Proceedings of the 9th World Congress on Mechanical, Chemical, and Material Engineering (MCM'23)
Brunel University, London, United Kingdom –August 06-08, 2023
Paper No. HTFF 206
DOI: 10.11159/htff23.206

Numerical Research on a Drag Reduction Technique Used For the NACA 0012 Airfoil

Amine AGRISS[1], Mohamed AGOUZOUL, and Abdeslem ETTAOUIL
Mohammed V University in Rabat, ERG2(ME), Rabat, Morocco
[1]amineagriss@research.emi.ac.ma

Abstract **-** The objective of this study is to propose a new method of drag reduction applied to a NACA 0012 aerodynamic airfoil. A device has been installed at a point in the area where flow separation is likely to occur. It allows reducing drag of the airfoil. The numerical simulations have been carried out in the two−dimensional case. The computational fluid dynamics (CFD) software Ansys Fluent 17.0 has been used. An incompressible and laminar flow has been considered for a Reynolds number of Re = 1000. An angle of attack of 5° has been considered. Results show that in certain device configurations, drag could be reduced while the lift−to−drag ratio increases. Future works concern studying this drag reduction device for other angles of attack.

Keywords**:** NACA 0012, device, separation point, Ansys Fluent, laminar, drag, lift

1. Introduction

Drag reduction constitutes a very vast field of research due to its great contribution in several fields (aeronautics, automotive, etc.). In fact, the future designs performances are increasingly improved by reducing drag. In that sense, lot of strategies have been developed over the years in order to reduce drag of aerodynamic structures.

Initially, the techniques of drag reduction are tested by the use of simplified model whether numerically or experimentally. Thanks to their simplicity and their usefulness, these models make it possible to reproduce the phenomena which occur in reality. In this context, the NACA 0012 airfoil has been widely used in numerical or experimental researches for testing the techniques of drag reduction used on an aircraft wing.

Among the drag reduction strategies proposed by several authors, controlling the flow using a micro−riblet film decreases drag of the NACA 0012 airfoil [1]. Another technique consists on using surface grooves for a NACA 4415 airfoil. In certain grooves configurations, the airfoil aerodynamic performances might be enhanced [2]. The active flow control using pulsed blowing and a harmonic actuation could delay the flow separation on a NACA 0015 [3]. The aerodynamic performances of the NACA 0012 could be improved by using an attached Gurney flap [4]. The flow control using vortex generators on the NACA 4415 airfoil improves its efficacity [5].

Other simplified models are also used for testing drag reduction devices. Flat plates and simplified car models are among these models. Some of strategies, employed on these models, used to reduce drag, flat plate shape corrugations could reduce drag [6]. Injecting some airflow into the back end of the Ahmed body through a conduit might reduce drag [7]. All these drag reduction strategies and others are developed in the literature.

The objective behind this study is to reduce drag of the NACA 0012 airfoil. A device is placed in the separation point in order to control the separation and reduce drag of the airfoil. The CFD program Ansys Fluent 17.0 has been used to perform numerical simulations. Various device configurations are tested. In addition to reducing drag, the best configuration is found to increase the lift−to−drag ratio.

2. Numerical modelling

In this study, an incompressible and laminar airflow is considered for a number of Reynolds Re = 1000. Navier−Stokes equations for laminar flows are defined by:

$$(\mathbf{U}.\nabla)\mathbf{U} = \nu\nabla^2\mathbf{U} - \frac{1}{\rho}\nabla p \quad (1)$$

U (u, v) represents the air velocity vector, ν represents its kinematic viscosity, ρ represents its density and p represents its pressure.

The Reynolds number is:

$$\mathrm{Re} = \frac{U.c}{\nu} \tag{2}$$

c denotes the airfoil chord, while U denotes the inlet airflow velocity.

The drag coefficient is defined by:

$$C_d = \frac{F_d}{\frac{1}{2}\rho U^2 S} \tag{3}$$

The lift coefficient is defined by:

$$C_l = \frac{F_l}{\frac{1}{2}\rho U^2 S} \tag{4}$$

F_d and F_l are the drag and lift forces and S is the area of reference taken as the wing area. For two−dimensional flows, the reference area is equal to the wing chord.

