Investigation and Analysis on Racing Car Front Wings

Jin-Jing TAN\textsuperscript{1,a}, Peter MYLER\textsuperscript{1,b}, Wen-An TAN\textsuperscript{2,c,*}

\textsuperscript{1}Department of Engineering, Sports and Sciences, University of Bolton, BL3 5AB, Bolton, U.K
\textsuperscript{2}School of Computer and Information, Shanghai Second Polytechnic University, China
\textsuperscript{a}Jt2put@bolton.ac.uk, \textsuperscript{b}P.Myler@bolton.ac.uk, \textsuperscript{c}watan@sspu.edu.cn

*Corresponding author

Keywords: Aerodynamic, CFD, SolidWorks\textsuperscript{TM}, C15 wind tunnel.

Abstract. A great number of studies have been done on aerodynamic test for the racing car. Due to critical competition among the racing car teams, the results of the studies will only been published when they are obsolete. This study was conducted by using the Computational Fluid Dynamics (CFD) utilising SolidWorks\textsuperscript{TM} to perform simulations about the airflow on the front and rear wings of a racing car with different angles of attack. The analysis generates the theoretical Lift force, Down force, and Drag force for a racing car wing. Consequently, at higher velocity, the down force and interrelated induced drag increases. The maximum speed on a straight part is thus reduced due to the increase in induced drag. Finally, tests from the C15 wind tunnel also show similar trends to those derived from the simulations. Compared with results of Model geometry and wind tunnel tested, it is shown that an angle between -10° and -20° below the horizontal indicates the stalling conditions.

Introduction

The technology of Formula 1 has, however, almost completed the transformation from the “black art” of the 1950s and 1960s to the science of the 2000s [1]. Computer science enables every solution to be evaluated, with ultimate convergence on the solution. The way a race car works is nearly understood which can be modified in different models by using the software.

There are five main parameters to determine the performance potential of a race car: power, weight, tire grip (coefficient of friction), drag, and lift/down-force. Formula 1 technology is about how to maximize these five parameters as much of the time as possible. With the hundreds of technologies involved Formula 1 has become so competitive and the stakes have become so high.

Now, the computer can provide an easy way to understand though simulation and modelling, and to optimize highly complex systems. One of the most perplexing aspects of evolution is that the benefits of the aerodynamic down force is appreciated, as shown in Figure 1.

In recent years aerodynamic downforce has proven to be of far greater value to reduce lap-times than lower drag on high powered race cars [2]. In some forms of racing, however, fuel economy has recently gotten more emphasis, so the compromise between drag and downforce is become harder to make.

Two primary sets of aerodynamic forces affect stability in race cars – the ratio of downforce from front to rear and the ratio of lateral gust force from front to rear. According to Van Valkenburgh, 2000 [3], downforce stability is one of the major importance because of the high forces available and the fact that they change drastically with speed.
As mentioned in the literature review, the most of published studies in sports industry are based on the obsolete technologies in shapes and equipment. In this work the subject is a study of the aerodynamic characteristics of the front wing based on a real model of racing car in University of Bolton, as shown in Figure 2. Though the CFD and C15 wind tunnel test, the angle of attack (AOA), lift and drag characteristics will be studied.

**Methodology**

There are two main methods applied in this study, one is Computational fluid dynamics (CFD), the other is the Wind Tunnel method.

Computational fluid dynamics (CFD), makes use of the applied mathematics, physics and computational software to address the issues that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions.

The experiments described are performed in the C15 Wind Tunnel at the University of Bolton. Besides, also discuss the wind tunnel method, apparatus of C15 wind tunnel in the lab.

**The Computer Model Simulation-Front wing**

The front wing of an F1 car is typically a two-element high-lift wing with an adjustable flap to vary the downforce obtained from the wing, and hence balance the car with different rear-wing configuration. The front wings runs with a strong ground effect.

Due to the fact that the test of the car operates in the wake generated by this wing, and that this wake severely affects in particular the undertray and diffuser of the car. It is important to tune the wake profile of the front wing.

The front wing produces 25-30% of the total downforce of the racing car. The amount of downforce produced is very dependent on the front ground clearance, and it is this fact ha produces most of the aero dynamic stability problem. As the front of the car moves lower, the front wing produces more downforce due to increased “ground effect” and more effective sealing of the endplate. This in itself produces a forward shift in balance, but also the increased upwash from the front wing reduces the rear shift in balance, increasing the balance shift.

A two-dimensional symmetrical aerofoil model template is shown in Figure 3, which copies the shape size from the real front wing in Figure 2. At the end of the trailing edge of a wing also called Gurney Flap.

In order to simplify the simulation, in this work we just neglected the Gurney Flap.
Solidworks. In this design only one software model was used, which is SolidWorks Education Edition 2013-2014. The aerodynamic analyse was conducted by using SolidWorksTM CFD. SolidWorksTM is a 3D mechanical CAD (computer-aided design) application system running on Microsoft Windows and developed by DassaultSystèmes SolidWorks [4]. In Fluid Flow Simulation, we can use SolidWorks to discover how flow simulation can uncover potential design flaws without physical prototypes.

