

Replacing the Aileron of the High-Lift Common Research Model with Trailing-Edge Coanda Actuators

Evan J. McFadden¹ and Jeffrey P. Bons²
The Ohio State University, Columbus, OH, 43235

The aileron control surface of the High-Lift Common Research Model (CRM-HL) is replaced with a segment of trailing-edge Coanda actuators to increase the takeoff lift-to-drag ratio without employing any external moving parts. The 10% scale CRM-HL model is simulated with computational fluid dynamics at takeoff conditions with the aileron at four deflection angles. A wide variety of trailing-edge Coanda actuator designs are tested in two-dimensional simulations for identifying the dominant geometric variables of the active flow control (AFC) technique. These variables include airfoil truncation, blowing direction, Coanda surface shape and size, and slot height. The optimal Coanda surface design, equipped with upper and lower slots, is installed on the CRM-HL model for simulations at takeoff conditions. Without blowing applied, the Coanda flow control design increases takeoff drag by 12 counts. At a fixed takeoff angle of attack, the Coanda actuators increase lift by 10.5%. At a fixed takeoff lift coefficient, the Coanda actuators increase the L/D by 2.5%. Throughout the study, results are compared to simulations and experiments of an AFC-augmented aileron on the CRM-HL.

I. Nomenclature

A_{AFC}	=	total exit area of flow control actuators
b	=	wing semispan
c	=	chord
C_D	=	drag coefficient
C_d	=	section drag coefficient
C_L	=	lift coefficient
C_l	=	section lift coefficient
C_M	=	pitching moment coefficient
C_p	=	pressure coefficient
C_Q	=	mass flow coefficient
C_μ	=	momentum coefficient
C_π	=	power coefficient
h	=	height
L/D	=	lift-to-drag ratio
M	=	Mach number
$\dot{m}_{AFC}$	=	mass flow rate supplied to flow control actuators
NPR	=	nozzle pressure ratio
NTR	=	nozzle temperature ratio
$P_{T,AFC}$	=	total pressure supplied to flow control actuators
P_∞	=	freestream static pressure
q_∞	=	freestream dynamic pressure

¹ Graduate Research Assistant, Mechanical and Aerospace Engineering Department, AIAA Student Member.

² Professor, Mechanical and Aerospace Engineering Department, AIAA Associate Fellow.

r	=	radius of Coanda surface
S	=	model reference area
V_{AFC}	=	mean velocity at the exit of flow control actuators
V_∞	=	freestream velocity
α	=	angle of attack
δ_a	=	aileron deflection angle
η	=	percent semispan, y/b
ρ_∞	=	freestream density

II. Introduction

Active flow control (AFC) is a heavily researched topic in aerodynamics and has the potential to be one of the most impactful focus areas for the future of aviation. In support of NASA’s Aeronautics Research Mission Directorate strategic thrust for ultra-efficient subsonic transports [1], active flow control can significantly enhance a vehicle’s fuel efficiency, weight management, and airframe drag. A NASA contractor report evaluated the impact and feasibility of applying separation control on large commercial aircraft high-lift devices and control surfaces [2]. The most promising applications from this report included simplifying the trailing-edge flaps and leading-edge slats to reduce aircraft weight and complexity. More recent research initiatives driven by NATO and DARPA have encouraged the community to develop flow control techniques that augment or replace the traditional control surfaces, especially on demonstrator aircraft [3, 4]. These studies have the capability to alter the trajectory of commercial transport design by incorporating active flow control systems early in the design process of a lighter and more fuel-efficient vehicle.

For example, teams from Boeing and NASA collaborated on full-scale flight testing of an AFC-enhanced vertical tail [5]. The vertical tail is typically oversized to support controllability in an engine-out situation. The active flow control system on the modified vertical tail of the Boeing 757 ecoDemonstrator produced up to a 14% higher side force, indicating the tail could be significantly downsized with AFC augmentation. The success of this study opened the door for further exploration of active flow techniques on commercial aircraft targeting control surface size reduction.

The Common Research Model (CRM) was designed through combined efforts between NASA and Boeing as a commercial transport model for connecting computational and experimental results, particularly through AIAA computational fluid dynamics (CFD) prediction workshops [6, 7]. The CRM’s nonproprietary nature promotes collaboration between industry, government research labs, and academia alike. There are several iterations of the CRM designed for different flight conditions, such as the High-Lift Common Research Model (CRM-HL) [8, 9]. This high-lift configuration includes leading-edge slats, Fowler flaps, a flow-through nacelle, flap fairings, and support brackets for each control surface. The nominal deployed positions and angles for the slats and flaps have been defined for takeoff and landing configurations [10, 11]. However, as with most aircraft designs, separation and flow unsteadiness still exist on the flaps and the wing upper surface at some flight conditions, offering a buy-in for active flow control.

The report by McLean et al. designated simple hinge flap devices as a promising technology for implementation on commercial transport vehicles [2]. Extensive wind tunnel investigations showed that AFC blowing over a simple hinge flap can keep flow attached on flaps at deflection angles up to 40° [12, 13]. These studies inspired the design of a simple hinge flap installed on the CRM-HL with AFC blowing over the hinge, and this was compared to the traditional Fowler flap devices in the landing configuration through wind tunnel experiments and simulations [14, 15]. The simplified, AFC-augmented high-lift configuration at nozzle pressure ratios greater than 2.0 outperformed the lifting performance of the conventional high-lift system and would reduce vehicle weight and cruise drag due to the removal of complex flap support systems and fairings. After indications that this AFC implementation was not a near-term technology, Vatsa et al. performed simulations of simple blowing over the Fowler flaps to decrease the drag on the vehicle due to reduced separation but found limited success [16].

The aileron of the CRM-HL was another candidate control surface to augment with active flow control from initial investigations by Shmilovich et al., so Andino et al. implemented AFC blowing over the hinge of a highly deflected aileron in the takeoff configuration of the CRM-HL [17, 18, 19, 20]. Blowing over the highly deflected aileron with steady jets contributed to a 4.8% higher L/D at the nominal takeoff C_L [18]. This is a promising result because only a 1% increase in takeoff L/D could result in a 2,800 lb. payload increase or a 150 nm range increase, based on a system study by Garner et al. [21]. This technique was demonstrated on the 10% scale CRM-HL through wind tunnel testing but added to the complexity of the aileron control surface by incorporating blowing actuators and plumbing systems to supply air for augmentation. Though an increase in takeoff L/D can directly impact vehicle takeoff weight and range, the complexity and weight penalty of additional air supply systems may reduce these predicted benefits. In a systems analysis for localized AFC applications, Vijgen et al. found a 0.125% increase in the vehicle’s operating

empty weight for an APU-powered air supply system to the aileron [22]. McLean et al. reported that the aileron and supporting systems account for 0.61% of the empty weight of a Boeing 777 aircraft [2]. By removing the aileron control surface, support systems, and actuation mechanisms, these system studies indicate that complete replacement of the aileron control surface with trailing-edge circulation control actuation could have less weight than an AFC-augmented aileron. This is possible if the weight for the air supply systems do not exceed the weight of the removed aileron systems. Further, the AFC-augmentation to the aileron is unlikely to be utilized outside of takeoff and landing conditions since the aileron separation is not observed in other flight conditions. The circulation control approach could produce a similar takeoff L/D benefit while serving as a roll control effector across the vehicle's flight profile.

Circulation control is often applied to an airfoil by capitalizing on the Coanda effect and blowing air over a blunt, rounded trailing edge. These “Coanda actuators” can produce significant lift increments over airfoil sections, and they have become increasingly popular due to advancements in manufacturing techniques. Small scale circulation control research began on circulation control airfoils with thick cross-sections for rotorcraft applications [23]. More recent investigations have experimented with circulation control actuators for thinner sections and transonic airfoils [24, 25]. The applications have even moved beyond wind tunnel testing and onto flow control demonstrator vehicles. Warsop et al. applied circulation control actuators to the trailing edge of the MAGMA demonstrator aircraft for entirely fluidic flight controls [26]. Simon et al. employed trailing-edge circulation control for fast response flight control on a tailless delta wing vehicle [27]. Pisano et al. demonstrated that partial span Coanda-based flow control on a VTOL tail-sitter vehicle can provide the same yaw torque as a full-span hinged flap [28]. The success of these demonstrations motivates the implementation of circulation control effectors on the realistic CRM-HL geometry.

The objective of the current study is to examine the effect of replacing the aileron of the CRM-HL with trailing-edge Coanda actuators for enhanced L/D in the takeoff configuration. This is investigated with CFD simulations, starting with validation of the computational approach, design of the actuators at takeoff conditions using two dimensional CFD, and finally implementation of the three-dimensional Coanda actuators on the semispan model of the CRM-HL.

III. Methodology

A. Computational Methods

Reynolds-Averaged Navier Stokes (RANS) CFD simulations are conducted using NASA’s Fully Unstructured 3D (FUN3D) solver. FUN3D uses a node-based finite-volume discretization. The low-diffusion flux-splitting scheme and the stencil-based Van Leer flux limiter with a heuristic pressure limiter govern the flux constructions. The Hierarchical Adaptive Nonlinear Iterative Method (HANIM) contains a preconditioner, a generalized conjugate residual (GCR) method, realizability checks, and an adaptive CFL controller to accelerate simulation convergence [29]. Turbulence is modeled with the negative Spalart-Allmaras one-equation turbulence model with rotation and curvature corrections applied (SA-neg-RC) [30, 31]. The computational domain is designed to simulate the test article in free flight with Riemann-invariant farfield boundary conditions applied 100 chord lengths away from the model. Additionally, a symmetry plane bisects the model for reduced computational cost. The mesh is coarsened near the farfield boundary conditions to reduce reflections of pressure waves and to numerically diffuse the turbulent eddy viscosity.

All simulations in this study use the adaptive mesh refinement algorithm Sketch-to-Solution (S2S) coupled with FUN3D, as described by Kleb et al. [32]. S2S’s geometry-linked mesh adaptation is performed according to the discretization error from the reconstruction of the Hessian matrix of the Mach number. Each mesh is initialized with the previous mesh’s solution and then passed through 500-2000 flow solver iterations before adapting the next mesh to the newest mean flow solution. Iterative convergence for this study is defined by less than 1% variation in the lift coefficient over the last 10% of flow solver iterations. Formal mesh convergence is the point at which the lift coefficient changes by less than 1% when the cell count is doubled. This mesh convergence criteria could not be achieved in all cases for this study due to the complexity of the CRM-HL, so the adaptive mesh refinement algorithm was stopped once the mesh exceeded 1×10^9 tetrahedral cells ($\sim 2 \times 10^8$ nodes) due to prohibitive simulation costs.

Matching the experimental wind tunnel conditions, the freestream Mach number was 0.2, and the Reynolds number based on the reference chord of the CRM-HL was 3.0×10^6 . Most of the simulations in the current study were performed at the nominal takeoff angle of attack: 4.6° . Wind tunnel corrections through the Transonic Wall Interference Correction System (TWICS) were applied to the experimental data resulting in the geometric angle of attack of 4° being corrected to 4.6° [18, 33]. These freestream conditions were applied for the 3D semispan simulations as well as the 2D characteristic airfoil simulations.

B. High-Lift Common Research Model

The High-Lift Common Research Model used for this study is the 10% scale model of the CRM-HL tested in the 14- by 22-Foot Subsonic Wind Tunnel (14x22) experiments [18, 19, 20].