To measure the airfoil aerodynamic efficiency, the lift−to−drag ratio called also the L/D ratio is calculated, it is defined as:

$$L/D = \frac{C_l}{C_d} \tag{5}$$

A greater L/D ratio airfoil is more efficient than a lower ratio airfoil.

3. Studied configuration

Figure 1 shows the NACA 0012 airfoil, with a chord of c = 1 m, used in this study. The numerical simulations are performed for Re = 1000.

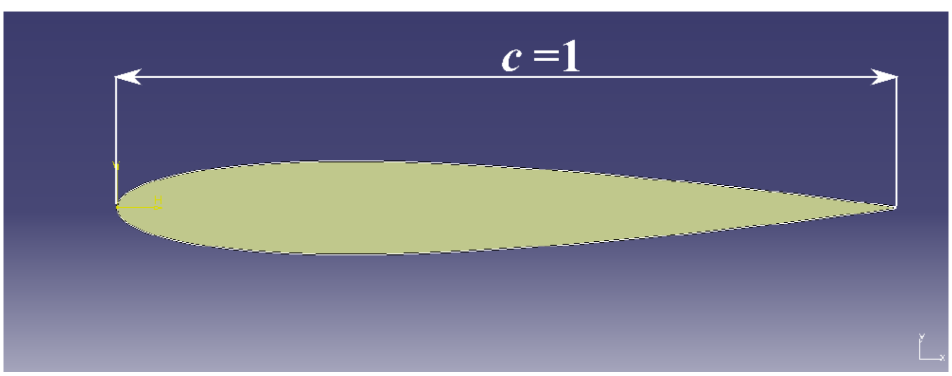


Fig. 1: Airfoil NACA 0012 description.

Using a 5° angle of attack, the separation point is found to be at a distance of 0.6 m from the point of attack. That's why, our idea is to place a device in this separation point in order to reduce drag. This is shown in the figure 2. Various device configurations are evaluated depending on the distance *h*.

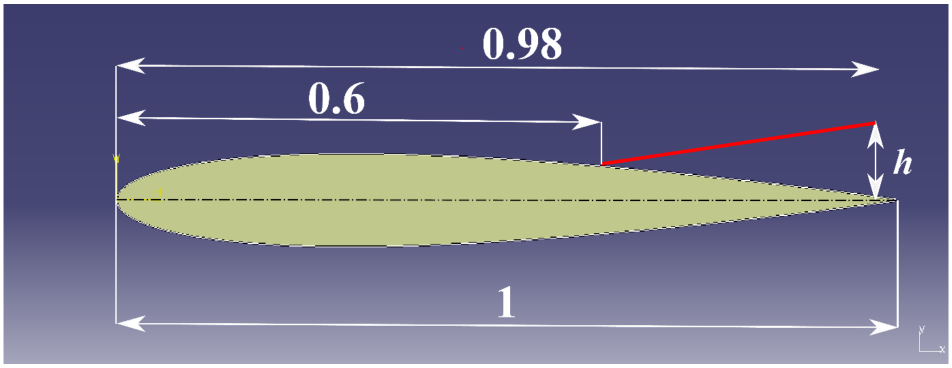


Fig. 2: Emplacement of the device of drag reduction.

The flow domain and boundary conditions of the current numerical simulations are shown in the figure 3. The velocity inlet is used around the airfoil. The airfoil is considered as fix and the air is moving around it. α denotes the angle of attack. The flow velocity is $U = 1$ m/s.