Digital Model. We built a computer model in SolidWorksTM by Boss-Extrude. The process of building model includes some Boss-Extrudes through the shape mentioned in Figure 4. And it also demonstrates 3 views of the front wing model in CAD. The size is: Length 142mm, Width 86.48mm and the maximum thickness 11.2 mm. The width and thickness is based on a 3.28:1 contraction ratio of the real wing while the length is a controllable value in CFD simulation and without the effect to generate Lift force, Down force, Drag in C15 wind tunnel. Moreover, the Isometric view of the wing is highlight in grey.

Wind Tunnel Method

The basic idea of test experiment within a wind tunnel is simple: Instead of chasing a flying airplane or a moving car with all the measuring instrumentations, the test model and supporting instruments stays stationary while the air moves relative to it. Wind tunnel controls test conditions change to be suitable and, in principle, simulation of external atmospheric condition [5]

Facility Introduction Hardware. The Armfield C15 is a small benchtop wind tunnel with a transparent working section. A wide range of accessories and instrumentation are available, allowing the comprehensive study of subsonic aerodynamics [6].

Dimensions of C15 Wind Tunnel. Height: 0.700m; Width: 2.250m; Depth: 0.460m

Feature and Characteristics. There is a honeycomb flow straightener incorporated at the inlet. Moreover a 9.4:1 contraction ratio keeps well developed airflow through the working section. The basic wind tunnel is supplied with a USB interface, of which can connected to the PC to provide the data of air velocity, static pressure, lift/down and drag force. The working section incorporates a simple technique for flow analysis. A simple adjustment arrangement allows the Rotation of the racing car wing and position to be easily changed.
**Facility Introduction – Software.** Procedure of Equipment Set-Up. 
The apparatus must be set-up in accordance with the Help Text:
The C15 sensor cable should be connected to the SENSORS socket with the IFD USB interface.
The manometer should be connected to the tapping points so that the numbers on the labels correspond.
The USB port on the IFD7 should be connected to the PC after the application software is installed.

**Results**

**CFD Results**

Computer Aided Simulation is a unique tool that is extremely helpful and cost-effective in preparing an updating plan. In fact, computer aided simulation is the only feasible method for analyzing a proposed complex automated installation in a timely manner that is understandable to executives who are not trained in simulation methodology.

Assumptions:
A schematic of each angle of attack is shown in Figure 6, from -5° to 20°, with 5° one step included 5 steps. The highlighted blue one is the original front wing in AOA(angle of attack) 0°. And the numerical model has been generated through CFD simulation in twice with different velocity.
1. According to the various sources, the velocity of the racing car between 0 to 200mph (200mph=89.408m/s). Therefore the first velocity of air flow in X axis generated by CFD simulation is 60m/s as while as the second is 80m/s.
2. Generate a new case study to create a new configuration for the case study for the time being.
3. The analysis type is external, and we can select the air flow pre-defined as the project fluids in gases type.
4. The value of the flow type is laminar and turbulent.
5. The initial and ambient conditions is demonstrated in Figure 6.
6. Set computational domain in a suitable area.

Follow the assumption, the CFD results of different AOA are shown in appendix.

**Wind Tunnel Results**

The results gathered from the C15 wind tunnel test with different velocity of air flow, from Rotation -30° to Rotation 0° best results are shown in tables.

![Figure 6. Schematic of Each Angle of Attack.](image)
The general conditions of the C15 wind tunnel test are:
(1) Ambient Temperature 18 °C.
(2) Density of Air [kg/m³] 1.211.
(3) Rotation -30°, -20°, -10° and 0°.
Follow the assumption, the CFD results of different AOA are shown in appendix.

| Fan Speed [%] | Lift Force L [N] | Drag Force D [N] | Air Velocity v [m/s] |
|---------------|-----------------|-----------------|---------------------|
| 0             | 0.00            | 0.00            | 2.5                 |
| 10            | 0.00            | 0.00            | 2.9                 |
| 20            | 0.02            | 0.00            | 6.3                 |
| 30            | 0.15            | 0.04            | 9.1                 |
| 40            | 0.16            | 0.22            | 12.1                |
| 50            | 0.12            | 0.25            | 14.6                |
| 60            | -0.05           | 0.34            | 16.8                |
| 70            | -0.13           | 0.46            | 19.1                |
| 80            | -0.24           | 0.53            | 22.2                |

Discussion

CFD Results

Figures listed in the appendix summarize the pressure condition relationships with different AOAs. When the fan speed is at the high velocity of 60m/s, air flow will start to separate the trailing edge from the aerofoil surface. If we ignore the aerofoil interference of Gurney Flap, the numerical model is easy to be generated.

The results can be summarized as follows:

(1) The air flow accelerates though the downside of the numerical front wing model, creating a low-pressure area marked in blue.

(2) The air flow accelerates though the downside of the numerical front wing model, creating a low-pressure area marked in blue.

5° AOA: The Figure 7 illustrate the pressure for the front wing operating in a smooth airflow in AOA 5°. Obviously, the airflow has been separated from two parts. One is above while the other is close to the up surface. According to the Bernoulli’s Theorem, the negative lift is very small, when the upside pressure is similar to the downside pressure.

