JOURNAL OF MECHANICAL ENGINEERING AND SCIENCES (JMES)
ISSN: 2289-4659 e-ISSN: 2231-8380
VOL. 16, ISSUE 1, 8811 – 8819
DOI: https://doi.org/10.15282/jmes.16.1.2022.13.0696

ORIGINAL ARTICLE

Influence of different blowing parameters on flow control on an airfoil

S. Abbasi

Faculty of Mechanical Engineering, Arak University of Technology, 3818141167, Arak, Iran
Phone: +8433400663

ABSTRACT – A numerical study on flow separation control is conducted for an airfoil with a blowing jet. In this regard, the effect of different parameters of air blowing on stall controlling and flow structure over NACA 0012 is followed. RANS equations were employed in conjunction k-ω-SST turbulent model. To validate the numerical results, they are compared with reported experiments, and good agreement is observed. In this paper, the effects of different parameters like blowing location, angle of jets, and local jet velocity are investigated in various cases. Blowing location in the range of 0.1-0.9 of chord length from the leading edge, local jet velocities of 0.1, 0.3, and 0.5 of free stream velocity, and angles of 30°, 60°, and 90° are studied. Results reveal that blowing near airfoil ending increases the ratio of lift/drag coefficients. Furthermore, blowing near the angle of 30o prove a positive effect on aerodynamic characteristics. Using a jet velocity equal to the half of free stream velocity shows a favorable effect on blowing. Blowing the flow never revealed to weaken vortices, and therefore it is not recommended to use blowing jet.

ARTICLE HISTORY
Received: 13th Aug. 2021
Revised: 07th Dec. 2021
Accepted: 06th Jan. 2022

KEYWORDS
Flow control;
active control;
numerical simulation;
separation

INTRODUCTION

Stall occurs when a lifting surface loses its aerodynamic lift. A well-known reason for the stall is the flow separation from the surface. An airfoil would be the best example to describe the stall phenomenon. As the angle of attack increases beyond the stall angle a stall, the flow is detached from the airfoil surface. The flow may separate from the trailing edge or the leading edge, depending on the specific design of an airfoil and the flow condition including the Reynolds and Mach numbers [1]. To avoid this, flow control mechanisms are introduced. Flow control aims to change the flow field in a way that flow separation from the airfoil surface can be delayed so that the drag force decreases and the lift force increases. First, passive flow control methods like geometry modification were studied. Later, active flow control strategies, like suction and blowing control, were investigated. In these methods by increasing boundary layer thickness, the required adverse pressure gradient to separate flow is dramatically increased and therefore the separation is avoided [2]. The blowing method consists of adding energy to the lower boundary layer by blowing air through slots and energizing the flow near the wall which enables it to overcome a larger pressure gradient. Jet entrainment had been shown to enhance the lift generated by airfoils.

Prandtl was a pioneer in studying the blowing of the boundary layer to investigate its effect on the flow field. He applied blowing around a cylinder and succeeded to delay separation [3]. Primary experimental investigations on flow separation control on airfoils were reported in the 1930s when the effect of suction on boundary layer separation using slots was studied. In the first flight test, 17 suction slots were installed between 20-60% of B-18 plane chord length[4]. Theories of suction for boundary layer and inverse flow were studied by Abzalilov et al. [5]. Rosas [6] numerically simulated flow separation control using synthetic jets where lift coefficient increased to 93% for NACA 0012 airfoil. Wu et al. [7] controlled the flow by using a slot near NACA 0012 airfoil leading edge and by applying suction and blowing. Nae [8] at an angle of attack of 13o, investigated the effect of the compound jet at 10% of chord length. Results of these studies showed that suction and blowing near to the leading edge increase lift coefficient and decrease drag coefficient. Peiqing et al. [9] successfully delayed the flow separation by applying a slot on the airfoil and by blowing.

Ortmann and Kahler [10] investigated the turbulent boundary layer using high-speed flow blows. It was found that high speed blowing in the flow direction is not effective in the increase of the lift force. Genc et al. [11] numerically investigated the effect of suction and blowing on NACA 2415 airfoil in a transient state. The separation bubble in suction and blowing simulation was not completely vanished, it was reduced. They also showed that if several blowing jets were used, better results would be obtained than one jet use. You and Moin [12] used the LES simulation method to study flow separation with a synthetic jet on NACA 0015 and they could increase the lift coefficient by 70% and decrease the drag coefficient by 18%. Yousefi et al. [13] studied the jet length effect on the NACA 0012 aerodynamics coefficients. They found that with an increase in jet length, the ratio of lift to drag raises, and the best length equals 2.5% of airfoil chord length.

Most of the reported studies concentrated on applying a high-speed jet near the leading edge with low angles of attack. But, the effect of influential parameters of jet-like location, velocity, and angle has not been investigated yet. In this paper, the effect of controlling parameters such as location, velocity, and jet angle is studied in a wider range. Moreover, to