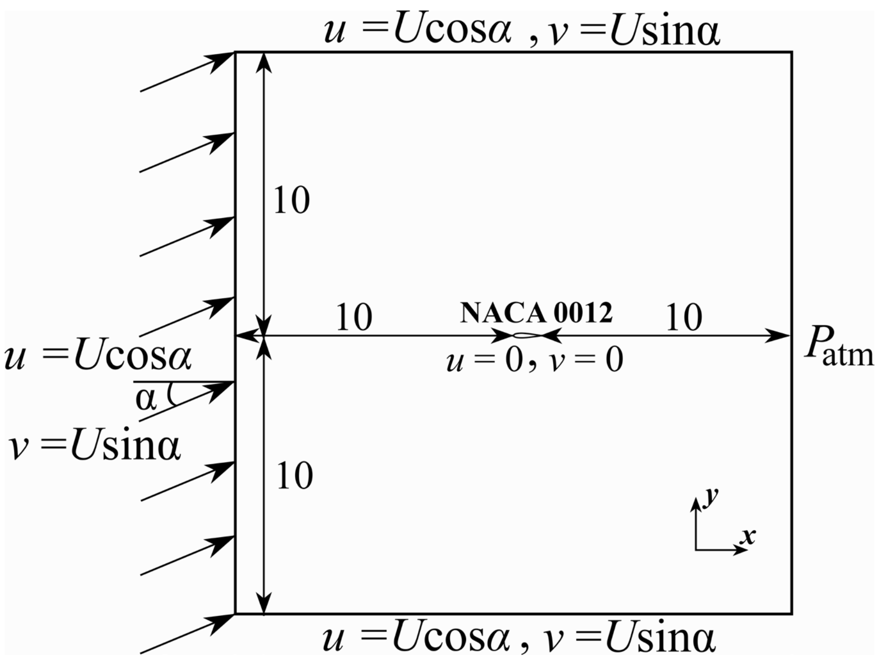


Fig. 3: Boundary conditions of current simulations.

The mesh used in this study is shown in the figure 4. It is a non−uniform structured mesh of 65 800 elements built using the ICEM 17.0 tool.

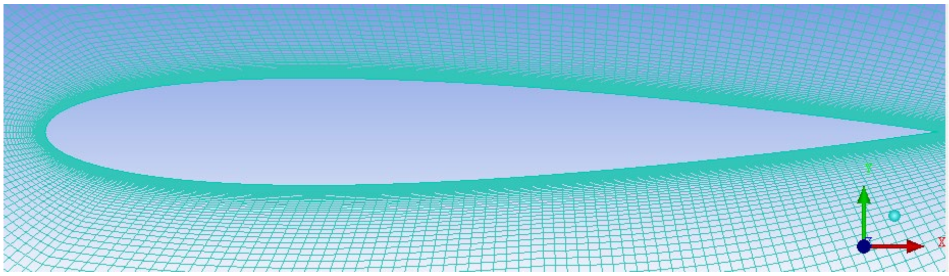


Fig. 4: Non−uniform structured mesh used in this study.

The residuals evolution of continuity and momentum equations is shown in the figure 5. The solution convergence is obtained from the 900 iteration.

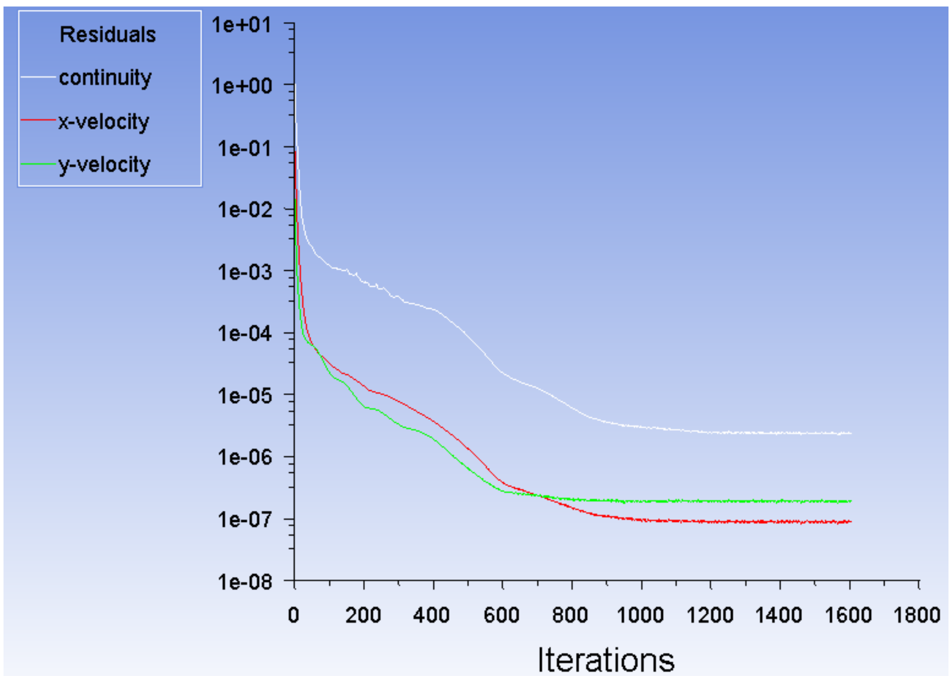


Fig. 5: Evolution of continuity and momentum equations residuals.