1. CRM-HL semispan model

In the takeoff configuration, the leading-edge slats are deployed to 22° , and both flaps are deflected by 25° . The leading-edge slat is sealed so that no flow can cross from the slat cove to the suction surface of the main element. This eliminates the effect of the slat bracket wakes on the upper surface of the wing's main element. The model also has a flow-through nacelle mounted to the wing with a pylon near 35% span. The nacelle has a chine to help reduce flow unsteadiness and separation on the inboard section of the main element at high angles of attack. The aileron is installed on the outboard wing section from approximately 73% to 95% span, and it can be deflected to angles of 0° , 7.5° , 16° , and 25° . The 10% scale semispan model installed in the 14x22 is pictured in Fig. 1. In simulations, a symmetry plane was applied at the centerline of the fuselage so that the peniche and wind tunnel floor did not need to be modeled. The simulation geometry included all 15 support brackets for the slats as well as the 3 flap fairings that aerodynamically shield the flap support brackets. The 10% scale model has a reference area of 1.918 m^2 and a mean aerodynamic chord of 0.7005 m . The moment reference center in the model coordinate system is at $(3.368 \text{ m}, 0 \text{ m}, 0.4520 \text{ m})$.

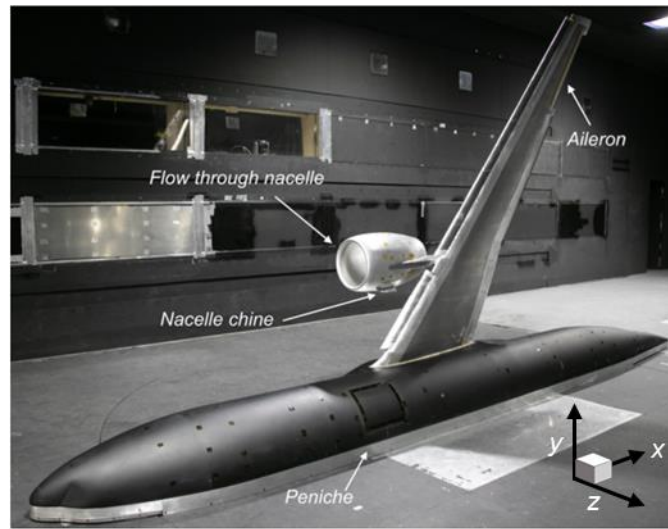


Fig. 1 CRM-HL installed in NASA Langley 14- by 22-Foot Subsonic Wind Tunnel, adapted from [18]

2. CRM-HL characteristic airfoil

Design of the Coanda actuators was completed with a two-dimensional (2D) characteristic airfoil section of the CRM-HL. This supercritical airfoil section was extracted from the CRM-HL in the takeoff configuration at 82% span. This spanwise location is near the middle of the aileron control surface, and the wind tunnel model has a row of pressure taps at this spanwise location. The drawings in Fig. 2 depict this airfoil modified for different configurations including the leading-edge slat deflected to 22° . The flow direction is left to right. The first cross-section is the original airfoil at this spanwise location of the CRM-HL without the aileron deflection. The second cross-section of Fig. 2 shows the aileron deflected to 16° with a jet installed to blow over the aileron hinge, as used in wind tunnel experiments of Andino et al. [18]. The third cross-section demonstrates the CRM-HL airfoil section modified to accommodate a trailing-edge Coanda actuator. The airfoil was thickened linearly about the mean camber line while keeping the chord length the same. The depicted example required a trailing-edge thickness nearly 20 times larger than the original airfoil—the main contributor to increased drag. The circular Coanda surface was installed such that the slot blows tangential to the camber line from both the upper and lower sides of the Coanda surface, which have independent internal plenums.

The shape of the Coanda surface and size of the slot are the confounding variables for performance of the circulation control actuators. Candidate shapes included circular, biconvex, and elliptical surfaces. The Coanda surface is typically characterized by the radius of curvature at the slot exit normalized by the airfoil chord (r/c). Recommended values for r/c from the literature range from 0.02 to 0.06 [24]. The height of the actuator over the Coanda surface is typically normalized by the radius of the Coanda surface (h/r). The literature recommends values of h/r in the range of 0.01 to 0.08 [24]. The reference chord of this CRM-HL characteristic airfoil measures 0.402 m from the stowed positions of the control surfaces. The simulations of CRM-HL characteristic airfoil used the same freestream Mach

number and unit Reynolds number as the full vehicle simulations. Additionally, the 2D airfoil simulations were performed at the nominal takeoff angle of attack ($\alpha = 4.6^\circ$).



Fig. 2 Two-dimensional cross-sections of the CRM-HL at 82% span for (top) original geometry, (middle) AFC-augmented aileron, and (bottom) installed trailing-edge Coanda actuator

IV. Simulation Validation

A. Validation Setup

Validation of the simulation methodology was completed by recreating the circular wall jet experiments of Novak et al. [23]. A study by Swanson et al. described the validation simulations that were performed in the CFL3D and FUN2D codes with multiple turbulence models to match the experimental results of Novak et al. [34]. From these simulations, Swanson et al. recommended best practices for turbulence model choices which guide the current work.

1. Circular wall jet model

The circulation control airfoil of Novak et al. (Fig. 3) has a chord length of 0.381m and a maximum thickness (t/c) of 15.6% of the chord length [23]. It has a supercritical leading edge and a Coanda surface at the trailing edge with a radius (r) of 0.0254 m ($r/c = 0.067$). The blowing slot has a height (h_{slot}) of 0.76 mm, characterized by $h/r = 0.030$, and the trailing edge of the upper surface has a thickness (h_{TE}) of 0.56 mm. The wind tunnel experiments of Novak et al. were performed with the airfoil at a geometric angle of attack of 0° , and the airfoil spanned the entire 0.610 m width of the wind tunnel. The model contained 67 surface static pressure taps, and these were used to calculate the lift coefficient and pitching moment coefficient with numerical integration. The drag coefficient was calculated from integration of the total pressure wake rake measurements. Two momentum coefficients ($C_\mu = 0.03$ and 0.10) were applied for flow control through the jet, and LDV measurements were taken at many locations on the model.

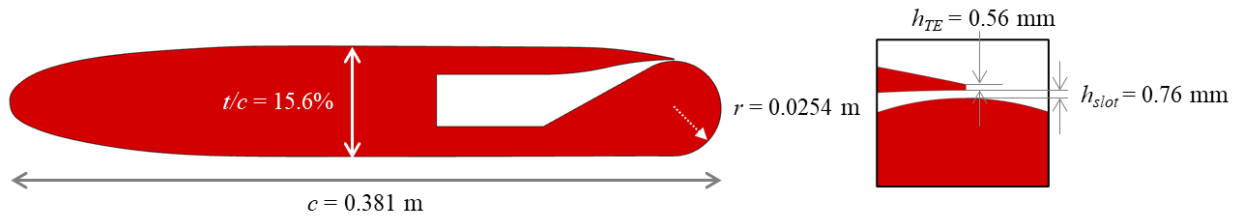


Fig. 3 Diagram of circulation control airfoil used in experiments of Novak et al. [23]

2. Simulation Setup

The airfoil was modeled with steady-state RANS simulations with the SA-neg-RC turbulence model, as demonstrated by Swanson et al. [34]. The freestream Mach number was 0.12, and the Reynolds number based on the chord length was approximately $1e6$. Though the geometric angle of attack was 0° , Swanson et al. reported that a significant downwash was imposed on the model due to the interaction of the Coanda actuation and the viscous wind tunnel walls. The corrected angles of attack for the computations in a free domain were -2.46° and -5.86° for the applied momentum coefficients of 0.03 and 0.10, respectively. The jet was activated in the simulation with a subsonic inflow boundary condition at the entrance to the plenum, where total pressure and total temperatures were specified as ratios to the freestream static pressure and static temperature. Table 1 summarizes the boundary conditions applied to target each momentum coefficient setpoint, as defined by Swanson et al. [34].

Table 1 Summary of AFC boundary conditions used for simulations of circulation control airfoil in Fig. 3

C_μ	NPR	NTR	α_{corr}
0.03	1.089	1.000	-2.46°
0.10	1.275	1.000	-5.86°

B. Validation Results

The current study's simulation validation results were compared directly to the experimental values of Novak et al. [23]. The airfoil surface pressures from experiments and FUN3D computations are compared in Fig. 4 for both momentum coefficients. Additionally, Table 2 reports the differences between the experimental and simulated lift coefficients and momentum coefficients by using the Swanson et al. boundary conditions. There is agreement in pressure coefficients across much of the airfoil, with the CFD demonstrating more curvature in the distribution that is not captured with the discrete taps. Overshooting the target momentum coefficient of 0.03 results in accurate prediction of leading-edge suction but overprediction of Coanda surface suction. The opposite is true for undershooting the target momentum coefficient of 0.10 where the Coanda surface pressure is accurate, but the leading-edge values are underpredicted.

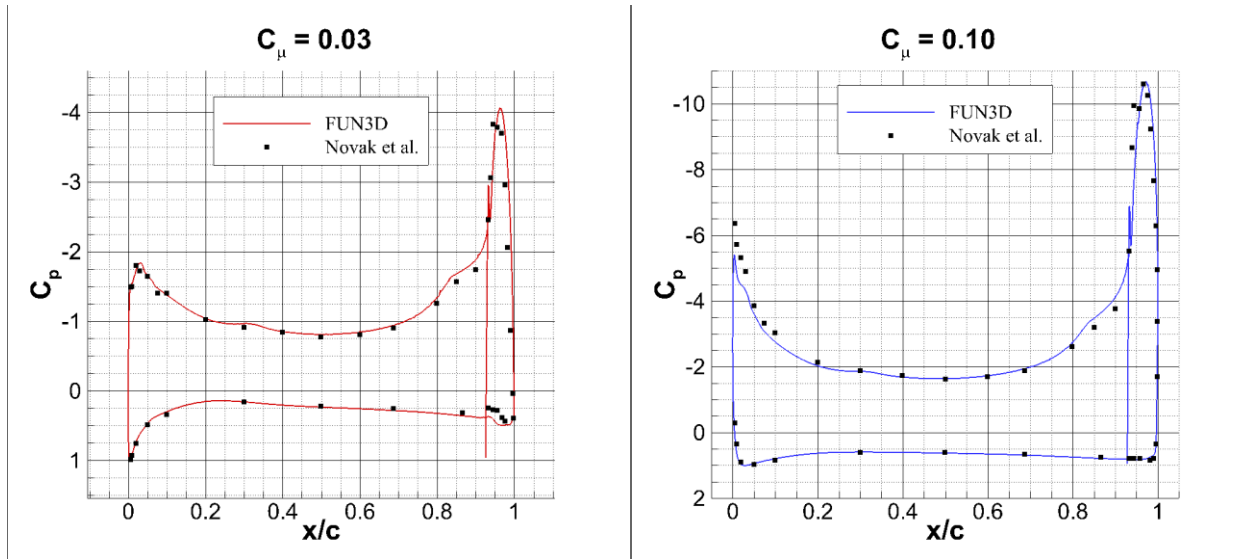


Fig. 4 Surface pressure comparisons between experiment and FUN3D simulations for (left) $C_\mu = 0.03$ and (right) $C_\mu = 0.10$ [23]

Table 2 Target and experimental circulation control airfoil values compared to CFD values [23]

Target C_μ	Experimental C_L [23]	CFD C_μ (present study)	CFD C_L
0.03	1.50	0.036	1.562 (+4.1%)
0.10	3.575	0.094	3.502 (-2.0%)

The streamlines of the flowfield around the Coanda surface are plotted for the experiment and FUN3D simulations for both momentum coefficients in Fig. 5 and 6. In the lower momentum coefficient case (Fig. 5), the simulation accurately predicted the presence of a vortex pair at the separation point of the wall jet. The higher momentum coefficient simulation (Fig. 6) demonstrated a cleaner separation which matches the experiment very well. The location of this trailing-edge stagnation point is correlated with the circulation produced by the airfoil. The leading-edge stagnation is found to move toward the pressure surface of the airfoil with increased circulation. Circulation and sectional lift coefficient are directly related by the Kutta-Joukowski theorem. As blowing increases, the jet “wraps” further around the Coanda surface, and the section lift coefficient increases significantly. The lower lift coefficient for the validation simulation at $C_\mu = 0.10$ is supported by the earlier separation point of the jet on the Coanda surface. The validation simulations provided sufficient confidence in the computational methods to extend the work to the CRM-HL characteristic airfoil and the 3D CRM-HL semispan model.