*CORRESPONDING AUTHOR | S. Abbasi | ✉ s_abbasi@arakut.ac.ir

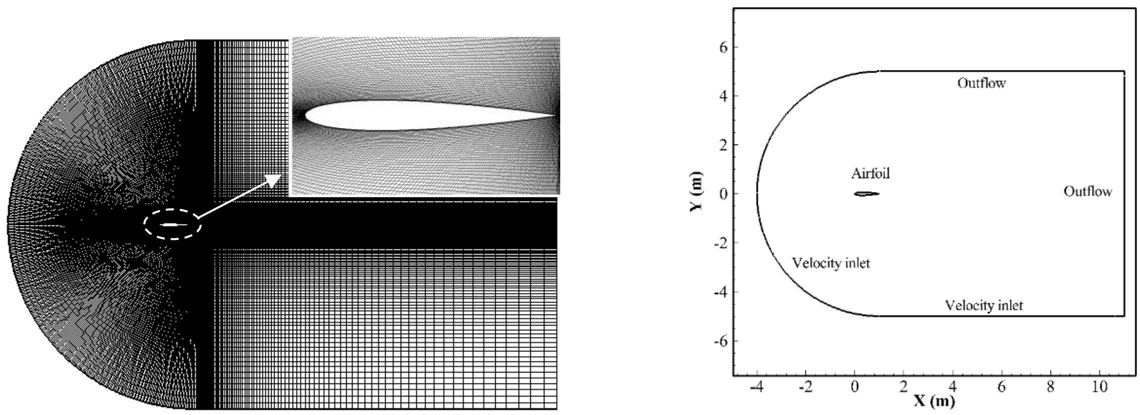
understand the effect of applying jet, flow structure, aerodynamics coefficients variation, shear stress variation, and velocity field over airfoil are critically analyzed.

NUMERICAL SIMULATION

The solver selected in this study is Ansys- Fluent 19 commercial software capable of simulating viscous flows. The software uses the finite volume method to solve the governing equations including momentum and continuity. In this study, the flow is supposed to be steady-state, incompressible, and two-dimensional. The continuity and momentum equations are used as governing equations.

a structured grid system has been employed. in order to reduce the effect of numerical diffusion on the solution, for the spatial discretization of the governing equations, a second-order upwind scheme is employed for the discretization of momentum, turbulent kinetic energy, and specific dissipation rate. Stability, economy, and appropriate precision for a wide range of turbulent flows justifies k-ω-sst turbulence model popularity in the simulation of turbulent heat transfer and flow in the industry[14-16]. The SIMPLE algorithm is used to couple the velocity field with the pressure field [17].

The geometry of this study is NACA 0012 with 1 meter of chord length. Airflow with Reynolds number of 5×10^5 is assumed. Flow analysis is performed using commercial Ansys Fluent software. Boundary conditions and structured grids are shown in Figure 1. To couple velocity with pressure field, the Simple method is used.

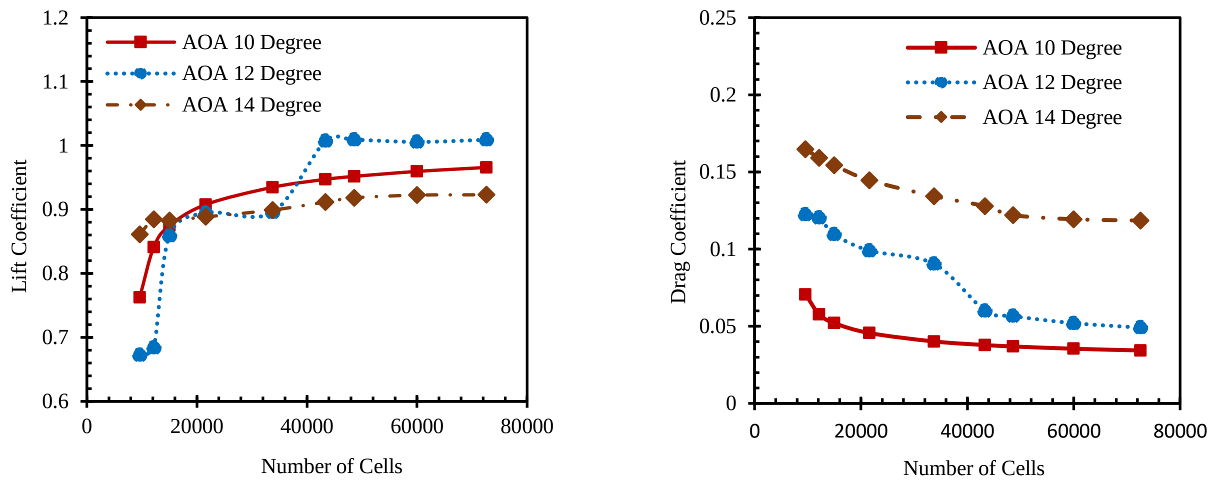


(a) (b)

Figure 1. (a) Airfoil mesh and (b) boundary conditions of flow over the airfoil

To verify the results, the experimental output of two reported studies available in the literature is considered. These experimental results include lift and drag coefficients for various angles of attack. Figure 2 shows a comparison between numerical results and experimental reports. Good agreement can be observed from both results.

To make results independent from the number of cells of a given mesh, different fine and coarse mesh is used to perform a flow analysis with various angles of attack (10, 12 and 14 degrees). The results are showed in Figure 2. It can be observed that exceeding the number of cells from 50000 does not affect the lift and drag coefficients. Therefore, mesh with cells number of 50000 is used for the analysis.



(a) (b)

Figure 2. Mesh sensitivity analysis for: (a) lift Coefficient and (b) drag coefficient

VALIDATION OF THE RESULTS

To verify the results, experimental output of two reported studies available in the literature is considered [18, 19]. These experimental results include lift and drag coefficients for various angles of attack. Figure 3 shows the comparison between numerical resuslts and experimental reports. Good agreement can be observed from both results.

