EPJ Web of Conferences 114, 02069 (2016)
DOI: 10.1051/epjconf/201611402069

Numerical study of flow around NACA0015 in ground effect

Khadidja Malti[1], Hanaa Hebow[2] and Bachir Imine[1,a]

[1]*LASP Laboratory, Faculty of Mechanical Engineering, USTO, Algeria.*
[2]*University of Aleppo, Faculty of mechanical Engineering, Syria.*

Abstract. The aim of this work is to present a numerical simulation of flow around a wing profile NACA0015 under the ground effect. CFD software has been used to determine the aerodynamic performance for different angles of incidence. The flow is considered two-dimensional and the adopted meshing considered the effects of the boundary layer. The Spalart-Almaras turbulence model was adopted for the investigation of complex flow around the profile. The results obtained by CFD were compared to those obtained by the literature.

1 Introduction

When a wing approaches the ground two phenomena are actually involved in the increase of the lift force and reduction of the drag. The effects of proximity to the ground on the aerodynamic performances of the wing were investigated since 1920. Previous studies have shown that the decrease in altitude generates a rapid increase in lift and drag reduction. So favorable aerodynamic characteristics are obtained when a wing is close to the ground. For an open-wheel racing car, the flow around a number of components including the front wings, distributor, and wheels are subjected to the direct influence of the ground effect. The improved aerodynamic response can have a significant effect on the performance of the down force. For example, the down force can reach three times the weight of the car, one-third from the front wing, a third fairing and the rear diffuser and the rest of the rear fender.

During the first decades of the twentieth century [1], aerodynamic research has been carried in the automotive field and in particular to reduce drag to achieve the performance of advanced vehicle speed. During the second half of this century, development continued in the same direction, with reduction or reversal of the lift motor to increase their roll stability.

The flow characteristics over a symmetrical airfoil—NACA 0015—are studied experimentally in a low speed wind tunnel by Ahmed an al.[2]. The pressure distribution on the airfoil surface was obtained, lift and drag forces were measured and mean velocity profiles were obtained over the surface. As a result, they obtained a higher values of lift coefficient when the airfoil is close to the ground. Also, the wing in ground effect has been studies by Moure and al.[3]., they detailed experimental results from wind tunnel studies of a DHMTU and NACA 0012 section operating in ground effect. It was found that the drag of the DHMTU 12-35.3-10.2-80.12 increased with decreasing altitude, contrary to expectations and superior lift performance to the NACA 0012 baseline section was generated. In a numerical tier, Abramowski [4] proposed numerical simulations of the ground effect which affects an air flow when a wing approaches the ground or sea surface. The two-dimensional viscous flow problem is solved and the results of CFD are presented. On the basis of numerical calculations, an empirical formula has been proposed for quantitative assessment of the chord dominated ground effect.

In 2006, Hebow conducted an experimental investigation on the ground effect on a turbulent flow around a profile NACA0015 with a negative incidence. It has been proven that the aerodynamic performances of the airfoil degrade when clearance decreases.

In this present study, a NACA0015 profile type was chosen in order to simulate the operating conditions of the work of Hebow [5] to validate the numerical model used. As it shown in figure 1, the incidence α is defined as an angle between the direction of the undisturbed flow profile and the axis of symmetry, H is the height between the floor and the underside: determined from the extreme point of the edge leak. The variation of the angle of attack α and the parameter H to estimate the coefficient of drag and lift in several cases studied. Also in this study, a negative angle of attack are considered in order to investigate the roll stability.

