Available online www.jsaer.com

Journal of Scientific and Engineering Research, 2023, 10(1):30-36

Research Article

ISSN: 2394-2630
CODEN(USA): JSERBR

Aerodynamic Simulations of An Airfoil Wake Reduction System

Birame Sy[1*], Mamadou Kabirou Toure[1], Abdoulaye Sarr[1,2], Ousseynou Diao[1]

[1]École Polytechnique de Thiès, Sénégal
[2]Air Force Academy, Thiès, Sénégal
*Email: bsy@ept.sn

Abstract The current project is focused on reducing the wake of an aerodynamic profile. In fact, reducing the wake of a body results on reducing its drag, its proper aerodynamic noise and also interactions with the aircraft surfaces downstream. To reduce the wake, suction and blowing devices are integrated on the intrados and the extrados of the airfoil. The objective of this numerical study is to analyze the behavior of this wake reduction system. The analysis is done via Computational Fluid Dynamics (CFD) simulations with RANS and URANS methods. The numerical results of the velocity profile for different positions around the airfoil are shown to illustrate the effectiveness of the control system.

Keywords Aerodynamics, flow control, wake reduction, suction, blowing, boundary layer

Introduction

Currently, billions of liters of fossil oil are consumed to defeat the overall drag opposing the forward movement of aircraft, increasing the environmental footprint of the aeronautical industry. In this context, the ability to optimize aerodynamics represents one of the most important current issues. Moreover, each technological advancement resulting in fuel economy goes by example, to reduce the size of the tanks, and therefore to lighten the device, thus leading to a drop in consumption. In addition, controlling the flow around an aircraft allows significant improvement in the overall comfort associated with the world's aircraft fleet, in terms of noise pollution, reduction of polluting gases and maneuverability. The efforts of research devoted to the study of these means of control are considerable and continue to increase because they constitute a major scientific challenge, due to its complexity and the extreme diversity of the ways investigation associated with it.

The current project is focused on reducing the wake of an aerodynamic profile. In fact, reducing the wake of a body results on reducing its drag, its proper aerodynamic noise and also interactions with the aircraft surfaces downstream [1].

To reduce the wake, suction and blowing devices are integrated on the intrados and the extrados of the airfoil. The objective of this numerical study is to analyze the behavior of this wake reduction system. The analysis is done via Computational Fluid Dynamics (CFD) simulations with RANS and URANS methods. The numerical results of the velocity profile for different positions around the airfoil are shown to illustrate the effectiveness of the control system.

Most flow control studies in aeronautics are based on flow stabilization and separation control. Solutions related to wake control are rare, only found in turbomachinery when rotor-stator interaction is analyzed. Kohlhaas, M., K. Bamberger, and T. Carolus [2] designed a trailing edge blowing solution to reduce the rotor-stator interaction noise. In this application, the boundary layer is tangentially blown, receiving an amount of momentum. This amount balances the velocity deficit in the boundary layer which is responsible for the wake thickness.

Another approach is to remove the natural boundary layer by suction through a wall slot. The transversal vorticity diffusion, from wall to free flow, produces a vertical component of velocity which is responsible for the boundary layer development. Suction inhibits this velocity by an opposed aspiration velocity. This idea has been used for the boundary layer stabilization on high-lift airfoil since the 60s [3] but never been experimented in wake reduction purposes. In this project the authors propose to combine suction and blowing technologies for the purpose of wake reduction.

Flow control on the airfoil

Control device modeling

The wake is the signature of a boundary layer developed around a body in a flow. Hence, the wake thickness is minimized when the boundary layer is minimized at the trailing edge. In order to reduce it, two complementary devices are used: suction and blowing.

Suction velocity has an important component in wall tangential direction, so that the boundary layer is locally accelerated. Just downstream, a new boundary layer restarts developing. Suction device, upstream the blowing slot, benefits from much more space, and it is exploited to extend its width. Hence, its flow rate capability is rather higher. After suction, the new boundary layer is tangentially blown in order to reduce the velocity deficit near the wall. If the blowing flow rate is too high, the flow can be destabilized leading to a mixing layer. Ideally, both equipment should be positioned at the trailing edge to avoid a boundary layer development, but spatial constraints make it unfeasible. Therefore, the final configuration is an equilibrium between flow optimization and technological limitations. To illustrate it, analytical calculations are given in the case of a flat plate of length L in a uniform flow of velocity U_0.