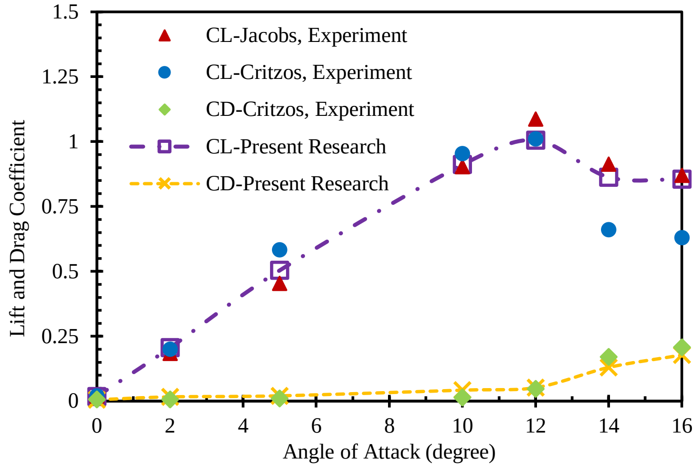


Figure 3. Comparison between numerical and experimental lift and drag coefficients [18, 19].

INVESTIGATED PARAMETERS

The effect of different influential parameters of flow blowing including flow jet location (Lj), jet angle (θ), and jet intensity (I) on flow behavior and aerodynamics coefficients is scrutinized (Eqs. (1)-(6)). Figure 4 shows introduced parameters. Jet intensity is the ratio of jet velocity to free stream velocity. Location of the jet is specified related to the airfoil chord. θ shows the angle between jet velocity vector and local jet surface and α is the angle between local jet surface and horizontal direction. positive θ is used for blowing.

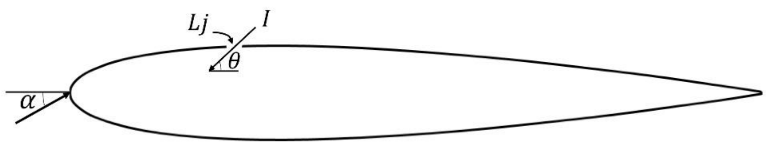


Figure 4. Jet parameters on NACA 0012 airfoil

$$\boldsymbol{I = \frac{u_{jet}}{u_\infty}} \tag{1}$$

$$\boldsymbol{u = u_{jet}\ cos(\theta + \alpha)} \tag{2}$$

$$\boldsymbol{v = u_{jet}\ sin(\theta + \alpha)} \tag{3}$$

$$\boldsymbol{C_\mu = \frac{\rho.h.{u_{jet}}^2}{\rho.c.{u_\infty}^2} = \frac{h}{c} \times (\frac{u_{jet}}{u_\infty})^2 = \frac{h}{c} \times I^2} \tag{4}$$

In Eq. (4), C_μ is jet momentum coefficient, ρ is density, h is jet length and c is airfoil chord length.

Dannenberg and Weiberg [18] findings state that optimum jet length is about 2.5% of the chord length. They also found that increasing jet length more than 2.5% of the chord length has a neglecting effect on lift coefficient. Therefore, here h is assumed to be constant. So, Eq. (4) is rewritten as presented below.

$$H = \frac{h}{c} = 0.025 \tag{5}$$

$$C_\mu = H.I^2 = 0.025\ I^2 \tag{6}$$

From Eq. (4) it can be inferred that momentum coefficient is a function of jet intensity and local jet length. Since jet length is assumed to be constant, momentum coefficient is only dependent on jet intensity.

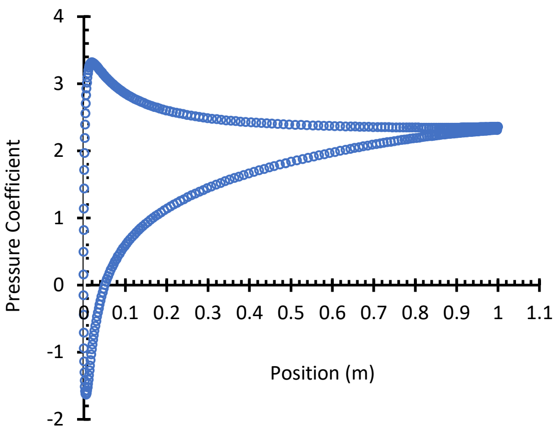
RESULTS

The aim of this paper is to study and modify the variation of aerodynamics coefficients. Therefore, the effect of blowing of the flow is investigated. Influential parameters of flow blowing and their effect on flow structure and aerodynamics coefficients are analyzed**.**

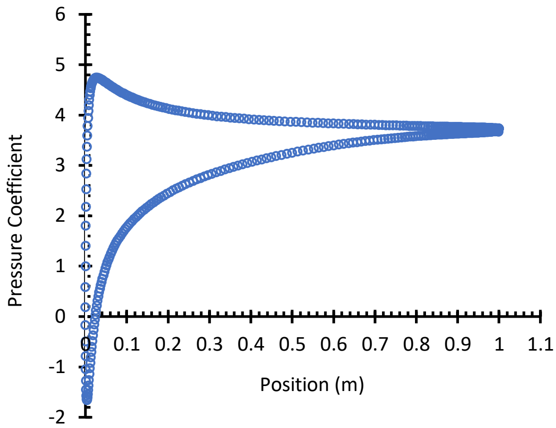
Effect of Blowing on Aerodynamics Coefficients

One of the prominent factors in aerodynamics is the ratio of lift to drag (L/D) which directly affects many functional properties of systems. In a given condition, the higher ratio is desired [20] . Therefore, the objective is to increase the lift to drag ratio. The lift coefficient increases by increasing the angle of attack. When stall is reached, more increase in angle of attack reduces the lift coefficient. It is attributed to flow separation from the surface of the airfoil.

