Numerical Investigations of Injection-Slot-Size Effect on the Performance of Coflow Jet Airfoils

Baoyuan Wang,* Bahaa Haddoukessoumi,† Jonathan Levy,† and Ge-Cheng Zha‡

University of Miami, Coral Gables, Florida 33124

DOI: 10.2514/1.37441

Three coflow jet airfoils with twice-doubled injection-slot sizes are calculated using a Reynolds-averaged Navier-Stokes computational fluid dynamics solver with the one-equation Spalart–Allmaras model. At the same angle of attack, the twice-larger injection-slot-size airfoil passes the (about twice-greater) jet mass flow rate, with the momentum coefficients also nearly doubled. The coflow jet airfoil with the largest slot size has the lowest stall angle of attack. When the injection-slot size is reduced from the maximum by half, the stall angle of attack and the maximum lift coefficient are increased. When the injection-slot size is further reduced by half, the stall angle of attack is still increased, but the maximum lift coefficient is lower due to the smaller momentum coefficient. The trends of the stall angle of attack and maximum lift coefficient agree fairly well with the experiment. At high angles of attack, both the computed lift and drag coefficients agree fairly well with the experimental data. The Reynolds-averaged Navier–Stokes model cannot handle the turbulence mixing at high angles of attack.

Nomenclature

\[ A = \text{area} \]
\[ c = \text{speed of sound} \]
\[ D = \text{drag} \]
\[ d = \text{distance to the closest wall} \]
\[ d_i = \text{distance of the field point to the trip location} \]
\[ e = \text{total energy per unit mass} \]
\[ F = \text{reactionary force} \]
\[ J = \text{Jacobian of transformation} \]
\[ L = \text{lift} \]
\[ l, m, n = \text{normal vectors on } \xi, \eta, \text{and } \zeta \text{ surfaces with their magnitudes equal to the elemental surfaces and pointing to the directions of increasing } \xi, \eta, \text{and } \zeta \]
\[ l_i, m_i, n_i = \text{grid moving velocities} \]
\[ m = \text{mass flow rate} \]
\[ Pr = \text{Prandtl number} \]
\[ Pr_t = \text{turbulent Prandtl number} \]
\[ P = \text{pressure} \]
\[ q_t = \text{total heat flux in Cartesian coordinates} \]
\[ R = \text{gas constant} \]
\[ Re = \text{Reynolds number} \]
\[ S = \text{force from the airfoil surface integral} \]
\[ T = \text{temperature} \]
\[ t = \text{time} \]
\[ U, V, W = \text{contravariant velocities in } \xi, \eta, \text{and } \zeta \text{ directions} \]
\[ u, v, w = \text{velocity components in the } x, y, \text{and } z \text{ directions} \]
\[ v = \text{velocity vector} \]
\[ x, y, z = \text{Cartesian coordinates} \]
\[ \alpha = \text{angle of attack} \]
\[ \gamma = \text{ratio of specific heats} \]
\[ \Delta U = \text{difference of the velocities between the field point and the trip location} \]
\[ \Delta x_t = \text{grid spacing along the wall at the trip location} \]
\[ \theta = \text{angle between slot surface and the line normal to the airfoil chord} \]
\[ \mu = \text{molecular viscosity} \]
\[ \mu_t = \text{turbulent viscosity} \]
\[ \nu = \text{kinematic viscosity} \]
\[ \xi, \eta, \zeta = \text{generalized coordinates} \]
\[ \rho = \text{density} \]
\[ \tau_{ik} = \text{shear stress in Cartesian coordinates} \]
\[ \omega_{ij} = \text{wall vorticity at the wall boundary-layer trip location} \]

Subscripts

\[ i, j, k = \text{indices} \]
\[ L, R = \text{left- and right-hand sides of the interface} \]

