Numerical Investigation of Flow Control Via Blowing And Suction on NACA4415 Airfoil

1st AMRANI Djedjiga, 2nd IMINE Bachir, 3rd ADJLOUT Lahouari, 4th MEFTAH Sidi Mohammed ElAmine

Laboratory of aeronautics and propulsive systems, departement of mechanical engeneering
Sciences and Technology University of Oran (usto-MB)
Oran, Algeria
djedjiga.amrani@hotmail.com
imine_b@yahoo.fr
adjlout@yahoo.fr

***Abstract*— a blowing flow control method was studied in the present work around an asymmetrical NACA 4415 profile. The effect of many parameters are investigated (jet angle (0°, 30°and 45°), jet amplitude (A= 0.1 to 1)), combinations of two actuators jet and an effect of suction was compared to the blowing. Flow was fully turbulent for Reynolds numbers of 2.18x10^5. A commercial code, fluent was used in the current simulations. The turbulence model used is SPALLART ALLMARAS. The results show that the lift coefficient increases and drag coefficient decreases while the amplitude rose. The optimum location of jet is near to the stall position (30% c) and a tangential jet is more beneficial. The maximum lift coefficient increases by 13 % and a flow separation is delayed to 17°for the dual jet who seems to be the best.**

Keywords—flow control, blowing, suction, lift, drag, stall, CFD.

I. Introduction

Delaying or manipulate a separation flow is most important research subject in recent years. In this aim, it's necessary to reduce skin friction by eliminating transition of laminar boundary layer to turbulent. As is known, laminar boundary layer is more resistant to the stall then turbulent one. To prevent a separation, many flow control techniques exist (passive/ active) passive control devices don't require an external energy, contrary to the active methods which use an energy. Controlling the flow, affect on the aerodynamic characteristics. The lift coefficient is increased and drag coefficient is reduced (higher velocity gradient). The loss on drag and suppression of turbulence are translated to less consumption of energy.

Huang and al[1], Goodarzi and al[2] achieved a computational study of diverse parameters influence (location, jet angle and velocity ratio) on lift and drag coefficient. A numerical research of suction and injection effect on aerodynamic characteristics is carried out by Shojarfard[3]. Four slots are installed on the trailing edge. They concluded that suction increase lift and injection decrease skin friction drag.

Piperas[4] published the results of investigation of boundary layer suction on wind turbine airfoil using CFD by an arrangement of different suction section. Akcayoz and tuner [5] examined the optimization of synthetic jet parameters on a NACA 0015 profile at different angle of attack. Their results indicate that the optimum location of the jet is better when approaching the leading edge and the angle of the optimum jet increases with the angle of incidence. The suction surface is more advantageous than adding a mass on the flow from the aircraft engine which outcome on higher lift and lower drag [6]. As well, Yousefi and al [7] reviewed an investigation for increase or decrease the lift and drag coefficient on the airfoil with suction and blowing.

The objective of this paper is studying the dynamic stall around a wind turbine blade and optimizing the aerodynamics performances numerically.

II. Governing equations

In this study, the flow is assumed to be steady, incompressible and two dimensional. So momentum and continuity equations become:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

$$\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial y}\left[u\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right]$$

$$\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[u\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right]$$

The viscous model used for modeling the turbulence is SPALLART ALLMARAS. This model is one equation model for the turbulent viscosity . It solves a transport equation for the kinematic eddy (turbulent) viscosity. In this original form, the SPALLART ALLMARAS model is effectively a low Reynolds number model and was designed specifically for aerospace applications involving wall bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients.

The turbulent viscosity is computed from [8]:

$$v_t=\tilde{v}f_{v1},\quad f_{v1}=\frac{\chi^3}{\chi^3+C_{v1}^3},\quad \chi:=\frac{\tilde{v}}{v}$$

And the transport equation for   is:

$$\frac{\partial \tilde{v}}{\partial t} + u_j \frac{\partial \tilde{v}}{\partial x_j} = \frac{1}{\sigma}\frac{\partial}{\partial x_k}\left[(v+\tilde{v}) + \frac{\partial \tilde{v}}{\partial x_k}\right] +$$
$$c_{b1}(1-f_{v2})\tilde{S}\tilde{v} - c_{w1} f_w \left(\frac{\tilde{v}}{d}\right)^2 + \frac{c_{b2}}{\sigma}\frac{\partial \tilde{v}}{\partial x_k}\frac{\partial \tilde{v}}{\partial x_k}$$