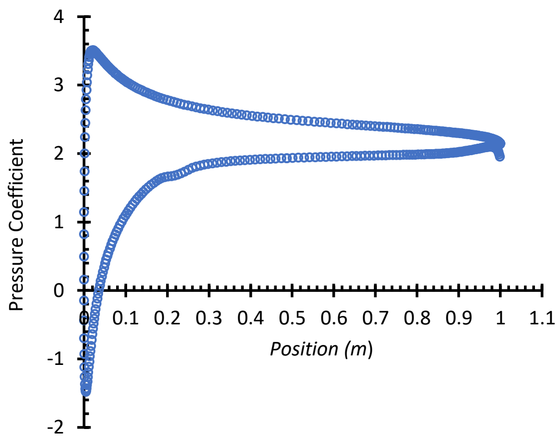
According to Figure 3 by increasing the angle of attack up to 12°, the lift coefficient increases. After 12°, the lift coefficient has a descending behavior. Pressure coefficients are illustrated in Figure 5, for various angles of attack (before, during and after the stall).



(a) Angle of attack $\mathbf{10}°$



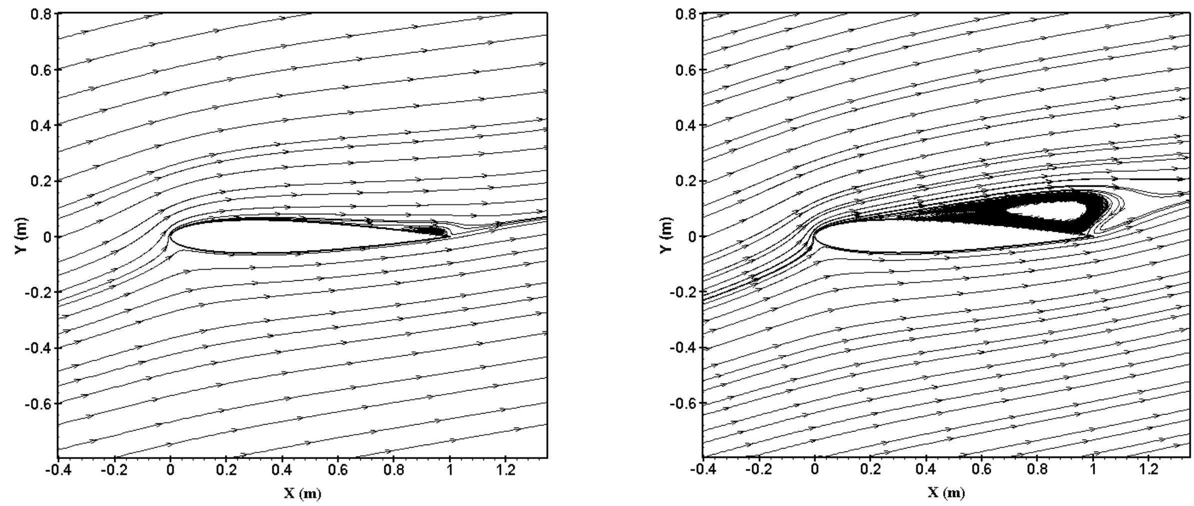
(b) Angle of attack $\mathbf{12}°$



(c) Angle of attack $\mathbf{14}°$

Figure 5. Variation of pressure coefficients on the pressure and suction sides of the airfoil

Figure 6 shows streamlines over airfoil at attack angles of 12 and 14 degrees. As illustrated in Figure 6, by increasing the attack angle after 12 degree, flow separation intensifies which leads to an increase in drag coefficient and decrease in lift coefficient.



(a) angle of attack $\mathbf{12°}$ (stall) (b) angle of attack $\mathbf{14°}$

Figure 6. Streamlines over airfoil

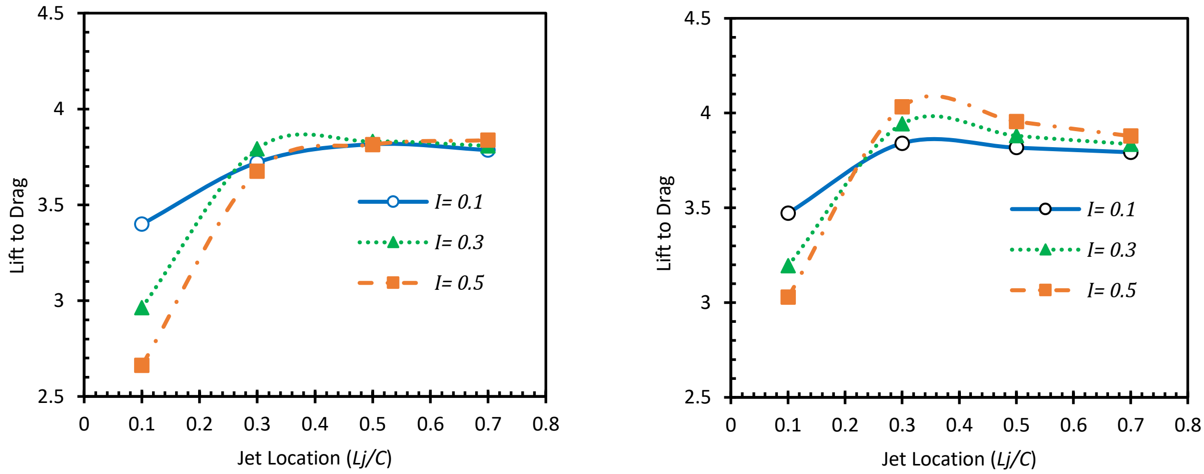
Flow Separation Control

Flow separation has a negative effect on aerodynamics characteristics and it is desired to apply methods to delay flow separation and increase L/D ratio separation. One flow control is blowing method.