The boundary layer is supposed to be fully turbulent. For turbulent boundary layer, there is no auto-similar solution as in laminar case. The von Karman equation [4] provides a relation between the mean friction coefficient and the mean characteristics of the boundary layer:

$$\frac{C_f}{2} = \frac{d\theta}{dx} + (2\theta + \delta^*)\frac{1}{U_0}\frac{dU_0}{dx} \qquad (1)$$

When this equation is coupled to the experimental value of the friction coefficient:

$$C_f(x) = 0.0592\, {Re_x}^{-1/5} \qquad (2)$$

And the classical power function approximation of the velocity profile:

$$\frac{U}{U_0} = \left(\frac{y}{\delta}\right)^{1/7} \qquad (3)$$

It leads to an expression of thickness:

$$\delta(x) = 0.38\left(\frac{U_0}{\nu}\right)^{-1/5} x^{4/5} \qquad (4)$$

Thus, if the boundary layer is supposed to be totally aspirated in the suction slot at x, the suction flow rate needed is:

$$Q_S = \int_0^{\delta(x_S)} \rho U(x_S)dyL = \frac{7}{8}\rho U_0 L\delta(x_S) \qquad (5)$$

The blowing device must balance out the velocity deficit in the boundary layer so the flow rate injected by the blowing slot at x_B is:

$$Q_B = \rho L\left[U_0\delta(x_B - x_S) - \int_0^{\delta(x_B - x_S)} U(x_B - x_S)dy\right] \qquad (6)$$

$$Q_B = \frac{1}{8}\rho U_0 L\delta(x_B - x_S) \qquad (7)$$

For a 1 m flat plate, with the suction device placed at $2/3$ of the chord and the blowing device at the trailing edge, in a 40 m/s flow, this computation leads to $Q_S = 0.63\ kg/s$ and $Q_B = 0.05\ kg/s$.

Wake quantifier

The main objective of this control system is to reduce the airfoil wake. In fact, the velocity deficit downstream the airfoil is an important noise source. C_2 criterion is a global quantifier, measuring the wake intensity at a fixed position after the trailing edge and it is defined by:

$$C_2 = \int_0^\infty \left(U_0 - U(y)\right)dy \qquad (8)$$

One of the keys of this research is to minimize the C_2 criterion in the wake. It is important to remark that this criterion is global: the integral expression does not take into account local velocity fluctuations due to turbulence.

Simulation of the control system

Numerical domain

RANS and URANS computations are done in a rectangular domain with a 2D multi-block structured grid with 2,650,000 quad elements. Figures 1-3 show some mesh details of specific parts. Although this unconventional geometry, the structured approach enables high quality criteria of the mesh.

The far-field boundaries are located at a distance of 10 chord. The surfaces of the airfoil are modeled as no-slip (wall) boundaries. The size of the first grid rows is sufficiently low to obtain $\mathbf{y^+}$ around unit at the profile wall. For the turbulence we use a **k-ω** turbulence model [5,6].

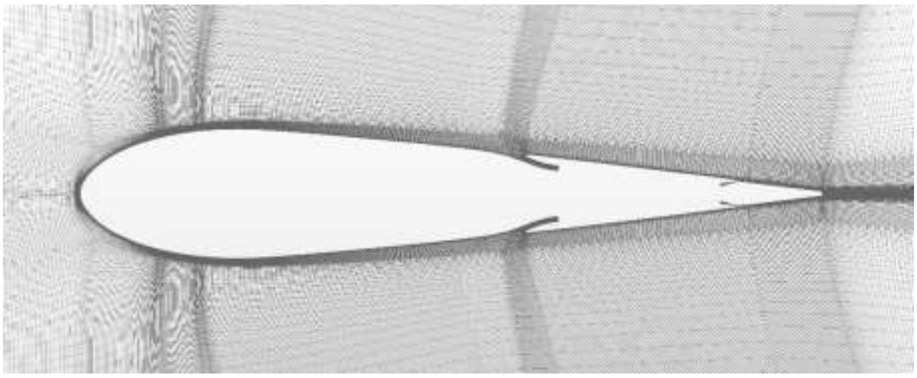

Figure 1: The mesh around the airfoil

Figure 2: Suction device mesh (extrados)

