

CFD and Experimental Data Comparisons for Conventional and AFC-Enabled CRM High-Lift Configurations

Veer N. Vatsa*

NASA Langley Research Center, Hampton, VA 23681

John C. Lin[†], Latunia P. Melton[‡], and David P. Lockard[§]

NASA Langley Research Center, Hampton, VA 23681

Ryan Ferris[¶]

Dassault Systemes Simulia Corp., 301 E Ocean Blvd, Suite 1200, Long Beach, CA 90802

Numerical simulations have been performed for a simplified high-lift (SHL) version of the Common Research Model (CRM) configuration, where the Fowler flaps of a representative conventional high-lift (CRM-HL) configuration are replaced by a set of simple hinged flaps. These hinged flaps are equipped with integrated modular active flow control (AFC) cartridges on the suction surface of the flap shoulder, and the resulting geometry is known as the CRM-SHL-AFC configuration. The main objective is to make use of AFC devices on the CRM-SHL-AFC configuration to produce the aerodynamic performance (lift) comparable to that of the CRM-HL configuration over a large angle of attack range encompassing maximum lift conditions. For comparison purposes, computations are also performed for the CRM-HL configuration. In the current paper, PowerFLOW[®], a CFD code based on the Lattice Boltzmann method (LBM), is used to simulate the entire flow field associated with the CRM-SHL-AFC configuration equipped with a combination of two different types of AFC actuators. The transonic version of the PowerFLOW[®] code that has been validated for high-speed flows is used to simulate the flow field generated by the high-momentum actuators required to mitigate reversed flow regions on the suction surfaces of the main wing and the flap. This study is focused on the AFC systems and actuator arrangements that had emerged based on the parametric studies conducted previously at the nominal landing condition. Comparisons of the numerical solutions for lift and surface pressures are presented here with the experimental data, demonstrating the usefulness of CFD for predicting the flow field and lift characteristics of AFC-enabled high-lift configurations over a broad angle of attack range. The numerical solutions predict the expected trends in aerodynamic forces with angle of attack variation.

*Senior Research Scientist, Computational AeroSciences Branch, Research Directorate; Associate Fellow AIAA

[†]Senior Research Scientist, Flow Physics Branch, Research Directorate; Associate Fellow AIAA

[‡]Senior Research Scientist, Flow Physics Branch, Research Directorate; Associate Fellow AIAA

[§]Senior Research Scientist, Computational AeroSciences Branch, Research Directorate; Associate Fellow AIAA

[¶]A&D Solution Consultant Specialist, Member AIAA

Nomenclature

C_D	drag coefficient, $F_x/(0.5\rho_\infty V_\infty^2 S)$
C_L	lift coefficient, $F_z/(0.5\rho_\infty V_\infty^2 S)$
C_p	pressure coefficient, $(p - p_\infty)/(0.5\rho_\infty V_\infty^2)$
F_x	x -component of integrated force
F_z	z -component of integrated force
Gpp	autospectrum of pressure
$\dot{m}$	total mass flow rate through actuators
p	static pressure
Q	dynamic pressure, $0.5\rho_\infty V_\infty^2$
Re	Reynolds number based on mean aerodynamic chord
S	wing reference area
V	total velocity
3-D	three-dimensional
x, y, z	Cartesian coordinates
y^+	turbulence length scale
α	angle of attack
η	normalized semispan location
ρ	density

Subscripts:

∞	freestream quantity
----------	---------------------

Abbreviations:

AFC	active flow control
CFD	computational fluid dynamics
CRM	Common Research Model
DNS	direct numerical simulation
HL	high-lift
LaRC	Langley Research Center
LBM	Lattice Boltzmann Method
MAC	mean aerodynamic chord
NASA	National Aeronautics and Space Administration
NPR	nozzle pressure ratio
PSD	power spectral density
SHL	simplified high-lift
VLES	very large eddy simulation
VR	variable resolution

I. Introduction

The high-lift systems of modern transport aircraft strongly influence the sizing, economics, and safety of the aircraft design [1, 2]. The NASA Advanced Air Transport Technology (AATT) Project is seeking to demonstrate the potential benefits of reducing the cruise drag (and therefore fuel burn) associated with modern high-lift systems without sacrificing aerodynamic and acoustic performance during takeoff and landing operations. One possible approach is to use active flow control (AFC) [3, 4] to provide the required lift performance while reducing the cruise drag associated with the external mechanisms used to deploy a conventional slotted flap during high-lift operations [5, 6].

A recent system integration study by Hartwich et al. [7] indicated that up to a 2.25% fuel burn reduction is possible if an AFC-enabled simplified high-lift system (i.e., simple hinged flaps inboard and outboard) could provide the necessary lift recovery at the approach angle of attack. The AFC-related performance gains are primarily due to the 3.3-count excrescence drag reduction from the removal of the external fairings for the Fowler flap mechanism [7] (see Fig. 1, left side for a typical example). However, the main challenge here is to develop an AFC system that can provide the necessary lift recovery for a simple hinged flap high-lift system (Fig. 1, right side) while minimizing the pneumatic power usage. Pneumatic-powered AFC concepts, such as

steady blowing through discrete convergent-divergent nozzles [6] or unsteady blowing through fluidic oscillators (often referred to as sweeping jet actuators) [8–18], have been used in the past to mitigate reverse flow regions and enhance the lift characteristics of high-lift systems.

Based on recent experimental work of Lin et al. [19], a combination of fluidic actuators and steady blowing along two rows on the flap shoulder region of an AFC-enabled simple hinged high-lift configuration was shown to be very effective for achieving lift comparable to an equivalent conventional high-lift system at nominal landing conditions. It is worth noting that such experimental work is extremely expensive and time consuming, and practical considerations limit the design space that can be explored experimentally. Also, as reported by Vatsa et al. [20], CFD can play an important role in downselecting more efficient configurations by conducting simulations of several potential candidates before investing in expensive model fabrication and testing. The work of Ref. [20] clearly demonstrated that CFD can predict the correct qualitative and quantitative trends for integrated forces and pressure distributions for such complex configurations in the presence of AFC. In addition, such a methodology was shown to be very useful for comparing the aerodynamic performance of the AFC-enabled high-lift configurations with the conventional high-lift configurations at nominal landing conditions. The current work focuses on extending the simulations to higher angles of attack near maximum lift, and on comparisons with unsteady data acquired by Kulite[®] pressure transducers from the experimental study.

II. Configuration and Computational Setup

The geometries under consideration here correspond to variants of the 10%-scale, semispan, high-lift configurations that are based on the “open” NASA Common Research Model (CRM) [21, 22]. The conventional CRM-HL configuration considered here, and shown in Fig. 2 (a), is based on the geometry reported by Lacy and Sclafani [23]. It consists of a semispan wing, fuselage, nacelle/pylon, and slats and flaps, where the slats and flaps are attached to the main wing by slat and flap brackets, respectively. The gap and overlap settings are those specified in the initial design and obtained on the experimental model using fixed brackets. The AFC-enabled simplified high-lift configuration shares most of the components with the conventional CRM-HL configuration. However, the Fowler flap is replaced by a simple hinged flap geometry with embedded AFC actuators, and this AFC-enabled hinged flap geometry, shown in Fig. 2 (b), is also known as the AFC-enabled CRM-SHL or CRM-SHL-AFC configuration. Two rows of active flow control (AFC) devices, such as fluidic actuators and discrete blowing slots, are embedded in the flap shoulder to control flow separation on the highly deflected flap (50° inboard, and 55° outboard flap deflections), and to recover lift characteristics comparable to the conventional CRM-HL configuration. Interchangeable AFC cartridge chambers (4 on the inboard flap and 6 on the outboard flap) contain a set of AFC devices, and compressed air is fed to these chambers through separate feed tubes. Both of these configurations have been tested over a wide angle of attack range in the 14-by 22-Foot Subsonic Tunnel (14×22) at the NASA Langley Research Center (LaRC). For the CRM-SHL-AFC configuration, CFD simulations and wind tunnel tests have also been performed for several variants of actuator arrangements as documented in Refs. [19, 20, 24]. Based on these studies, the HELP (High Efficiency Low Power) configuration consisting of sweeping (unsteady) jets in the upstream row and discrete steady jets in the second row were found to be most effective and efficient for lift enhancement, and the performance of this configuration will be examined computationally at higher angles of attack in this paper.

