Simulation of 3D Co-Flow Jet Airfoil with Integrated Micro-Compressor Actuator

Yan Ren *, Paula Alejandra Barrios † Gecheng Zha ‡
Dept. of Mechanical and Aerospace Engineering
University of Miami, Coral Gables, Florida 33124
E-mail: gzha@miami.edu

Abstract

This paper presents the simulations of 3D Co-Flow Jet (CFJ) active flow control airfoil with an integrated micro-compressor. This is an important step to simulate the CFJ flow control airfoil by controlling the RPM of the micro-compressor actuator. The simulations are performed at freestream Mach number 0.3 to simulate the cruise condition of a general aviation electric aircraft. The airfoil used in this work is CFJ-NACA-6421. The simulations employ 3D RANS solver with Spalart-Allmaras (S-A) turbulence model, 3rd order WENO scheme for the inviscid fluxes, and 2nd order central differencing for the viscous terms.

The micro-compressor actuator is designed to achieve high efficiency at a required mass flow rate to maintain a desirable momentum coefficient ($C_{\mu}$) of the CFJ airfoil at a certain flight condition. The aerodynamic performance, CFJ mass flow rate, energy expenditure, and 3D flow field are studied for the CFJ airfoil at different micro-compressor working points, which is achieved by fixing the compressor RPM and modifying the CFJ airfoil Angles of Attack (AoA). The results show that the compressor mass flow rate linearly increases with the rise of the CFJ airfoil AoA, except at a high AoA of 12°, which leads to the stall of the airfoil. When the AoA increases, the CFJ airfoil stalls before the compressor chokes. The stall of the airfoil decreases the mass flow passing through the compressor, which prevent the compressor from achieving higher mass flow. For the aerodynamic performance of the CFJ airfoil, the maximum $C_L/C_D$ achieved is 122.6. The maximum $(C_L/C_D)_c$ achieved is 52.3, with the compressor efficiency $\eta = 74.3\%$. The simulation results suggest that for large AoA leading to airfoil stall, the compressor RPM need to be increased to move the compressor operating line up to have higher mass flow rate and $C_{\mu}$. The present study establishes a very important simulation tool, which can mimic the whole CFJ airfoil aerodynamic performance with integrated micro-compressor actuators to control the airfoil within a flight envelop at different operating conditions.

Nomenclature

$CFJ$ Co-flow jet
$AoA$ Angle of attack
$LE$ Leading Edge
$TE$ Trailing Edge
$S$ Planform area
$s$ Wing Span length
$c$ Profile chord

* Postdoc Researcher, Ph.D., AIAA member
† Ph.D. student, AIAA member
‡ Professor, ASME Fellow, AIAA associate Fellow

Approved for public release; distribution is unlimited.
Introduction

Co-Flow Jet (CFJ) flow control method [1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11] is demonstrated to achieve radical lift augmentation, stall margin increase, drag reduction for stationary and pitching airfoils. In the CFJ airfoil concept, an injection slot near the leading edge (LE) and a suction slot near the trailing edge (TE) on the airfoil suction surface are created. As shown in Fig. 1, a small amount of mass flow is withdrawn into the suction duct, pressurized and energized by a micro compressor, and then injected near the LE tangentially to the main flow via a injection duct. The whole process does not add any mass flow to the system and hence is a zero-net mass-flux (ZNMF) flow control. The validity of CFJ flow control method in CFJ-NACA-6421 airfoil is demonstrated in our previous publication [8, 12]. However, the micro compressor interfaces with the injection and suction ducts are simulated by profile boundary conditions extracted from the compressor. The simulation only reflects a specified operating condition of the compressor and does not have any interaction effect between the micro-compressor actuator and the airfoil, which will determine the CFJ wing system operating line (or map) at different flow conditions in a flight envelop such as the variation of angle of attack, flow speed, etc. The aerodynamic performance of CFJ airfoil with micro-compressor actuator integrated need to be studied.