III. PARAMETERS SELECTION

A. *Blowing and suction devices*

The blowing and suction devices are modeled as a velocity conditions. The jet velocity as defined as:

$$u = A\cos(\theta_{jet} + \beta)$$

$$u = A\sin(\theta_{jet} + \beta)$$

$$A = \frac{u_{jet}}{U_\infty}$$

Where **A** is the amplitude (velocity ratio), **θ_{jet}** is the angle of jet between the local surface jet and jet entrance velocity direction, **β** is the angle between the free stream velocity direction and the local jet surface.

B. *Grid generation*

The grid around the NACA 4415 is generated by the gambit program. In order to get more details of the jet, a clustering grid is adopted near the jet orifice. A c-type grid around the airfoil is generated. The grid is depicted in fig 2.The total number of cells is adopted as 400 000 cells after checking the mesh independence with deferent size mesh (mesh1=120 456, mesh2=400 000, mesh3=638 032 cells) as is shown in the figures 3 and 4.

The airfoil geometry, slot position and dimensions are as follows: the chord length of the airfoil is 152 mm, single and dual jet width of 2.5% chord length is placed on the upper surface at 30% and 35%, respectively which simulating the blowing and suction under Re=2.18x10^5.

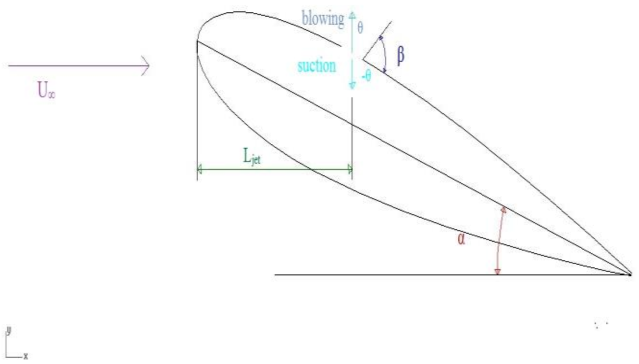

Fig1. Blowing and suction mechanism

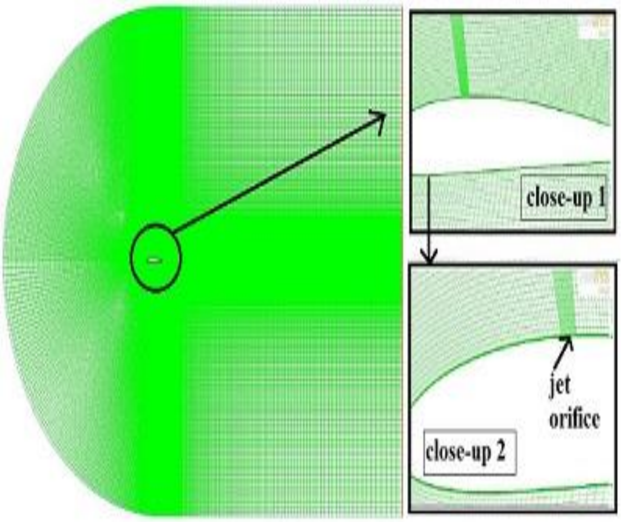

Fig 2. C-type grid

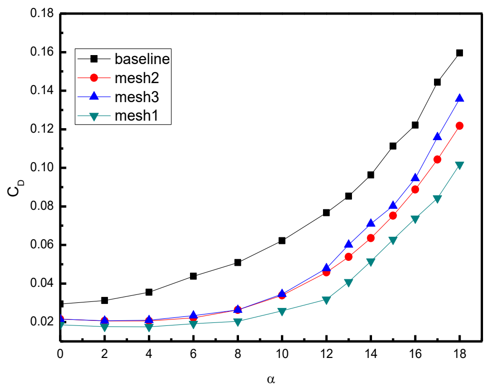

Fig 3. C-type grid

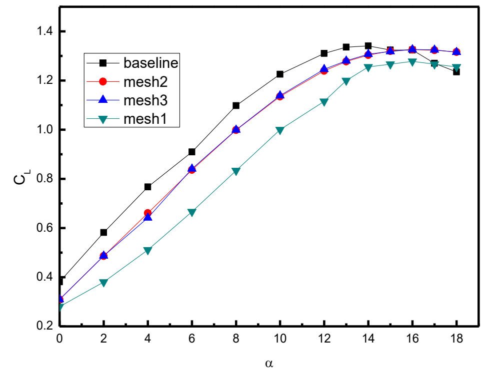

Fig 4. C-type grid

C. *Flow solver*

