Numerical simulation of flow control in lift-increase effect using air-blowing

Xu Kao, Zhu Xiaojun*, Zhu Jianghong and Li Feng
China Academy of Aerospace Aerodynamics, Beijing, 100074, China

*E-mail: zxj011010826@163.com

Abstract. In this study, numerical simulation method is applied to analyze the aerodynamic characteristics of the airfoil influenced by the parameters of the jet port. The NACA0012 airfoil is studied by using the incompressible Navier-Stokes equations with the structural grid. The influence of air-blowing speed, width, position of the jet and angle of attack on the aerodynamic characteristics of the airfoil is presented. The results show that, for the airfoil, the lift coefficient can be impacted by blowing speed, width and position of the jet to a certain extent. Jet control can reduce the stall angle of attack of the airfoil, but before the stall angle of attack, the effect of lift-increase is almost same at different angle of attack. A basis for the jet design and the selection of jet position along the wing is provide.

1. Introduction
In terms of reducing drag and increasing lift, the application of flow control have attracted the attention of a large number of scholars [1]. Jet blowing vertically or aslant from the lower surface of the wing, can change the pressure distribution of wing, increase the pressure difference of upper and lower surfaces, resulting in the increase of lift. At present, experiment and numerical simulation are two main analysis methods for research of jet control [2]. To accurately simulate the effect of jet control on lift, the first thing is to be able to simulate the flow field with a jet control accurately and meticulously [3]. As the flow field with a jet control is quiet complicated and disturbing, the numerical simulation is confronted with many problems. Most of the research are based on wind tunnel experiment so far. With the fast development of the computer technology, the numerical simulation of the flow field around airfoil with high angle of attack become possible [4]. Based on this issue, numerical simulation can be conducted on the research of jet effect on the wing, and the mechanism of the lift-increase.

This kind of jet control has been applied in AV-8 Harrier aircraft preliminarily. It can help aircraft achieve short-range vertical takeoff and landing, so the length of track is reduced. There are two factors influencing the application of jet control method so far. The first is to get the steady gas source. The second is that full wingspan jet can bring big a significant impact on the safety of the wing structure [5]. The first question can be solved by entraining gas from the engine. But the second question haven’t been solved well yet. Based on the above, only a part of wingspan jet control have been put forward in this paper. Local jet control can avoid problems of structural safety caused by the full wingspan jet control. What’s more, the best location of the jet port on the wingspan can be found and the effects of different jet velocity on the aerodynamic characteristics of the wing can be studied through the simulation of the flow field with jet control.
2. Jet control numerical simulation method

2.1. Control equation and turbulent model

In this paper, the impressive RANS Navier-Stokes equation is adopted as control equation to the flow field numerical simulation [6], it can be expressed as

Continuous equation \( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = 0 \) \hspace{1cm} (1)

Momentum equation \( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_i \partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} \) \hspace{1cm} (2)

The finite volume method is used to carry out the space discretization, and the convection terms adopt two order upwind schemes, diffusion terms use central difference scheme. Pressure based simple algorithm is used for pressure-velocity coupling, the discrete algebraic equations are solved by Gauss-Seidel algebraic method [7]. Spalart-Allmaras (SA) turbulent model is used as turbulent model. SA turbulent model is a relatively simple one equation model which solved a transport equation for the eddy viscosity. This model is more suitable for the flow problem with a wall limit [8]. It can give a good result to the boundary layer problem with the inverse pressure gradient. It often used in aerodynamic problems, such as the flow field analysis to aircraft, airfoil and so on.

2.2. Boundary conditions

Airfoil and wing surface adopt wall boundary condition, meet the conditions of no-slip, insulation, surface normal gradient is zero. Jet port adopt the mass flow inlet as the boundary condition. Mass flow is defined as \( \dot{m} = \rho u s \). Among them \( \rho \) is the density of the fluid of jet, which is equal to the air flow, \( s \) is the area of the jet, \( u \) is the velocity of the jet. For wing, the plane of symmetry adopt symmetric boundary condition. In addition to airfoil surface, symmetric plane and jet port, the other boundaries of flow field all be set as pressure far field.

