

Racecar Front Wing Aerodynamics

Wael A. Mokhtar and Jonathan Lane

Lake Superior State University

Copyright © 2008 SAE International

ABSTRACT

A numerical study of a racecar front wing is presented. The focus of the study is to investigate the aerodynamics characteristics of a wing operating in a small ground clearance. A finite-span wing with a symmetric airfoil section is used. It was found that as the wing gets closer to the ground it generates more downforce and more drag. As the wing gets very close to the ground, the downforce reaches a maximum value and after that the wing generated less downforce as it gets closer to the ground. The drag force follows a similar trend of dependency on the ground clearance. The lower surface of the wing and the ground form a convergent-divergent-nozzle shape that is responsible of all the changes in the generated forces. For very small ground clearances, the boundary layers developed on the wing lower surface and the ground get closer to each other, decrease the airflow below the wing and ultimately decrease the generated downforce significantly. The study was preformed using the CFD package STAR CCM+, developed by CD Adapco.

INTRODUCTION

Racecar designers use several tools to generate downforce. The objective of this force is to have better wheel traction during the acceleration. Also the downforce allows racecars to take corners at higher speeds. Wings have been one of the effective tools. They are installed in the rear and the front of the vehicle. For each race, organizers put specific rules for the use of wings in terms of number of elements, relative size to the vehicle, clearance from the ground and the vehicle, etc.

Aerodynamics of wings has been an active topic of research for more than one hundred years since the successful flight by the Wright brothers. This has been very much well developed and well established for aircraft applications. The main difference in using wings for racing cars is the existence of the ground near to the wing, which significantly affects the airflow around the wing. The interaction between the wing and the vehicle body and small aspect ratio of the wing are also two difference between wings used for airplanes and racecars as report by Katz [1][2].

Several experimental and numerical studies exist in the literature. They can be categories into two main groups. The first group studied the wing with a ground effect as a simplified two-dimensional flow around an airfoil. Both single element and multi-element airfoil sections were used in the studies. As examples for single element airfoils, Ranzenbach et al. used NACA0015 and NACA4412 to study the ground effect [3][4] and Zhang et al. used LS(1)-0413 to study small ground clearance [5]. For multi-element airfoils, Jasinski et al. used UIUC700, a two-element airfoil section, in an experimental study [6]. Ranzenbach et al also used a two-element airfoil NACA 63₂-215 Mod B [7]. A good review can be found in the work presented by Coulliette and Plotkin [8]. The parameters studied in this two-dimensional flow approach include the effect of the airfoil thickness, camber, and angle of attack for an airfoil with ground effect.

The second group studied the three-dimensional flow around a wing with ground effect. Both experimental and numerical methods were used. Katz used a panel method to study a wing with ground effect for open-wheel racecars [9][10]. A two-element finite-span wing was studied by Zhang and Zerihaan [11]. Wiedemann investigated the effect of using a fixed or a moving ground when testing a wing with ground effect [12]. Katz and Dykstra studied low aspect ratio wings for Indy car rear wing [13]. Gogal et al presented a numerical study for a wing with end plates during yaw conditions [14]. His study was for rear wing with no ground effect included. The main focus of the three-dimensional study of a wing with ground effects is to investigate the effects on the flow structure round the actual wing such as the wake roll up and the wing-tip vortices.

PRESENT WORK

The current author previously presented numerical studies for low Reynolds number flow around wings with and without ground effect [15][16][17]. The first study focused on using low Reynolds number wings for aeronautical applications such as UAV's. In the second study, a high-lift airfoil was used to study the ground effect for racecar application. The study included ground clearance from 20 to 120% chord length. It also included the effect of the angle of attack and Reynolds number.

The study showed that both the downforce and the drag increased with decreasing the ground clearance for the studied range. It also showed that the Reynolds number has nearly no effect on the generated forces. This study was extended to three-dimensional flow around wing with ground effect [17]. It included for ground clearances ranging from 20 to 250% chord length for a finite span wing with and without end plates. It was found that, similar to the two-dimensional study, both the down force and the drag increase with decreasing the ground clearance for the studied range.