1. Introduction

TO IMPROVE aircraft performance, innovative technologies should be pursued to dramatically reduce the weight and consumption of the aircraft and to significantly increase its mission payload and stall margin. Both military and commercial aircraft will benefit from these technologies. Flow control is the most promising route to bring significant performance improvements to aircraft [1–7]. Recently, Zha et al. [8–11] developed a promising airfoil flow control technique using a coflow jet, which significantly increases lift, stall margin, and drag reduction. The coflow jet airfoil is designed with an injection slot near the leading edge and a suction slot near the trailing edge on the suction surface. The slots are opened by translating a large portion of the suction surface downward. A high-energy jet is injected tangentially near the leading edge and the same amount of mass flow is drawn in near the trailing edge. The turbulent layer between the main flow and the jet causes strong turbulence diffusion and mixing under the severe adverse pressure gradient, which enhances lateral transport of energy from the jet to the main flow and allows the main flow to overcome the severe adverse pressure gradient that remains attached at high angles of attack (AOA). The high-energy jet induces high circulation and hence generates lift. The energy main flow fills the wake and therefore reduces drag. The coflow (CFJ) airfoil achieves net-zero mass-flux flow control and a significantly zero jet mass flow.

In [8–12], Zha et al. compared the performance of coflow jet airfoils with the performance of conventional airfoils and showed that the coflow jet airfoils are more effective in increasing lift and reducing drag.

The first of the geometrically non-optimized designs was completed in 2006. In [13], the design of the coflow jet airfoil was refined using advanced turbulence models and geometry optimization techniques.

The second design was completed in 2008. In [14], the coflow jet airfoil was designed with a linear actuator providing a maximum jet mass flow rate of 1% of the main flow mass flow rate. In [15], the coflow jet airfoil was designed with a linear actuator providing a maximum jet mass flow rate of 2% of the main flow mass flow rate.

Figure 1: The test sections for the three airfoils. (a) CFJ0025-060L with 0.1527 m airfoil chord and a 0.610 m span, (b) CFJ0025-060R with 0.1527 m airfoil chord and a 0.610 m span, and (c) CFJ0025-060L/50% with 0.1527 m airfoil chord and a 0.610 m span.

Received 7 March 2008; revision received 11 June 2008; accepted for publication 24 June 2008. Copyright © 2008 by the authors of this article. Published by the American Institute of Aeronautics and Astronautics, Inc., with permission. Copies of this paper may be made for personal or internal use, on condition that the copier pay the $10.00 per-copy fee to the Copyright Clearance Center, Inc., 222 Rosewood Drive, Danvers, MA 01923; include the code 0021-8669/08 $10.00 in correspondence.

*Ph.D. Candidate, Department of Mechanical and Aerospace Engineering, Member AIAA.
†Visiting Senior Undergraduate Student, Ecole d’Ingénieur en Modélisation Mathématique et Mécanique, France.
‡Associate Professor, Department of Mechanical and Aerospace Engineering, gzha@miami.edu. Senior Member AIAA.
II. CFJ Airfoil Geometry

Figure 1 shows the baseline airfoil, NACA0025, and the other airfoils with coflow jet slots. The chord length of the airfoil is 127 m and the span is 0.3 m. The coflow jet airfoils are defined using the following convention: CFJ4dig-INJ-SUC, where 4dig is the same as the NACA 4-digit convention, INJ is the percentage of injection-slot size to the chord length, and SUC is the percentage of suction-slot size to the chord length. For example, the CJ0025-065-196 airfoil has an injection-slot height of 0.65% of the chord and a suction-slot height of 1.96% of the chord. The suction-injector shape is a downward translation of the portion of the original suction surface between the injection and suction slots. The injection of suction slots are located at 7.11 and 83.18% of the chord from the leading edge. The slot faces are normal to the suction surface to make it tangent to the main flow.

The CJ0025-131-196 airfoil is designed with an injection-slot size twice larger than that of the CJ0025-065-196 airfoil to examine the effect of injection-slot size. The suction-slot size is unchanged. The slot locations are also the same as those of the CJ0025-065-196 airfoil.

The paper simulates a new CFJ airfoil, CJ0025-033-065, for which both the injection and suction sizes are half of the CJ0025-065-196 airfoil. In other words, for the three CFJ airfoils shown in fig. 1, the injection-slot sizes are doubled twice. The purpose is to study the injection geometry effect on the CFJ airfoil performance. The injection-slot-size design criterion is to be able to suck in the same region mass flow without being choked.