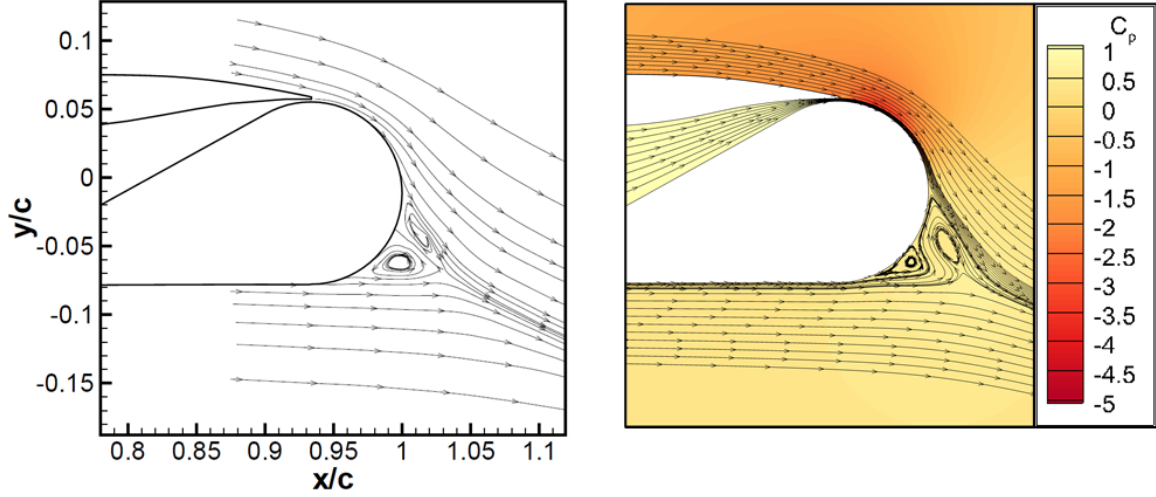


Fig. 5 Trailing-edge streamlines for $C_\mu = 0.03$ from (left) experiment [23, 34] and (right) FUN3D simulation, including contours of C_p

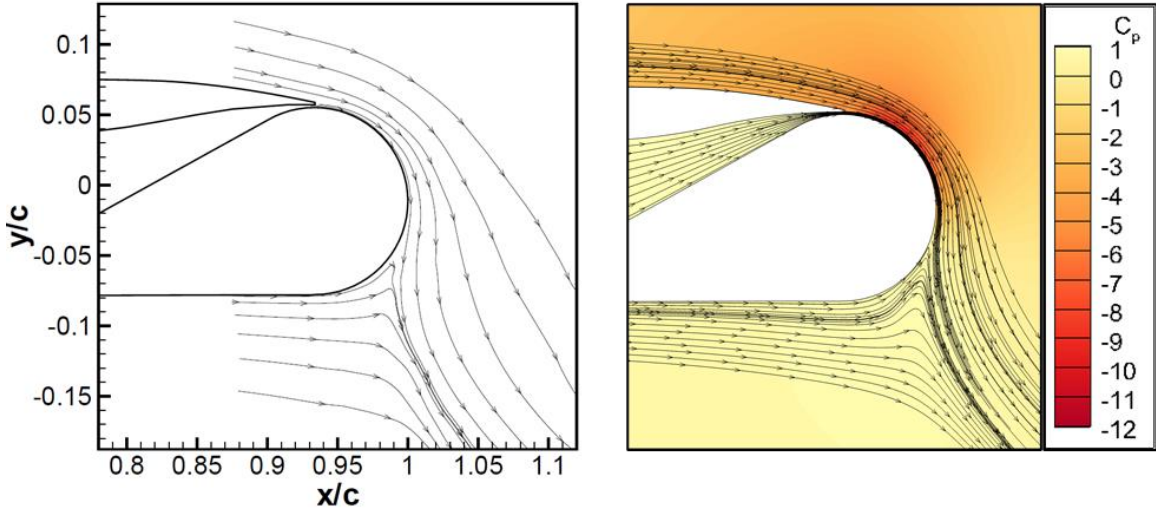


Fig. 6 Trailing-edge streamlines for $C_\mu = 0.10$ from (left) experiment [23, 34] and (right) FUN3D simulation, including contours of C_p

V. Results and Discussion

A. 2D CFD Coanda actuator design

A large set of design variables was explored with 2D RANS simulations to determine the most influential geometric variables on performance for the Coanda actuators. First, the characteristic airfoil at the bottom of Fig. 2 was compared to the airfoil truncated at 90% and 95% chord. The supercritical airfoil has a large amount of camber at the trailing edge, making it aft loaded. Truncating it resulted in lift losses up to 70%, demonstrating the necessity of retaining as much of the airfoil's original chord and camber as possible. Next, camber-aligned blowing was compared to chord-aligned blowing. The non-circular Coanda surfaces essentially have fixed separation points at the tip of the surface, which are significantly more effective when aligned with the camber of the airfoil. For the circular Coanda surface, aligning the jet with the airfoil's camber increases the wrapping angle by approximately 15° since the jet is blowing tangential to the upper surface flow instead of redirecting it upward prior to wrapping.

The C_p contour and streamline plots in Fig. 7 depict the trailing-edge flow behavior for circular, biconvex, and elliptical Coanda surfaces at $\text{NPR} = 1.3$. The circular and biconvex surfaces have a fixed radius of curvature around the entire surface. However, the curvature of the biconvex surface is not continuous at the tip, as the trailing edge thickness cannot accommodate a circular surface with the larger radius of curvature specified with the biconvex

surface. This discontinuity can be advantageous for providing a fixed separation point around the airfoil in the non-blowing case. The elliptical surface has a large radius of curvature at the exit of the slot, but the shape provides a variable radius of curvature so that there is no discontinuity in the curvature at the tip of the surface. This provides slightly more wrapping potential than a biconvex surface. The chord length changes only slightly between the three Coanda surface shapes. All the non-blowing cases demonstrate very similar trailing-edge flow patterns even though the biconvex and elliptical Coanda surfaces are supposed to promote a fixed trailing-edge stagnation point. The jet is unable to wrap around the tips of the biconvex and elliptical surfaces, significantly limiting their performance compared to the circular surface. The circular surface demonstrated a much higher range in C_L and L/D benefits across the NPR values simulated, making it the obvious choice for circulation control on this airfoil. Schlecht and Anders similarly found that the Coanda surfaces with smaller aspect ratios (near circular) had larger lift increments in low-speed flight conditions ($M = 0.3$) [24]. Forster et al. also generated a near circular Coanda surface from their adjoint optimization algorithm at their low-speed design point ($M = 0.4$) [35].

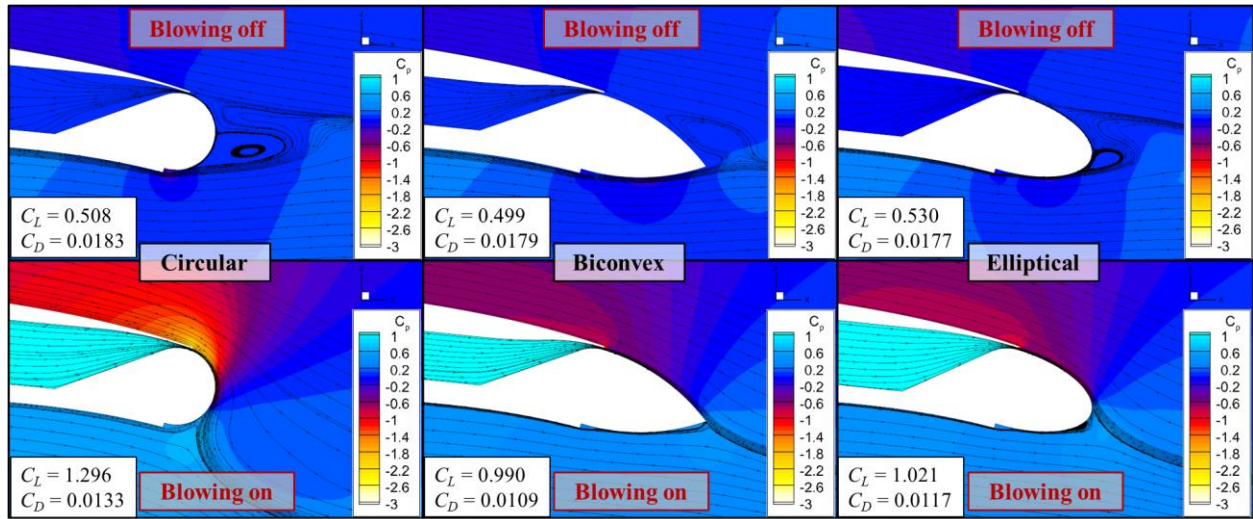


Fig. 7 Contour maps and streamlines of (left) circular, (middle) biconvex, and (right) elliptical Coanda surface shapes for blowing (top) off and (bottom) on at NPR = 1.3

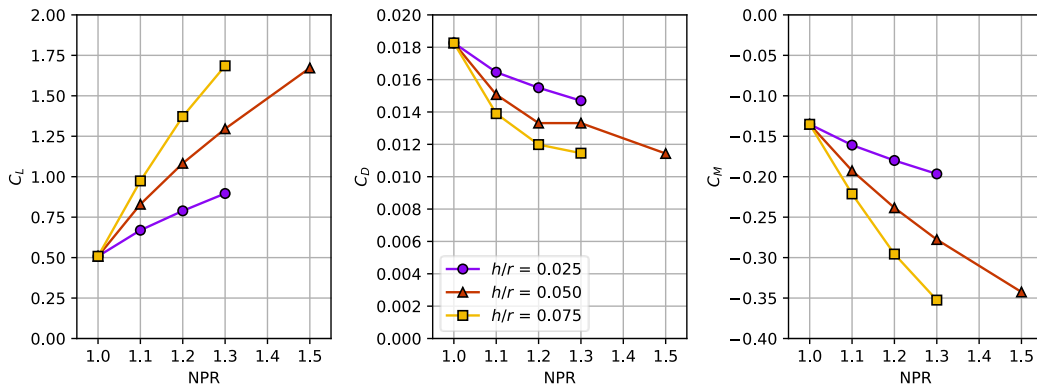


Fig. 8 Effect of slot height on (left) lift, (middle) drag, and (right) pitching moment coefficients with respect to NPR ($r/c = 0.0076$)

The slot height (h/r) was varied over the circular Coanda surface within the literature's recommended range for values of 0.025, 0.05, and 0.075. The lift, drag, and pitching moments for each are plotted in Fig. 8. The larger slots provided more mass flow (and momentum) across smaller ranges of pressure ratios, demonstrating that slot height is the most influential variable for the Coanda actuators on this airfoil. In a similar manner, the radius of the circular actuators was varied for $r/c = 0.0076$, 0.014, and 0.020 (not presented). The larger radii contributed more to baseline drag than to better L/D performance when actuated. The central r/c value (0.014) was selected for its closeness to the literature's recommendations but with slightly less impact on baseline drag than the highest r/c value. Table 3

summarizes the values of each Coanda actuator geometric variables that were tested in simulations. This design space was selectively sampled instead of explored with a full factorial design.