Flow Blowing

In order to investigate the effect of blowing, a slot with 2.5% of length of airfoil chord was produced at different locations and the effect of injection on flow structure and aerodynamics coefficients was studied. L_j is jet location from airfoil leading edge, I is suction jet intensity and θ is suction jet angle corresponding to horizontal direction.

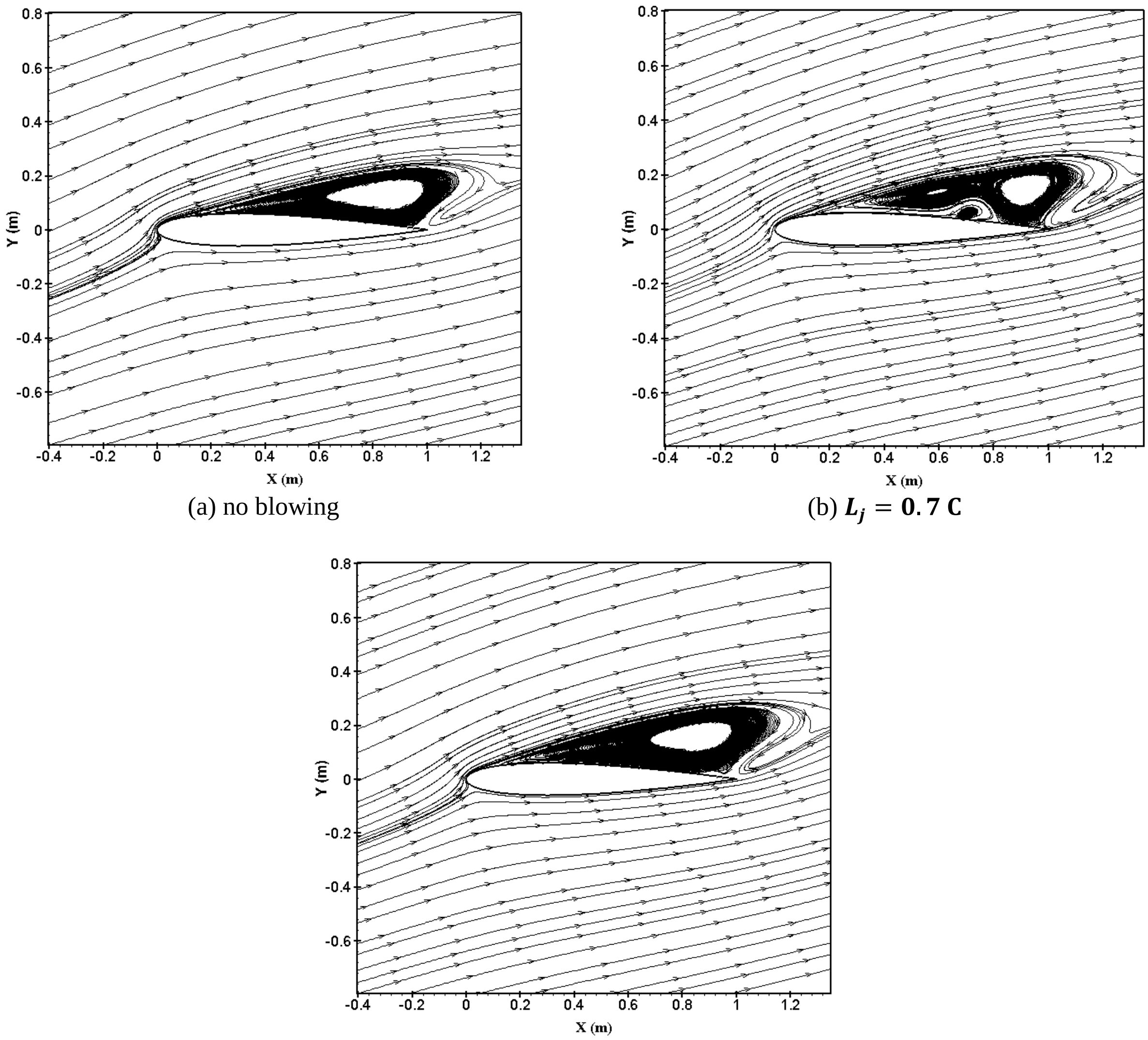
Figure 7 shows the lift to drag ratio variation for $0.1\text{C} \leq L_j \leq 0.7\text{C}$, $0.1 \leq I \leq 0.3$ and $30° \leq \theta \leq 90°$. It is clear that blowing near to the leading edge not only doesn't improve the lift to drag ratio but also it decreases L/D in comparison with no blow $(\text{L/D} = 4.8)$. However as jet angle decreases and jet location becomes closer to the trailing edge, the ratio of lift to drag increases. Increasing jet intensity near to leading edge has an adverse effect and leads to a decrease in L/D ratio. But, around ending of the airfoil, increasing jet intensity raises the lift coefficient and reduces the drag coefficient. As a whole, in $0.1\text{C} \leq L_j \leq 0.3\text{C}$ applying blowing jet increases L/D and in $L_j > 0.3\text{C}$ no considerable effect of blowing jet is observed.



(a) $\boldsymbol{\theta = 90°}$ (b) $\boldsymbol{\theta = 60°}$

Figure 7. Lift to drag ratio variation at different blowing angles

Figure 8 depicts the effect of blowing jet on streamlines at $\theta = 30°$ and $I = 0.5$. As can be observed from Figure 13, using blowing jet increases vortices in flow separation area and therefore applying blowing jet does not make a positive effect on weakening vortices. Figure 9 shows static pressure coefficient distribution on suction- blowing sides of the airfoil in different states with no- blowing and blowing jets. It can be conclude that when blowing jet locates at $L_j = 0.7$, static pressure coefficient increases. Moreover, pressure coefficients for $L_j = 0.7$ and $L_j = 0.9$ are almost the same and they both show lower values comparing with no- blowing state. This proves their ineffectiveness.