The baseline airfoil, CJ0025-065-196, and the CJ0025-131-196 airfoil are tested in the wind-tunnel tests [12], and the experimental results are used for comparison with the CFD simulation in this paper.
\[ \tau_{ik} = (\mu + \mu_\text{t}) \left[ \frac{\partial u_i}{\partial x_k} + \frac{\partial u_k}{\partial x_i} - \frac{2}{3} \delta_{ik} \frac{\partial u_l}{\partial x_l} \right] \]

\[ q_k = -\left( \frac{\mu}{\rho R_T} + \frac{\mu_\text{t}}{\rho R_T} \right) \frac{\partial T}{\partial x_k} \]

where \( \mu \) is determined by Sutherland’s law, and \( \mu_\text{t} \) is determined by the S-A model:

\[ \mu_\text{t} = \rho \overline{v} f_v \]

The kinetic viscosity \( \nu \) is defined as

\[ \nu = \frac{\mu}{\rho} \]

In Eqs. (6–8), (42), (17), and (18), the repeated subscript indices represent the coordinates \( x, y, \) and \( z \) following Einstein summation convention. Equations (17) and (18) are transformed to a generalized coordinate system in computation.

The sixth equation of the governing equations (1–9) is the one-equation turbulence model [13]. The functions in the equation are given as

\[ f_{v1} = \frac{\chi}{\chi + C_{v1}} \]

\[ S = \frac{\bar{S}}{R_E \kappa^2 c_T^2 f_{v2}} \]

\[ f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \]

\[ f_w = g \left[ \frac{1 + C_{w1} \alpha}{\alpha^2 + C_{w1}^2} \right] \]

\[ g = r + C_{w2}(r^2 - r) \]

\[ f_{\text{nl}} = C_{\text{nl}} g_e \exp \left[ -C_{\text{nl}} \frac{a^2}{\Delta U^2} \left( d^2 + g^2 d_i^2 \right) \right] \]

\[ f_{\text{lin}} = C_{\text{lin}} \exp(-C_{\text{lin}} \chi^2) \]

where

\[ \chi = \frac{v}{\bar{v}} \]

\[ S = \sqrt{ \left( \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \right)^2 + \left( \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)^2 } \]

\[ r = \frac{\bar{v}}{R_E \kappa^2 c_T^2} \]

The constants in the S-A model are set to have the following values:

The shear stress \( \tau_{ik} \) and total heat flux \( q_k \) in Cartesian coordinates can be expressed as

\[ \mathbf{R} = \begin{bmatrix} 0 & 0 & 0 \\ l_1 \tau_{zz} & l_2 \tau_{zz} & l_3 \tau_{zz} \\ l_4 \tau_{zz} & l_5 \tau_{zz} & l_6 \tau_{zz} \\ \xi (v + \overline{v})(m \cdot \nabla \overline{v}) \\ \end{bmatrix} \]

\[ \mathbf{S} = \begin{bmatrix} 0 & 0 & 0 \\ n_1 \tau_{zz} & n_2 \tau_{zz} & n_3 \tau_{zz} \\ n_4 \tau_{zz} & n_5 \tau_{zz} & n_6 \tau_{zz} \\ \xi (v + \overline{v})(m \cdot \nabla \overline{v}) \\ \end{bmatrix} \]

\[ \mathbf{T} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \\ \xi (v + \overline{v})(n \cdot \nabla \overline{v}) \\ \end{bmatrix} \]

\[ D = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ S \end{bmatrix} \]

where

\[ \beta_k = u_k \tau_{kk} - q_k \]

\[ S_v = \rho C_{h1}(1 - f_{v2}) \overline{v} + \frac{1}{Re} \left[ -\rho \left( C_{w1} f_w - \frac{C_{w1}}{k_0^2 f_{v1}} \right) \left( \frac{\overline{v}}{d} \right)^2 \right] + \frac{p}{\sigma} C_{h2} (\nabla \overline{v})^2 - \frac{1}{\sigma} (v + \overline{v}) \nabla \overline{v} \cdot \nabla \rho + Re \left[ pf_{\text{nl}} \left( \Delta U \right)^2 \right] \]

\[ U = l_1 + l_2 \mathbf{V} l_3 + l_4 u + l_5 v + l_6 w \]

\[ V = n_1 + n_2 \mathbf{V} n_3 + n_4 u + n_5 v + n_6 w \]

\[ W = m_1 + m_2 \mathbf{V} m_3 + m_4 u + m_5 v + m_6 w \]