Table 3 Range of Coanda actuator variables investigated for 2-D design space exploration

Coanda Actuator Variable	Range tested
Actuator placement as x/c	0.90, 0.95, 1.0
Surface shape (Aspect Ratio)	Circular (1), Elliptical (2.3), Biconvex (3.1)
Blowing direction	Chord-aligned, camber-aligned
Slot height as h/r	0.010 to 0.075
Coanda radius as r/c	0.0076 to 0.039
NPR	1.0 to 1.5

The matrix of simulations of the 2D Coanda actuator designs were gathered into performance maps (Fig. 9) to determine the direct relationships between performance gains (ΔC_L) vs. NPR and slot height. A quadratic loss function $\mathcal{G}$ (defined below) was created to quantify the cost of designs for pressure, mass flow, and momentum usage. Each term in the loss function is designed to be equally weighted. The pressure term of the loss function ($\text{NPR} - 1$) indicates the target value for NPR is 1.0, and this produces a cost $\mathcal{O}(0.1)$. The mass flow equation is directly related to the height of the slot, so the mass flow term in the loss function is simply the height of the slot in mm ($\mathcal{O}(0.1)$). The momentum term of the loss function is the momentum coefficient multiplied by 10 to elevate this cost to $\mathcal{O}(0.1)$. Target values for the mass flow and momentum terms are both 0. The full quadratic loss function is the sum of the squares of each of these terms.

$$\mathcal{G}(h, \text{NPR}) = (\text{NPR} - 1)^2 + h^2 + \left(10 * C_\mu(h, \text{NPR})\right)^2 \quad (1)$$

Plotting the performance map with respect to the loss function identifies the least expensive slot height for each lift coefficient gain (ΔC_L), highlighted with the markers and dashed line on the right of Fig. 9. The red marker indicates an infeasible design point while the green markers are manufacturable design points, making this a constrained optimization problem. It is observed that the optimal design would have an adjustable slot height to minimize the loss function at each desired ΔC_L , but this would be mechanically complex and defeat the purpose of a simplification of the CRM-HL control surface. Instead, a fixed slot height was selected at the minimum manufacturable design point (on the constraint), which was considered $h_{\text{slot}} = 0.25\text{mm}$.

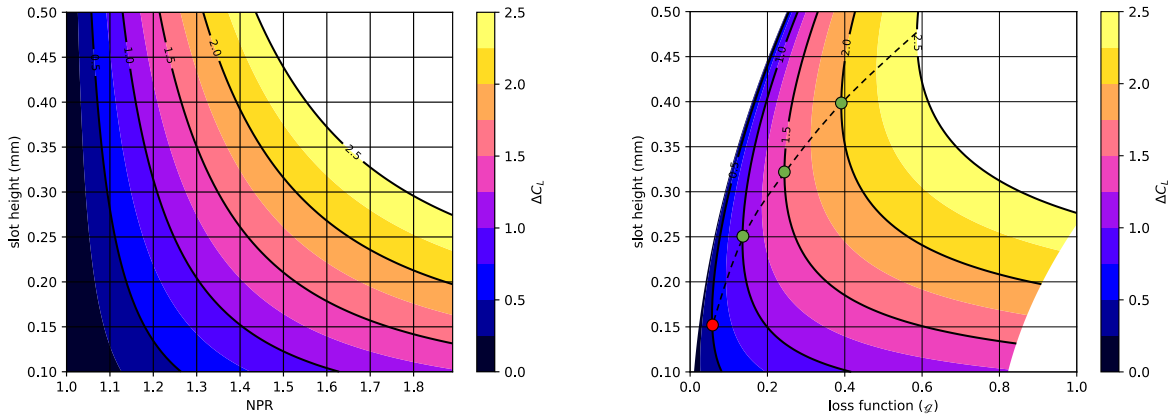


Fig. 9 Contours of ΔC_L plotted against slot height and (left) nozzle pressure ratio and (right) the quadratic loss function, $\mathcal{G}$, to determine the optimal slot height

B. Traditional aileron performance on CRM-HL

Before the Coanda actuators were tested on the CRM-HL, the performance of the traditional aileron control surface was evaluated in the CFD simulations of the current study. This provided context for the simulated performance of the baseline CRM-HL in takeoff conditions before modifications were made to the wing surfaces, and it determined the threshold for improving the takeoff L/D with active flow control applied. Additionally, the active flow control

augmentation of Andino et al. was simulated to compare the L/D benefits produced in experiments to those attained with the simulation methodology of the current study [18].

The four aileron positions (δ_a) of 0° , 7.5° , 16° , and 25° were simulated at the nominal takeoff angle of attack (α) of 4.6° to compare to the TWICS corrected experimental wind tunnel measurements of Koklu et al. [19]. The direct comparison between FUN3D simulations and experimental values are presented in Fig. 10 for lift coefficient, drag coefficient, pitching moment coefficient, and lift-to-drag ratio against aileron deflection. The simulations underpredict lift by approximately 5% and drag by 7.5%. This contributes to higher lift-to-drag ratios in the simulations by an average of 2.7%. The magnitude of the pitching moment coefficient varies between simulation and experiment by just over 11%.

In these simulations, surface pressures across the main element agree well. However, flap circulation was often underpredicted, and separation on the flap behind the nacelle and pylon was more severe. Additionally, the wind tunnel model did not have boundary layer trips on the leading-edge slat whereas the RANS simulations assumed turbulent flow over the entire model. This contributes to slightly lower suction on the simulated slat upper surface, leading to less lift on this control surface.

The complexity of this wind tunnel model limits the predictive power of simulations. Even with adaptive mesh refinement, RANS models frequently lead to these underpredictions in lift and drag coefficients with fully built-up models. Test Case 2 in AIAA's 5th High-Lift Prediction Workshop demonstrated this exact issue, where participant simulation accuracy declined as the commercial transport model contained more devices such as the slats, flaps, and nacelle [36]. Solutions utilizing adaptive mesh refinement were closer to the experimental values than fixed-grid RANS solutions but still experienced this decline in accuracy as the model was built-up, even at low angles of attack.

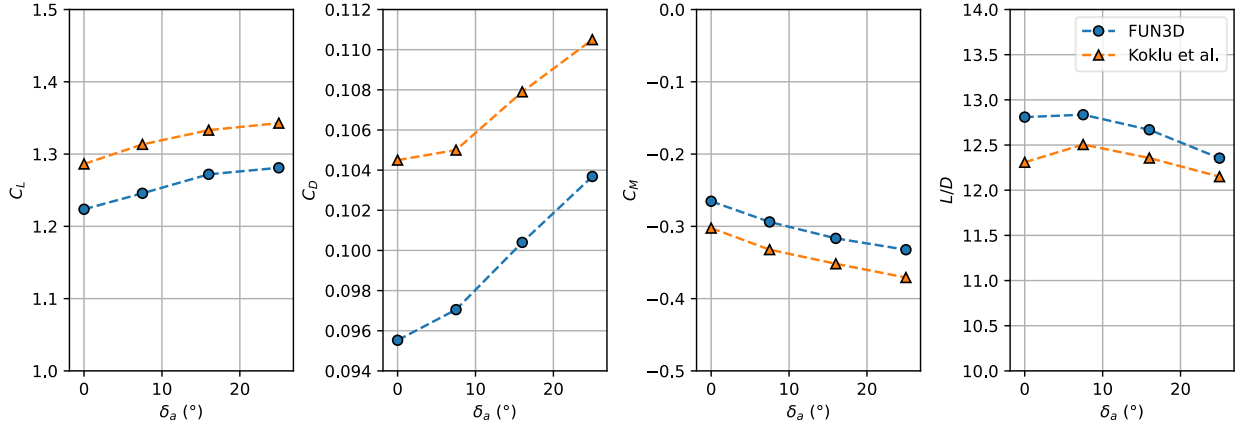


Fig. 10 Comparison of aerodynamic coefficients produced from simulation and experiment for aileron deflection on CRM-HL, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$ [19]

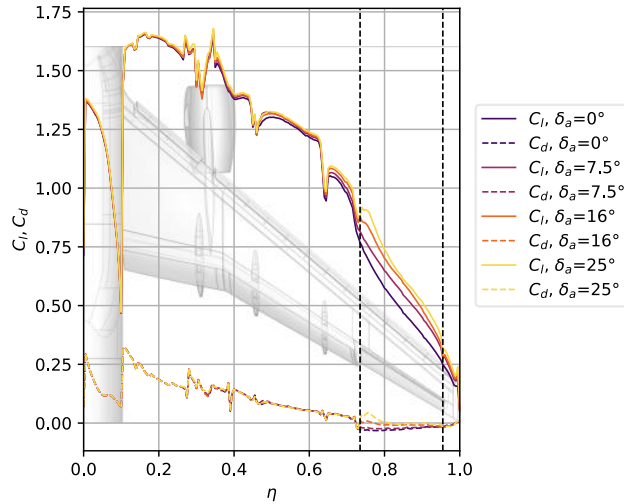


Fig. 11 CFD-extracted section lift and drag coefficient distributions due to aileron deflections at takeoff, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

As observed in the simulation results (Fig. 10), positioning the aileron at 25° produces a lift increase of 4.7% and a drag increase of 8.5%. The maximum L/D is found with the aileron deflected to 7.5° (also true for experiments), which is the prescribed takeoff aileron position for the CRM-HL. The simulation enables the calculation of section lift and drag coefficients across the entire span of the vehicle (normalized by the mean aerodynamic chord) as presented in Fig. 11, with vertical dashed lines indicating the spanwise limits of the aileron control surface. Even though the aileron only spans from 73% to 95% span, it still benefits lifting performance on the outboard flap ($0.65 < \eta < 0.73$) and near the wingtip ($\eta > 0.95$). The section drag coefficient penalties for the highest aileron deflections outweigh the lift benefits, leading to a lower L/D . The aileron's position further out on the span is due to its primary use for roll control, but this control surface clearly produces a nontrivial lift benefit that can be utilized during takeoff and landing.

The work of Shmilovich et al. and Andino et al. targeted the use of a 16° aileron deflection augmented with active flow control to attain even higher L/D values at takeoff [17, 18]. CFD simulations complementing the experimental work of Andino et al. were completed in the current study with the full array of converging-diverging jets (CD1), and the spanwise loading distributions of the AFC-augmented aileron are presented in Fig. 12 for two nozzle pressure ratios. The AFC actuators reduce separation and realign surface flow to enable higher section lift coefficients on the aileron. With AFC activated, surface flow is further improved over most of the wing but is most pronounced over the aileron section. With $NPR = 3$ at this fixed angle of attack, lift is increased by 5.5% compared to the nominal takeoff aileron position of 7.5° [18]. For these AFC simulations, the lift, drag, and pitching moment coefficients underpredicted Andino et al. experimental values by 4.9%, 5.1%, and 12.4%. The increased accuracy in drag for the AFC cases compared to the non-blowing CRM-HL simulations highlights how RANS simulations struggle to predict drag values in highly separated flows, such as the 16° -deflected aileron without AFC augmentation.