The semispan of the wing is 115.675 inches, and the mean aerodynamic chord (MAC) in the stowed (cruise) configuration is 27.58 inches at a span station of 46.875 inches. The wing reference area (S) used for normalizing force coefficients is 2973.6 square inches. The computations are performed at a freestream Mach number of 0.2 and a Reynolds number of 3.27 million based on the MAC of the model, corresponding to the test conditions in the 14×22. The model is installed on top of a 3.5-inch-thick peniche (standoff), (see Fig. 2), and can be rotated to change the angle of attack (α) of the model relative to the incoming stream. The force balance does not include any force contribution from the peniche, which is a nonmetric component. Computations are performed with the model in the test section to facilitate a direct comparison with the experimental data acquired in the solid wall tunnel. Slip boundary conditions are imposed on the tunnel ceiling and side walls, and on the tunnel floor upstream of the peniche. Velocity and temperature are imposed on the tunnel inflow boundary. No-slip boundary conditions are applied on the peniche and all of the exposed solid surfaces of the configuration. Static pressure is specified at the wind tunnel outlet for the downstream boundary condition.

To simulate the flow in the actuators, total pressure and temperature inflow boundary conditions are imposed at the inlet faces of the feed tubes that are connected to the AFC cartridges, and the complete flow field developing inside the chambers and actuators is computed as an integral part of the overall solution for the entire configuration. Due to the bistable nature of the flow inside the fluidic actuators, unsteady oscillating jets

emanate from the actuator exit nozzles. For the HELP configuration, the flow also exits through an array of discrete blowing slots that are situated downstream of the fluidic actuators. The high-momentum flow exiting the actuator nozzles (and slots) energizes and interacts with the external flow over the flap surface, which results in reduced reversed flow regions and increased lift. The synergistic benefit of these exiting jets, which results from the two rows actuating together to produce a lift that is greater than the sum of each row actuating individually, was reported by Lin et al. [19].

III. Simulation Method

Numerical simulations presented in this paper have been performed using the PowerFLOW[®] code, which is a compressible flow solver based originally on the three-dimensional 19-state (D3Q19) Lattice Boltzmann Method (LBM). The PowerFLOW[®] code represents LBM-based CFD technology developed over the last 30 years [25–29], and has been extensively validated for a wide variety of applications ranging from academic direct numerical simulation (DNS) cases to industrial flow problems in the fields of aerodynamics [30] and aeroacoustics [31]. In contrast to methods based on the classical Navier-Stokes (N-S) equations, the LBM uses a simpler and more general physics formulation at the mesoscopic level [25]. As shown in Refs. [32, 33], the LBM equations recover the macroscopic hydrodynamics of the Navier-Stokes equations through the Chapman-Enskog expansion.

The PowerFLOW[®] code can be used to solve the Lattice Boltzmann equation in a DNS mode [34], where all of the turbulent scales are spatially and temporally resolved. However, the DNS mode is very compute intense, and because of compute resource considerations, this approach is not practical for Reynolds numbers above 10,000 or so. Therefore, for most engineering problems at higher Reynolds numbers ($>10,000$), the simulations are usually conducted in conjunction with a hybrid turbulence modeling approach where the smaller scales are modeled and larger, energy-containing scales are directly resolved. For the current work, the Lattice Boltzmann Very Large Eddy Simulation (LB-VLES) approach described in Refs. [27, 28, 35] is used. Because the flow inside the actuator nozzles is expected to choke and become supersonic for higher input pressure cases, all of the results presented here were simulated with the fully-compressible (transonic) version of the PowerFLOW[®] code [36–38] for consistency.

The PowerFLOW[®] code is used for solving the Lattice Boltzmann equation on embedded Cartesian meshes, which are generated within the PowerFLOW[®] Software Suite. Variable Resolution (VR) regions can be defined to allow for local mesh refinement of the grid by successive factors of two in each direction [25]. To reduce the resolution requirement near solid surfaces for higher Reynolds number flows, a hybrid wall function is used to model the near-wall region of the boundary layer [30, 39]. The PowerFLOW[®] code uses explicit time marching to solve the governing equations and scales efficiently on modern computer clusters consisting of thousands of processors, making it suitable for large-scale applications.

IV. Results

The computations of the three-dimensional (3-D) flow field for the conventional CRM-HL and the AFC-enabled CRM-SHL (CRM-SHL-AFC) configurations enclosed within the 14×22 test section have been performed using the transonic version of the PowerFLOW[®] code. The computational setup has the flexibility to turn the air supply on or off to any of the 10 chambers (4 on the inboard flap and 6 on the outboard flap), thus allowing us to examine the effect of flow control on either of the flaps as well as on selected portions of the flaps. This is easily accomplished by modifying the inflow boundary conditions on the feed tube inlets during the discretization and setup stage. Based on the work of Lin et al. [19], the CRM-SHL-AFC configuration was able to match the target lift of the CRM-HL at the higher angles of attack near the maximum lift condition only with constant spanwise NPR. Therefore, the simulations reported in this paper were performed with (nominally) constant NPR for all of the AFC actuators.

As mentioned in the previous section, the computations are performed on embedded Cartesian meshes, which are generated within the PowerFLOW[®] Software Suite. Finer resolution was specified for high-gradient regions, such as on the suction surface of the slats, main wing, and flaps. A total of 12 VR regions were used for the CRM-HL configuration. On the other hand, 13 VR regions were used for the CRM-SHL-AFC configuration, where the additional VR was added for improved resolution of the flow inside the actuator nozzle throats. The variation in grid density for lifting surfaces is illustrated in Fig. 3 with a planar cut of the grid at the span station of $y=27.75$ inches, where every other voxel is shown for improved clarity. The minimum spacing normal to viscous surfaces was chosen to produce an average value of 120 for y^+ , which has been found adequate for

resolving boundary layers with the LBM-VLES approach used in the PowerFLOW[®] code [40].

The finest VR was used for discretizing the internal domain in the vicinity of the actuator throat nozzles that are embedded in the flap shoulders of the CRM-SHL-AFC configurations. For these simulations, a minimum of 8×16 voxels (cells) were used at the throat, based on the experience gained during the grid sensitivity study conducted for a single actuator [41]. The voxels in the finest VR region have an edge length of 0.00567 inches. The grid resolution on major components and in the farfield is comparable to the resolution used for previous CRM-SHL-AFC simulations by Vatsa et al. [42]. In the current work, additional VR regions similar to the ones reported in Ref. [20] were also created to capture the vortical regions emanating from the slat edges, fuselage/wing and nacelle/pylon junctures, and the wing tip. The resulting grid consisted of approximately 436 million voxels (cells) for the CRM-HL configuration, and 583 million voxels for the CRM-SHL-AFC configuration.

Since the flow field is inherently unsteady, the computations are run long enough in time for the flow to achieve a quasisteady state, as determined by the variation of the integrated forces (e.g., lift) in time for a given case. Computations are typically run for a physical time of 0.5-0.6 seconds, which corresponds to the time required for traveling a distance of approximately 50 chord lengths. This allows us to extract time-averaged solutions for the integrated forces, surface pressures and streamlines, and to make quantitative comparisons among different configurations and flow conditions. A parametric study was conducted by Vatsa et al. [20], where the pressure ratio at the inlet feed tube was varied from 1.25 to 3.5. Examination of the flow inside the actuator nozzles indicated the presence of sonic flow for pressure ratios of 2.0 and higher. Simulations were also performed with the actuation turned off completely (AFC-OFF), to serve as a baseline for assessing the effectiveness of AFC for lift augmentation.

Unfortunately, the feed tube pressures required for boundary condition specification were not measured in the wind tunnel tests. Instead, pressure levels inside the AFC cartridge chambers were measured in the experimental work of Ref. [19], and these chamber pressures are lower than the feed tube pressures because of the expansion between the feed tubes and chambers. An iterative procedure was devised by Vatsa et al. [20] to vary the feed tube pressures to match the measured chamber pressures, and the feed tube pressures determined in this manner were used for boundary condition specification at the feed tube inlets. The lift coefficients resulting from computations performed in this manner for the nonactuated (AFC-OFF, NPR=1.0) and actuated (NPR=1.46, 2.0 and 2.41) cases are compared with the experimental data in Figs. 4 and 5 as a function of chamber pressure (NPR) and mass flow rate ($\dot{m}$), respectively, for the nominal landing approach configuration at 8° angle of attack. The lift coefficient corresponding to the conventional high-lift configuration based on the experimental data is also plotted in these figures as a green dotted line to serve as the target lift for the CRM-SHL-AFC configuration. It is worth mentioning that the predicted lift levels shown in Ref. [20] were higher compared to the ones presented here for the same conditions. This was caused by including the force contribution of the peniche in the CFD results, not realizing that the force balance used for the experimental study was not connected to the peniche. The CFD results presented in this paper have excluded contribution of forces from the peniche to provide a one-to-one comparison with the experimental data. The computed values of lift shown in Fig. 4 are in excellent agreement with the measured data, the predicted values being only slightly higher than the experimental data for the actuated cases at a given value of NPR. Similar agreement is seen when the lift coefficient is plotted as a function of mass flow rate (Fig. 5), except for a slight underprediction at NPR=1.46. Nonetheless, the overall agreement of the predicted lift levels is considered quite good, given various uncertainties incurred during the simulation of such a complex configuration in a wind tunnel environment. A similar trend is observed for the increment in lift with respect to the AFC-OFF values (ΔC_L) for these cases, as shown in Fig. 6. It is clear from these results that AFC with the current HELP actuator arrangement not only achieves the goal of matching the target lift of the CRM-HL configuration at $\alpha = 8^\circ$, the nominal landing approach angle of attack, but actually exceeds it for $\text{NPR} \geq 2.0$.