In the present work, the previous study is extended to include small ground clearances (less than 20% chord length). Figure 1 shows the symmetric (NACA0012) finite-span wing used in the study. Table 1 shows the studied range of ground clearance. H is the vertical distance from the leading edge to the ground; c is the wing chord length, α is the angle of attack, and Re is the Reynolds number based on the wing chord length. Below is the definition of Reynolds number:

$$Re = \frac{\rho U_{\infty} c}{\mu} \quad (1)$$

Where ρ and μ are the air density and viscosity respectively. U_{∞} is the free stream velocity (racecar relative speed). The CFD code STAR CCM+ , developed by CD Adapco, is used in the study.

Table 1: Studied parameters

Angle of attack (α)	6°
Wing aspect ratio	6
Reynolds number (Re)	1.5×10^6
Ground clearance (H/c)	0.05 to 2.0 with a step of 0.01

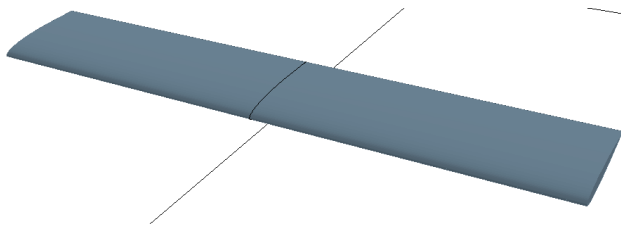


Figure 1: Three-dimensional view of the finite-span wing (NACA 0012) used in the study.

A low angle of attack ($\alpha = 6^{\circ}$) was selected to avoid the development of separation regions that start to appear near to stall condition around 12° for NACA0012. The Reynolds number ($Re = 1.5 \times 10^6$) was chosen to represent an average value for the operating conditions for a racecar wing. Also it was shown in the previous studies that the overall performance of the wing is not significantly affect by Reynolds number. Both values are kept constant during the study to focus on the ground clearance.

COMPUTATIONAL METHOD

The present study is done using STAR CCM+ code. The code is a computational finite volume method for structured and unstructured grids developed by CD-Adapco Inc. In the current study, a segregated flow scheme is used to solve the three dimensional Reynolds Averaged Navier Stocks Equations (RANS). The code uses AMG SIMPLE implicit solver. Gauss-Seidel relaxation scheme was used in solving both the velocity and the pressure. The $k-\epsilon$ two-equation turbulence model is used.

COMPUTATIONAL DOMAIN – The left, right and upper boundaries of the computational domain are ten times the chord length upstream, downstream and above the wing respectively. A half wing is used in the study with vertical symmetric plane defined at the wing mid span to save the computation time. The half domain extends ten times the chord length in the span-wise direction. A polyhedral unstructured grid is generated for each case. The domain is divided into near field and far field regions to allow more grid clustering near to the wing surface and the ground. Figure 3 shows a close up near to the wing for one of the grids used in the study. About half a million cells are used in the numerical simulation.

BOUNDARY CONDITIONS – Wall boundary conditions are applied on both the wing surface and the ground. No-slip condition is applied to the wing surface ($U=V=0$). Also a no-slip boundary condition is used for the moving ground with zero relative velocity to the moving air ($U=U_{\infty}$, $V=0$). The symmetric boundary condition is applied on the symmetric plane. The far field boundary condition is used to defined the rest of the domain boundaries