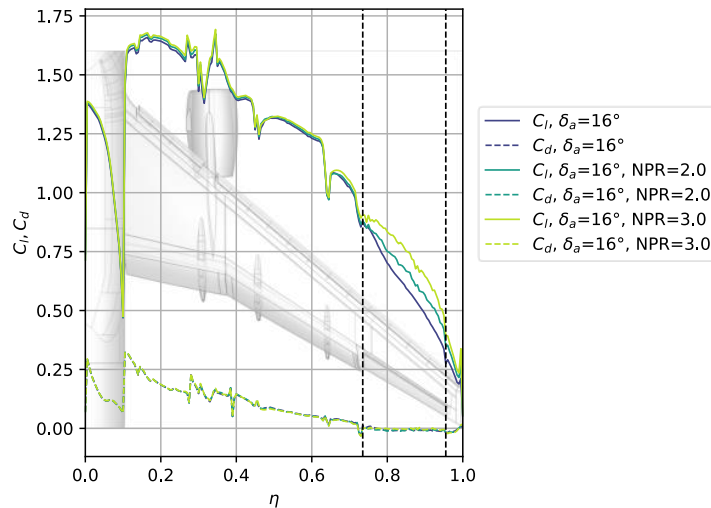


Fig. 12 CFD-extracted section lift and drag coefficient distributions due to AFC-augmentation over 16° -deflected aileron, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

C. Coanda flow control on the 3D CRM-HL model

The optimal Coanda actuator from the 2D simulation analysis was a circular surface characterized by $r/c = 0.014$ and $h_{slot} = h_{TE} = 0.25\text{mm}$. For installation on the CAD model of the CRM-HL (Fig. 13), the blowing was aligned normal to the trailing edge. Therefore, the trailing-edge-normal chord was the reference chord for the nondimensional parameters. Due to the CRM-HL's tapered wing, the trailing-edge-normal chord length decreases further out on the span, requiring the Coanda surface (cyan in Fig. 13) radius to decrease with span from 4.85mm to 3.53mm. The Coanda AFC segment was installed entirely within the aileron-cutout section of the model (red in Fig. 13), from 73% to 95% semispan, so that it could realistically be installed on the 10% scale CRM-HL wind tunnel model. The original CRM-HL airfoil shape was thickened symmetrically about the mean camber line to accommodate the circular Coanda surface and blowing slots at the trailing edge. The GEOLAB group performed this shape alteration while retaining curvature continuity on the upper and lower surfaces of the wing and without introducing inflection points in the airfoil shape.

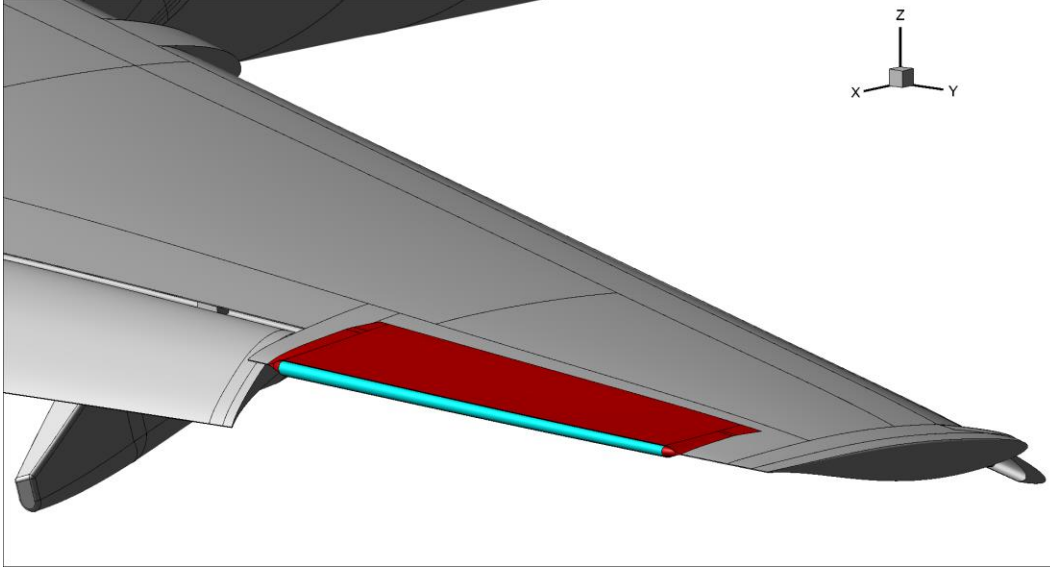


Fig. 13 Visualization of Coanda flow control installed in aileron cutout section of CRM-HL

For manufacturing simplicity and flow uniformity, the slot height (h_{slot}) is held constant across the span of the AFC segment, so the h/r characterization increases along the span from 0.052 to 0.072 as r decreases. Slots were installed at the upper and lower sides of the Coanda surface with independent plenums to enable blowing from one or both slots for control like the modified airfoil in Fig. 2. Alternate designs were created with the slot discretized into straight jets with a 10:1 width-to-height ratio at the exit plane. There are therefore unactuated regions on the Coanda surface between each discrete straight jet. These two alternate models had slots with 25.4% and 50.3% of the original slot exit area for applying flow control at lower mass flow rates.

The impact that these modifications have on the baseline model at takeoff without AFC are tabulated below in Table 4. In comparison to the aileron at 0° deflection, lift marginally increases, and drag increases by 12 counts. There is then a slight reduction in L/D . Since the aircraft is now unable to deflect an aileron to 7.5° , as prescribed for the takeoff configuration, the AFC must now provide these benefits.

Table 4 Effect of Coanda actuator modifications to CRM-HL for performance at takeoff (NPR = 1)

	C_L	C_D	C_M	L/D
$\delta_a = 0^\circ$	1.224	0.0955	-0.265	12.81
$\delta_a = 7.5^\circ$	1.246	0.0971	-0.294	12.84
Coanda, no blowing	1.229	0.0967	-0.265	12.71

1. Fixed angle of attack results ($\alpha = 4.6^\circ$)

The 2D design simulations focused on control at low pressure ratios, so the nozzle pressure ratios supplied separately to the actuators on the semispan CRM-HL were 1.25, 1.5, 1.75, and 2.0. At a constant angle of attack, the effects on the aerodynamic coefficients of the vehicle are plotted in Fig. 14. By actuating the upper slot flow control at NPR = 2, the lift coefficient is increased by 10.5%. This demonstrates that the flow control has significantly more range in lift and pitching moment coefficients than any position of the aileron previously tested (Fig. 10). From linear interpolation, the upper surface can recover the nominal takeoff lift coefficient with just NPR = 1.09. The maximum L/D is found at the lowest NPR setting for the upper slot blowing (NPR = 1.25)—a 1.9% increase over the L/D produced by the 7.5° aileron deflection at the same angle of attack.

The lower slot blowing demonstrates behavior that is nearly symmetric to the upper slot in lift and pitching moment coefficients. The drag increases similarly for upper and lower slot blowing, leading to significant L/D reductions for lower slot actuation. The upper slot blowing clearly contributes to performance benefits while the role of lower slot blowing is to mimic the negative deflection angles of the aileron. The vehicle would utilize these negative angles for asymmetric deflection of the ailerons to induce a strong rolling moment. This would be an uncommon event during takeoff, so this lower slot has a more significant role in other flight conditions. Even so, it is still valuable to characterize its performance at takeoff.

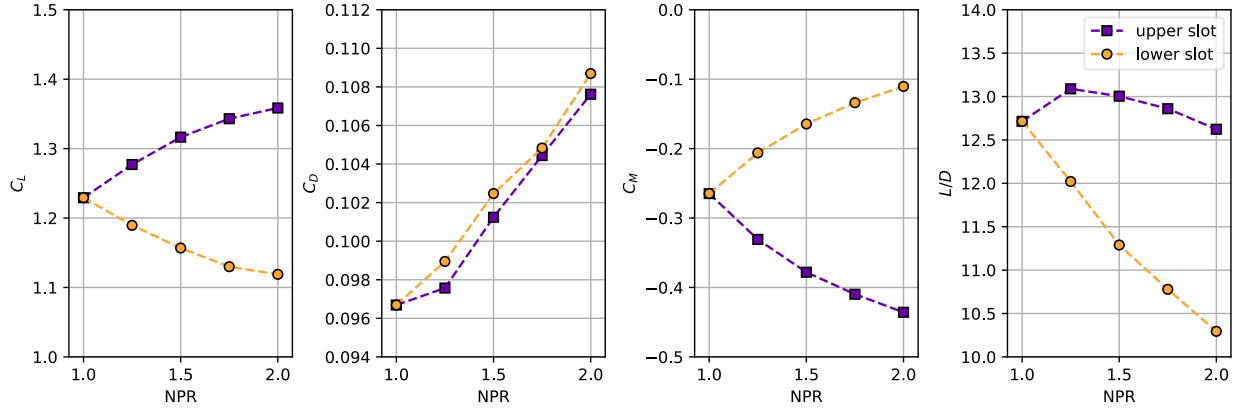


Fig. 14 Aerodynamic effect of the Coanda flow control through upper and lower slots at a range of pressure ratios, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

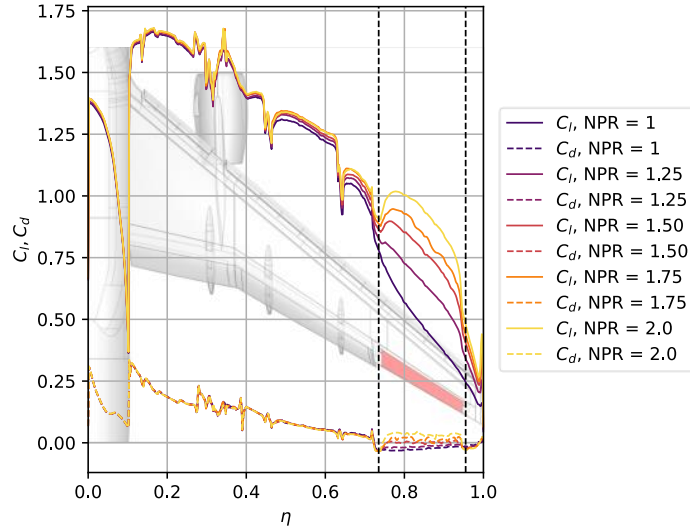


Fig. 15 CFD-extracted section lift and drag coefficient distributions due to Coanda flow control through the upper slot at varying nozzle pressure ratios, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

The section lift and drag coefficients were extracted from the CFD solutions for upper slot blowing and are presented in Fig. 15. The $NPR = 1$ condition represents no blowing and closely traces the $\delta_a = 0^\circ$ distribution in Fig. 11, consistent with the aerodynamic coefficients in Table 4. With the AFC activated, the wing experiences significant increases to the section lift distribution around the Coanda section as well as on the outboard wing. These “controlled” section lift coefficients are sometimes double those of the non-blowing condition, which is a much larger impact than deflecting the aileron or augmenting the aileron with AFC. These lift gains are competing with the drag penalties for overall L/D benefits, and the drag coefficients along the Coanda flow control segment increase significantly more than AFC augmentation of the 16° aileron.