The commercial code fluent is used as a calculator. The Spallart Allmaras model was employed for modeling turbulence. The fluent code solves the Reynolds averaged Navier-stokes equations using the finite volume discretization. A second order upwind method was used to discretize the governing equations and the resulting system of equations is then solved using the SIMPLEC coupled solution procedure until convergence criteria of 10^{-6} reduction in all dependent variable residuals is satisfied.

IV. Results and Discussion

A. Amplitude effect

Simulations shows that increasing the jet velocity have an amplifying effect on lift generation mechanism and decreasing the production in drag. This is because the blowing has covered more surfaces and changing the velocity profiles in the boundary layer. The maximum change in lift is obtained by blowing for the amplitude of 1, which caused increase in the coefficient of lift about 2.7% on the incidence angle17°. The maximum reduction in the drag also obtained by blowing for the amplitude of 1, which reduces drag coefficient about 31% at 17° .

B. Effect of jet angle

The effects of jet blowing direction are assessed by locating the angle between the jet and the surface of the airfoil (jet angle) 0°, 30°and 45°. The fig7 and 8 show the magnitude of changed aerodynamic forces coefficients versus jet angle θ_{jet} with amplitude of 1. When the jet convects tangentially downstream, the jet control is almost effective than 30° and 45°. This because the high angles of jet causes a vertical component of blowing to the airfoil surface which generates an early separation of flow layers from the airfoil boundary. As it shown in the figure 7, the lift for high angles jet still increase with the incidence angles but a notable loss in lift force is about 32.03% and 91.82% for 30° and 45° comparing to the clean airfoil, respectively. But an enhancement in the drag coefficient is observed (figure8).