A. Angle of attack variation

Based on the results presented so far, and the ones that were reported recently by Vatsa et al. [20], the PowerFLOW[®] -code-based CFD methodology has been successful in accurately predicting the trends in integrated forces and detailed pressure distributions for the CRM high-lift configurations at the nominal landing approach condition of $\alpha = 8^\circ$. An attempt is made here to assess the accuracy of the PowerFLOW[®] code for predicting lift for such configurations at higher angles of attack, being fully aware that predicting the flow near maximum lift is a very challenging task for most CFD methods [43].

Computations of the conventional CRM-HL configuration [19] were performed over a wide range of angle of attack that encompasses the maximum lift condition. The time-averaged lift based on these computations is

compared with the experimental data in Fig. 7. Agreement of predicted lift with the measured data is quite good up to about $\alpha = 12^\circ$. Small, but perceivable differences start appearing beyond this angle of attack. Nonetheless, the PowerFLOW[®] code is able to predict the correct qualitative trends over a broad angle of attack range. For example, the CFD results replicate the dip in the lift curve slope near $\alpha = 15^\circ$, and the maximum lift is predicted between $\alpha = 16^\circ$ - 17° . Similar comparisons for the drag coefficient are shown in Fig. 8 for the same angle of attack range. Once again, the agreement of drag values with the measured data is quite good for the complete angle of attack range, with the largest differences being seen in the vicinity of maximum lift conditions.

The next set of results presented in Fig. 9 show the variation in integrated lift for the CRM-SHL-AFC configuration. The CFD results are compared with the integrated lift from the experimental data of Ref. [19] for the AFC-OFF condition, and with the AFC at NPR of 1.46, 2.0 and 2.41 over a broad angle of attack range. Excellent agreement is observed between predicted and measured lift up to $\alpha = 12^\circ$ for the actuated cases, and up to $\alpha = 18^\circ$ for the nonactuated (AFC-OFF) case corresponding to the maximum lift condition. In general, the CFD results predict the correct trends over the entire angle of attack range considered here, although the CFD slightly underpredicts the maximum lift levels for the actuated cases. The experimental results for the conventional CRM-HL configuration are also shown in this figure to serve as a target for reference. It is clear from this figure that the lift produced by the CRM-SHL-AFC configuration matches or exceeds the target levels over the entire angle of attack range for $\text{NPR} \geq 2.0$.

B. Sectional pressure comparisons

Time-averaged surface pressure coefficient (C_p) distributions were extracted at the sectional cuts of the model corresponding to the spanwise rows shown in Fig. 10 (a), with the flaps and slats being in the stowed position. The experimental data were acquired at a discreet number of pressure taps placed along these rows (see Fig. 10 (b)). The computed values of C_p are compared with the measured data at the nominal landing approach angle of attack, $\alpha = 8^\circ$, and near the maximum lift condition at $\alpha = 16^\circ$. These comparisons are presented in Figs. 11 and 12 at the inboard ($y=27.75$ inches, $\eta = 0.240$) and outboard ($y=63.8$ inches, $\eta = 0.552$) midflap span stations, respectively, for different NPR levels, including the AFC-OFF case. In these figures, the results for the slat were shifted upstream by approximately 1.2 inches to avoid overlap between the slat and main wing leading edge regions.

The computational results are in good agreement with the measured data at both the inboard and outboard midflap stations, and the effect of increasing the angle of attack and actuation levels (NPR) is well predicted over most of the slat, wing, and flap surfaces. Whereas the lower surface of the wing remains relatively insensitive to flow control, the suction pressure increases on the upper surface with increasing values of NPR. The most significant increase in suction pressure is observed in the vicinity of the slat and main wing leading edges. Increasing the angle of attack from 8° to 16° results in much higher values of suction pressure on the upper surface, especially in the vicinity of the slat and wing leading edges, resulting in higher lift. These effects are captured very accurately by the simulations, as seen in Figs. 11 - 12. Other than slight overpredictions of the slat leading edge suction peaks, the computed pressure distributions agree quite well with the measured data on the slat and wing surfaces.

The computational results indicate sharp suction peaks at the flap leading edge for AFC cases at $\alpha = 16^\circ$. Unfortunately, there are not enough pressure taps on the model to capture such peaks in the experiment. The CFD also predicts the correct trends for the pressure recovery in the vicinity of the flap trailing edge with increases in AFC input pressures. The main difference between the inboard and the outboard span stations is the presence of even sharper peaks at the slat leading edge for higher NPR levels at the outboard ($y=63.8$ inches) midflap station. The slight discontinuity in C_p for the $y=63.8$ inch station (Figs 12) at the wing/flap interface is due to the misalignment of spanwise rows used for extraction of surface pressure for the main wing and the outboard flap (see Fig. 10 (b)).

It is interesting to note that the effect of AFC can be observed even at further outboard span sections, which do not contain any flaps or AFC actuators. This is apparent in C_p distributions presented in Fig. 13 for $y=94.74$ inches ($\eta = 0.819$), where one can clearly see the increase in the suction peaks on the slat and wing leading edges with increasing levels of NPR, even though this station does not contain any actuators. Apparently, such an increase in suction peaks at the outboard stations is caused by an increase in the overall circulation levels in the presence of AFC. The computed C_p distributions shown in Fig. 13 accurately track the experimentally observed trends, and are found to be in excellent agreement with the experimental data, including the pressure recovery at the flap trailing edge.

C. Flow visualization

The results presented so far are based primarily on the surface pressure distributions, and one needs to look at the surface, and off-surface flow characteristics to fully appreciate the effect of AFC on the resulting flow field. Previous studies [42, 44] indicated that flow behavior on the lower (pressure) side of the lifting surfaces is relatively insensitive to AFC actuation levels. Therefore, attention will be focused here on examination of the flow characteristics over the upper (suction) surfaces. The suction surface streamlines and skin friction contours based on time-averaged solutions are included in this section for $\alpha = 8^\circ$ and 16° to assess the affect of actuation levels on separation control at moderate and high angles of attack. The results based on the nonactuated (AFC-OFF) and AFC (NPR=2.41) simulations are compared in Fig. 14 for $\alpha = 8^\circ$, and in Fig. 15 for $\alpha = 16^\circ$. It appears that at $\alpha = 8^\circ$, the flow is attached over most of the slat and main wing surfaces even for the AFC-OFF case, where the wing leading edge shows the maximum skin friction levels. The most prominent effect of AFC is on the flaps, as seen in Fig. 14(b). Whereas massive separation is observed over the entire suction surface of the flaps in the absence of AFC (AFC-OFF), a significant reduction is observed in the extent of the reversed flow region when AFC is applied. Small pockets of reversed flow and low skin friction levels are still present between the chamber boundaries and at the Yehudi break between the inboard and outboard flaps. This is mainly due to the absence of actuators in those regions. As expected, the AFC has minimal effect on the flow characteristics for the outboard section of the wing beyond the outboard flap, since there are no actuators in that region.

The surface streamline patterns and skin friction contours for the $\alpha = 16^\circ$ case are presented in Fig. 15. The primary effect of increased angle of attack seems to be the presence of more extensive reversed flow regions on the entire flap, and the outboard wing surface for the AFC-OFF case. In addition, a pocket of low skin friction appears on the wing, downstream of the inboard nacelle juncture. Using AFC with an NPR=2.41 results in significant reduction in the reversed flow regions on the flaps. In addition, the AFC reduces the extent of separated flow upstream of the flaps, as well as in the outboard portion of the wing. These results clearly demonstrate the effectiveness of flow control to mitigate flow separation even at high angles of attack, and to improve the lift characteristics, as was observed in Fig. 9.