The C_p traces at 82% span depict changes to the surface flow and overall suction produced on the wing (Fig. 16). The wing cross-sectional shape is also shown at the bottom of each plot to better understand the shape of the C_p distribution. There is a notable increase to slat suction with actuation, indicating a change in the leading-edge stagnation point location. This leading-edge stagnation point on the slat is found to move up to 2 mm with the highest blowing rates. The largest change in surface pressure is found on the aft section of the chord and on the circular Coanda surface, producing more lift and a more negative pitching moment. With lower surface blowing, the C_p curve self-intersects, which leads to negative section lift coefficients at some stations on the span for the higher blowing rates. The Coanda flow control is very effective in altering the circulation around the wing section. As observed in the validation simulations and 2D design simulations, upper slot blowing increases circulation and lower slot blowing drastically reduces circulation.

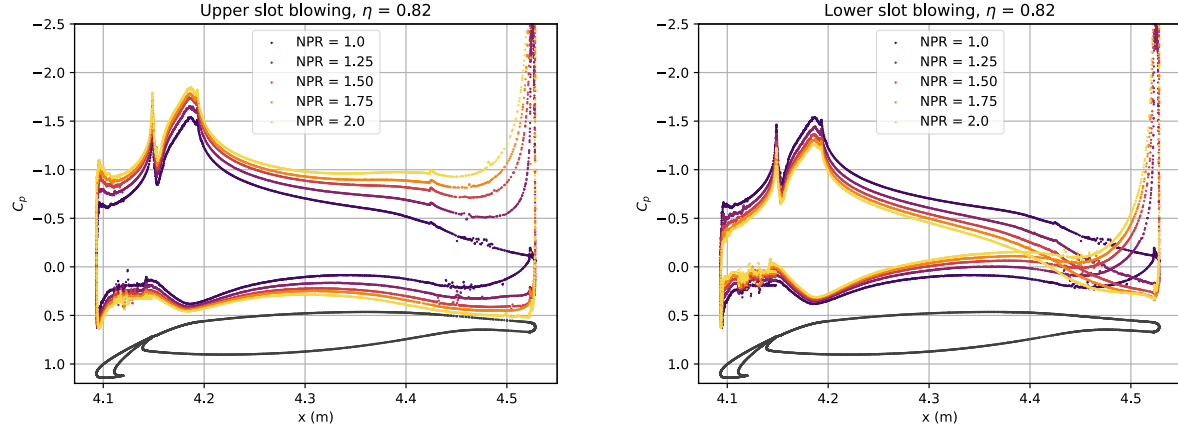


Fig. 16 Pressure coefficient traces at 82% semispan for Coanda flow control (left) upper surface blowing and (right) lower surface blowing, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

The velocity profiles around the Coanda surfaces from two AFC cases in Fig. 17 depict how the jet entrains freestream flow, mixes out, and separates while traversing the Coanda surface. The wrapping angle (red curved arrow) is characterized by the point on the Coanda surface where the jet separates from the surface (wall shear goes to zero) and can be visualized with these profiles where the wall-tangent velocity changes direction. This wrapping angle decreases with span due to the decreasing Coanda surface radius with span, since it is more challenging to maintain an attached jet to a smaller Coanda radius (larger h/r). For the upper surface blowing, the average wrapping angle increases from 81° to 105° with NPR's from 1.25 to 2.0. With this range of wrapping angles, the flow control has more wrapping potential (and lift benefit) if higher nozzle pressure ratios could be applied at takeoff.

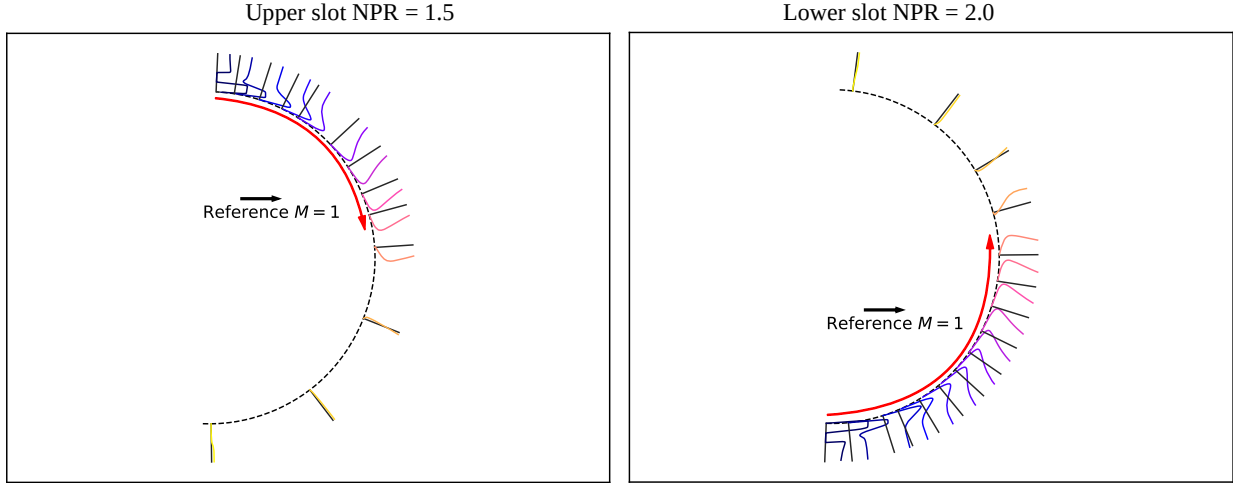


Fig. 17 Velocity profiles of Coanda jet wrapping at $\eta = 0.82$ for (left) upper surface blowing at NPR = 1.5 and (right) lower surface blowing at NPR = 2.0, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

Though the design of the control actuators prioritized low pressure ratios, it is important to characterize the active flow control system by the other coefficients used in the community which capture the mass flow (C_Q), momentum (C_μ), and power (C_π) required for the control, as calculated in the equations below.

$$C_Q = \frac{\dot{m}_{AFC}}{\rho_\infty V_\infty S} \quad (2)$$

$$C_\mu = \frac{\dot{m}_{AFC} V_{AFC}}{q_\infty S} \quad (3)$$

$$C_\pi = C_Q \frac{P_{T,AFC}}{q_\infty} \quad (4)$$

Table 5 summarizes these three coefficients for the upper slot blowing at each NPR tested and $\alpha = 4.6^\circ$. The mass flow coefficient required for control can be contextualized by understanding how much mass flow is available to the aircraft in this flight condition. The flow-through nacelle at this takeoff condition captures a mass flow rate characterized by a mass flow coefficient of 0.0274. If the mass flow through the nacelle was supplying the active flow control, then it is important to note that the lowest nozzle pressure ratio setting is using a mass flow rate equivalent to 0.93% of the mass flow captured by the nacelle. Shmilovich et al. indicated that an Auxiliary Power Unit could potentially supply a mass flow coefficient (C_Q) of 0.000167 for a wide-body twin-engine aircraft [17]. Although air is supplied to the AFC system at a low pressure, vehicle designers need to carefully consider the AFC air source and account for the impacts that this would have on thrust production and fuel burn at takeoff.

Table 5 Cost coefficients for upper slot Coanda flow control, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

NPR	C_Q	C_μ	C_π
1.25	0.000257	0.00144	0.0115
1.50	0.000355	0.00268	0.0190
1.75	0.000428	0.00363	0.0268
2.00	0.000493	0.00438	0.0352

To address the mass flow requirement concerns, the models with the discretized slots were tested at a fixed NPR, and the aerodynamic performance of these configurations are plotted in Fig. 18 against the mass flow coefficient (C_Q), which scales directly with slot area. These configurations require mass flows below the estimated APU limit of Shmilovich et al. [17]. However, the lifting performance drops off precipitously with these discretized slots. The jet emanating from the full slot separates off the Coanda surface in a continuous sheet, but the discretized slot experiences stall cells that form around the edges of each of the individual actuators, separating at a point instead of a uniform sheet. This significantly reduces the wrapping and therefore the effectiveness of the circulation control, leading to much lower section lift coefficients. This is reflected in the spanwise lift distributions of Fig. 19 and the surface flow visualizations of Fig. 20. For implementation on a physical model, much care will need to be taken so that physical supports for the full slot do not lead to the formation of these stall cells on the Coanda surface.

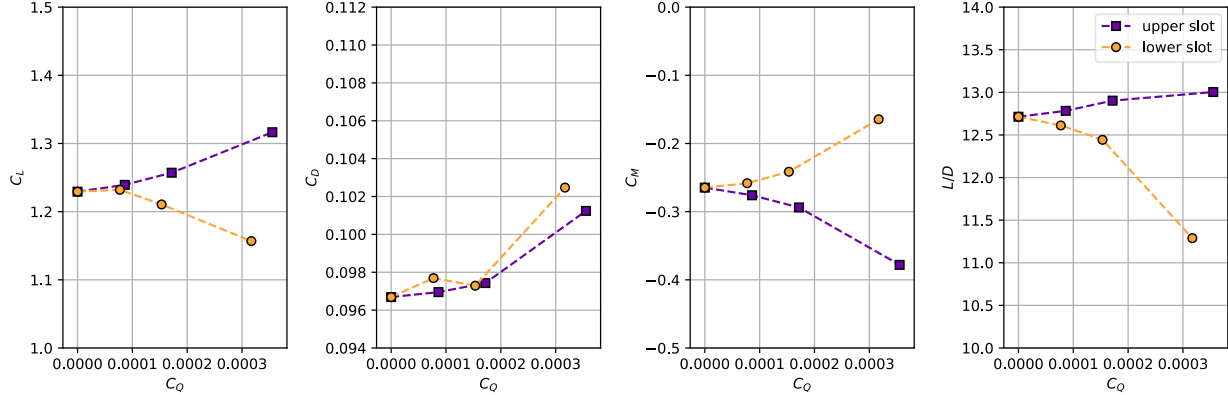


Fig. 18 Aerodynamic effect of discretizing the slot (changing C_Q) at $NPR = 1.5$, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

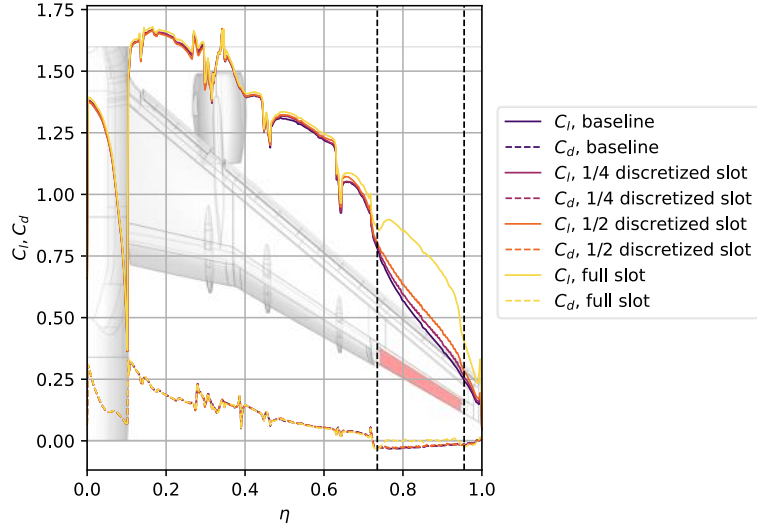


Fig. 19 CFD-extracted section lift and drag coefficient distributions due to Coanda flow control through the discretized slot models at constant NPR = 1.5, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

