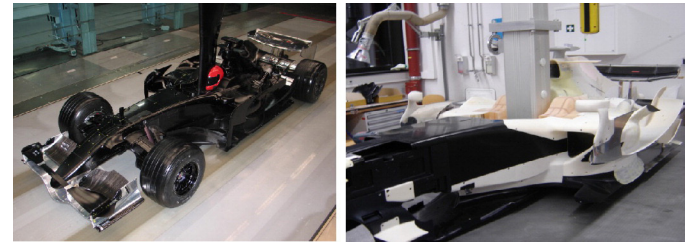


Figure 7. A 60%-scale wind-tunnel model of the 2008 F1 car (left) and the same model during preparation in the workshop (right).

PIV System

As described earlier, the task of the aerodynamicists is to improve KPI's during the test session, usually by adjusting and replacing parametrically designed parts on the model to build a huge database. Because of the large number of possible combinations, aerodynamicists used to need to conduct as many wind-tunnel runs as possible to improve the chance of finding new optimizations. Results were analyzed purely using the numbers of KPI, which refer to aerodynamic load measurements. Development was also monitored using these numbers. In other words, experiments were conducted systematically to accomplish the goal of achieving the target KPI numbers, and understanding or confirming the actual flow structure produced by each configuration was usually a secondary task in the wind tunnel. Instead, the work conducted to understand the flow was solely based on CFD simulation.

To a certain extent, this mass-production-style testing procedure functioned well in terms of meeting the KPI target. Consequently, it took some time to realize that there were discrepancies between the results from the wind tunnels and CFD simulation, which required immediate rectification at a certain stage. With this motivation, it was decided to equip both wind tunnels with a PIV system. This system has already been used for several years, not only to validate results of CFD simulation but also to understand the “actual flow phenomena” in the wind tunnel. This section explains how the system was developed to allow use in the development loop.

Overview of the Permanently-Installed System

As the access to the test section from the plenum was limited owing to the test-section structures that hold the side glass, it was not possible to simply install a commercially available PIV system. Instead, a tailor-made system was required. Applying PIV in the wind tunnel with a closed test section has become more common in recent years (e.g., Passmore [36]), but was not common in the early 2000s. When the first demonstration was made in 2003 at TMG, some literature on open test-section facilities was available, such as the study of Cogotti and De Gregorio [37], but to the best of the authors' knowledge, there were no previous examples of PIV systems used in automotive wind tunnels with closed test sections at that time. Thus, a new system had to be built from scratch, through trial and error.