\[
U \quad \text{Flow velocity}
\]
\[
q \quad \text{Dynamic pressure } 0.5 \rho U^2
\]
\[
p \quad \text{Static pressure}
\]
\[
\rho \quad \text{Air density}
\]
\[
\dot{m} \quad \text{Mass flow}
\]
\[
M \quad \text{Mach number}
\]
\[
\omega \quad \text{Pitching Moment}
\]
\[
P \quad \text{Pumping power}
\]
\[
\infty \quad \text{Free stream conditions}
\]
\[
C_L \quad \text{Lift coefficient } L/(q_\infty S)
\]
\[
C_D \quad \text{Drag coefficient } D/(q_\infty S)
\]
\[
C_\mu \quad \text{Jet momentum coef. } \dot{m}_j U_j/(q_\infty S)
\]
\[
P_c \quad \text{Power coefficient } L/(q_\infty S V_\infty)
\]
\[
\eta \quad \text{Micro-compressor total-to-total efficiency}
\]
\[
(C_L/C_D)_c \quad \text{CFJ airfoil corrected aerodynamic efficiency } C_L/(C_D + P_c)
\]
The purpose of this paper is to simulate the 3D CFJ active flow control airfoil with an integrated compressor actuator along with suction and injection ducts as a complete system shown in Fig. 1. As indicated in Fig. 1, the suction duct is connected to the micro-compressor actuator inlet and the injection duct is connected to the micro-compressor outlet. The micro-compressor is designed to achieve high efficiency at a specified mass flow rate range, which is required to maintain a desirable momentum coefficient \( C_\mu \) of the CFJ airfoil within a certain range. In order to reduce the energy loss inside the ducts, center-bodies connecting to the inner circles of the micro compressor inlet and outlet are used to guild the flow as shown in Fig. 1. Parametric studies are performed to study the CFJ airfoil and micro-compressor performance at different compressor working points. The conclusion of this work could provide guidance for future high-efficiency CFJ wing aircraft design and control.

2 Methodology

2.1 Lift and Drag Calculation

The momentum and pressure at the injection and suction slots produce a reactionary force, which is automatically measured by the force balance in wind tunnel testing. However, for CFD simulation, the full reactionary force needs to be included. Using control volume analysis, the reactionary force can be calculated using the flow parameters at the injection and suction slot opening surfaces. Zha et al. [2] give the following formulations to calculate the lift and drag due to the jet reactionary force for a CFJ airfoil. By considering the effects of injection and suction jets on the CFJ airfoil, the expressions for these reactionary forces are given as:

\[
F_{x_{cfj}} = (\dot{m}_j V_{j1} + p_{j1} A_{j1}) \cos(\theta_1 - \alpha) - (\dot{m}_j V_{j2} + p_{j2} A_{j2}) \cos(\theta_2 + \alpha)
\]  
\[
F_{y_{cfj}} = (\dot{m}_j V_{j1} + p_{j1} A_{j1}) \sin(\theta_1 - \alpha) + (\dot{m}_j V_{j2} + p_{j2} A_{j2}) \sin(\theta_2 + \alpha)
\]

where the subscripts 1 and 2 stand for the injection and suction respectively, and \( \theta_1 \) and \( \theta_2 \) are the angles between the injection and suction slot’s surface and a line normal to the airfoil chord. \( \alpha \) is the angle of attack.
The total lift and drag on the airfoil can then be expressed as:

\[ D = R'_x - F_{x,cfj} \]  
\[ L = R'_y - F_{y,cfj} \]  

where \( R'_x \) and \( R'_y \) are the surface integral of pressure and shear stress in \( x \) (drag) and \( y \) (lift) direction excluding the internal ducts of injection and suction. For CFJ wing simulations, the total lift and drag are calculated by integrating Eqs.(3) and (4) in the spanwise direction.

### 2.2 Jet Momentum Coefficient

The jet momentum coefficient \( C_\mu \) is a parameter used to quantify the jet intensity. It is defined as:

\[ C_\mu = \frac{\dot{m}V_j}{\frac{1}{2}\rho_\infty V_\infty^2 S} \]  

where \( \dot{m} \) is the injection mass flow, \( V_j \) is the mass-averaged injection velocity, \( \rho_\infty \) and \( V_\infty \) denote the free stream density and velocity, and \( S \) is the planform area. In this study, the CFJ injection momentum coefficient is controlled by the compressor RPM that also determines the compressor power.

### 2.3 Micro-compressor Power Coefficient

CFJ is implemented by mounting a pumping system inside the wing that withdraws air from the suction slot and blows it into the injection slot. The power consumption is determined by the jet mass flow and total enthalpy change as the following:

\[ P = \dot{m}(H_{t1} - H_{t2}) \]  

where \( H_{t1} \) and \( H_{t2} \) are the mass-averaged total enthalpy in the injection cavity and suction cavity respectively, \( P \) is the Power required by the pump and \( \dot{m} \) the jet mass flow rate. Introducing \( P_{t1} \) and \( P_{t2} \) the mass-averaged total pressure in the injection and suction cavity respectively, the compressor efficiency \( \eta \), and the total pressure ratio of the pump \( \Gamma = \frac{P_{t1}}{P_{t2}} \), the power consumption is expressed as:

\[ P = \frac{\dot{m}C_pT_{t2}}{\eta}(\Gamma^{\gamma-1} - 1) \]  

where \( \gamma \) is the specific heat ratio equal to 1.4 for air. The power coefficient is expressed as:

\[ P_c = \frac{P}{\frac{1}{2}\rho_\infty V_\infty^3 S} \]
2.4 Aerodynamic Efficiency