A more complete perspective of the overall global flow field emerges by examination of the 3-D streamlines to supplement the surface flow patterns that were presented in Figs. 14–15. Such images are especially helpful in visualizing the vortices generated in the juncture regions. Simulated 3-D flow patterns at $\alpha = 8^\circ$ are compared in Figs. 16 (a)–(b) for nonactuated (AFC-OFF) and AFC (NPR=2.41) cases, respectively. Identical seeding was used to generate the streamlines shown in these figures. The streamline patterns shown in Fig. 16 (a) indicate massively separated flow on both the inboard and the outboard flaps in the absence of actuation. These streamlines also indicate the presence of vortices generated by the wing/fuselage and nacelle/pylon junctures that travel in the spanwise direction toward the wing tip. A significant reduction in the reversed flow region, and weakening of these vortices on the inboard flap are observed for the NPR=2.41 (Fig. 16 (b)) simulations. For example, the spanwise vortices from the wing root disappear completely, and flow on the inboard flap surface becomes attached. The outboard flap also appears to be mostly attached except in the vicinity of the Yehudi break, as seen in Fig. 16 (b). Small pockets of off-body separation are also seen near the outboard flap trailing edge, even in the presence of AFC; however, these separation regions are much smaller compared to the AFC-OFF case.

Similar 3-D off-surface streamline patterns at $\alpha = 16^\circ$ are compared in Figs. 17 (a)–(b) for AFC-OFF and AFC (NPR=2.41) simulations, respectively. In the absence of actuation, massive flow separation and strong corner vortices are evident at this higher angle of attack. The spanwise vortex on the inboard flap travels further downstream compared to the $\alpha = 8^\circ$ case. The AFC case (NPR=2.41) successfully reduces the extent of reversed flow to a large degree. Compared to the lower angle of attack, a small pocket of reversed flow is observed near the inboard flap trailing edge. Along with the results presented earlier in Figs. 14–15, the 3-D streamline patterns of Figs. 16–17 reveal the overall flow characteristics associated with high-lift configurations, and help demonstrate the effectiveness of AFC for reducing the extent of accompanying reversed flow regions for such high-lift configurations.

D. Unsteady data comparison

A total of 12 Kulite[®] unsteady pressure transducers were placed on the upper inboard and outboard flap surfaces, as indicated by the blue rectangles in Fig. 18. As seen in this figure, half of the transducers were placed just downstream of the flap shoulders, whereas the others were placed just upstream of the trailing edges. Numerical probes were placed at these locations in the CFD simulations to collect unsteady pressure

data. Power spectral density (PSD) distributions at the midspan of the inboard flap shoulder and at the trailing edge Kulite® transducer locations are compared in Figs. 19 (a) and (b), respectively, for the actuated case at NPR=2.41, and 8° angle of attack. Notice that the PSD levels show rapid decay (roll off) with frequency in the 0.1 - 10 kHz. range for the trailing edge Kulite® transducers (Fig. 19 (a)). On the other hand, the PSD distributions show a much more gradual decay with frequency for the Kulite® transducers at the flap shoulder (Fig. 19 (b)), and indicate a slight increase in PSD levels between the frequencies of 5-10 kHz. A tonal frequency at 1.1 kHz. is seen in both the experimental data, and the CFD predictions for PSD in Fig. 19 (b), which corresponds to the oscillation frequency of the fluidic actuators placed along the flap shoulder. The Kulite® transducers used in this study are only deemed accurate up to a frequency of 10 kHz., which is the maximum frequency used in these figures. The CFD results replicate the correct qualitative trends, and the overall comparison between the CFD predictions and the experimental data is quite good. Similar agreement is observed for the PSD at the outboard midspan locations, as seen in Figs. 20 (a) and (b). Because the fluidic actuators used on the outboard flap are smaller in size compared to the ones on the inboard flap, their oscillating frequency is higher (1.83 kHz.), as seen in Fig. 20 (b). Although not shown here, similar agreement has been observed for the lower NPR levels.

V. Concluding Remarks

Computational results simulated with the fully-compressible, transonic version of a Lattice-Boltzmann-based PowerFLOW® CFD code are presented here for 10% scale models of the conventional high-lift (CRM-HL) and the simplified high-lift (CRM-SHL-AFC) configurations. The latter configuration implemented the HELP actuation concept that consists of two rows of actuators (unsteady sweeping jets for row 1, and steady discrete jets for row 2) embedded in the flaps for active flow control. The entire flow field, including the flow inside the actuators, is simulated in the current approach.

The initial focus of this work was to examine the effect of varying the AFC feed tube pressure (NPR) levels on lift enhancement of the CRM-SHL-AFC configuration at the nominal landing approach angle of 8°. Excellent agreement with the experimental data was obtained at this angle of attack for the lift coefficient as a function of NPR and mass flow rate ($\dot{m}$).

Simulations were also performed to assess the accuracy of the current numerical scheme for predicting the flow characteristics over these configurations at higher angles of attack, including the maximum lift condition. Excellent agreement with the measured data was obtained for the lift and drag coefficients for the CRM-HL configuration for α varying from 4° to 19°. The computed results accurately replicated the experimentally observed features, such as a drop in the lift curve slope near $\alpha = 15^\circ$, and maximum lift near $\alpha = 18^\circ$.

Next we examined the effect of AFC on lift enhancement for the CRM-SHL-AFC configuration with angle of attack variation. Other than a slight underprediction of the maximum lift, the CFD results were found to be in good agreement with the experimental data, especially in the linear range of the lift curve slope, i.e., for $\alpha \leq 12^\circ$. The computed sectional pressure distributions for selected cases were also shown to agree well with experimental data at 8° and 16° angles of attack. In addition, time-averaged surface skin friction contours and surface streamline patterns, along with 3-D (off-surface) streamlines based on simulations were also presented to understand the effect of AFC and angle of attack variation on the flow structure. These results clearly demonstrate the beneficial effect of AFC on separation control and lift enhancement, and add to our understanding of the flow physics for high-lift configurations.

Finally, the power spectral density distributions obtained from the current simulations were compared with the Kulite® pressure transducer data to assess the accuracy of the current methodology for resolving unsteady features of the flow field. The CFD results were shown to replicate the experimental data accurately over a broad frequency range. In summary, the CFD results presented in this paper were found to be in good agreement with the experimental data, and the current CFD methodology is shown to be a very useful tool for conducting parametric studies for such complex high-lift configurations consisting of embedded AFC actuators.

Acknowledgments

This work was supported by the NASA Advanced Air Vehicles Program through the Advanced Air Transport Technology Project. The authors would like to thank Judith A. Hannon, Marlyn Y. Andino, Mehti Koklu, and Keith B. Paschal who supported the 14x22 wind tunnel test, as well as provided data reduction and analysis from the test.

References

- [1] Van Dam, C., “The Aerodynamic Design of Multi-Element High-Lift Systems for Transport Airplanes,” *Progress in Aerospace Sciences*, Vol. 38, 2002, pp. 101–144.
- [2] Loftin, L., “Quest for Performance - The Evolution of Modern Aircraft,” NASA SP-468, <https://www.hq.nasa.gov/pao/History/SP-468/ch10-5.htm>, 1985.
- [3] Greenblatt, D., and Wygnanski, I., “The Control of Flow Separation by Periodic Excitation,” *Progress in Aerospace Sciences*, Vol. 36, 2000, pp. 487–545.
- [4] Cattafesta III, L., and Sheplak, M., “Actuators for Active Flow Control,” *Annual Review of Fluid Mechanics*, Vol. 43, 2010, pp. 247–272.
- [5] McClean, J., Crouch, J., Stoner, R., Sakrai, G., Feifel, W., and Rush, H., “Study of the Application of Separation Control by Unsteady Excitation to Civil Transport Aircraft,” NASA CR-1999-209338, 1999.
- [6] Hartwich, P., Shmilovich, A., Lacy, D., Dickey, E., Sclafani, A., Sundaram, P., and Yadlin, Y., “Refined AFC-Enabled High-Lift System Integration Study,” NASA CR-2016-219170, 2016.
- [7] Hartwich, P., Camacho, P., El-Gohary, K., Gonzales, A., Lawson, E., and Shmilovich, A., “System-Level Trade Studies for Transonic Transports with Active Flow Control (AFC) Enhanced High-Lift Systems,” AIAA Paper 2017-0321, January 2017.
- [8] DeSalvo, M., Whalen, E., and Glezer, A., “Enhancement of a High-Lift Airfoil using Low-Power Fluidic Actuators,” AIAA Paper 2010-0863, June 2010.
- [9] DeSalvo, M., Whalen, E., and Glezer, A., “High-Lift Enhancement using ActiveFlow Control,” AIAA Paper 2011-3355, June 2011.
- [10] Woszidlo, R., and Wygnanski, I., “Parameters Governing Separation Control with Sweeping Jet Actuators,” AIAA Paper 2011-3172, June 2011.
- [11] Vatsa, V., Koklu, M., Wygnanski, I., and Fares, E., “Numerical Simulation of Fluidic Actuators for Flow Control Applications,” AIAA Paper 2012-3239, June 2011.
- [12] Seele, R., Graff, E., Gharib, M., Taubert, L., Lin, J., and Wygnanski, I., “Improving Rudder Effectiveness with Sweeping Jet Actuators,” AIAA Paper 2012-3244, June 2012.
- [13] Seele, R., Graff, E., Lin, J., and Wygnanski, I., “Performance Enhancement of a Vertical Tail Model with Sweeping Jet Actuators,” AIAA Paper 2013-0411, January 2013.
- [14] Graff, E., Seele, R., Lin, J., and Wygnanski, I., “Sweeping Jet Actuators - A New Design Tool for High Lift Generation,” NATO Workshop on Innovative Control Effectors for Military Vehicles (AVT-215), 2013. Stockholm, Sweden, May 20–22.
- [15] Gregory, J., and Tomac, M., “A Review of Fluidic Oscillator Development and Application for Control,” AIAA Paper 2013-2474, June 2013.
- [16] Melton, L., and Koklu, M., “Active Flow Control Using Sweeping Jet Actuators on a Semi-Span Wing Model,” AIAA Paper 2016-1872, January 2016.
- [17] Melton, L., Koklu, M., Andino, Y., M., Lin, J., and Edelman, L., “Sweeping Jet Optimization Studies,” AIAA Paper 2016-4233, June 2016.
- [18] Jones, G., Milholen, W., Chan, D., and Goodliff, S., “A Sweeping Jet Application on a High Reynolds Number Semi-Span Supercritical Wing Configuration,” AIAA Paper 2017-3044, June 2017.
- [19] Lin, J., Melton, L., Hannon, J., Andino, K. M., M.Y., Paschal, K., and Vatsa, V., “Wind Tunnel Testing of Active Flow Control on High Lift Common Research Model,” AIAA Paper 2019-3723, June 2019.
- [20] Vatsa, V., Duda, B., Lin, J., Melton, L., Lockard, D., O’Connell, M., and Hannon, J., “Comparative Study of Active Flow Control Strategies for Lift Enhancement of a Simplified High-Lift Configuration,” AIAA Paper 2019-3724, June 2019.
- [21] Vassberg, J., DeHaan, M., Rivers, S., and Wahls, R., “Development of a Common Research Model for Applied CFD Validation Studies,” AIAA Paper 2008-6919, August 2008.