[a] Corresponding author: imine_b@yahoo.fr

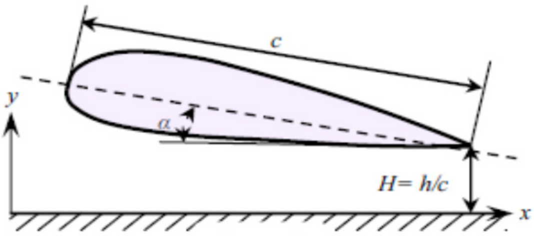


Figure 1. Geometry of physical problem

2 Transport equations and turbulence model

The flow around the profile is considered turbulent symmetrical. The general forms of the transport equations can be written in Cartesian coordinates as:

Continuity equation

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{U}_j\right)=0 \qquad (1)$$

Momentum conservation equation

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{U}_i\tilde{U}_j\right)=\bar{\rho}g_i-\frac{\partial\bar{p}}{\partial x_i}-\frac{\partial}{\partial x_j}(\rho u_i'' u_j'')$$
$$+\frac{\partial}{\partial x_j}\left[\bar{\mu}\left(\frac{\partial\tilde{U}_i}{\partial x_j}+\frac{\partial\tilde{U}_j}{\partial x_i}\right)-\frac{2}{3}\bar{\mu}\frac{\partial\tilde{U}_k}{\partial x_k}\delta_{ij}\right] \qquad (2)$$

Turbulence model

The model of turbulence used in this present work is a model with a transport equation for the ν quantity suggested by Spalart and Allmaras (3):

$$\frac{\partial(\bar{\rho}\tilde{v})}{\partial t}+\vec{\nabla}.\left(\bar{\rho}\tilde{\vec{V}}\tilde{v}\right)=\vec{\nabla}.\left[\frac{\bar{\mu}+\bar{\rho}\tilde{v}}{\partial SA}\vec{\nabla}\tilde{v}\right]+P_v-D_v \qquad (3)$$

Or the terms of production and destruction are defined as follows:

$$P_v=C_{b1}C_{\mu}\tilde{S}\bar{\rho}\tilde{v}+C_{b2}\frac{\bar{\rho}(\vec{\nabla}\tilde{v})^2}{\partial SA} \qquad (4)$$

$$D_v=C'_{\omega 1}f_{\omega}\frac{\bar{\rho}(\vec{\nabla}\tilde{v})^2}{\rho d^2} \qquad (5)$$

With

$$f_x=g\left(\frac{1+C_{\omega 3}^6}{g^6+C_{\omega 3}^6}\right)^{1/6} \text{ and } g=r+C_{\omega 2}\left(r^2-r\right) \qquad (6)$$

And

$$M_t=\bar{\rho}\tilde{v}\frac{\chi^3}{\chi^3+C_{v1}} \qquad \chi=\frac{\bar{\rho}\tilde{v}}{\mu} \qquad (7)$$

And the coefficients of closing are given by the following values:

$$\sigma_{SA}=\frac{2}{3};\quad C_{b1}=0.1355;\quad C_{b2}=0.622;$$
$$C_{v1}=7.1;\ C_{\omega 1}=0.3;\ C_{\omega 3}=2. \qquad (8)$$

(For more information see reference 5)

3 Meshing

For the airfoil case, NACA 0015 airfoil with 0.1 m of chord length c and as it shown in figures 2 and 3, the 2D structured mesh are generated The geometry of our model being variable according to the parameters h / c and the angle of attack α. Our main concern was to develop a grid that ensures resolutions in agreement with the available literature [5]. The geometry considered is composed of four distinct surfaces and a profile NACA 0015. The separation into four areas is to enable to refine the mesh without imposing a total number of items too high.