(a) no blowing (b) $L_j = 0.7$ C

(c) $L_j = 0.9$ C

Figure 8. Streamlines at $I = 0.5$, $\theta = 30^o$ and angle of attack 16°

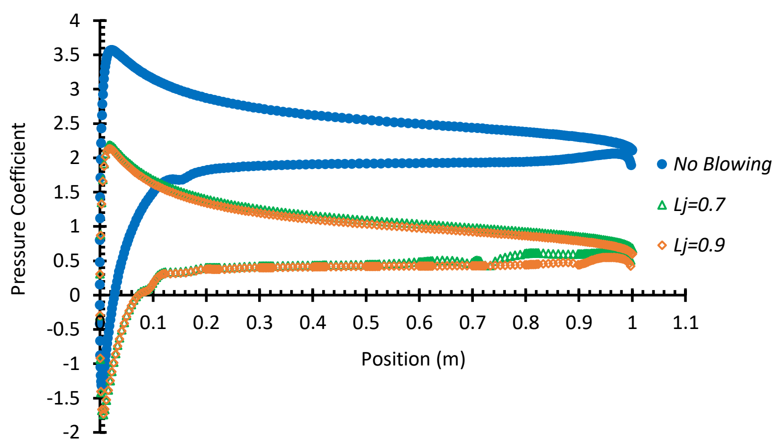
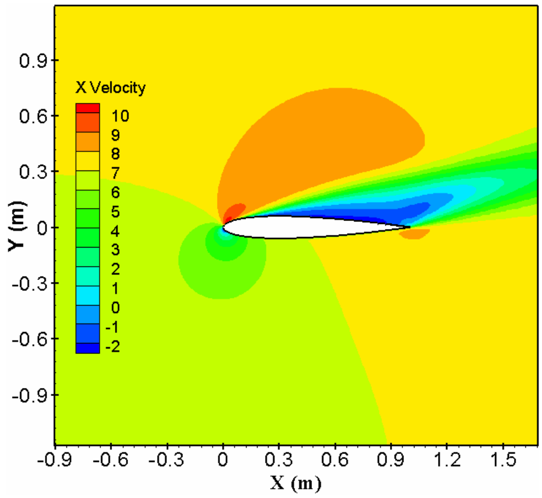
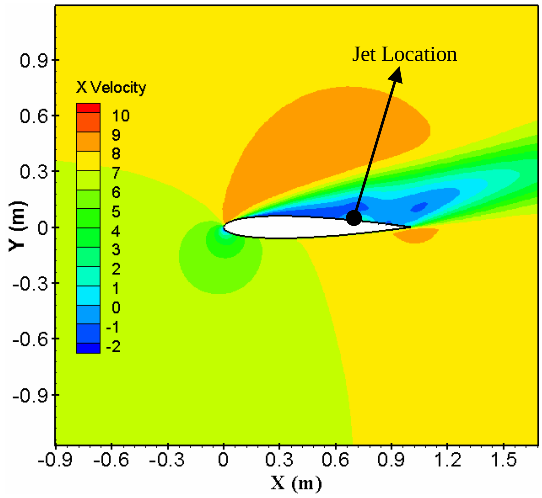


Figure 9. Pressure distribution over airfoil at angle of attack 16°

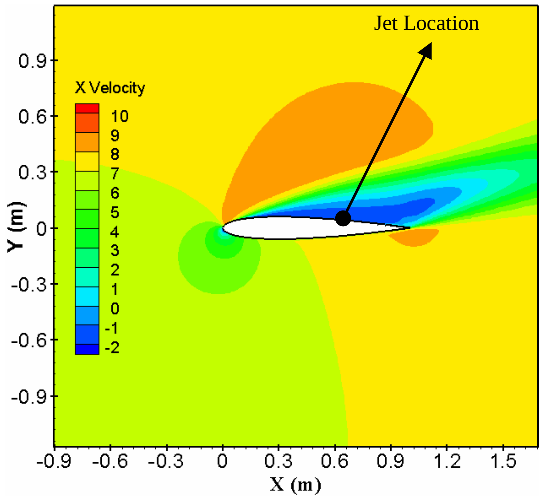
Figure 10 shows airfoil velocity field with and without blowing jet at angle of attack 16°. Unlike the suction jet, the blowing jet does not have a considerable effect on the velocity field, and at its location it just slightly boosts up the flow velocity.