\[ \mathbf{L} = \frac{\nabla \xi}{J} \frac{\partial \eta}{\partial \zeta}, \quad \mathbf{M} = \frac{\nabla \eta}{J} \frac{\partial \xi}{\partial \zeta}, \quad \mathbf{N} = \frac{\nabla \zeta}{J} \frac{\partial \xi}{\partial \eta} \]

\[ I = \xi \frac{\partial \eta}{\partial \zeta}, \quad M_x = \xi \frac{\partial \xi}{\partial \eta}, \quad N_x = \xi \frac{\partial \xi}{\partial \eta} \]

When the grid is stationary, \( l_1 = m_1 = z_1 = 0. \)

Because \( \Delta \xi = \Delta \eta = \Delta \zeta = 1 \) in the current discretization, Eqs. (13) and (14) are written as follows in the solver:

\[ I = \frac{\nabla \xi}{J}, \quad M = \frac{\nabla \eta}{J}, \quad N = \frac{\nabla \zeta}{J} \]

\[ I_e = \frac{\xi}{J}, \quad M_e = \frac{\eta}{J}, \quad N_e = \frac{\zeta}{J} \]

The flow computations are performed by simply tripping the boundary-layer model, the vortices and large vortices are present in the inlet as the main parameters.

To study the pressure distribution, the static pressure is evaluated at the injection point.
\[ \sigma = \frac{1}{\kappa}, \quad C_{b1} = 0.1355, \quad C_{b2} = 0.622, \quad \kappa = 0.41 \]
\[ C_{u1} = 7.1, \quad C_{u2} = C_{b1}/k^2 + (1 + C_{b2})/\sigma \]
\[ C_{u2} = 0.3, \quad C_{u3} = 2, \quad C_{u4} = 1 \]
\[ C_{u2} = 2, \quad C_{u3} = 1.2, \quad C_{u4} = 0.5 \]

The full turbulent boundary-layer assumption is used in the formulation by setting \( C_{u1} = 0 \) and \( C_{u4} = 0 \) to be consistent with the swept boundary layer in the experiments. Because CFD solutions obtained from the steady-state calculations based on the RANS model, the unsteady details of the shear-layer mixing entrainment and large coherent vortex structures are not able to be captured. The total pressure and total temperature are given at the injection-duct exit as the boundary conditions.

To study the effect caused only by the geometry, the injection total pressure is iterated to match the experimental momentum coefficient. The static pressure at the suction-duct entrance is iterated to match injection jet mass flow rate.

### Scheme for Inviscid Flux

The low-diffusion E-CUSP (LDE) scheme given in [14] is used to evaluate the inviscid fluxes. The basic idea of the E-CUSP scheme is split the inviscid flux into the convective flux \( E^c \) and the pressure flux \( E^p \). With the one extra equation from the S-A model, the splitting virtually the same as the original scheme and is straightforward. In the generalized coordinate system, the flux \( E \) can be split as follows:

\[
E = E^c + E^p = \begin{pmatrix}
\rho U \\
\rho uU \\
\rho vU \\
\rho wU \\
\rho u \dot{U} \\
\rho v \dot{U}
\end{pmatrix} + \begin{pmatrix}
l_p \\
l_p \\
l_p \\
l_p \\
l_p \\
\dot{U}
\end{pmatrix}
\]

\[ E^c = \rho U f^c, \quad f^c = \begin{pmatrix}
u \\
w \\
e
\end{pmatrix}
\]

\[ E^p = \begin{pmatrix}
\rho l_p \\
\rho l_p \\
\rho l_p \\
\rho l_p \\
\rho U \dot{U}
\end{pmatrix}
\]

The convective term \( E^c \) is evaluated as follows [14]:

\[ E^c = \rho U f^c, \quad f^c = \begin{pmatrix}
u \\
w \\
e
\end{pmatrix}
\]

\[ C = c\left(l_p^2 + l_p^2 + l_p^2\right)^2
\]

Then the convective flux at interface \( \frac{1}{2} \) is evaluated as

\[ E_k^c = C\left[\rho C^+ f_k^c + \rho C^- f_k^c\right]
\]

where the subscripts \( L \) and \( R \) represent the left- and right-hand sides of the interface.