- [22] NASA Common Research Model, URL: <http://commonresearchmodel.larc.nasa.gov>, last accessed, July 2019.
- [23] Lacy, D., and Sclafani, A., “Development of the Common Research Model (HL-CRM): A Representative High Lift Configuration for Transonic Transports,” AIAA Paper 2016-0308, January 2016.
- [24] Melton, L., Lin, J., Hannon, J., Koklu, M., Andino, M., and Paschal, K., “Sweeping Jet Flow Control on the Simplified High Lift Version of the Common Research Model,” AIAA Paper 2019-3726, June 2019.
- [25] Chen, H., “Volumetric Formulation of the Lattice Boltzmann Method for Fluid Dynamics: Basic Concept,” *Physical Review A*, Vol. 58, 1998, pp. 3955–3963.
- [26] Chen, H., Teixeira, C., and Molvig, K., “Realization of Fluid Boundary Conditions via Discrete Boltzmann Dynamics,” *Intl. J. Mod. Phys. C*, Vol. 9, No. 8, 1998, pp. 1281–1292.
- [27] Yakhot, V., and Orszag, S., “Renormalization Group Analysis of Turbulence. I. Basic Theory,” *J. Sci. Comput.*, Vol. 1, No. 2, 1986, pp. 3–51.
- [28] Chen, H., Kandasamy, S., Orszag, S., Shock, R., Succi, S., and Yakhot, V., “Extended Boltzmann Kinetic Equation for Turbulent Flows,” *Science*, Vol. 301, No. 5633, 2003, pp. 633–636.
- [29] Chen, S., and Doolen, G., “Lattice Boltzmann Method for Fluid Flows,” *Ann. Rev. Fluid Mech.*, Vol. 30, 1998, pp. 329–364.
- [30] Fares, E., and Nölting, S., “Unsteady Flow Simulation of a High-Lift Configuration using a Lattice Boltzmann Approach,” AIAA Paper 2011-869, January 2011.
- [31] Khorrami, M., Fares, E., and Casalino, D., “Towards Full-Aircraft Airframe Noise Prediction: Lattice-Boltzmann Simulations,” AIAA Paper 2014-2481, June 2014.
- [32] Chen, H., Chen, S., and Matthaeus, W., “Recovery of the Navier-Stokes Equations Using a Lattice-gas Boltzmann Method,” *Physical Review A*, Vol. 45, 1992, pp. 5339–5342.
- [33] Qiana, Y. H., D’Humiéres, D., and Lallemand, P., “Lattice BGK Models for Navier-Stokes Equations,” *Europhysics Letters*, Vol. 17, 1992, pp. 479–484.
- [34] Li, Y., Shock, R., and Chen, H., “Numerical Study of Flow Past an Impulsively Started Cylinder by Lattice Boltzmann Method,” *J. Fluid Mech.*, Vol. 519, 2004, pp. 273–300.
- [35] Chen, H., Orszag, S., Staroselsky, I., and Succi, S., “Expanded Analogy between Boltzmann Kinetic Theory of Fluid and Turbulence,” *J. Fluid Mech.*, Vol. 519, 2004, pp. 301–314.
- [36] Shan, X., Yu, X.-F., and Chen, H., “Kinetic Theory Representation of Hydrodynamics: a Way Beyond the Navier-Stokes Equation,” *Physics. Rev. Lett.*, Vol. 80, 1998, pp. 65–88.
- [37] Zhuo, C., Zhong, C., Li, K., Xiong, S., Chen, X., and Cao, J., “Application of Lattice Boltzmann Method to Simulation of Compressible Turbulent Flow,” *Commun. Comput. Physics*, Vol. 8, 2010, pp. 1208–1223.
- [38] Fares, E., Wessels, M., Li, Y., Gopalakrishnan, P., Zhang, R., Sun, C., Gopalaswamy, N., Roberts, P., Hoch, J., and Chen, H., “Validation of a Lattice Boltzmann Approach for Transonic and Supersonic Simulations,” AIAA Paper 2014-0952, January 2014.
- [39] Fares, E., “Unsteady Flow Simulation of the Ahmed Reference Body using a Lattice Boltzmann Approach,” *Comput. Fluids*, Vol. 35, No. 8-9, 2006, pp. 940–950.
- [40] Fares, E., Duda, B., Ribeiro, A., and König, B., “Scale-resolving simulations using lattice Boltzmann-based approach,” *CEAS Aeronautical Journal*, Vol. 9, 2018, pp. 721–733. <http://dx.doi.org/10.1007/s13272-018-0317-0>.
- [41] Duda, B., Fares, E., Wessels, M., and Vatsa, V., “Unsteady Flow Simulation of a Sweeping Jet Actuator Using a Lattice-Boltzmann Method,” AIAA Paper 2016-1818, January 2016.
- [42] Vatsa, V., Duda, B., Lin, J., Melton, L., and O’Connell, M., “Numerical Simulation of a Simplified High-Lift CRM Configuration Embedded with Fluidic Actuators,” AIAA Paper 2018-3063, June 2018.
- [43] Rumsey, C., Slotnick, J., and Sclafani, A., “Overview and Summary of the Third AIAA High Lift Prediction Workshop,” *Journal of Aircraft*, Vol. 56, No. 2, 2019, pp. 621–644.
- [44] Vatsa, V., Casalino, D., Lin, J., and Appelbaum, J., “Numerical Simulation of a High-Lift Configuration with Embedded Fluidic Actuators,” AIAA Paper 2014-2142, June 2014.

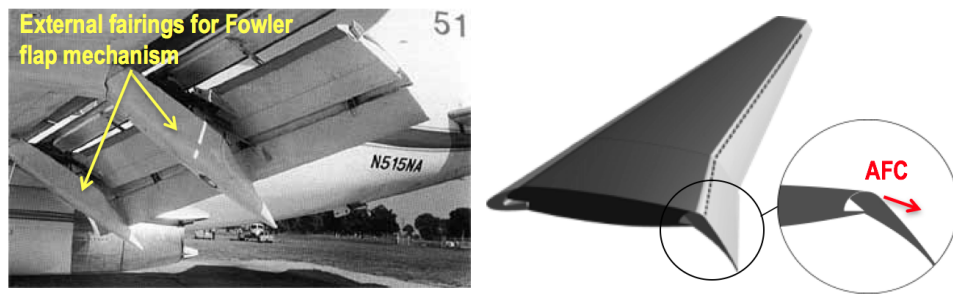
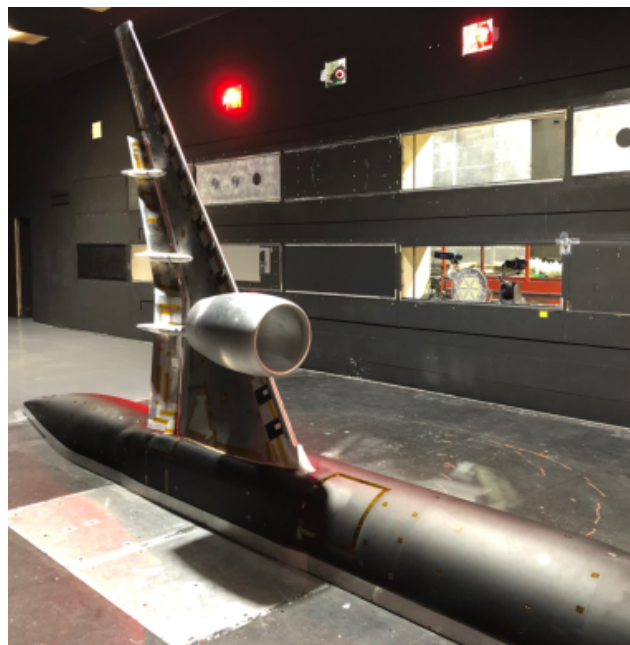


Fig. 1 Schematic of Fowler flap and simple hinged flaps.