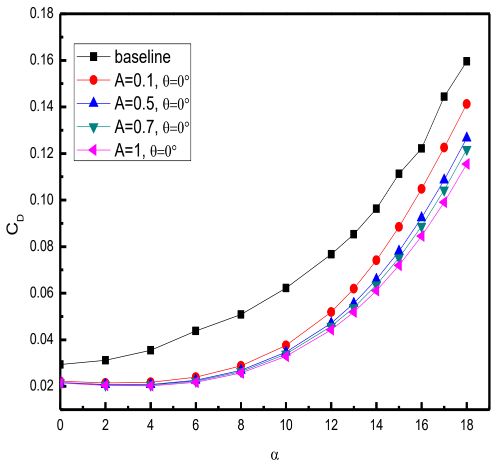

Fig5. The drag coefficient versus an incidence angle α

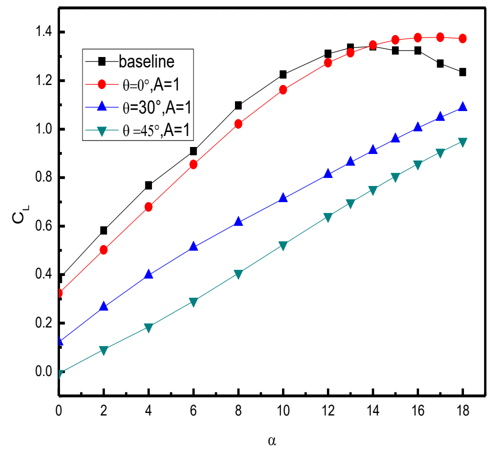

Fig7. The lift coefficient versus an incidence angle α

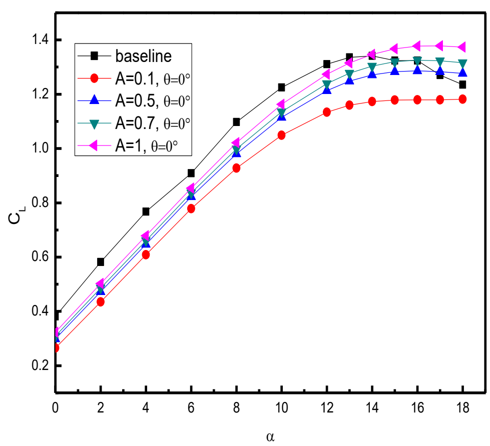

Fig6. The lift coefficient versus an incidence angle α

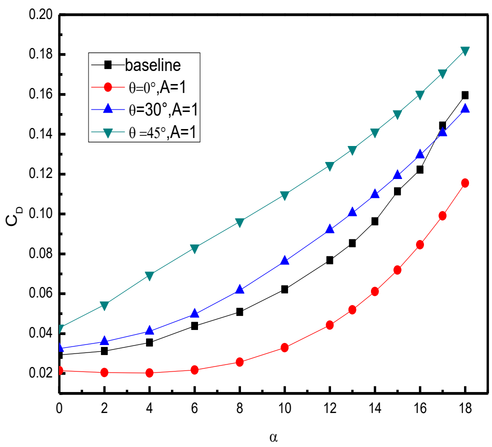

Fig8. The drag coefficient versus an incidence angle α

C. Dual jet effect

The fig. 9 and 10 Show the variation on lift and drag coefficients of the airfoil under dual and single jet, and the jet actuators have the same length (2.5% c), amplitude (0.1) and the jet angle (0°). Compared to single jet actuators, the combination of two jets actuators can lead to the improvement of aerodynamics characteristics of the airfoil, such as the maximum increment on lift coefficient 13% And maximum decrement of drag coefficient of about 13.9%, respectively. The stall angle is delayed from 14° to 17°. This improvement on lift becomes from the interaction of blowing momentum and the thickness of boundary layer.

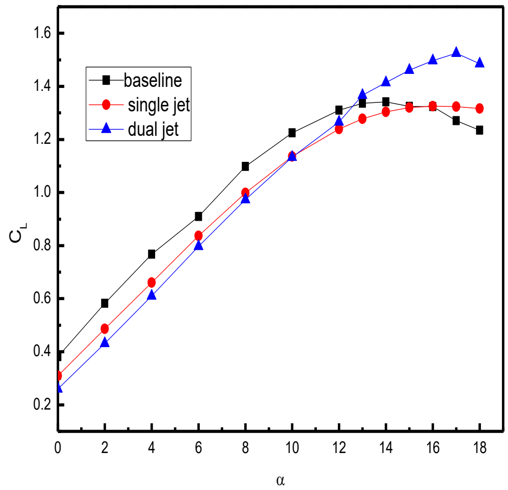

Fig9. The lift coefficient versus an incidence angle α

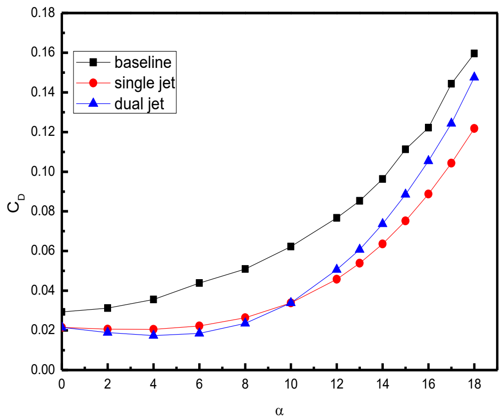

Fig10. The drag coefficient versus an incidence angle α

D. Comparison between suction and blowing

A study of controlling the flow via suction are inducted in this paper, in reason to compare it with a blowing one. For thus, a suction with an angle of -180° and for low amplitude (A=0.1) is studied.