The interface speed of sound is

\[ C_1 = \frac{1}{2}(C_L + C_R)
\]

The following relations to express the formulations from \(-\infty < \alpha < \infty \) are used [14]:

\[ C^+ = C_1^+ (1 + \beta_L)M_L - \beta_L M_L^+ - M_L^+ \]
\[ C^- = C_1^- (1 + \beta_R)M_R - \beta_R M_R^- + M_R^- \]

The pressure flux \( E^p \) is evaluated as follows:

\[ E_k^p = \begin{pmatrix}
0 \\
\rho l_p \\
\rho l_p \\
\rho l_p \\
\rho U \dot{U}
\end{pmatrix}
\]

\[ E_k^p = \begin{pmatrix}
D^+_{L,R} \\
D^+_{L,R} \\
D^+_{L,R} \\
D^+_{L,R}
\end{pmatrix}
\]

where

\[ D^+_{L,R} = [\alpha(1 + \beta) - \beta D^+]_{L,R}
\]

The pressure-splitting coefficient borrowed from Van Leer [15] is

\[ P^+_{L,R} = \frac{1}{2}(M_{L,R} + 1)^2(2 + M_{L,R})
\]

For the pressure term in the energy equation, the contravariant speed of sound \( \tilde{C} \) is consistent with \( \tilde{U} \) and is calculated as

\[ \tilde{C} = C - l_i
\]

\[ S_{L,R} = \left[\tilde{\alpha}(1 + \tilde{\beta})M - \tilde{\beta} \tilde{M}_{L,R}\right]
\]

where

\[ \tilde{M} = \frac{\tilde{U}}{\tilde{C}}
\]

and \( \tilde{\alpha} \) and \( \tilde{\beta} \) are evaluated based on \( \tilde{M} \) using the formulations given in Eqs. (41) and (42). The use of \( \tilde{U}, \tilde{C}, \) and \( \tilde{M} \) in the pressure term for the energy equation is to take into account the grid speed so that the flux will transit from subsonic to supersonic smoothly. When the grid is stationary, \( l_i = 0, \tilde{C} = C, \) and \( \tilde{U} = U \).

The LDE scheme can accurately resolve wall boundary-layer profiles and capture crisp shock profiles and exact contact surfaces [14] with low diffusion. The third-order MUSCL-type differencing of Van Leer is used with the LDE scheme. The viscous terms are discretized using second-order central differencing. The implicit
Gauss-Seidel line relaxation is used for time marching to achieve a high convergence rate.

C. Jet Effects on CFJ Airfoil Performance

By using a control volume analysis, Zha et al. [10] derived an expression for the force effect of the injection and suction jets on the CFJ airfoil. Based on Newton’s third law, the momentum and pressure at the injection and suction slots produce a reactionary force, which must be taken into account in the drag and lift calculations. The expressions for these reactionary forces are given as

\begin{align}
F_{x,i} &= (\rho_i V_{j,x} + p_{j,x} A_{j,x}) \times \cos(\theta_1 - \alpha) \\
&\quad - \gamma (\rho_i V_{j,y} + p_{j,y} A_{j,y}) \times \cos(\theta_2 + \alpha) \\
F_{y,i} &= (\rho_i V_{j,y} + p_{j,y} A_{j,y}) \times \sin(\theta_1 - \alpha) \\
&\quad - \gamma (\rho_i V_{j,x} + p_{j,x} A_{j,x}) \times \sin(\theta_2 + \alpha)
\end{align}

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 slots’ surfaces and a line normal to the airfoil chord [10]. The total lift and drag on the airfoil can then be expressed as

\begin{align}
D &= R_y - F_{x,i} \\
L &= R_x - F_{y,i}
\end{align}

where \(R_x\) and \(R_y\) are the surface integral of pressure and shear stress in the \(x\) (drag) and \(y\) (lift) directions. For the CFD simulation, the total lift and drag are calculated using Eqs. (57) and (58).