2.3. Physical model and grid division

![Figure 1. Local refinement of 2D grids.](image)

![Figure 2. 3D mesh model.](image)

Two-dimensional model adopts the NACA0012 airfoil. The chord length of the airfoil is one meter. Computational grid adopt the C-H type grid, which is shown in figure 1. Around the boundaries of the computational domain have a distance of 20 times the chord length from the airfoil surface, airfoil downstream direction is taken as 20 times the chord length also. The total number of the grid is 43586 after the encryption of jet. The height of the most inner mesh is \( 10^{-2} m \), \( y^+ \approx 1 \).

Three-dimensional model adopts finite wingspan model based on the NACA0012 airfoil. As is shown in the figure 2, wing root tangential section(z=0) is the symmetric plane, the reference area of the semispan is 3 square meters. Half length of the computational domain is 12 meters. So the wing tip(z=3m) is in the flow field, the effect of the wing tip vortex is taken in to the consideration.

3. 2D model numerical simulation result and analysis

3.1. Numerical simulation algorithm validation
When $Ma = 0.2, Re = 4.6 \times 10^6$ and there is no jet control on NACA0012 airfoil, the lift coefficient (Cl) change for variation of angle of attack gotten by numerical simulation and experiment is shown in the figure 3. From the picture, it shows that before stall angle of attack, the results of numerical simulation are in agreement with the experimental results. The maximum deviation between two methods is 3.25%. Another very important parameter to validate the accuracy is the pressure distribution. When $Ma = 0.2, Re = 4.6 \times 10^6$, and the angle of attack $\alpha = 10^\circ$, the pressure coefficient distribution got by numerical simulation and experiment is shown in figure 4. From the picture, it shows that the pressure coefficient of numerical simulation is also in agreement with the experimental results. Thus conclusion can be drew that this numerical simulation algorithm meets to the requirement of engineering application.

![Figure 3. The comparison of lift coefficients.](image)

![Figure 4. The comparison of pressure coefficients.](image)

3.2. The mechanism of jet’s effect on lift-increase

The following study compares the airfoil pressure distribution between whether have jet or not. The basic airfoil is NACA0012. The angle of attack $\alpha = 0^\circ$, Mach number $Ma = 0.2$, Reynolds $Re = 4.6 \times 10^6$. The location of jet $x_j = x_j/e = 0.5$, The width of jet $\Delta x_j = 0.04$. The jet vertically blowing from the lower surface of the airfoil, the velocity of jet $v_j = v_j/v_\infty = 1$, the pressure around jet is $p_j = p_j/p_\infty = 1$. Figure 5 shows the pressure contours plot around the airfoil with jet or not. Figure 6 shows the pressure coefficient distribution around the airfoil with jet or not. From the picture, it shows that there is a positive value of the integral of the pressure coefficient of the airfoil surface when there is a jet, and the pressure of the lower surface of the airfoil is bigger than the upper surface. Figure 7 shows the velocity vector diagram near the jet port and the streamline of the flow near the airfoil. From the picture, a separation zone is formed in front of the jet port and behind the jet port respectively can be seen by us. The separation zone in front of the jet is very small while the separation zone behind the jet is relatively large.

![Figure 5. The pressure contours plot around the airfoil with jet or not.](image)
3.3. Influence of jet velocity in the flow field

Figure 8. The effect of jet velocity on the Cl.

Figure 9. The pressure coefficients distribution of different jet velocity.

Based on the calculation conditions above, the following research only change the magnitude of $\bar{v}$. Figure 8 shows the effect of jet velocity on the Cl. Figure 9 shows the pressure coefficients distribution of different jet velocity. As is shown in the picture, when $\bar{v}$ is less than a fixed value, the lift increases with the raising of jet velocity. But beyond this value the lift decreases with the raising of jet velocity. The reason is that when $\bar{v} < 7$, as the jet velocity raising, the pressure differences before and behind the jet port are all climbing up, the pressure difference on the jet port only reduce a little; when $\bar{v} > 7$, the pressure difference on the jet port changes drastically and drops a little before and behind the jet port, which causes the lift coefficient to reduce.

3.4. Influence of jet location in the flow field

Similarly, the following research only change the chordwise location of the jet. All other parameters are equal to case B. The effect of jet location $\bar{x}_j = 0.5, 0.6, 0.7, 0.8$ on the Cl is shown in figure 10. The
Cp difference between different jet location is shown in figure 11.

**Figure 10.** The effect of jet velocity on the Cl.  **Figure 11.** The pressure coefficients distribution of different jet location.