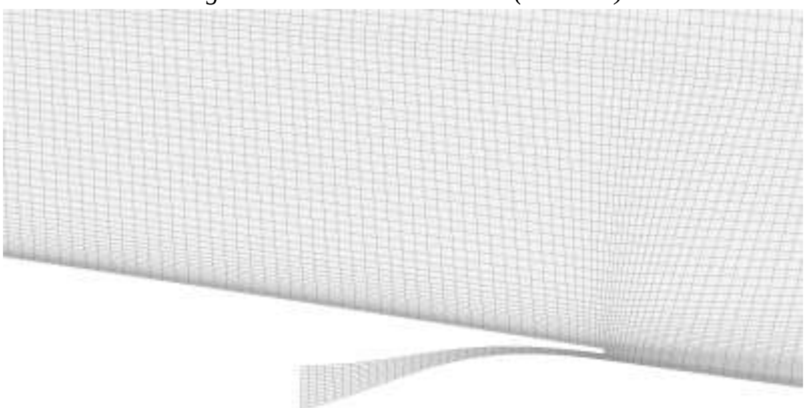

Figure 3: Blowing device mesh (extrados)

Results and Analysis

In these simulations, two configurations are compared at different positions: a baseline case (no flow control) and a control solution case (at optimal control parameters). In order to perceive the suction and blowing effects, velocity profiles are plotted before and after each device. The local velocity is represented in a non-dimensional form by the freestream velocity; the normal distance is non-dimensional as well, in this case, by the airfoil chord.

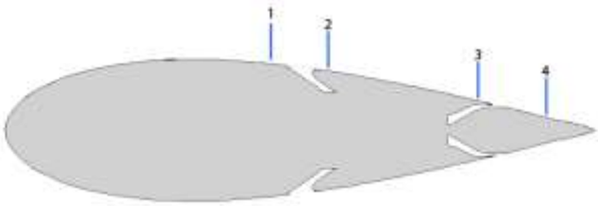

Figure 4: Numerical velocity probes locations

Concerning suction (Figure 5), before suction it is possible to distinguish an acceleration of the boundary layer due to tangential velocity suction. This is the only difference because the baseline and control case have the same shape. After suction, the effect is more evident; the former boundary layer has been absorbed and a new boundary layer restarts developing. Shapes are no longer the same.