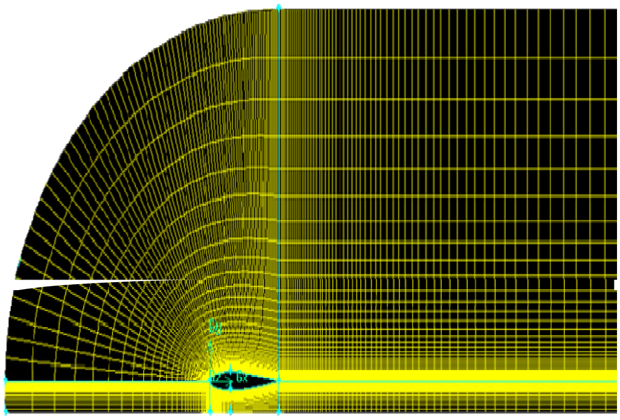


Figure 2: Meshing of computational domain

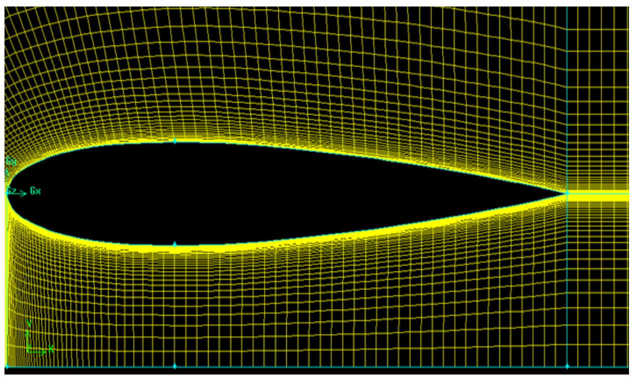


Figure 3: Meshing around the profile

Table I: Quality of meshing

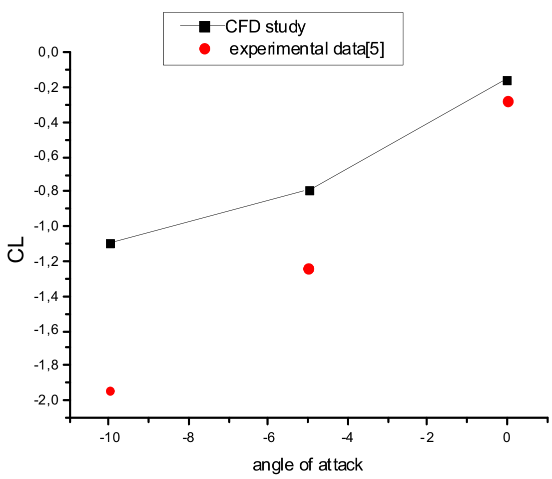
h/c	Angle of attack α	Elements
0.25	0°	14475
	-5°	15349
	-10°	16683
0.35	0°	14475
	-5°	15073
	-10°	16407
0.45	0°	14475
	-5°	14475
	-10°	16177

4 Results

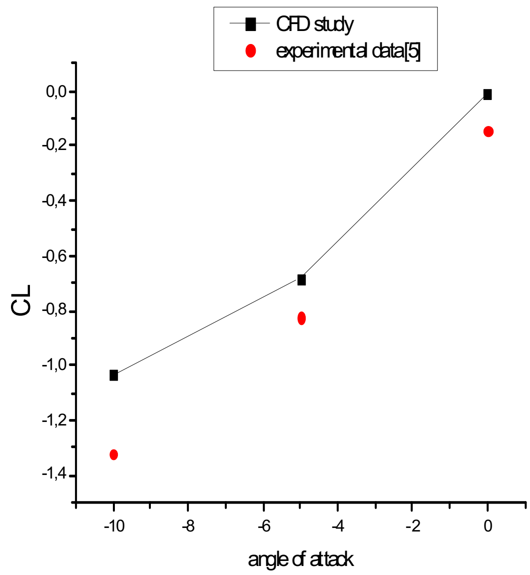
Several compilations were carried out for different negative angles of attack and different ground clearance to determine the lift and drag. The negative angles of attack were used in order to simulate the downforce. Each case was carried out at the same speed: 20 meters per second. The angle of attack extended from the following values: α =0, -5 and -10 degrees.