(a) no blowing



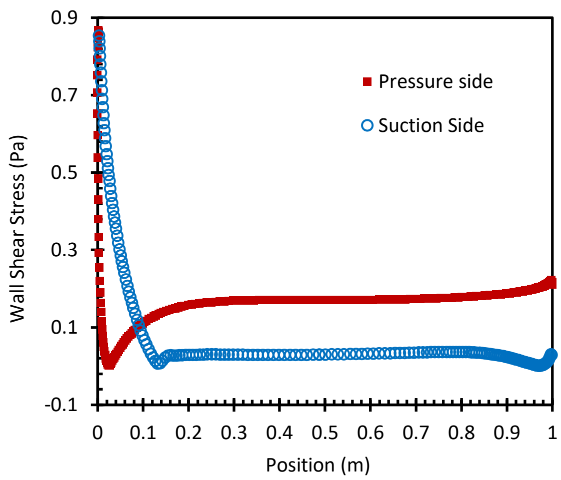
(b) $L_j = 0.7$ C



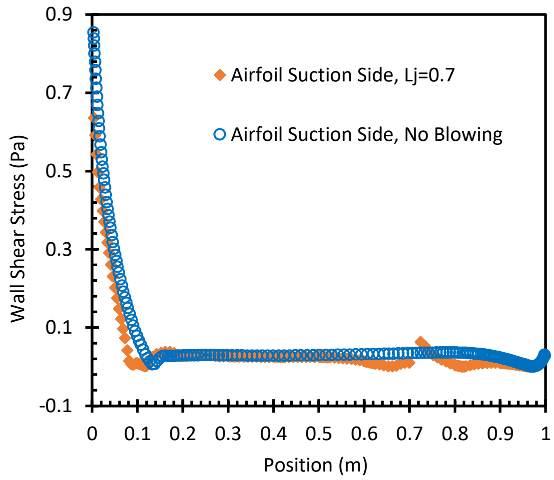
(c) $L_j = 0.9$ C

Figure 10. Velocity contour at $I = 0.5$, $\theta = 30^o$ and angle of attack 16°

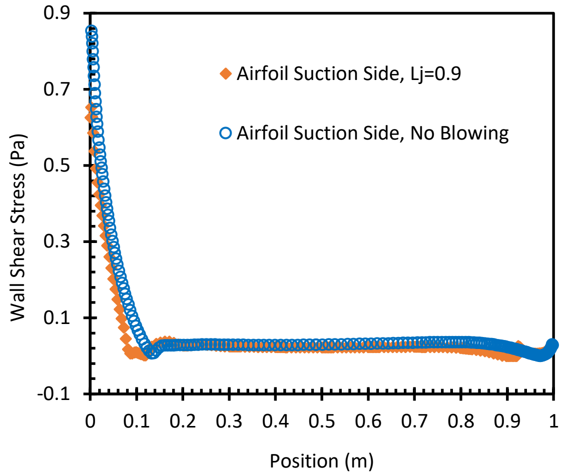
Findings show that shear stress on the pressure side does not change due to the suction jet location on the airfoil suction side. Figure 11 shows shear stress of airfoil wall on the suction side for both with- and without-blowing-jet conditions at different locations. As can be seen there are oscillations on the diagram at blowing jet points. This is attributed to an increase in the velocity gradient at injection points. Applying blowing can remarkably reduce shear stress near to the leading edge. Unlike suction, blowing jet reduces shear stress in entire airfoil even though this reduction is very small.



(a) no blowing



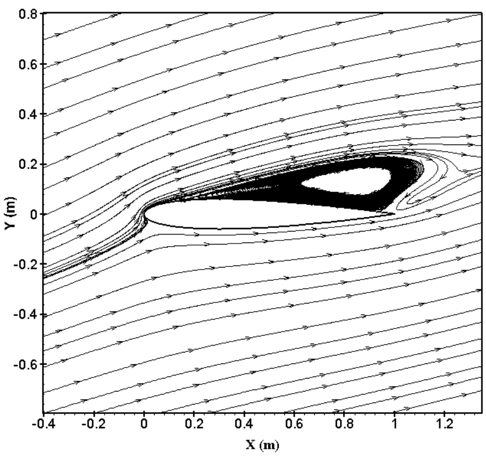
(b) $L_j = 0.7$ C



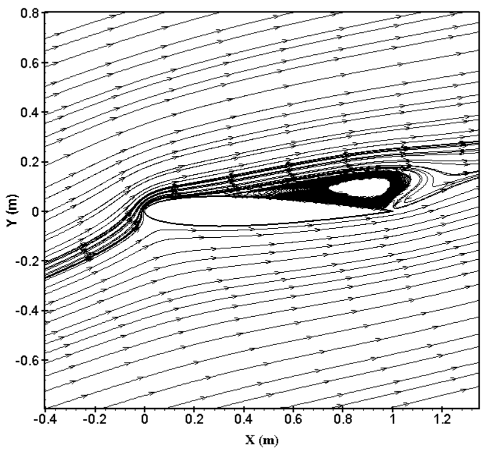
c) $L_j = 0.9$ C

Figure 11. Shear stress of airfoil wall on suction side

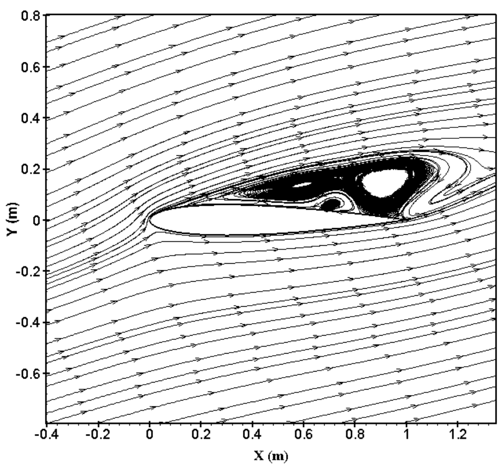
Figure 12 shows streamlines for both suction and blowing in their best conditions and compares them with no-suction and no-blowing states. As can be seen, the use of suction makes vortices smaller. Perpendicular suction with intensity $I = 0.5$ possesses the highest improvement in aerodynamics coefficients so that L/D increases up to 32%. However, blowing does not have a positive effect on the flow. Results show that the best case of blowing is the one with the angle of $\theta = 30°$ and the intensity of $I = 0.5$ where, nevertheless, reduces L/D about 17%.



(a) no jet



(b) suction at $L_j = 0.1$ C and $\theta = -90^o$



(c) blowing at $L_j = 0.7$ C and $\theta = 30^o$

Figure 12. Streamlines at $I = 0.5C$ and angle of attack 16°.