IV. Results and Discussion

The freestream Mach number is about 0.11 and the Reynolds number is about \(3.8 \times 10^5\), which is in the laminar/transitional region. To make the boundary layer fully turbulent to mimic the realistic flight conditions, the airfoil leading edge is tripped to trigger the turbulence in the wind-tunnel tests. In CFD simulation, the boundary layer is assumed to be fully turbulent starting from the leading edge. The different boundary conditions between the experiment and CFD is that the CFD simulates the airfoil in an open field with no wind-tunnel wall. Such a difference is a common practice of CFD simulation and is expected to cause little effect on simulation accuracy. A typical 2-D computational mesh is shown in Fig. 2 with 5 blocks. The dimensions of the blocks in the tangential and radius directions are \(33 \times 49\) (chamber), \(97 \times 97\), \(17 \times 97\), \(23 \times 97\), and \(59 \times 193\), respectively.

Figure 3 shows the momentum coefficients vs the AOA. In the wind-tunnel experiment, the injection total-pressure coefficients for the CFJ0025-065-196 and CFJ0025-131-196 are the same. Because the injection-slot size is increased by 2 times, the mass flow rate and the momentum coefficients are also about 2 times different, as shown in Fig. 3. The CFD computation matches the experimental momentum coefficients very well. For the CFJ0025-033-065 airfoil created in this paper, because no experiment is done, the momentum coefficients are determined by using the same injection total pressure as that of the CFJ0025-065-196 airfoil, which generates about half of the momentum coefficients of the CFJ0025-065-196 airfoil.

Figure 4 is the lift-coefficient comparison for the airfoils with the different slot sizes. The experiment shows that the CFJ0025-131-196 airfoil with the maximum slot size and momentum coefficients only generate slightly higher lift than the CFJ0025-065-196 airfoil before it stalls. The stall AOA and the maximum lift of the CFJ0025-131-196 airfoil is even less than that of the CFJ0025-065-196 airfoil, which has half the injection-slot size. The CFD simulations also predict the same trend. Quantitatively, the computed lift coefficients agree quite well with the experiment before AOA = 20\(^\circ\). When the AOA is greater than that, the CFD underpredicts the lift. It may be because that the RANS model cannot accurately predict the mixing process, which is inherently unsteady and may also have large vortex structures generated. The stall AOA is predicted quite well, except that the trend of the stall is more gradual instead of being abrupt shown in the experiment.

The CFD simulations of the CFJ0025-065-196 and CFJ0025-131-196 airfoil indicate that the smaller injection size airfoil has a higher maximum stall AOA and lift. This trend agree very well with the experiment. When the injection-slot size is further reduced by half, the CFJ0025-033-065 airfoil, the stall AOA is also greater than that of the CFJ0025-065-196 airfoil, as shown in Fig. 4. However, the maximum lift of the CFJ0025-033-065 airfoil is lower than that of the CFJ0025-065-196 airfoil because the jet momentum is less energy is lower.

Figure 5 is the drag coefficient of the CFJ airfoils with different injection-slot sizes. Similar to the lift prediction, the computed drag agree quite well with the experiment at low AOA. At high AOA, the drag is significantly underpredicted. Again, this may be attributed to the RANS turbulence model, which cannot well simulate the turbulence mixing at high AOA. In the experiment, the large injection-slot airfoil has slightly lower minimum drag, but the computation does not generate such a difference.

A mesh refinement study for the CFJ0025-065-196 airfoil at AOA = 10 and 35\(^\circ\) with the mesh size doubled in two directions. Both the lift and drag predicted with the refined mesh
Fig. 4 Lift coefficient of the CFJ airfoils with different injection-slot sizes.

Fig. 5 Drag coefficient of the CFJ airfoils.

Fig. 6 Wake profile of the CFJ airfoil with injection.

Fig. 7 Isotropic Mach number distribution on the surface of the CFJ airfoil with injection.

Fig. 8 Mach number contours of the NACA0025 airfoil at AoA = 20 deg.

Fig. 11 Difference, as shown in Figs. 4 and 5, respectively. This means that the baseline mesh size used is sufficient for the solution convergence.

Figure 6 shows the wake profiles of the baseline NACA0025 airfoil and the three CFJ airfoils one chord length downstream of the airfoil trailing edge. It shows that the baseline NACA0025 airfoil has the deepest velocity deficit. The CFJ airfoils have shallower wake profiles due to the CFJ energizing the main flow. The shallower wake profile generates smaller drag than the baseline airfoil [11].