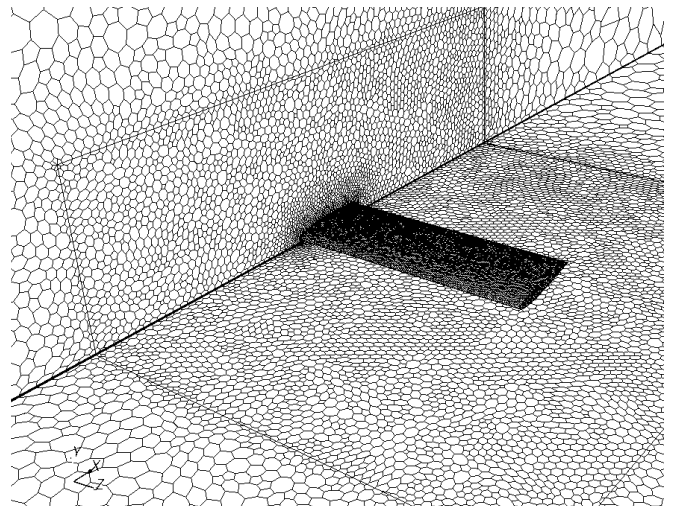


Figure 2: A sample of the unstructured grid used in the study, $H/c=0.05$.

RESULTS AND DISCUSSION

Since the main purpose of the wing is to generate downforce, the overall effect of the ground clearance

the aerodynamics forces will be discussed first. Then the interpretation will be expanded in the following sections to focus on several aspects of the flow structures around the wing and the interaction with ground. The forces will be presented as lift and drag coefficients. They are defined as follow:

$$C_L = \frac{L}{\frac{1}{2} \rho U_{\infty}^2 s} \quad (2)$$

$$C_D = \frac{D}{\frac{1}{2} \rho U_{\infty}^2 s} \quad (3)$$

Where, L and D are the lift (negative down force) and the drag respectively. s is the wing reference area.

AERODYNAMIC FORCES – Figure 3 shows the effect of the ground clearance on the generated lift (negative downforce). Within the studied range, for ground clearance higher than 10% ($H/c > 0.1$) the generated downforce is decreasing with increasing the ground clearance. This trend of dependency is expected to exist for higher values of ground clearance until the wing reaches the free flying conditions (no ground effect) as was reported by Mokhtar [16][17]. However, for ground clearance less than 10%, the generated downforce decreases significantly as the ground clearance decreases. For example, the magnitude of the downforce at ground clearance $H/c = 0.05$ is 28% of its value at $H/c=0.1$. The reduction rate also increases significantly as the ground clearance decreases. 3.8% reduction is observed when H/c is reduced from 0.1 to 0.09 while it is 57% when H/c is reduced from 0.06 to 0.05. It is clear that this range of ground clearance has a different flow structure around the wing and the ground effect is creating a stall-like condition, which is not expected for a NACA0012 wing at this low angle of attack ($\alpha = 6^\circ$) and Reynolds number ($Re = 1.5 \times 10^6$).

Figure 4 shows the effect of the ground clearance on the generated drag force. The trend of dependency is somewhat similar to the downforce shown in Figure 3. The drag increases with nearly a constant rate as the ground clearance decreases. The boarder value of this increase seems to be $H/c = 0.08$. Then the drag decreases with a higher rate as the ground clearance decreases below $H/c = 0.08$. The reduction rate of the drag is lower than the down force one for this range. The generated drag at $H/c = 0.05$ is about 83% its value at $H/c = 0.08$.

Figure 5 shows the change of the lift to drag ratio with the ground clearance. As the H/c increases the lift to drag ratio increases. This indicates that the downforce is more sensitive to the ground clearance than the drag force. The rate of the increase is higher for ground cleance less than 10% ($H/c < 0.1$).