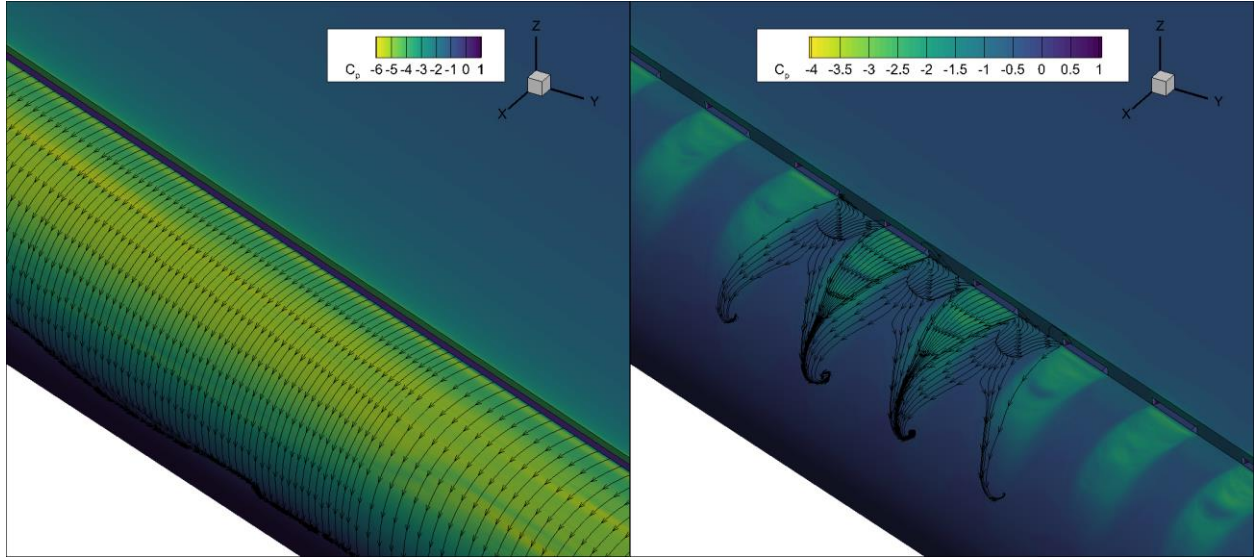


Fig. 20 Coanda surface streamline visualizations and pressure contours at NPR = 1.5 for (left) full slot and (right) $\frac{1}{2}$ discretized slot, $M = 0.2$, $Re = 3.0 \times 10^6$, $\alpha = 4.6^\circ$

2. Angle of attack sweep results

Although a nominal takeoff angle of attack is discussed throughout the current study, it can be beneficial to perform this analysis at fixed values of lift coefficient. This is because the coupling between increased drag due to increased lift makes changes to L/D less significant at a fixed angle of attack. For this analysis, the vehicle was simulated at a range of angles of attack between 0° and 6° with $\delta_a = 0^\circ$, non-blowing with Coanda actuators installed, and Coanda flow control at NPR = 2. The results of these angle of attack sweeps are plotted in Fig. 21.

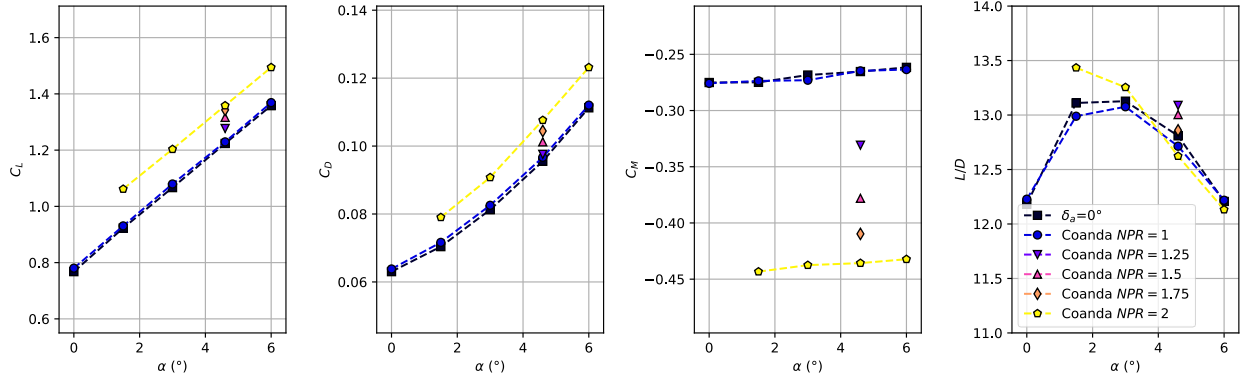


Fig. 21 Angle of attack sweep for upper slot blowing and zero aileron deflection, $M = 0.2$, $Re = 3.0 \times 10^6$

The vehicle remains in the linear lift regime for this angle of attack range, and the drag curve demonstrates quadratic behavior, enabling linear and quadratic fits with R^2 values greater than 0.9995. With changes to NPR at a constant angle of attack, the lift coefficient varies linearly with respect to momentum coefficient (not shown), and the drag coefficient varies linearly with respect to NPR (Fig. 14). This allows lift and drag coefficients to be interpolated from the simulations in Fig. 21 to any NPR at each angle of attack. Then, the linear and quadratic fits for lift and drag coefficients, respectively, can be used to calculate the L/D at a range of lift coefficients, as plotted in Fig. 22. There is a reduction in L/D for each C_L with the Coanda flow control installed compared to the $\delta_a = 0^\circ$ configuration. Applying flow control through the upper slot over the Coanda surface increases the L/D for most of the computed lift coefficients. Additionally, there is a slight increase in the lift coefficient at which the maximum L/D is produced.

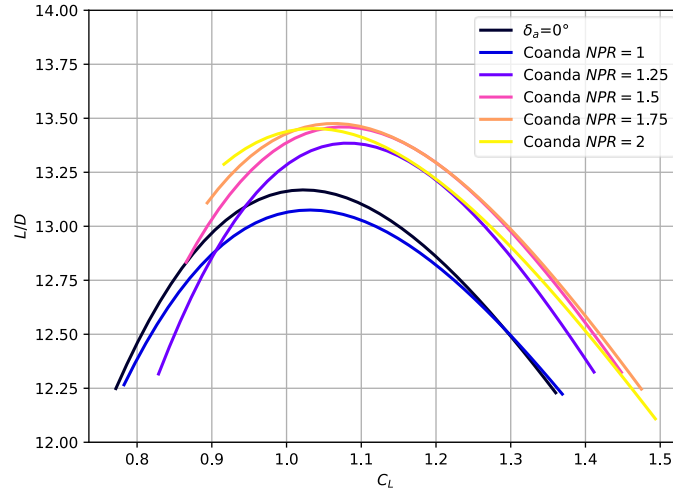


Fig. 22 Variation in L/D with C_L due to different magnitudes of upper slot blowing, $M = 0.2$, $Re = 3.0 \times 10^6$

For takeoff, it is appropriate to compare these results at the lift coefficient produced by the $\delta_a = 7.5^\circ$ configuration for the nominal takeoff angle of attack ($C_L = 1.246$ at $\alpha = 4.6^\circ$). The L/D that each AFC setting produces at this takeoff lift coefficient is extracted from Fig. 22. Select angle of attack sweep simulations were completed to determine the takeoff L/D for the AFC-aileron at NPR = 2. Finally, these datasets are plotted in Fig. 23 with the L/D measured at the takeoff lift coefficient for experiments of Andino et al. [18]. The Coanda flow control produces up to 2.5% increase in L/D compared to the nominal takeoff configuration at the takeoff lift coefficient. The L/D decreases at the highest blowing rate, and this is likely due to the higher induced drag caused by the outboard shift in lift distribution that strengthens the wingtip vortex. The experiments for the AFC-aileron produce nearly a 5% increase in takeoff L/D at NPR = 3. The limited CFD simulations of the AFC-aileron for the current study produce just greater than 1.5% L/D benefit at NPR = 2. The main benefit to installing the Coanda flow control is that it does not utilize any moving control surfaces, requiring less weight and number of parts for a notable L/D enhancement at takeoff. McLean et al. attributed 0.61% of the operating empty weight of a Boeing 777 vehicle to the aileron system. With this kind of weight reduction,

a vehicle designer can possibly afford to use this next generation roll control effector for heavier payloads, increased vehicle range, and/or shorter takeoff field lengths.

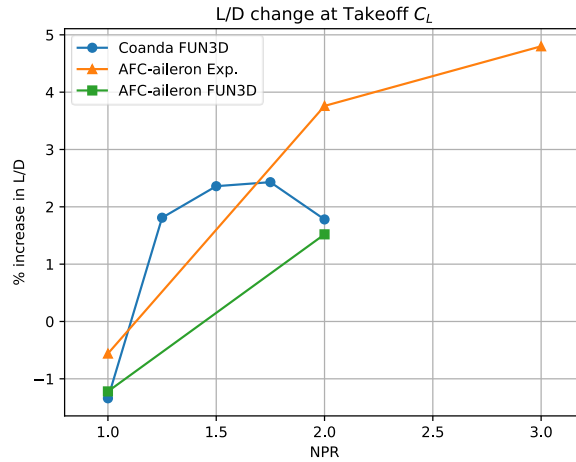


Fig. 23 Observed change to takeoff L/D at nominal takeoff lift coefficient for upper slot Coanda flow control and the AFC-aileron from simulations and experiments [18], $M = 0.2$, $Re = 3.0 \times 10^6$

VI. Conclusions

CFD simulations were utilized to replace the aileron control surface on the High-Lift Common Research Model with trailing-edge circulation control actuators (Coanda actuators). The simulation procedure for predicting Coanda actuator performance was validated against experimental data and complementary CFD simulations. This enabled the 2D simulations that investigated each geometric variable of the Coanda actuators on the characteristic airfoil and informed the design implemented on the 3D model. The supercritical airfoil produces a significant amount of lift on the aft chord and could not be truncated to accommodate the Coanda surface. It instead needed to be thickened about the mean camber line. The circular Coanda surface had greater performance and higher wrapping potential than the biconvex and elliptical surfaces. Larger slots produced more lift but required much more mass flow to do so. The Coanda surface radius had a weak effect on performance other than creating more drag in the non-blowing condition. A quadratic loss function was assigned to the 2D simulation set to determine the optimal Coanda actuator design.

The CRM-HL was simulated with the aileron in four positions, and these results were compared to experimental data to quantify the accuracy of the simulation techniques for such a complex vehicle model. Lift, drag, and pitching moment coefficients were underpredicted on average by 5%, 7.5%, and 11%, respectively. This produced L/D values larger than experiments by 2.7%. The AFC-aileron at $\delta_a = 16^\circ$ was simulated for $NPR = 2$ and 3, producing L/D predictions within 1% of experiments.

Installing the Coanda actuators in place of the CRM-HL aileron produced a takeoff drag increase of 12 counts without blowing. Actuating up to $NPR = 2$ produced a lift increase of 10.5%, which was much larger than deflecting the aileron or augmenting it with blowing. When compared at a constant takeoff lift coefficient, the Coanda actuators produce up to 2.5% increase in L/D compared to the nominal takeoff aileron setting. Most notably this benefit is achieved without any external moving parts.

The weight of the aileron and supporting systems accounts for 0.61% of a Boeing 777-class vehicle [2]. Replacing this control surface with the trailing-edge Coanda actuators could lead to a significant weight reduction. Additionally, the takeoff L/D is increased by 2.5% which contributes further to increased vehicle range, heavier payloads, and shorter takeoff field lengths [21].