As the fig.13 represents the lift to drag ratio, the suction , and blowing enhance the lift to drag ratio about 36.89% and 56.86%, respectively

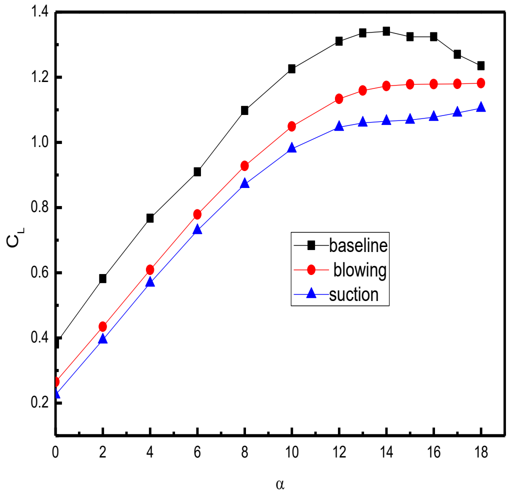

Fig11. The lift coefficient versus an incidence angle α

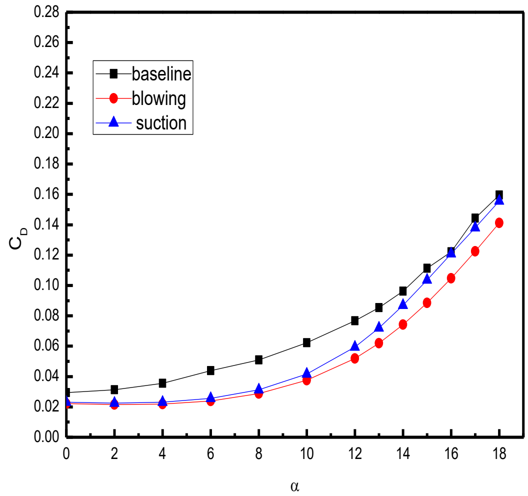

Fig12. The drag coefficient versus an incidence angle

jet angle. Another outstanding analyzed factor was the effect of two array jet on the aerodynamics characteristics. It's noteworthy elevation in lift coefficient for a dual jet.

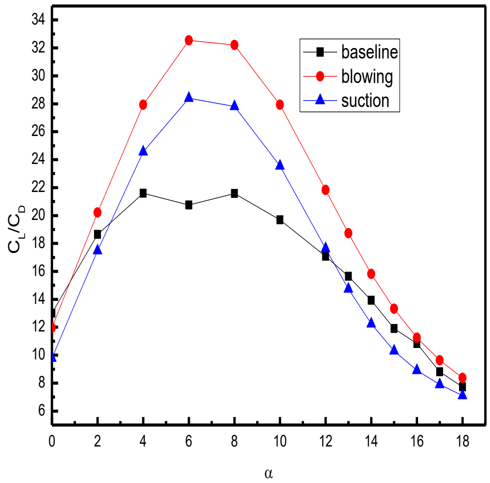

Fig13. The lift to drag ration versus the incidence angle α

V. CONCLUSIONS

This investigation focus on the optimization of blowing parameters as angle jet and amplitude. For this parametric study, the results show that lift coefficient enhance with raising the amplitude, and reducing the drag coefficient. In contrast, the lift decrease and drag increase with raising the

REFERENCES

[1] L.Huang, P.G.Huang, andR.P.Le Beau, "numerical study of blowing and suction control mechanism on NACA0012 airfoil", journalof aircraft, vol.41,2004.

[2] M.Goodarzi, M.Rahimi et R. Fereidouni, "investigation of active flow control over NACA 0015 airfoil via blowing", International Journal of Aerospace sciences, Iran,2012,1(4):57-63.

[3] M.H.Shojaefard,A.R.Avanesians and M.Ghaffarpour,"numerical investigation of flow control by suction and injrction on a subsonic airfoil",American journal of applied sciences2(10):1474-1480,2005.

[4] A.T. Piperas, "investigation of bouandary layer suction on a wind turbine airfoil using CFD", master thesis, technical university of denmark, denmark, 2010.

[5] E. Akcayoz and I.H Tunner, "numerical investigation of flow control over an airfoil using synthetic jets and its optimization", presented at International Aerospace Conference,Turkey,2009.

[6] Wong C. and K. Kontis,"flow control by spanwise blowing on NACA0012", journal of aircraft, 2007.44(1):p.337-340.

[7] K.Yousefi, S.R. Saleh et P. Zahedi,"Numerical investigation of suction and length of suction jet on aerodynamic characteristics of NACA 0012 airfoil",international journal of materials, mechanics and manufacturing, vol.1,no.2,may 2013.

[8] Spalart, P.R. and S.R. Allmaras, Aone equation turbulence model for aerodynamic flows. La recherche aerospatial, 1994