Figure 7 shows the surface isotropic Mach number for the airfoils at AoA = 0 deg. It can be seen that the surface loading, or the distribution, of the CFJ airfoils is much larger than that of the NACA0025 airfoil. The leading-edge suction peak Mach number of the CFJ airfoil is higher and the stagnation point is more downstream with the increase of the slot size. It can be seen that the injection ratios are located downstream of the peak Mach number to make use of the adverse pressure gradient to enhance mixing [16].

Figures 8-11 shows the Mach number contours with streamlines of the baseline NACA0025 airfoil and the three CFJ airfoils at AoA = 20 deg, respectively. The baseline airfoil has a massive separation, which is consistent with the experiment [12]. The J0025-033-065 airfoil also experiences a small separation at the
CFJ0025-065-196 airfoil with the injection-slot size reduced by half has higher stall AOA and maximum lift coefficient, which is consistent with the experiment. For the CFJ0025-033-065 airfoil which is created in this paper with the injection-slot size further reduced by half of the CFJ0025-065-196 airfoil, the CFD simulation indicates that the stall AOA is further increased, but the maximum lift coefficient is lower due to the relatively weaker CFJ. The predicted lift and drag at low AOA agree fairly well with the experiment. At high AOA, both the lift and drag are underpredicted (particularly the drag). This may be attributed to the RANS turbulence model, which cannot capture the inherent unsteady mixing process of the airfoil at high AOA.

Acknowledgment
This research is supported under the U.S. Army Research Office and U.S. Air Force Office of Scientific Research grant 50827-F10 monitored by Rhett Jefferie and Jerry Haire.

References

V. Conclusions
Three coflow jet airfoils with twice-doubled injection-slot sizes are calculated by using a RANS CFD solver with the one-equation Spalart–Allmaras (S-A) model. At the same angle of attack (AOA), the twice-larger injection-slot-size airfoil passes the (about twice-larger) jet mass flow rate, with the momentum coefficients also nearly doubled. However, just as with the trend shown in the wind-tunnel experiment, the CFJ0025-131-196 airfoil with the largest injection-slot size has the smallest predicted stall AOA. The trailing edge because the jet momentum is not strong enough. Both the CFJ0025-065-196 and CFJ0025-131-196 airfoils do not have any separation at AOA = 20 deg due to the stronger CFJ, which is also the same as demonstrated in the wind-tunnel tests [8, 12]. It is interesting to note that even though the CFJ0025-033-065 airfoil has the separation at AOA = 20 deg, the airfoil stall does not occur until AOA = 50 deg, as shown in Fig. 4. The airfoil may work under dynamically stable flow conditions, which may only be confirmed if unsteady simulation is conducted.
Flight Performance of Fixed and Rotary Wing Aircraft

Antonio Filippone

An essential resource for all students of aerospace engineering, practicing aircraft engineers, aircraft operations managers and organizations working in air traffic control.

This book presents an unrivalled range of advanced flight performance models for both transport and military aircraft. Topics covered include the numerical solution of supersonic acceleration, transient roll, optimal climb of propeller aircraft, propeller performance, long-range flight with en route stop, fuel planning, zero-gravity flight in the atmosphere, VSTOL operations, ski jump from aircraft carrier, optimal flight paths at subsonic and supersonic speed, range-payload analysis of fixed and rotary wing aircraft, performance of tandem helicopters, lower bound noise estimation, sonic boom, and more.

This book will be a valuable text for undergraduate and postgraduate students of aerospace engineering. It will also be an essential reference and resource for practicing aircraft engineers, aircraft operations managers and organizations handling air traffic control, flight and flying regulations, standards, safety, environment, and the complex financial aspects of flying aircraft.

AIAA Education Series
2006. 416 pages, Hardback
58-13: 978-1-56347-839-0
58-10: 1-56347-839-0

List Price: $84.95
AIAA Member Price: $59.95

Copublished with Butterworth-Heinemann. To order or for more information, contact AIAA by phone: 1-630-862-2422, fax: 703/661-1501, e-mail: aiaamail@presswarehouse.com, or order online at www.aiaa.org/books

Outside the United States, Canada, and South America, order from Butterworth-Heinemann, United Kingdom, at 44 1865 310 366 or fax: 44 1865 310 898.