(a) conventional CRM-HL



(b) AFC-enabled CRM-SHL

Fig. 2 Global view of conventional and simplified high-lift CRM configurations.

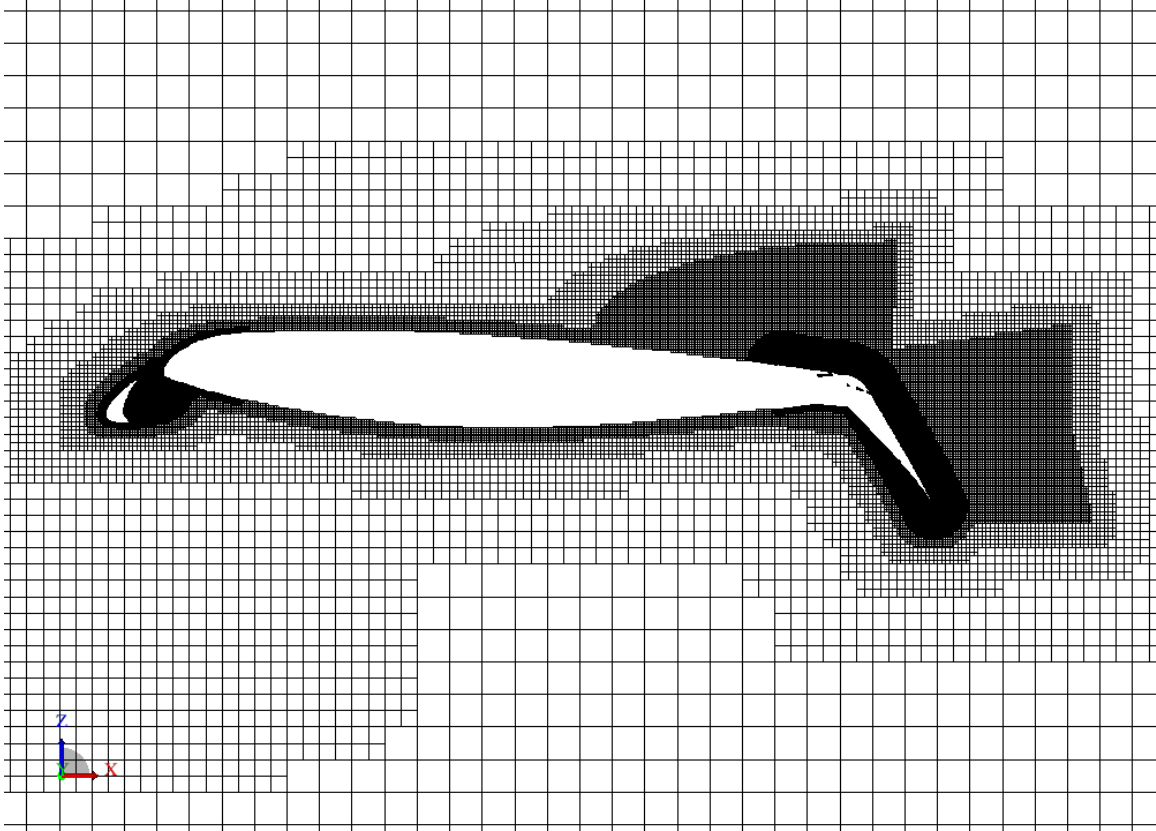


Fig. 3 Grid density variation at a planar section, $y=27.75$ in.

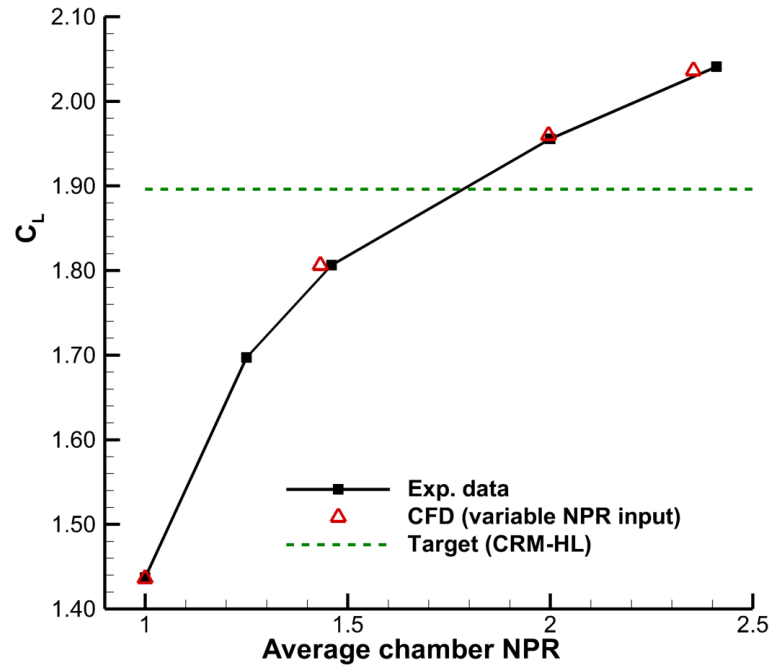


Fig. 4 Lift coefficient variation with NPR for variable input pressures, $Mach=0.2$, $Re=3.27 \times 10^6$, $\alpha = 8^\circ$.

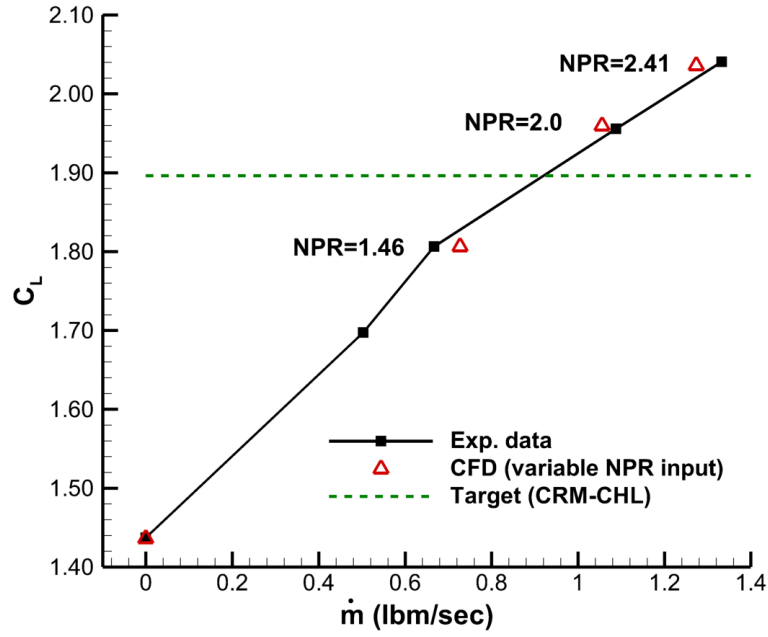


Fig. 5 Lift coefficient variation with input mass flow rate for variable input pressure, Mach=0.2, $Re=3.27 \times 10^6$, $\alpha = 8^\circ$.