Simulations are conducted in the laminar steady state. Gradients are solved with cell−based least−squares discretization. Pressure and momentum equations are solved by the second−order scheme. The precision of simulations is about 10^{-6}.

4. Results and discussions

The velocity contour obtained by the present simulation with a zero angle of attack is shown in the figure 6. The obtained velocity contour is in accordance with previous results [8].

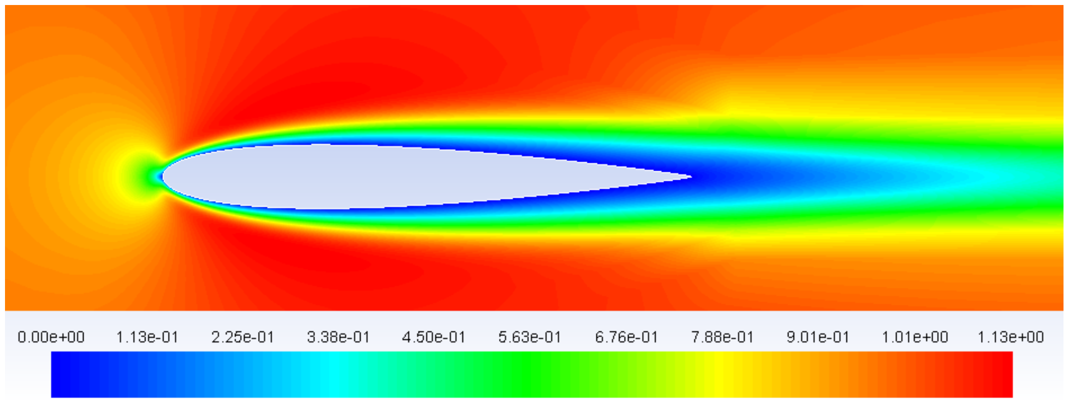
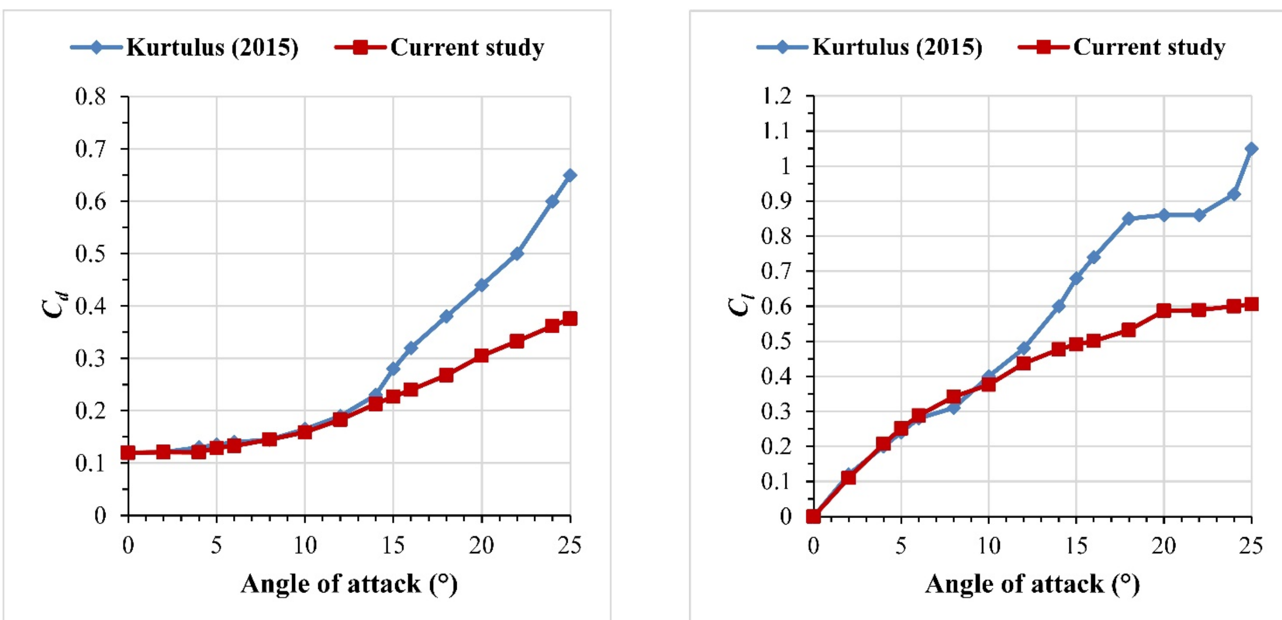