Acknowledgments

The authors would like to acknowledge J. Henry Korth, Gabriel Gutierrez, and Bil Kleb for their assistance in setting up and becoming proficient in Sketch-to-Solution and their constant willingness to discuss CFD roadblocks. Additionally, the authors would like to acknowledge the GEOLAB group and Scott Brynildsen for his CAD expertise and attention to detail in CRM-HL model preparation and design. The authors would also like to acknowledge Scott Anders for assistance in setting up and acquiring the appropriate information and geometries for the validation simulations. The authors would like to acknowledge the Advanced Air Vehicles Program Fellowship Opportunities through the Advanced Air Transport Technology project for financial support and advising from the Flow Physics and

Control Branch at NASA's Langley Research Center, including Latunia Melton, Marlyn Andino, and Mehti Koklu. Finally, computing resources were provided by the NASA Advanced Supercomputing (NAS) facility under the High-End Computing Capability (HECC) project.

References

- [1] "NASA Aeronautics Strategic Implementation Plan 2023," NASA, 2023. <https://www.nasa.gov/directorates/armd/armd-strategic-implementation-plan/>.
- [2] McLean, J. D., Crouch, J. D., Stoner, R. C., Sakurai, S., Seidel, G. E., Feifel, W. M. and Rush, H. M., "Study of the Application of Separation Control by Unsteady Excitation to Civil Transport Aircraft," 1999. ntrs.nasa.gov/citations/19990061938.
- [3] Forster, M., Seidel, J. and Warsop, C., "NATO AVT-350 Task Group: Innovative Control Effectors for Manoeuvring of Air Vehicles - Advanced Concepts - Introduction and Overview," in *AIAA SciTech 2025 Forum*, Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-1270>.
- [4] Feldstein, A., Uleck, K., Floyd, J., Gray, C., Intravartolo, N., Stauffer, M. and Garrett, T., "Preliminary Design of Coplanar Joined Wing Aircraft with Integrated Active Flow Control," in *AIAA SciTech 2023 Forum*, National Harbor, MD, 2023. <https://doi.org/10.2514/6.2023-2627>.
- [5] Whalen, E. A., Shmilovich, A., Spoor, M., Tran, J., Vijgen, P., Lin, J. C. and Andino, M., "Flight Test of an AFC Enhanced Vertical Tail," in *8th AIAA Flow Control Conference*, Washington, D.C., 2016. <https://doi.org/10.2514/6.2016-3927>.
- [6] Vassberg, J. C., DeHaan, M. A., Rivers, S. M. and Wahls, R. A., "Development of a Common Research Model for Applied CFD Validation Studies," in *26th AIAA Applied Aerodynamics Conference*, Honolulu, HI, 2008. <https://doi.org/10.2514/6.2008-6919>.
- [7] Rivers, M. B., "NASA Common Research Model: A History and Future Plans," in *AIAA Aviation 2019 Forum*, Dallas, TX, 2019. <https://doi.org/10.2514/6.2019-3725>.
- [8] Lacy, D. S. and Sclafani, A. J., "Development of the High Lift Common Research Model (HL-CRM): A Representative High Lift Configuration for Transonic Transports," in *54th AIAA Aerospace Sciences Meeting*, San Diego, CA, 2016. <https://doi.org/10.2514/6.2016-0308>.
- [9] Fell, J. S., Webb, S. R., Laws, C. T., Snyder, M. L. and Rhew, R. D., "Development of the 10% NASA High-Lift Common Research Model (CRM-HL)," in *AIAA SciTech 2021 Forum*, Virtual Event, 2021. <https://doi.org/10.2514/6.2021-0385>.
- [10] Lacy, D. and Clark, A. M., "Definition of Initial Landing and Takeoff Reference Conditions for the High Lift Common Research Model (CRM-HL)," in *AIAA Aviation 2020 Forum*, Virtual Event, 2020. <https://doi.org/10.2514/6.2020-2771>.
- [11] Evans, A., Lacy, D., Smith, I. and Rivers, M., "Test Summary of the NASA Semi-Span High-Lift Common Research Model at the QinetiQ 5-Metre Low-Speed Wind Tunnel," in *AIAA Aviation 2020 Forum*, Virtual Event, 2020. <https://doi.org/10.2514/6.2020-2770>.
- [12] Melton, L. P., Koklu, M., Andino, M. and Lin, J. C., "Active Flow Control for Trailing Edge Flap Separation," in *2018 AIAA Aerospace Sciences Meeting*, Kissimmee, FL, 2018. <https://doi.org/10.2514/6.2018-1799>.
- [13] Melton, L. P., "Active Flow Separation Control on a NACA 0015 Wind Using Fluidic Oscillators," in *7th AIAA Flow Control Conference*, Atlanta, GA, 2014. <https://doi.org/10.2514/6.2014-2364>.
- [14] Lin, J. C., Melton, L. P., Hannon, J. A., Andino, M. Y., Koklu, M., Paschal, K. B. and Vatsa, V. N., "Wind Tunnel Testing of Active Flow Control on High-Lift Common Research Model," in *AIAA Aviation 2019 Forum*, Dallas, TX, 2019. <https://doi.org/10.2514/6.2019-3723>.
- [15] Vatsa, V. N., Lin J. C., Melton, L. P., Lockard, D. P. and Ferris, R., "CFD and Experimental Data Comparisons for Conventional and AFC-Enabled CRM High-Lift Configurations," in *AIAA Aviation 2020 Forum*, Virtual Event, 2020. <https://doi.org/10.2514/6.2020-2939>.
- [16] Vatsa, V. N., Lin, J. C., Melton, L. G. P., Lockard, D. P. and Ferris, R., "CFD Simulations of Landing and Takeoff CRM High-Lift Configurations," in *AIAA Aviation 2021 Forum*, Virtual Event, 2021. <https://doi.org/10.2514/6.2021-2499>.

- [17] Shmilovich, A., Yadlin, Y., Vijgen, P. and Wozidlo, R., "Applications of Active Flow Control over the Deflected Aileron of the High-Lift Common Research Model," in *AIAA Aviation Forum and Ascend 2024*, Las Vegas, NV, 2024. <https://doi.org/10.2514/6.2024-3671>.
- [18] Andino, M. Y., Melton, L. P., Koklu, M. and Hannon, J., "Active Flow Control Enhanced Aileron of the High-Lift Common Research Model at Takeoff Condition," in *AIAA SciTech 2025 Forum*, Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-1491>.
- [19] Koklu, M., Hannon, J., Melton, L. P. and Andino, M. Y., "Testing of High-Lift Common Research Model at Takeoff Conditions," in *AIAA SciTech 2025 Forum*, Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-1490>.
- [20] Melton, L. P., Koklu, M., Andino, M. Y. and Hannon, J. A., "Wind Tunnel Testing of AFC over a Deflected Aileron on the High-Lift Common Research Model," in *AIAA Aviation Forum and Ascend 2024*, Las Vegas, NV, 2024. <https://doi.org/10.2514/6.2024-3672>.
- [21] Garner, P. L., Meredith, P. T. and Stoner, R. C., "Areas for future CFD development as illustrated by transport aircraft applications," in *10th Computational Fluid Dynamics Conference*, Honolulu, HI, 1991. <https://doi.org/10.2514/6.1991-1527>.
- [22] Vijgen, P., Ziebart, A., Shmilovich, A. and Wozidlo, R., "Low-Speed Performance Enhancement using Localized Active Flow Control," NASA CR-20220006733, 2022. <https://ntrs.nasa.gov/citations/20220006733>.
- [23] Novak, C. J., Cornelius, K. C. and Roads, R. K., "Experimental Investigations of the Circular Wall Jet on a Circulation Control Airfoil," in *25th AIAA Aerospace Sciences Meeting*, Reno, NV, 1987. <https://doi.org/10.2514/6.1987-155>.
- [24] Schlecht, R. and Anders, S., "Parametric Evaluation of Thin, Transonic Circulation-Control Airfoils," in *45th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, 2007. <https://doi.org/10.2514/6.2007-272>.
- [25] Forster, M. and Steijl, R., "Numerical Simulation of Transonic Circulation Control," in *53rd AIAA Aerospace Sciences Meeting*, Kissimmee, FL, 2015. <https://doi.org/10.2514/6.2015-1709>.
- [26] Warsop, C., Crowther, W. J. and Shearwood, T., "NATO AVT-239 Taks Group: Flight Demonstration of Fluidic Flight Controls on the MAGMA Subscale Demonstrator Aircraft," in *AIAA SciTech 2019 Forum*, San Diego, CA, 2019. <https://doi.org/10.2514/6.2019-0282>.
- [27] Simon, S., Williams, D. R., Brandt, S., Osteros, R. and Munson, M., "Modeling the Aircraft Dynamics and Active Flow Control Effects of a Tailless Subscale UCAS Aircraft in 6 Degrees-of-Freedom," in *AIAA SciTech 2024 Forum*, Orlando, FL, 2024. <https://doi.org/10.2514/6.2024-0491>.
- [28] Pisano, A. P., McCrink, M. H. and Bons, J. P., "Hover Flight Testing of a Coanda Circulation Control Actuated VTOL Tail-Sitter UAV," in *AIAA SciTech 2025 Forum*, Orlando, FL, 2025. <https://doi.org/10.2514/6.2025-2197>.
- [29] Wang, L., Diskin, B., Nielsen, E. J. and Liu, Y., "Improvements in Iterative Convergence of FUN3D Solutions," in *AIAA SciTech 2021 Forum*, Virtual Event, 2021. <https://doi.org/10.2514/6.2021-0857>.
- [30] Allmaras, S. R., Johnson, F. T. and Spalart, P. R., "Modifications and Clarifications for the Implementation of the Spalart-Allmaras Turbulence Model," in *Seventh International Conference on Computational Fluid Dynamics (ICCFD7)*, Big Island, HI, 2012. https://www.iccfd.org/iccfd7/assets/pdf/papers/ICCFD7-1902_paper.pdf.
- [31] Shur, M. L., Strelets, M. K., Travin, A. K. and Spalart, P. R., "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction," *AIAA Journal*, vol. 38, no. 5, pp. 784-792, 2000. <https://doi.org/10.2514/2.1058>.
- [32] Kleb, W. L., Park, M. A., Wood, W. A., Bibb, K. L., Thompson, K. B., Gomez, R. J. and Tesch, S. H., "Sketch-to-Solution: An Exploration of Viscous CFD with Automatic Grids," in *AIAA Aviation 2019 Forum*, Dallas, TX, 2019. <https://doi.org/10.2514/6.2019-2948>.
- [33] Iyer, V., Kuhl, D. D. and Walker, E. L., "Improvements to Wall Corrections at the NASA Langley 14x22-Foot Subsonic Tunnel," in *21st AIAA Applied Aerodynamics Conference*, Orlando, FL, 2003. <https://doi.org/10.2514/6.2003-3950>.
- [34] Swanson, R. C., Rumsey, C. L. and Anders, S. G., "Progress Towards Computational Methods for Circulation Control Airfoils," in *43rd AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, 2005. <https://doi.org/10.2514/6.2005-89>.

- [35] Forster, M., Biava, M. and Steijl, R., "Multipoint Optimisation of Coanda Surfaces for Transonic Circulation Control using the Adjoint Method," in *8th AIAA Flow Control Conference*, Washington, D.C., 2016. <https://doi.org/10.2514/6.2016-3773>.
- [36] "5th AIAA CFD High Lift Prediction Workshop Presentations," 28 August 2024. [Online]. Available: <https://hiliftpw.larc.nasa.gov/Workshop5/WorkshopPresentations/HLPW5-Overall-Summary.pdf>.