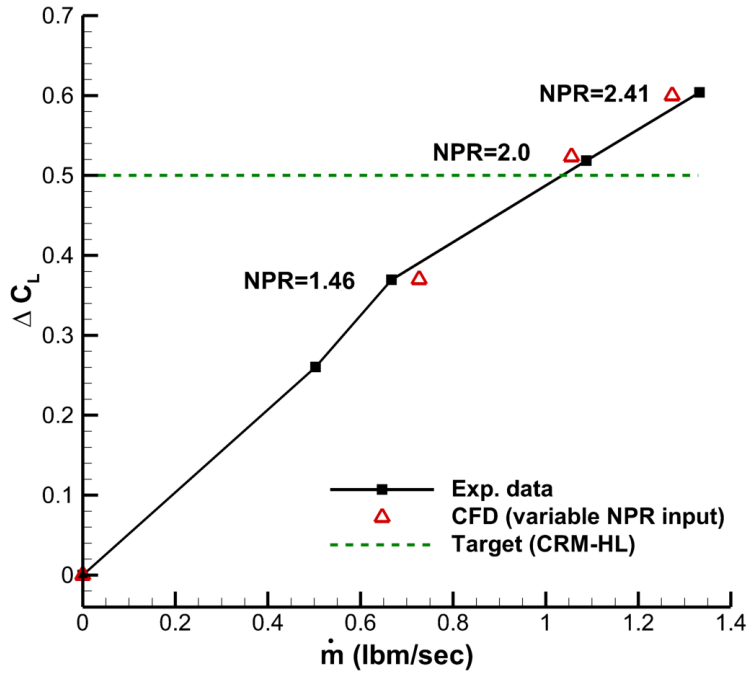


Fig. 6 Comparison of incremental lift coefficient variation with input mass flow rate, Mach=0.2, $Re=3.27 \times 10^6$, $\alpha = 8^\circ$.

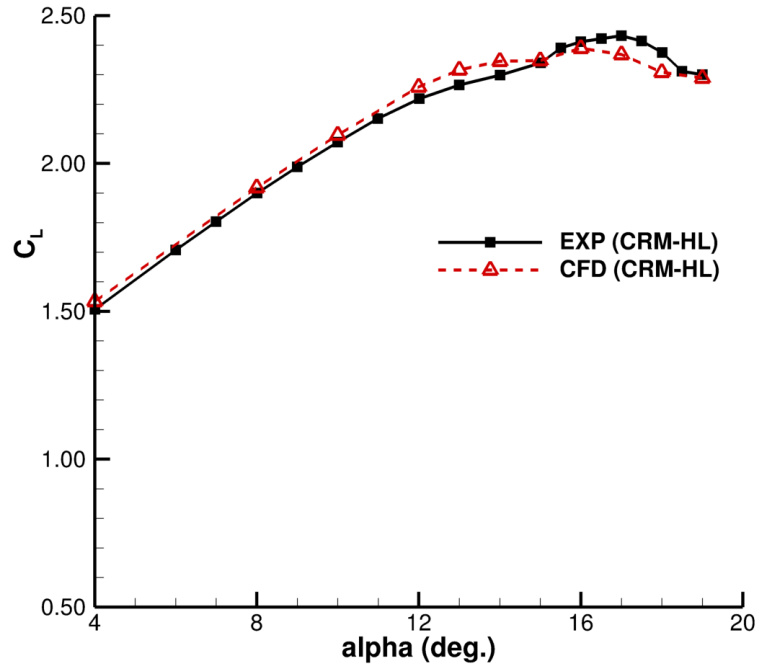


Fig. 7 Lift coefficient variation with angle of attack for conventional CRM-HL, Mach=0.2, $Re=3.27 \times 10^6$.

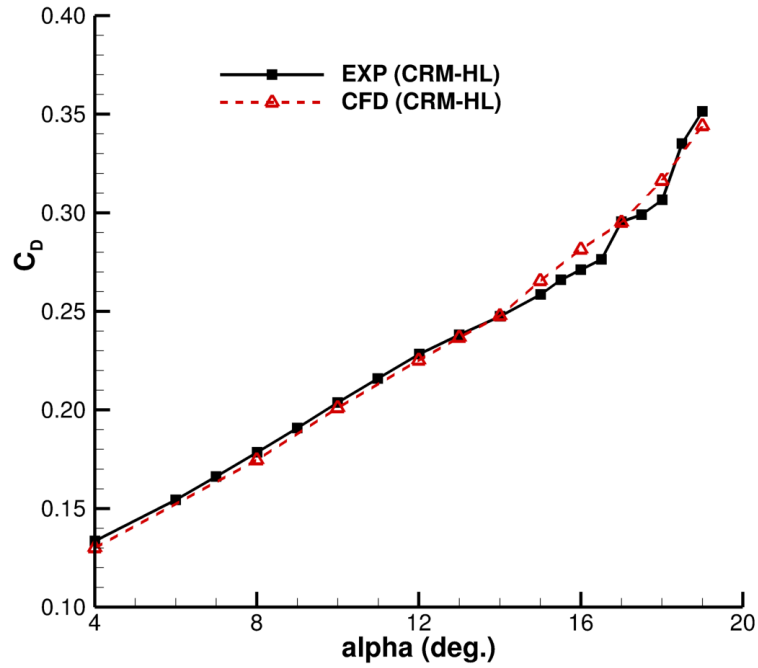


Fig. 8 Drag coefficient variation with angle of attack for conventional CRM-HL, Mach=0.2, $Re=3.27 \times 10^6$.

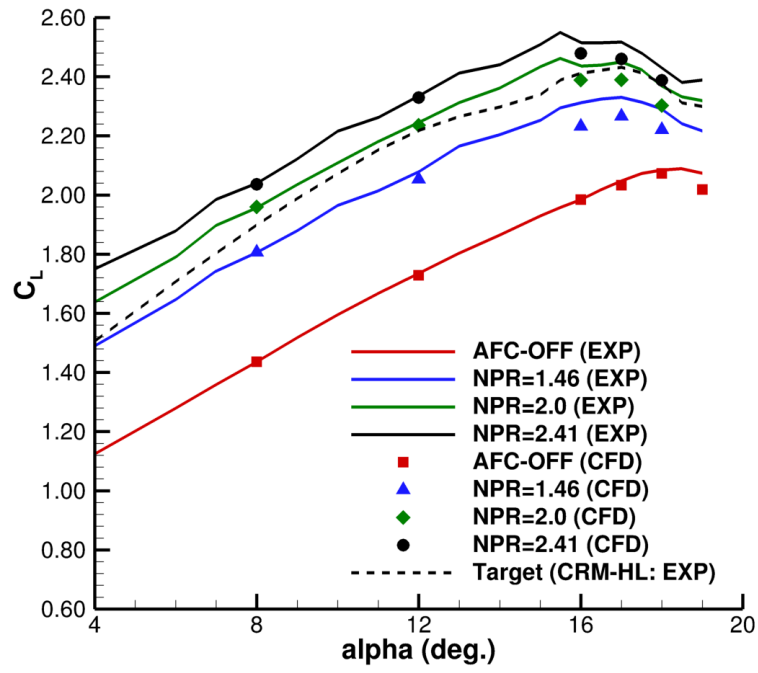
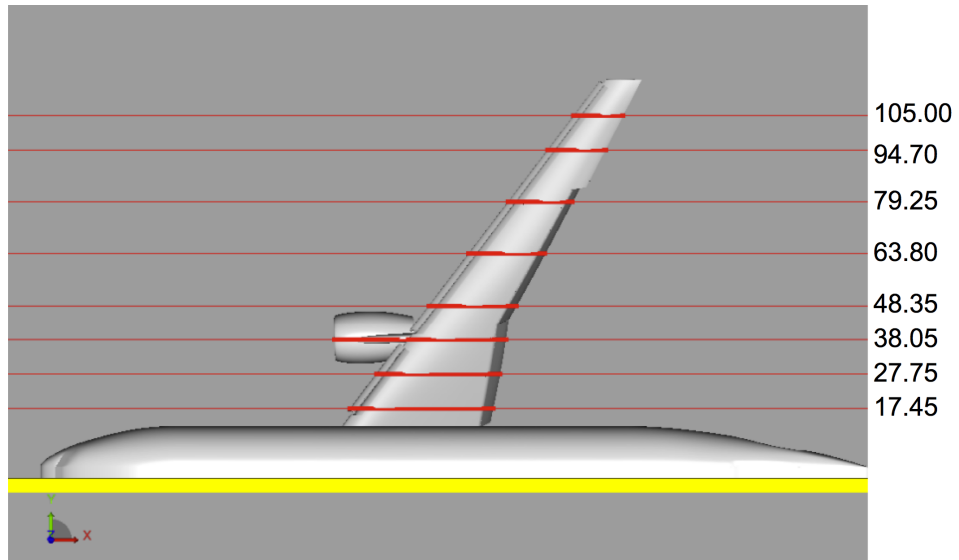
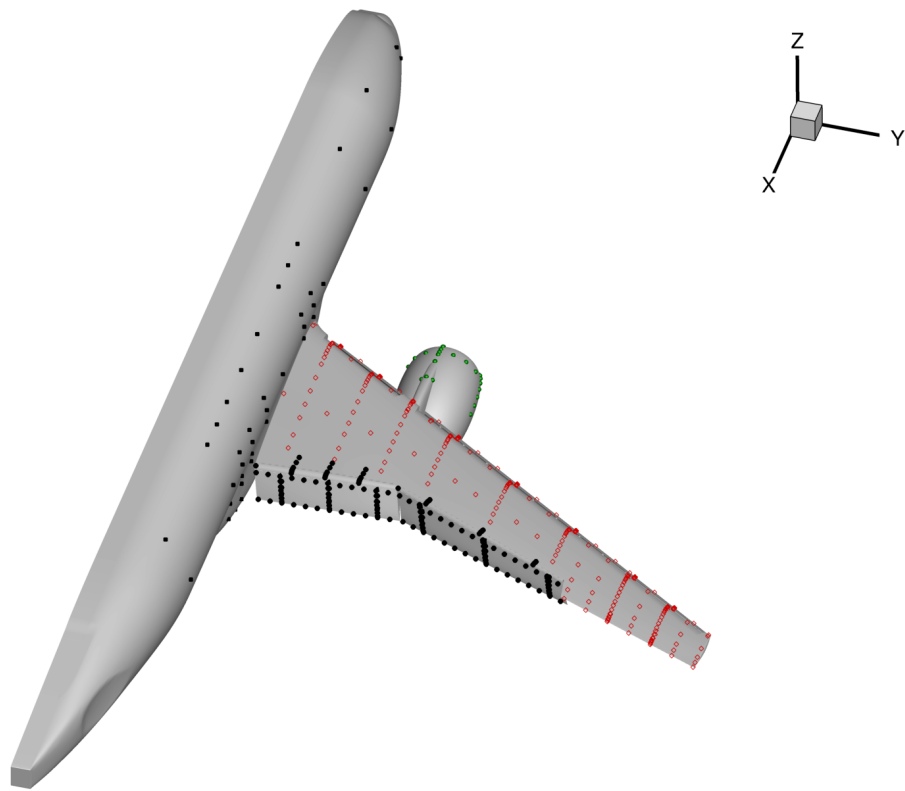