Figure 4 compares the computational evolution of the lift according to the angles of attack α with the experimentally data [5] for different cases of ground clearance h/c: A,B and C. An almost agreement of the two curves is noted in the linear evolution and the lift coefficient becomes more negative with decreasing angle of attack. However, examining the angle of attack values from α = - 5 to - 10, the differences between the numerical study and deviations from the measured curve become visible for all the ground clearance cases. It is clear that the numerical simulation is far from predicting the negative incidence, because the models currently used are not accurate for flows with separate streams. As it shown in the A,B and C cases, the downforce increases when the ground clearance h/c decreases.

a-



b-



c-

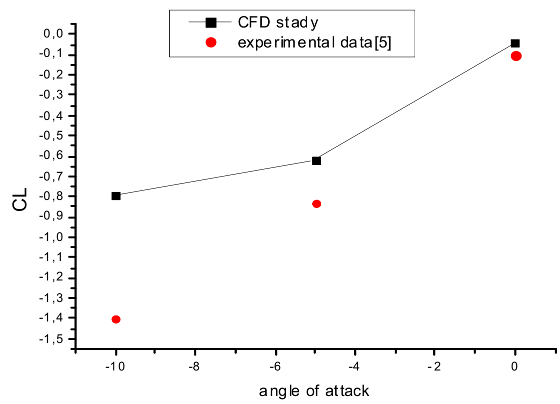
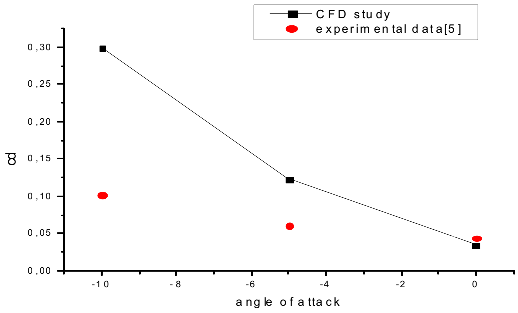


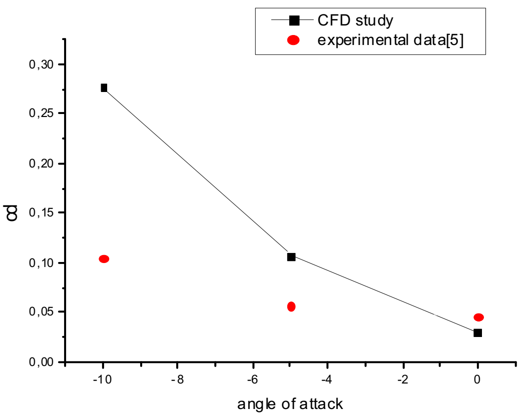
Figure 4: Variation of the lift coefficient with different ground clearance (h/c) **a** : h/c=0.25, **b** :h/c= 0.35 and **c**: h/c=0.45

The evolution of the drag coefficient according to the negative angle of attack with increasing ground clearance h/c=0.25, 0.35 and 0.45 was observed as illustrated in Figure 5. All curves are quasi linear type tendency similar to those found on the literature [5]. In particular, one notes the increase in the drag with the increase in the angle of attack. However, a slight conformity between the present study and experimental one was only partial. It was shown in the two graphs a reduction of the drag coefficient versus the angle of attack α, but depending on the ground clearance 'h / c', this analogy exists only at α = -10 °.