Fig. 6: Velocity contour for a zero angle of attack.

The drag and lift coefficients are compared to the results obtained in the literature for several angles of attack [9]. Figure 7 shows that for angles of attack ranging from 0° to 14°, the two aerodynamic coefficients are in line with previous results.

Starting from the angle α = 15°, drag and lift coefficients begin to diverge from prior studies' values. The reason for this is probably because of instability in the flow at those specific angles of attack.



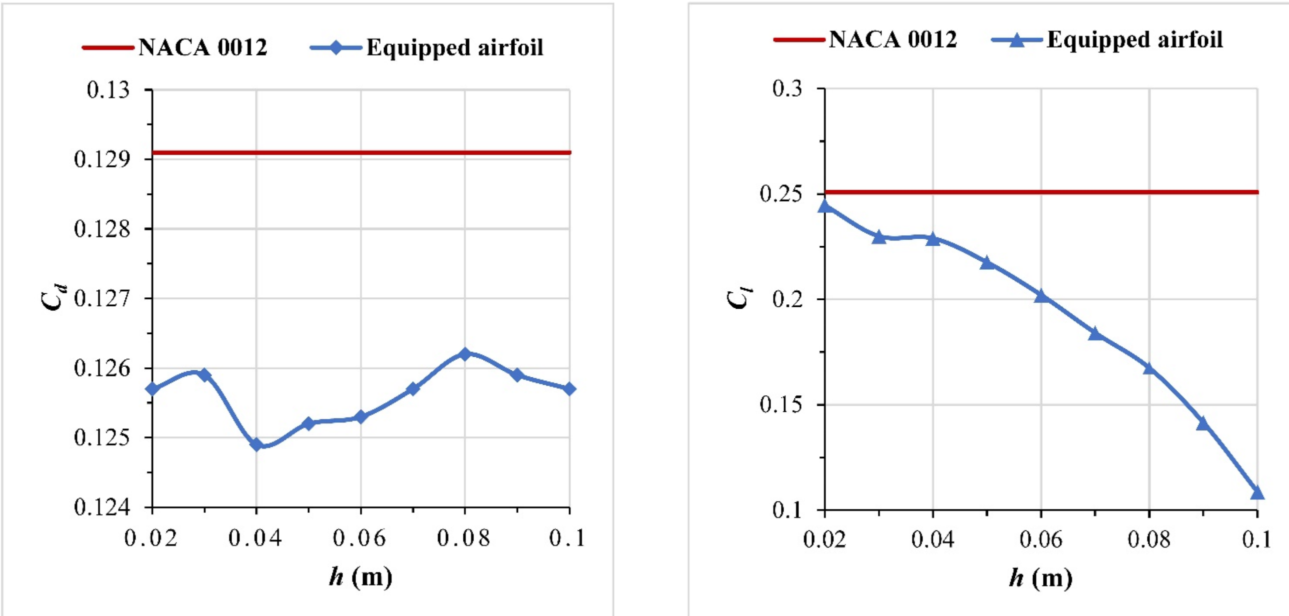
(a) (b)

Fig. 7: Comparison of aerodynamic coefficients with previous results.

Figure 8 depicts the evolution the aerodynamic coefficients, for the equipped airfoil, as a function of the device's height *h*.

For 0.02 m ≤ *h* ≤ 0.1 m, drag coefficients drop when compared to airfoil drag coefficient when the device is not attached. The maximum decrease is about 3.25% for *h* = 0.04 m.