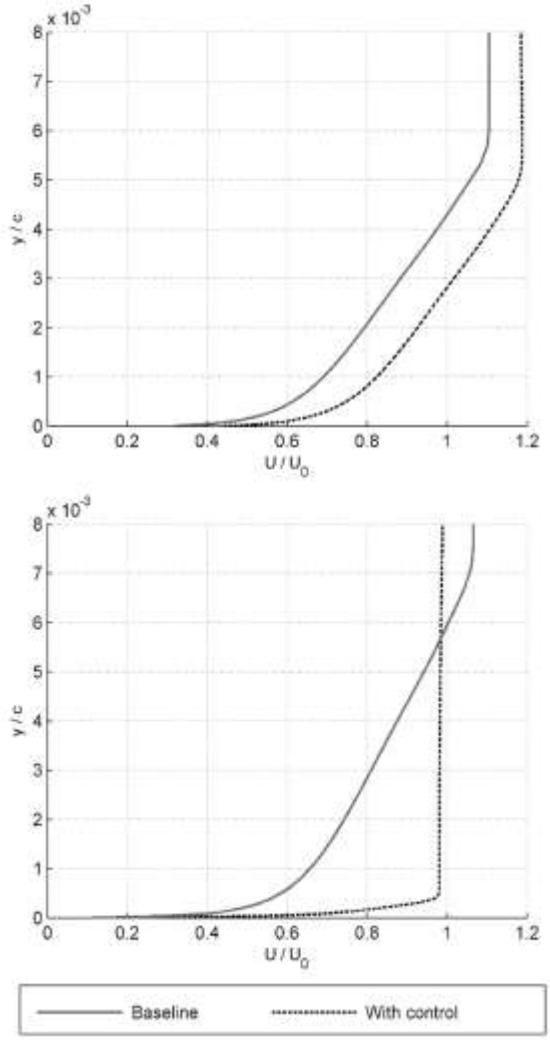

Figure 5: Velocity profile before (top), after (bottom) suction

On the other hand, blowing device effects are shown in Figure 6. Yet, it is possible to perceive suction effects on the control case boundary layer, which its velocity profile is more uniform, boundary layer thickness is much lower. Blowing introduces a momentum amount which accelerates the boundary layer near the wall; this acceleration causes a mixing layer, a peak of velocity, higher than the uniform velocity. This peak is intentionally provoked to obtain a uniform velocity profile at the trailing edge (the overspeed after the blowing slot is reduced while it travels the trailing edge distance). Table 1 contains parameters of the boundary layer of the presented cases.

At the trailing edge, intrados and extrados boundary layers join creating a wake. This wake could interact with another structure placed downstream the airfoil then generates more noise if the wake is long enough to interfere with the structure located downstream. Table 2 presents different characteristic parameters of wake such as the wake thickness, the C_2 criterion and the velocity deficit. The improvement in terms of C_2 criterion at 7.5 % chord is 11.0 %. At this position, Figure 6 represents the velocity profiles. An important reduction in velocity deficit is perceived for the control case, also presenting a wider uniform velocity range.