The conventional wing aerodynamic efficiency is defined as:

\[
\frac{C_L}{C_D}
\]

For the CFJ wing, the ratio above still represents the pure aerodynamic relationship between lift coefficient and drag coefficient. However since CFJ active flow control consumes energy, the ratio above is modified to take into account the energy consumption of the micro-compressor. The formulation of the corrected aerodynamic efficiency for CFJ wings is:

\[
\left(\frac{C_L}{C_D}\right)_c = \frac{C_L}{C_D} + P_c
\]

where \(P_c\) is the micro-compressor power coefficient defined in Eqn. 8 and \(C_L\) and \(C_D\) are the lift and drag coefficients of the CFJ wing. If the micro-compressor power coefficient is set to 0, this formulation returns to the aerodynamic efficiency of a conventional airfoil.

2.5 CFD Simulation Setup

The FASIP (Flow-Acoustics-Structure Interaction Package) CFD code is used to conduct the numerical simulation. The 3D Reynolds Averaged Navier-Stokes (RANS) equations with one-equation Spalart-Allmaras [13] turbulence model is used. A 3rd order WENO scheme for the inviscid flux [14, 15, 16, 17, 18, 19] and a 2nd order central differencing for the viscous terms [14, 18] are employed to discretize the Navier-Stokes equations. The low diffusion E-CUSP scheme used as the approximate Riemann solver suggested by Zha et al [15] is utilized with the WENO scheme to evaluate the inviscid fluxes. Implicit time marching method using Gauss-Seidel line relaxation is used to achieve a fast convergence rate [20]. Parallel computing is implemented to save wall clock simulation time [21]. The micro-compressor rotor flow is simulated in the rotating frame while the stator, the CFJ airfoil, and ducts are simulated in the stationary frame. In the rotating frame, the centrifugal and Coriolis forces are included as described in [22, 23].
2.6 Boundary Conditions

The 3rd order accuracy no slip condition is enforced on the solid surface with the wall treatment suggested in [24] to achieve the flux conservation on the wall. The computational mesh is shown in Fig. 2. Total pressure, total temperature and flow angles are specified at the upstream portion of the far field. Constant static pressure is applied at the downstream portion of the far field. Mixing plane boundary conditions are applied at the intersections of the ducts and the compressor. Symmetry boundary conditions are applied at the two sides in z-direction to ensure the effect of a segment of a 3D CFJ wing. The cross-section faces of the ducts are meshed using “H” topology while the parts around the airfoil are meshed using “O” topology. The total mesh size is 7.225 millions grid cells, split into 168 blocks for the parallel computation. The first grid point on the wing surface is placed at $y^+ \approx 1$.

3 Results

The parametric study results are presented in this section. The CFJ airfoil and micro-compressor performances are examined under different working conditions. The corresponding parameters of the CFJ airfoil and micro-compressor are listed in Table 1.
Table 1: Simulation parameters used in the current work.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Airfoil</td>
<td>CFJ-NACA-6421</td>
</tr>
<tr>
<td>Mach</td>
<td>0.3</td>
</tr>
<tr>
<td>AoA range</td>
<td>$-5^\circ \sim 12^\circ$</td>
</tr>
<tr>
<td>Compressor RPM</td>
<td>33,000</td>
</tr>
<tr>
<td>Compressor design RPM</td>
<td>30,000</td>
</tr>
<tr>
<td>Compressor design mass flow rate</td>
<td>0.165 kg/s</td>
</tr>
<tr>
<td>Compressor design total pressure ratio</td>
<td>1.04</td>
</tr>
<tr>
<td>Compressor design efficiency</td>
<td>84.4%</td>
</tr>
<tr>
<td>Compressor Inner Diameter</td>
<td>40 mm</td>
</tr>
<tr>
<td>Compressor Outer Diameter</td>
<td>64 mm</td>
</tr>
<tr>
<td># of stages</td>
<td>1</td>
</tr>
<tr>
<td># of rotor blade set</td>
<td>13</td>
</tr>
<tr>
<td># of stator blade set</td>
<td>15</td>
</tr>
</tbody>
</table>

Figure 3: Aerodynamic performance of the micro-compressor, (a) AoA and $\dot{m}$; (b) $C_\mu$ and $P_c$; (c) $P_{tr}$ and $\eta$.