While the lift coefficient decreases at all height *h* values. A maximum decrease is observed from *h* = 0.07 m.



(a) **(b)**

Fig. 8: Evolution of the aerodynamic coefficients with the device height *h*.

For all device configurations, the aerodynamic coefficients drop, resulting in a decline in *L*/*D* ratios. Just the first initial arrangement, with *h* = 0.02 m, provides for drag reduction while improving the *L*/*D* ratio (C_d decreases by 2.63% while C_l decreases by 2.47%).

5. Conclusion

This study deals with a device placed in the separation point of an aerodynamic airfoil NACA 0012 to minimize drag. Work has been conducted numerically with the CFD program Ansys Fluent 17.0. By placing the device of drag reduction at a point situated in the area where flow separation may occur, drag coefficient decreases for all configurations of the device. This is due to the suppression of the separation that occurs on the airfoil. However, the lift coefficient also decreases for all configurations. It is therefore necessary to take into consideration the lift−to−drag ratio which must be greater than that of the airfoil without use of the device. This is only possible for one design (the device with a height of *h* = 0.02 m).

Future studies will focus on studying this drag reduction device for other angles of attack. After that, 3D numerical simulations and experimental studies are suggested in order to approve the present results.

References

[1] S.−J. Lee and Y.−G. Jang, "Control of flow around a NACA 0012 airfoil with a micro−riblet film," Journal of Fluids and Structures, vol. 20, no. 5, pp. 659–672, Jul. 2005, doi: 10.1016/j.jfluidstructs.2005.03.003.

[2] Y. Liu, P. Li, W. He, and K. Jiang, "Numerical study of the effect of surface grooves on the aerodynamic performance of a NACA 4415 airfoil for small wind turbines," Journal of Wind Engineering and Industrial Aerodynamics, vol. 206, p. 104263, Nov. 2020, doi: 10.1016/j.jweia.2020.104263.

[3] L. Wang, L. Li, and S. Fu, "Numerical Investigation of Active Flow Control on a Pitching NACA 0015 Airfoil Using Detached−eddy Simulation," Procedia Engineering, vol. 79, pp. 49–54, 2014, doi: 10.1016/j.proeng.2014.06.308.

[4] Y. Amini, M. Liravi, and E. Izadpanah, “The effects of Gurney flap on the aerodynamic performance of NACA 0012 airfoil in the rarefied gas flow,” Computers & Fluids, vol. 170, pp. 93–105, Jul. 2018, doi: 10.1016/j.compfluid.2018.05.003.
[5] O. M. Fouatih, M. Medale, O. Imine, and B. Imine, “Design optimization of the aerodynamic passive flow control on NACA 4415 airfoil using vortex generators,” European Journal of Mechanics − B/Fluids, vol. 56, pp. 82–96, Mar. 2016, doi: 10.1016/j.euromechflu.2015.11.006.
[6] A. Agriss, M. Agouzoul, A. Ettaouil, and A. Mehdari, “Numerical study of new techniques drag reduction: application to aerodynamic devices,” Int. J. Simul. Multidisci. Des. Optim., vol. 12, p. 16, 2021, doi: 10.1051/smdo/2021015.
[7] A. Agriss, M. Agouzoul, A. Ettaouil, and A. Mehdari, “Numerical Investigation of a Drag Reduction Device Applied to the Ahmed Body,” International Review on Modelling and Simulations (IREMOS), vol. 15, no. 2, Art. no. 2, Apr. 2022, doi: 10.15866/iremos.v15i2.22103.
[8] R. C. Swanson and S. Langer, “Steady−state laminar flow solutions for NACA 0012 airfoil,” Computers & Fluids, vol. 126, pp. 102–128, Mar. 2016, doi: 10.1016/j.compfluid.2015.11.009.
[9] D. F. Kurtulus, “On the Unsteady Behavior of the Flow around NACA 0012 Airfoil with Steady External Conditions at Re=1000,” International Journal of Micro Air Vehicles, vol. 7, no. 3, pp. 301–326, Sep. 2015, doi: 10.1260/1756−8293.7.3.301.