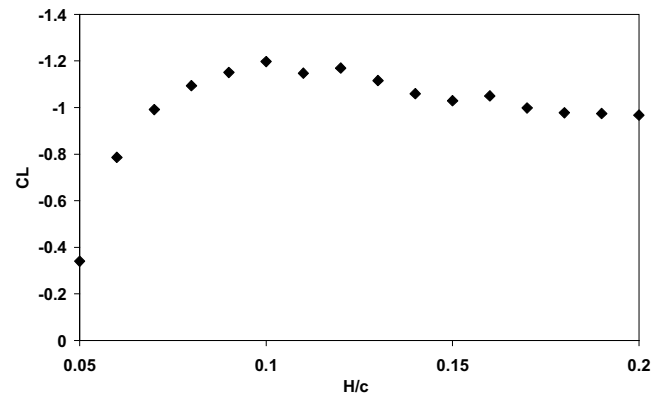


Figure 3: Effect of the ground clearance on the lift force coefficient

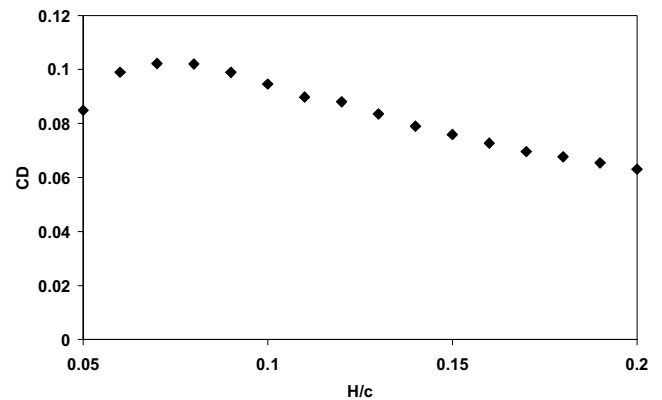


Figure 4: Effect of the ground clearance on the drag force coefficient.

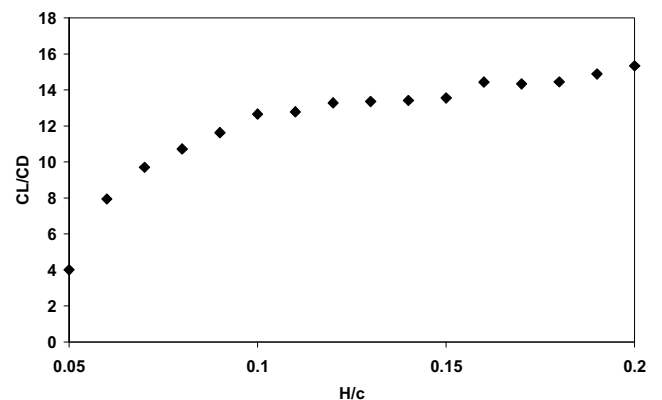


Figure 5: Effect of the ground clearance on lift to drag ratio.

Comparing Figures 3 and 4, as the ground clearance decreases from 10% to 8% ($H/c = 0.1$ to 0.08) the downforce decreases while the drag increases. This behavior supports the assumption of a stall-like condition where the generated adverse pressure gradient on the lower surface of the wing causes a massive flow separation. This separation decreases the downforce and increases the drag force. This condition is not expected for this range of angle of attack and Reynolds number. It seems that the existence of the ground near to the lower wing surface, the profile of the surface, and

angle of attack create a “diffuser-shape” under the wing and increase the adverse pressure gradient.

For ground clearance less than 8% ($H/c < 0.08$), both the drag and downforce decrease when H/c is decreased. This form of dependency suggests that the clearance between the wing and the ground have reached a value where the boundary layer developed on the moving ground started to affect the air flow rate under the wing. Further analysis of the airflow is presented in the following sections.

AIRFLOW AROUND THE WING – For a typical NACA0012 finite-span wing with the same studied flow conditions (Reynolds number $Re = 1.5 \times 10^6$), airflow will be attached to both surfaces. No downforce will be generated at zero angle of attack due to the symmetry of the wing profile (zero camber). As the angle of attack increases, positive pressure is created on the upper surface due to the air deceleration. While the air acceleration on the lower surface creates negative pressure distribution. The net vertical force is the downforce while the streamwise component will contribute to the generation of the drag force. The rest of the drag force comes from friction between the air the wing surface and as a secondary product from the wingtip vortices (induced drag). As the angle of attack increases both the downforce and the drag increase till the wing reaches a maximum value of downforce. Beyond this point, as the angle of attack increases the downforce decreases and the drag increases. This condition is known as wing stall where a massive separation occurs on the lower surface due to the increased adverse pressure gradient at this high angle of attack. For NACA 0012 finite-span wing stall usually happens at angle of attack higher than 12° .