Six cases (C1 - C6) are studied with fixed compressor RPM 33,000. Different compressor working points are
studied by changing the mass flow rate pass through the compressor, which is achieved by modifying the CFJ airfoil AoA. As shown in Fig. 3 (a), the compressor mass flow rate linearly increases with the rise of the CFJ airfoil AoA, except for the case C6, which presents airfoil stall due to too high AoA. Fig. 3 (b) shows the CFJ airfoil jet momentum coefficient $C_{\mu}$ and compressor power coefficient $P_{C}$ at different compressor working points. We can see that $C_{\mu}$ increases with the rise of compressor $\dot{m}$ except for the case C6. The airfoil stall leads to the drop of the mass flow. For the compressor $P_{C}$, case C4 shows the largest power consumption. Fig. 3 (c) presents the compressor total pressure ratio and efficiency at different working points. It is clear to see that the compressor stalls for the case C1, which leads to the lowest efficiency $\eta = 59.7\%$. High efficiency points corresponds to a $\dot{m}$ around 0.15kg/s, which is close to the compressor design mass flow rate. It is worth noting that for high AoA case C6, the CFJ airfoil stalls before the compressor chokes. The stall of the airfoil decreases the mass flow passing through the compressor, which prevent the compressor from achieving higher mass flow.

The aerodynamic performance of the CFJ airfoil is shown in Fig. 4. We can see from the figure that $C_L$ is almost linearly increased with the rise of airfoil AoA except for the case C6 due to the airfoil stall. Case C3 shows the maximum $C_L/C_D = 122.6$. Case C4 shows the maximum $(C_L/C_D)_c = 52.3$, which corresponds to the compressor efficiency $\eta = 74.3\%$.

The flow field around the CFJ airfoil at different compressor working points are described in the following. Case C1, C4, and C6 are chosen for the demonstration. The flow slices at the mid span of the airfoils are shown in Fig. 5. The simulation results show well attached flow for the case C1 and C4, while large flow separation on the airfoil suction surface can be observed for the case C6, which suggests that the airfoil stalls. The flow separation decreases the static pressure near the suction duct inlet and limits the mass flow going through the compressor.
In addition, non-uniform flow can be observed inside the suction duct for the case C1, which suggests that the compressor stalls.

Figure 5: 2D flow slices at the mid span of the case C1, C4, and C6.

Blade to blade flow slices of the micro-compressor for the three cases are shown in Fig. 6. For the case C1, flow separation can be observed for both rotor and stator blades, which suggests that the compressor stalls. For the case C4 and C6, no flow separation can be observed for both cases. The compressor efficiency $\eta$ is high (around 75%) for these two cases. Larger high speed zone near the rotor blade leading edge can be observed for the case C4 than the case C6 since the mass flow rate is greatly reduced for the case C6 due to the stall of the CFJ airfoil. The above study suggests that for large AoA cases that may lead to airfoil stall, the compressor RPM need to be increased to move the compressor operating line up to have higher mass flow rate and $C_\mu$. 
4 Conclusion

This paper presents the simulations of 3D Co-Flow Jet (CFJ) active flow control airfoil with an integrated micro-compressor. This is an important step to simulate the CFJ flow control airfoil by controlling the RPM of the micro-compressor actuator. The simulations are performed at freestream Mach number 0.3 to simulate the cruise condition of a general aviation electric aircraft. The airfoil used in this work is CFJ-NACA-6421. The simulations employ 3D RANS solver with Spalart-Allmaras (S-A) turbulence model, 3rd order WENO scheme for the inviscid fluxes, and 2nd order central differencing for the viscous terms.

The micro-compressor actuator is designed to achieve high efficiency at a required mass flow rate to maintain a desirable momentum coefficient ($C_\mu$) of the CFJ airfoil at a certain flight condition. The aerodynamic performance, CFJ mass flow rate, energy expenditure, and 3D flow field are studied for the CFJ airfoil at different micro-compressor working points, which is achieved by fixing the compressor RPM and modifying the CFJ airfoil Angles of Attack (AoA). The results show that the compressor mass flow rate linearly increases with the rise of the CFJ airfoil AoA, except at a high AoA of 12°, which leads to the stall of the airfoil. When the AoA increases, the CFJ airfoil stalls before the compressor chokes. The stall of the airfoil decreases the mass flow passing through the compressor, which prevent the compressor from achieving higher mass flow. For the aerodynamic performance of the CFJ airfoil, the maximum $C_L/C_D$ achieved is 122.6. The maximum $(C_L/C_D)_c$ achieved is 52.3, with the
compressor efficiency $\eta = 74.3\%$. The simulation results suggest that for large AoA leading to airfoil stall, the compressor RPM need to be increased to move the compressor operating line up to have higher mass flow rate and $C_\mu$. The present study establishes a very important simulation tool, which can mimic the whole CFJ airfoil aerodynamic performance with integrated micro-compressor actuators to control the airfoil within a flight envelop at different operating conditions.

References