a-



b-



c-

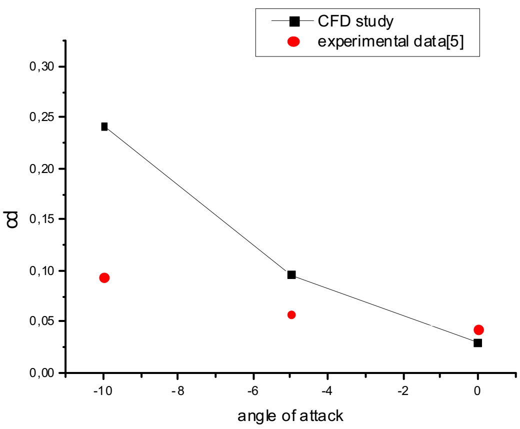
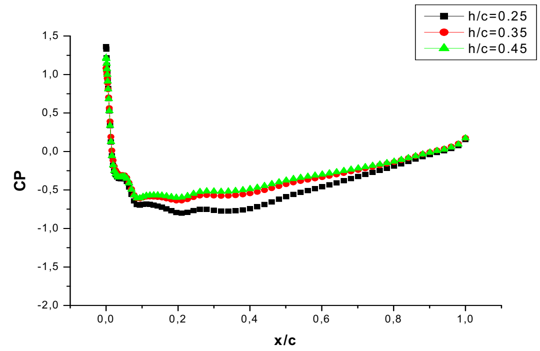
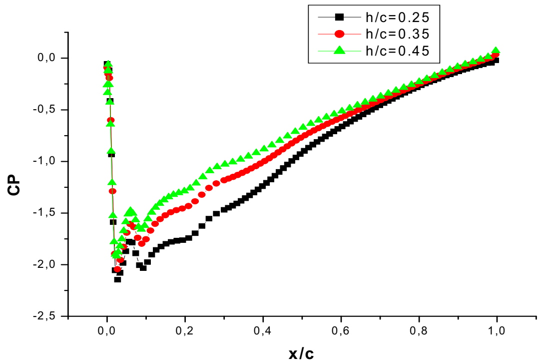


Figure 5 The evolution of the drag coefficient with different ground clearance. **a**: :h/c=0.25, **b**: h/c=0.35 and **c** : h/c = 0.45.

a-



b-



c-

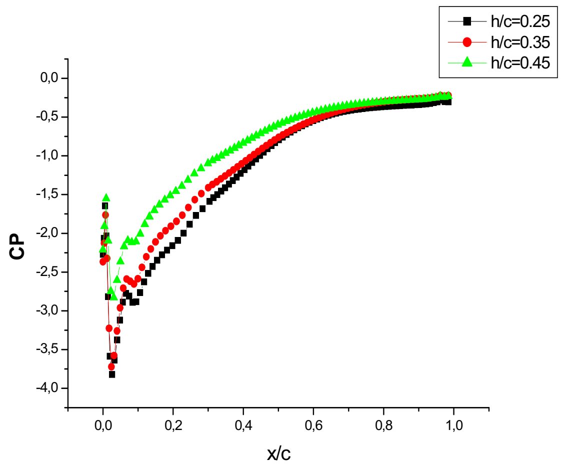


Figure 6: Pressure distribution on the surface of the airfoil for an angle of attack of (a) α = 0°, (b) α = -5°, (c) α = -10°.

Figures 6a, 6b and 6c shown the variation in the pressure on the airfoil surface for different values of attack angle: α = 0 °, - 5 ° and -10 ° and with different ground clearances h/c= 0.45, 0.35 and 0.25, For all cases, the pressure is high on the upper surface at points close to the leading edge. It was noted that for α = - 5° and -10°, the pressure coefficient is negative. This is because the area between the airfoil and the ground forms a divergent–convergent passage, and its effect is more prominent for lower ground clearances.

Conclusion

To study the performance of a NACA 0015 wing in ground effect, a detailed numerical investigation was carried out using CFD code with the Spalart-Allmaras turbulence model. It was found from the present numerical study that both the ground clearance and the angle of attack have a good effect on the aerodynamic performances of the used configuration. The comparison between the experimental and numerical fields, gives a reasonably good correlation.

References

1. Florent Mortel (2002) course ‘l’aerodynamique dans l’automobile’ Belford technology university, 2002.
2. M.R. Ahmed , S.D. Sharma , An investigation on the aerodynamics of a symmetrical airfoil in ground effect. International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:8, No:7, 2014.
3. Lange R, Moore J, “Large Wing-in-Ground Effect Transport Aircraft”,
   Journal of Aircraft, Vol.17, No.4, April 1980.
4. Tomasz Abramowski; Numerical investigation of airfoil in ground proximity. Journal of theoretical and applied mechanics 45, 2, pp. 425-436, Warsaw 2007.
5. Hebow. H, (2006). ‘Interferences between boundary layers on an Airfoil and a flate plate’. Thesis master degree in Mechanical Engineering. Aleppo University. 123p.