The existence of a moving ground near to the lower surface of the wing will change the characteristics of the airflow around the wing and ultimately affect the generated down and drag forces. Figures 6 through 21 show the streamlines on a streamwise plane passing through the wing mid-span for ground clearances $H/c = 0.05$ to 0.2 . The overall characteristics of the flow around the wing are nearly the same for all the cases with some changes in the region under the wing. The airflow is attached to the wing upper surface for all the cases. The location of the stagnation point is near to the leading edge. For ground clearances higher than 10% ($H/c > 0.1$), the location of the stagnation point is on the upper surface near to the leading edge. This creates positive pressure on the upper surface and slightly increases the generated downforce. As the ground clearance is decreased below 10%, the stagnation point moves toward the lower surface of the wing and acts in the opposite direction to the downforce. This will slightly decrease the generated downforce for this range of ground clearance. Beside the stagnation point location, no other major differences can be observed on the wing upper surface. The stagnation point location may slightly affect the generated force. However, it cannot cause all

the significant downforce reduction shown in Figure 3 for ground clearance less than 10%.

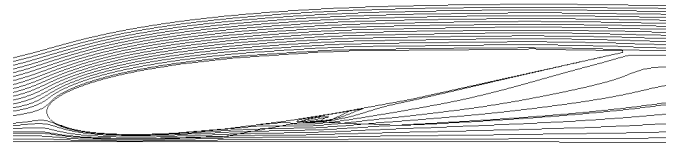


Figure 6: Streamlines on a streamwise plane located at the wing mid-span for $H/c = 0.05$.

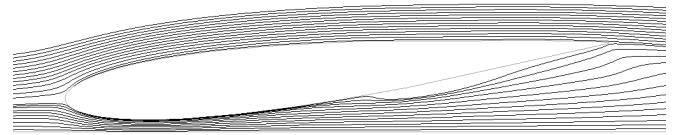


Figure 7: Streamlines on a streamwise plane located at the wing mid-span for $H/c = 0.06$.

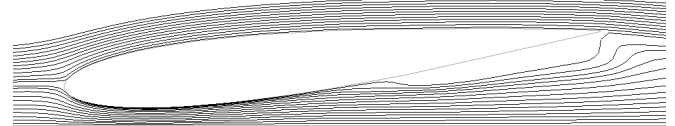


Figure 8: Streamlines on a streamwise plane located at the wing mid-span for $H/c = 0.07$.



Figure 9: Streamlines on a streamwise plane located at the wing mid-span for $H/c = 0.08$.

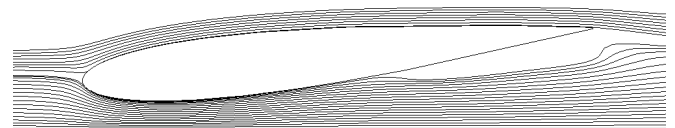


Figure 10: Streamlines on a streamwise plane located at the wing mid-span for $H/c = 0.09$.

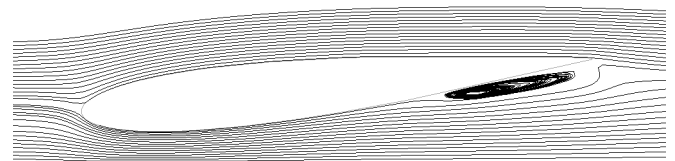


Figure 11: Streamlines on a streamwise plane located at the wing mid-span for $H/c = 0.1$.