Numerical simulation of the effect of a co-flow jet on the wind turbine airfoil aerodynamic characteristics

Heyong Xu*, Shilong Xing, Zhengyin Ye

Abstract

A co-flow jet active flow control method is applied to the wind turbine S809 airfoil, and numerically studied by solving Renolds-averaged Navier-Stokes equations. The co-flow jet airfoil is designed with an injection slot near the leading edge and a suction slot near the trailing edge on the airfoil suction surface. A high-speed jet is injected tangentially into the main flow, and in the meanwhile the same amount of mass flow is drawn into the suction slot. The solver is validated by comparing the computed results with the baseline experiment measurement. The flow around the co-flow jet airfoil is simulated at a jet momentum coefficient of 0.12, showing that the co-flow jet has a significantly positive effect in increasing lift, stall margin, and drag reduction. The present co-flow jet concept is proved to be an effective and promising active flow control method in the future wind turbine application.

Keywords: co-flow jet; active flow control; wind turbine; numerical simulation; jet momentum coefficient

1. Introduction

The power production of conventional wind turbine is limited at low wind speed conditions. Normally, natural wind farm may frequently experience adverse working conditions in which the wind speed is very low or very high. When the wind speed is very low, conventional wind turbines generate little or no usable power. This issue has limited the deployment of wind turbines to a relatively few sites where favourable wind conditions exist. At the other extreme, when the wind speeds is high, the turbine rotor will experience aerodynamic stall. The highly

* Corresponding author. Tel.: +86-15802935215; fax: +86-029-88491374.
E-mail address: xuheyong@nwpu.edu.cn
unsteady aerodynamic forces and moments resulted from the stall will inevitably shorten the life of wind turbine system and increase its maintenance frequency and cost. Therefore, there is a significant necessity to exploit effective flow control methods to extend the usable operating range of wind turbines by enhancing the lift of turbine blade, and to prolong the turbine life by suppressing the aerodynamic stall.

Circulation control and Gurney flap methods have been extensively investigated both experimentally [1-2] and numerically [3-4] over many years to enhance wing lift and alleviate dynamic stall. The numerical study performed by Tongchitpakdee et al. [3] indicates that under attached flow conditions, trailing-edge blowing and the Gurney flap both can produce a net increase in power generation. However, the result also indicates that under a high-wind speed condition where the flow is separated, the trailing-edge blowing and the Gurney flap both become ineffective in increasing the power output.

Zha et al. [5-6] have developed a new flow control technique by implementing a Co-Flow Jet (CFJ) on the suction surface. In present study, this concept is implemented on the NREL S809 airfoil. The baseline and CFJ conceptual geometry are shown in Fig.1 and Fig.2, respectively. The CFJ airfoil is designed with an injection slot near the leading edge and a suction slot near the trailing edge on the airfoil suction surface. A high-speed jet is injected tangentially into the main flow, and the same amount of mass flow is drawn into suction slot. Apparently, the working mechanism of CFJ airfoil is different from the conventional circulation control airfoil which relies on large-radius leading edge or trailing edge to induce Coanda effect. The CFJ airfoil relies on the jet mixing with the main flow to energize the main flow and overcome the adverse pressure gradient.

The present study conducts a preliminary investigation on the co-flow jet application to NREL S809 airfoil. The CFJ geometry simulation at a jet momentum coefficient of 0.12 is performed to investigate the effect of co-flow jet on the overall aerodynamic performance. The numerical simulation is carried out using an in-house Reynolds-averaged Navier-Stokes (RANS) solver based on structured grid developed by current research group.

2. Design of NREL S809 co-flow jet airfoil

Fig.1 shows the NREL S809 baseline airfoil with a 12% chord thickness, which is carefully designed and dedicated to horizontal axis wind turbine. Fig.3 shows the geometry of S809 CFJ airfoil in detail.

The injection and suction slots are set at the locations of 6.0%c and 80.0%c from the leading edge, respectively. The height of injection slot face AC is 0.65%c, and the suction slot face BD is 1.38%c. The size of suction slot is designed about twice of the size of injection slot to ingest the same amount of jet mass flow without being choked.
A jet channel linking the injection and suction slots is carefully designed by translating downward and slightly rotating the baseline suction surface portion $AB$ to the targeted location $CD$. The high-pressure cavity and low-pressure cavity are included in the CFJ airfoil simulation in order to make the simulation more realistic. In simulation, face $EF$ is set as a total pressure boundary and face $GH$ is set as a static pressure boundary. The structured C-grid is adopted to carry out the simulation, as shown in Fig.4.

3. Numerical methods

The integral form of Navier-Stokes equations for a bounded domain $\Omega$ with a boundary $\partial\Omega$ can be written as

$$\frac{\partial}{\partial t} \iiint_{\Omega} Q dV + \iint_{\partial\Omega} F(Q) \cdot n dS = \iint_{\partial\Omega} G(Q) \cdot n dS$$

(1)

where $Q$ is the conservative variables, and $F(Q), G(Q)$ are convective term and viscous term, respectively.