According to the Kutta condition mentioned, the flow will smoothly leave a sharp trailing edge, as shown in Figure 7.

0° AOA: From Figure 8, we can see the air flow thought the front wing smoothly. It means that the lift and drag keep the balance. The high pressure area is in front of the wing marked in yellow colour. In addition, the flow pressure in the trailing edge is higher than the middle area.

-5° AOA: Figure 9 demonstrates that the vortex occurred in the side of model in -5° AOA. But the size of vortex is relatively small. The downside pressure is smaller than the upside.
pressure, which produced the down force for the wing. The main high pressure area is still in front of wing.

-10° AOA: Figure 10 shows a greater pressure difference between the upper surface and the lower surface for the aerofoil, thus down force (negative lift) became more and more obvious. Additionally, the side vortex can be seen clearly.

-15° AOA: Figure 11 summarizes the pressure result as -15° AOA, the region of high pressure begins to separate from the aerofoil surface to the trailing edge. Down force (negative lift) became much heavier. Pressure of yellow area in front of wing is 102616.56 Pa while the pressure of the blue area below the aerofoil is 100610.86 Pa. Additionally, the side vortex is larger than that in -10° AOA.

-20° AOA: According to the theory, 19 degree could be approach to the stall condition. But the results from the Figure 12 are not similar to the expected. The vortex became the largest in -20° AOA compared with rest of the AOAs.

But the different result can be gathered from the second CFD simulation when changing the velocity of air flow in X axis (horizontal) to 80m/s.

Figure 13 demonstrates the flow trajectories at the 5° AOA (80m/s) from the top view. A vortex can be observed clearly on the top of the upper surface and in the middle of the numerical wing model in Figure 13. The air flow though the upper surface follows the round trajectories, then leave the sharp trailing edge smoothly.

The upper surface pressure is 100458.57 Pa, and it is very close to the lower surface pressure 100559.27 Pa.

Wind Tunnel Results

Compared with the 2 Tables above, a conclusion can be made that with the increasing of the air flow velocity, the drag force is rising gradually. In addition the negative Lift occurs when the air flow velocity reach to 16.8m/s in Table 2, which is confirmed that only enough speed can create effect down force (negative lift) in front wing model.

To summarize, it can be seen that at -10° AOA the drag force is very similar to that at 20° AOA in table 3. Moreover, the lift force is always larger than the drag force. The effective negative lift can keep the wing reach to a stable condition. In other words, racing car can move faster. The stall condition is likely to happen between -10° AOA and -20° AOA.

Conclusions

In CFD, Experiment Results Are As Follows

(1) Kutta condition can be confirmed at the 5° AOA when the velocity of air flow in X axis is 60m/s
(2). At the 10° AOA, separation occurs obviously, which is due to the velocity and pressure distributions.
At the 15° AOA, the up surface pressure is far larger than the downside pressure, which means the downforce (negative lift) reaches its peak.

In C15 Wing Tunnel, Experiment Results Are As Follows

At -10° AOA the drag force is very similar to that at 20° AOA in table 3. Moreover, the lift force is always larger than the drag force.
The effective negative lift can keep the wing reach to a stable condition. In other words, racing car can run faster.
The stall is likely to happen between -10° AOA and -20° AOA, which is close to the CFD results.

**Optimum Angle**

Compared with the CFD results and the result from C15 wind tunnel tests, a conclusion is that the optimum angle of attack is an angle between -10° and -20° below the horizontal for the front wing to reach stall condition. As well as Vortex could be occurred in any angle of attack above 5° of the horizontal. Due to the lift and drag, the down force (negative lift) in front wing can produce a high speed to the Racing cars.

**Appendix**

The CFD results of different AOAs are shown in following figures below:
Acknowledgement

This research was financially supported by the National Science Foundation. The paper is supported in part by the National Natural Science Foundation of China under Grant No. 61272036. We must thank very much for the following people who have provided advance technical supports, they are Karl Gregory, Garry Owen, Dave Lewtas, Tim Ward, and Erping Zhou.

References

[1] W. Kieffer, S. Moujaes, N. Armbya, 2005. CFD study of section characteristics of Formula Mazda race car. Mathematical and Computer Modelling 43 (2006) 1275–1287, 14 March.

[2] Houghton, E.L; Carpenter, P.W, 2003. Aerodynamics for engineering students. Oxford: Butterworth Heinemann.

[3] Van Valkenburgh, P., 2000. Race car engineering & mechanics. Seal Beach: P. Van Valkenburgh.

[4] Solidworks, 2014. Solidworks.co.uk. [Online] Available at: http://www.solidworks.co.uk/sw/3d-cad-whitepapers.htm [Accessed 10 2 2015].

[5] Katz, J., 1995. Race car aerodynamics: designing for speed. Cambridge: Robert Bentley.

[6] Armfield Ltd., n.d. Computer Controlled Subsonic Wind Tunnel. [Online] Available at: http://discoverarmfield.com/en/products/view/c15/computer-controlled-subsonic-wind-tunnel [Accessed 15 5 2015].