CONCLUSION

In this paper the effect of different parameters of a flow control method blowing- on flow structure and aerodynamics coefficients is studied. Numerical analysis on NACA 0012 airfoil in a steady state with Reynolds of 500000 was taken into account using commercial FLUENT solver. Results can be highlighted as fallows:

- Numerical results are in good agreement with reported experimental results.
- Location and intensity of jets have the highest and lowest effects on the flow structure respectively. In other words, these two parameters can be considered for flow control.
- Locating blowing jets near to the airfoil ending leads to better results in increasing the lift to drag coefficients ratio. However, generally, injecting jet does not have a favorable effect on boundary layer control and flow separation.
- Reducing blowing angle from 90º to 30° introduces an improvement on the flow control in a way that injection at the angle of 30º better increases the L/D ratio.
- Optimum state of blowing is at $L_j = 0.7$ C and $\theta = 30°$ even though L/D ratio decreases by 17%.
- Applying blowing does not have a remarkable effect on the velocity over the airfoil.

REFERENCES

[1] J. D. Anderson Jr, *Fundamentals of Aerodynamics*: 5th Edition, *McGraw-Hill Education*, 2010.

[2] J.A.D. Ackroyd, B.P. Axcell, and A. Ruban, "Early developments of modern aerodynamics," *Elsevier Science & Technology*, 2001.

[3] M. Gad-el-Hak, "Flow control: Passive, active, and reactive flow management," 1st Edition, *Cambridge University Press*, 2000.

[4] A.L. Braslow, "A history of suction-type laminar-flow control with emphasis on flight research," *CreateSpace Independent Publishing Platform*, 2013.

[5] D. Abzalilov, L. Aksent'Yev, and N. Il'Inskii, "The inverse boundary-value problem for an airfoil with a suction slot," *Journal of applied mathematics and mechanics,* vol. 61, pp. 75-82, 1997.

[6] C. R. Rosas, "Numerical simulation of flow separation control by oscillatory fluid injection," *PhD Theses*, Texas A&M University, 2005.

[7] J.-Z. Wu, X.-Y. Lu, A. G. Denny, M. Fan, and J.-M. Wu, "Post-stall flow control on an airfoil by local unsteady forcing," *Journal of Fluid Mechanics,* vol. 371, pp. 21-58, 1998.

[8] C. Nae, "Synthetic jets influence on NACA 0012 airfoil at high angles of attack," in *23rd Atmospheric Flight Mechanics Conference*, Boston, USA, 1998.

[9] P. Q. Liu, Y. X. Cui, L. Wang, and Q. L. Qu, "Computational investigation of the slat blowing control for high-lift airfoil," *Applied Mechanics and Materials*, vol. 138-139, pp. 223-228, 2012

[10] J. Ortmanns, C. Kähler, and B. Weg, "Investigation of pulsed actuators for active flow control using phase locked stereoscopic particle image velocimetry," in *International Symposium on Applications of Laser Techniques in Fluid Mechanics*, Lisbon, Portugal, 2004.

[11] M. S. Genç, Ü. Kaynak, and H. Yapici, "Performance of transition model for predicting low Re aerofoil flows without/with single and simultaneous blowing and suction," *European Journal of Mechanics-B/Fluids*, vol. 30, pp. 218-235, 2011.

[12] A. T. Piperas, "Investigation of boundary layer suction on a wind turbine airfoil using CFD," *Master Thesis, DTU Mechanical Engineering*, 2010.

[13] D. You and P. Moin, "Active control of flow separation over an airfoil using synthetic jets," *Journal of Fluids and Structures,* vol. 24, pp. 1349-1357, 2008.

[14] W.-T. Chong, W. K. Muzammil, K.-H. Wong, C.-T. Wang, M. Gwani, Y.-J. Chu, *et al.*, "Cross axis wind turbine: Pushing the limit of wind turbine technology with complementary design," *Applied Energy,* vol. 207, pp. 78-95, 2017.

[15] P. C. Rocha, H. B. Rocha, F. M. Carneiro, M. V. Da Silva, and A. V. Bueno, "k–ω SST (shear stress transport) turbulence model calibration: A case study on a small scale horizontal axis wind turbine," *Energy,* vol. 65, pp. 412-418, 2014.

[16] J. Zhang, W. Chu, H. Zhang, Y. Wu, and X. Dong, "Numerical and experimental investigations of the unsteady aerodynamics and aero-acoustics characteristics of a backward curved blade centrifugal fan," *Applied Acoustics,* vol. 110, pp. 256-267, 2016.

[17] L. Qiao, S. Wei, R. Gu, X. Quan, and Y. Yang, "The investigation of the airfoil for the small wind turbine based on the seagull airfoil," in *2011 Asia-Pacific Power and Energy Engineering Conference*, 2011, pp. 1-4.

[18] C. C. Critzos, H. H. Heyson, and R. W. Boswinkle, "Aerodynamic characteristics of naca 0012 airfoil section at angles of attack from 0 to 180," *National Aeronautics and Space Administration Washington DC*, Technical Note, 1955.

[19] E. N. Jacobs and A. Sherman, "Airfoil section characteristics as affected by variations of the Reynolds number," *NACA Technical Report*, vol. 586, pp. 227-267, 1937.

[20] R. E. Dannenberg and J. A. Weiberg, "Section characteristics of a 10.5-percent-thick airfoil with area suction as affected by chordwise distribution of permeability," *National Aeronautics and Space Administration Moffett Field Ca Ames Research Center*, Technical Note, 1952.