4. Results and Discussions

4.1. Baseline simulation and validation

The experiment [12] for NREL S809 airfoil is chosen to validate the present solver. The flow Mach number $Ma$ is 0.15, and Reynolds number is $2.0 \times 10^6$ for all the simulations in present study.

A structured C-grid of $440 \times 120$ is generated to conduct the simulation. The $y^+$ of the first layer is no more than 1, and the number of wrap-around point on the airfoil is 240. The grid is used to simulate a wide range of angles of attack from -1.04 deg to 20.15 deg, and the calculated aerodynamic force coefficients are compared with free transition measurement, fixed transition measurement, as well as the computation results from Reference [13] by commercial CFD-ACE solver, as shown in Fig.5.

In the range with AOA lower than 6.16 deg, all present numerical results agree well with the experiment. When the AOA is larger than 6.16 deg, the discrepancy between present numerical results and experiment measurements becomes larger but with a similar varying trend. The experiment didn't give the drag coefficients for AOA higher than 7.17 deg, and only computation result from Reference [13] is available to be compared, showing a fairly good
agreement. From an overall view, the present numerical results agree fairly well with experiment. The docile stall characteristic of the S809 airfoil is well reproduced by present simulation, although the lift for separated flow is constantly a little higher than experiment.

4.2. Co-flow jet geometry simulation

To study the co-flow jet effect on the CFJ airfoil performance, a case with a momentum coefficient of 0.12 is simulated. The momentum coefficient for two-dimensional case is defined as

\[ C_\mu = \frac{\dot{m}_j V_j}{0.5 \rho_{\infty} U_{\infty}^2} \]  

(2)

Here, \( \dot{m}_j \) is the co-flow jet mass flow rate, \( V_j \) the injection jet velocity, \( \rho_{\infty} \) the freestream density, \( U_{\infty} \) the freestream velocity.

Fig. 6 gives the comparisons of lift and drag coefficients between CFJ case and baseline. For the case with a co-flow jet, the total aerodynamic force is computed by summating the pressure forces and shear stresses on the airfoil surface parts of \( \overline{AOB} \) and \( \overline{CD} \), and pressure forces on slot faces \( \overline{AC}, \overline{BD} \), and the jet reaction forces on the slot faces \( \overline{AC}, \overline{BD} \). It is obvious to find that the co-flow jet generates a significant increase in the lift, stall margin, and drag reduction. As shown in Fig. 6 and Table 1, the lift coefficient increases by 1.27, nearly doubling that of the baseline. The stall AOA increases by 8.8 deg. It is really inspiring to find that for the co-flow jet case the drag coefficients even become negative (which is a thrust) when AOA is below 16 deg. The negative drag mainly results from the reaction force of high-speed jet flow.

![Fig.6 Aerodynamic coefficients comparisons for CFJ geometry.](image)

Table 1. Comparison of aerodynamic parameters between the baseline and CFJ case.

<table>
<thead>
<tr>
<th>Airfoil</th>
<th>AOA(_{CL=0})</th>
<th>AOA(_{CL,max})</th>
<th>(C_{L,max})</th>
<th>(C_{D,min}(AOA=0 \text{ deg}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>-1.3 deg</td>
<td>12.2 deg</td>
<td>1.28</td>
<td>0.012</td>
</tr>
<tr>
<td>Co-flow jet case</td>
<td>-2.0 deg</td>
<td>21.0 deg</td>
<td>2.55</td>
<td>-0.020</td>
</tr>
<tr>
<td>Discrepancy</td>
<td>0.7 deg</td>
<td>8.8 deg</td>
<td>1.27</td>
<td>0.032</td>
</tr>
</tbody>
</table>

Fig. 7 shows the Mach number contours and streamlines of baseline and CFJ case at AOA = 20.15 deg. For the baseline, massive separation occurs on most part of the suction surface, whereas for the co-flow jet case the separation is completely suppressed. Moreover, it is interesting to find that the lift level corresponds to the location of leading edge stagnation point, which complies with the circulation theorem. The stagnation point of the CFJ case with a higher lift is more downstream, which can be clearly observed by comparing Fig. 7a and Fig. 7b.
5. Conclusions

In the present paper, a co-flow jet concept is implemented on the wind turbine airfoil to enhance aerodynamic performance, and the Reynolds-averaged Navier-Stokes equations are solved to carry out present study. Some conclusions can be drawn as follows. (1) The present solver can give a fairly good prediction for attached flow and separated flow around the wind turbine airfoil. (2) The co-flow jet case with a momentum coefficient of 0.12 is simulated, showing that the CFJ implementation can significantly increase the lift, stall margin and reduce the drag. (3) The co-flow jet has a significant advantage of fully suppressing a massive separation which is hard to achieve by other conventional flow control methods. The present study demonstrates that the co-flow jet concept is an attractive and effective active flow control method for the future wind turbine application.

Acknowledgements

This work was supported by the National Natural Science Foundation of China (Grant Nos. 11202166 and 11472223), the Fundamental Research Funds for the Central Universities, and the Soaring Star Encouragement Program of Northwestern Polytechnical University.

References