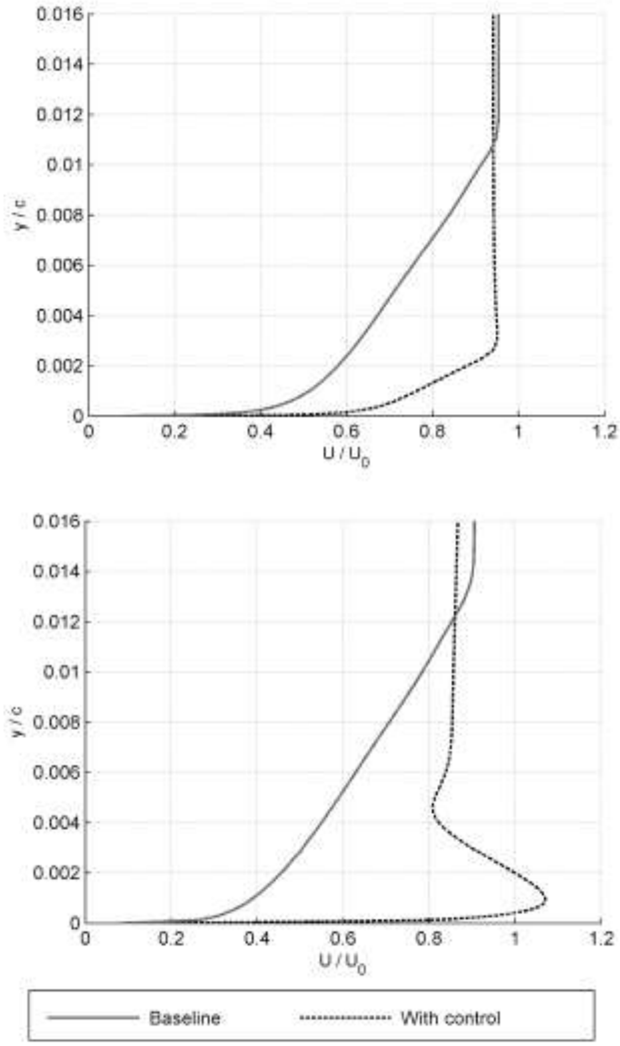

Figure 6: Velocity profile before (top), after (bottom) blowing

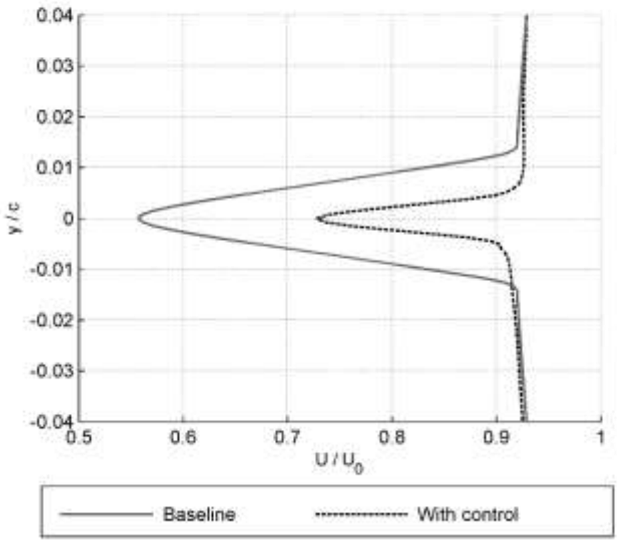

Figure 7: Velocity profile in the wake

Table 1: Boundary layer characteristics

Pos.	**Configuration**	$\boldsymbol{\delta}$	$\boldsymbol{\delta^*}$	$\boldsymbol{\theta}$	$\boldsymbol{H}$
1	Baseline	0.00555	0.00128	0.00086	1.49
	With control	0.00496	0.00098	0.00070	1.39
2	Baseline	0.00701	0.00157	0.00106	1.48
	With control	0.00041	0.00008	0.00005	1.53
3	Baseline	0.01093	0.00267	0.00176	1.52
	With control	0.00266	0.00046	0.00034	1.35
4	Baseline	0.01407	0.00396	0.00238	1.67
	With control	0.00047	0.00007	0.00004	1.59

Table 2: Wake characteristics

Position	**Configuration**	$\boldsymbol{\delta_{wake}}$	$\boldsymbol{C_2}$	$\boldsymbol{\Delta U_{max}}$	**Trend**
1% chord from TE	Baseline	0.387	0.0250	0.806	$\Delta U \downarrow$
	With control	0.437	0.0244	0.464	42.5%
3.75% chord from TE	Baseline	0.491	0.0273	0.562	$\Delta U \downarrow$
	With control	0.503	0.0253	0.429	23.7%
7.5% chord from TE	Baseline	0.608	0.0293	0.443	$\Delta U \downarrow$
	With control	0.600	0.0264	0.273	38.4%

Conclusion and perspectives

The boundary layer and wake analyses have shown the qualitative and quantitative influence of the control devices in flow performances. However, the wake intensity criterion C_2 results have not been as good as expected: only an 11.0 % of reduction. It might be due to an inaccurate application of C_2 criterion. In addition, there are possible noise sources not taken into account by this criterion, for instance the instantaneous velocity fluctuations, filtered by the RANS methods.

Large Eddy Simulations (LES) [7] would be the following step of the research. More accurate turbulence details offered by LES would provide other acoustic criteria to combine or complement the current results.

This research has studied control parameters for airfoil at zero degrees angle of attack. Another possible future step would be the airfoil analysis at other angle of attack and the determination of its optimal control parameters.

References

[1]. Pearcey, H. H. (1961). Shock induced separation and its prevention. *Boundary layer and flow control 2*: 1170-1344.

[2]. Kohlhaas, M., K. Bamberger, and T. Carolus. (2013). Optimization and validation of trailing edge blowing for the reduction of rotor-stator interaction noise. *10th European Conference on Turbomachinery Fluid dynamics & Thermodynamics*.

[3]. Hazen, David C. (1954). Boundary layer control. *PRINCETON UNIV NJ JAMES FORRESTAL RESEARCH CENTER*,

[4]. Van Driest, Edward R. (1951). Turbulent boundary layer in compressible fluids. *Journal of the Aeronautical Scienc*es 18.3: 145-160.

[5]. Menter, F. R. (1992). Improved two-equation k-omega turbulence models for aerodynamic flows. NASA (National Aeronautics and Space Administration). (No. A-92183).

[6]. Hellsten, A. (1998). Some improvements in Menter's k-omega SST turbulence model. *In 29th aiaa, fluid dynamics conference* (p. 2554).

[7]. Mason, Paul J. (1994). Large-eddy simulation: A critical review of the technique. *Quarterly Journal of the Royal Meteorological Society* 120.515: 1-26.