From the picture, it shows that due to the influence of the jet, a large positive pressure is generated in front of the jet, while the negative pressure is formed after the jet, and the lift increases with the jet moving near to the trailing edge. The reason is that while the jet moving near to the trailing edge, the pressure value of the lower surface of the airfoil is more positive, resulting in the increase of the lift.

### 3.5. Influence of jet width in the flow field

The location of jet is fixed in $x_l = 0.5$. Only change the width of jet, and all other parameters are the same with case B. The lift coefficient and pressure coefficient difference between different jet width $\Delta x_l = 0.005, 0.01, 0.02, 0.03, 0.04, 0.05$ are shown in figure 12 and figure 13.

**Figure 12.** The effect of jet width on the Cl.  **Figure 13.** The pressure coefficients distribution of different jet width.

As is shown in the picture, when the width of jet is in a certain range, the lift coefficient increases with the increase of the jet width. The reason can be said in two aspects. With the increase of jet with, on the one hand, the mass flow is increased also, so the reaction of the gas to airfoil is strengthened; one the other hand, the positive pressure of the lower surface before jet port is increased, while the negative pressure of the upper surface before jet port is increased also, so the pressure difference between lower surface and upper surface before jet port is increased, while the pressure difference between lower surface and upper surface behind jet port only change a little, resulting in the increase of the lift.

### 3.6. Jet effect on the lift at different angle of attack

Based on case B, the following research merely change the airfoil angle of attack. With the change of the angle of attack, the lift coefficient differences between using jet control or not are shown in figure 14. As is shown is the picture, the stall angle of attack is obviously reduced, but before stall angle of
attack, jet control can make the lift coefficient increase at any angle of attack, and the effect of lift-increase is almost same. In comparison, the maximum lift coefficient can increase by 0.411 approximately. Conclusion can be drew that the effect of lift-increase is very obvious when the angle of attack is not very big.

![Figure 14. The lift coefficients of different angle of attack.](image)

4. Conclusion
Based on the NACA0012 airfoil, a research is conducted on jet control on the lower surface of the airfoil and finite span model in order to increase the lift. The main conclusions are as follows:

1. With appropriate settings, the aerodynamic characteristics of the airfoil can be accurately simulated. The method is satisfied with the requirement of engineering application;

2. Jet control on the lower surface of NACA0012 airfoil can increase the lift coefficient of the airfoil by changing the airfoil surface pressure distribution. The effect of jet control have a certain relationship with the jet velocity, jet width and jet location, and the stall angle of attack is reduced, but before the stall angle of attack, the effect of lift-increase is almost the same at different angle of attack, while the maximum lift coefficient is increased by 0.411.

References
[1] Tong Zengjun, Liu Peiqing, Duan Huishen. Numerical simulation of the flap blowing and suction control for two dimension multi-element airfoil[J]. Aircraft Design, 2010, 30(4):10-15.
[2] Zhao Wanli, Liu Peiqing, Zhu Jianyong. Numerical simulation of the flow control using gurney flaps[J]. Power system and clean energy, 2011, 27(9).
[3] Li Feng, Deng Ningfeng, Zhou Weijiang, Ma Handong. The Numerical simulation of lift-increase effect on an airfoil with a jet from its surface.[J]. Acta Aeronautica et Astronautica Sinica.1994, 15(5).
[4] Guo Tongqing. Transonic unsteady aerodynamics and flutter computations for complex assemblies[D]. Nanjing: Nanjing University of Aeronautics and Astronautics, 2008, 48-59.
[5] Li Dawei, Qiu Jinhao, Ji Hongli, Nie Rui. Numerical simulation of blowing control based on limited wingspan model [J]. Chinese journal of applied mechanics, 2014, 31(4).
[6] Di Yachao, Gao Ge, Dong He, Xu Jinglei, Tang Yangyang, Xu Huan. Numerical investigation of impressible turbulent flows past airfoils[J]. Journal of Aerospace power, 2013, 28(12).
[7] Krathapalli A, Leopold D. An experiment investigation of flow around an airfoil with a jet exhausting from lower surface.[J] NASA-166131, 1981.
[8] Zhu Yixi, Lu Zhiliang, Guo Tongqing. Numerical simulation of multi-element airfoil in unsteady ground effect.[J]. Acta Aeronautica et Astronautica Sinica, 2014.