Fig. 9 Lift coefficient variation with angle of attack for AFC-enabled CRM-SHL, Mach=0.2, $Re=3.27 \times 10^6$.

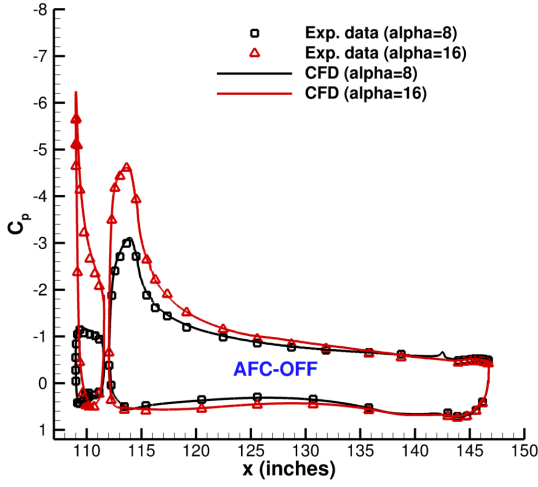


(a) Spanwise cuts

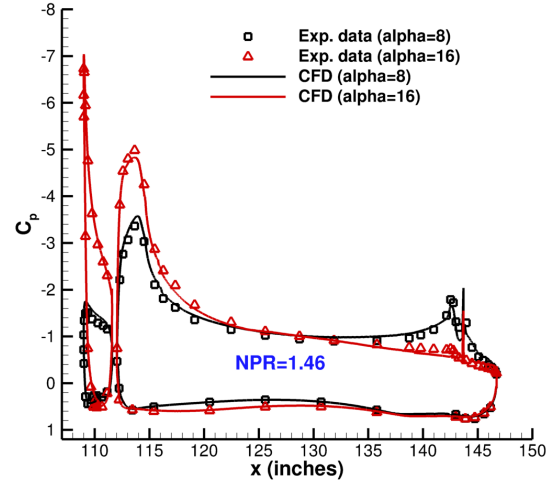


(b) Pressure ports

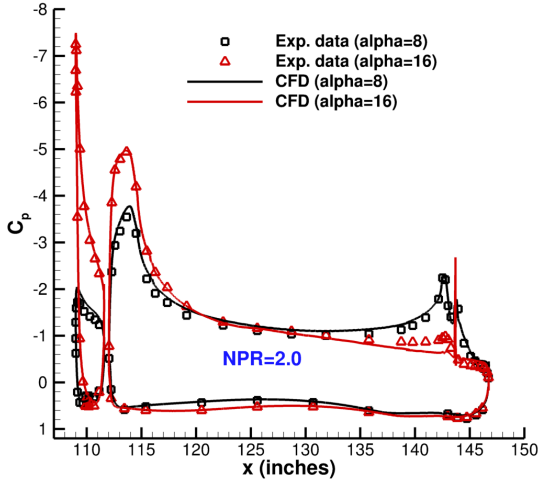
Fig. 10 Schematic of spanwise cuts and static pressure port locations.



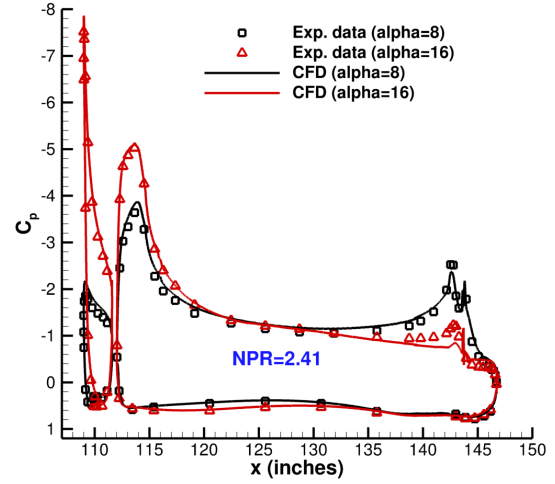
(a) AFC-OFF



(b) NPR=1.46

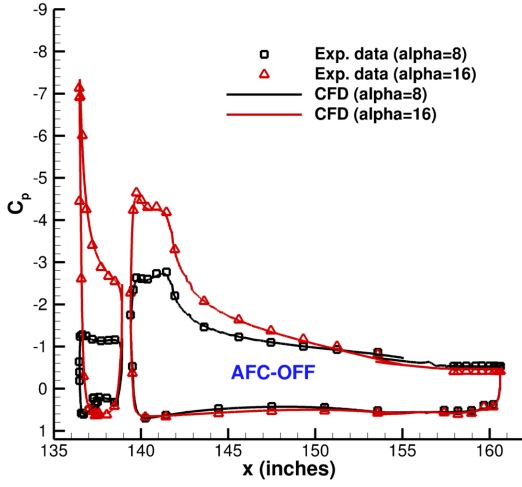


(c) NPR=2.0

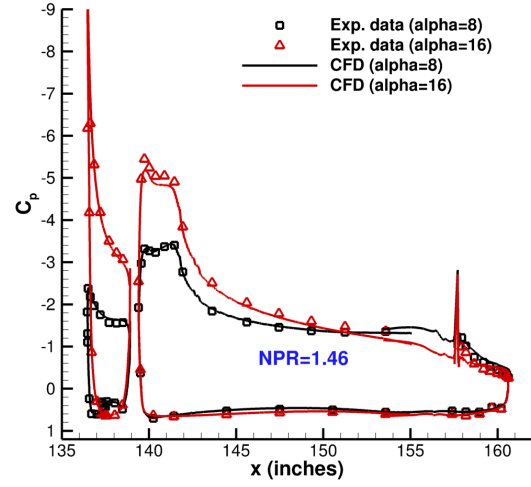


(d) NPR=2.41

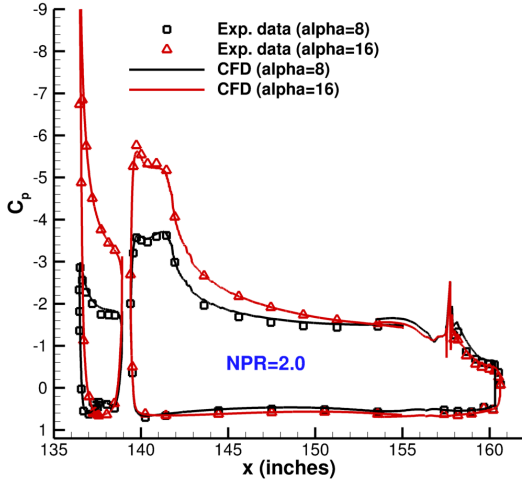
Fig. 11 Effect of pressure variation on sectional C_p at $y=27.75$ inches ($\eta = 0.240$)
Mach=0.2, $Re=3.27 \times 10^6$.



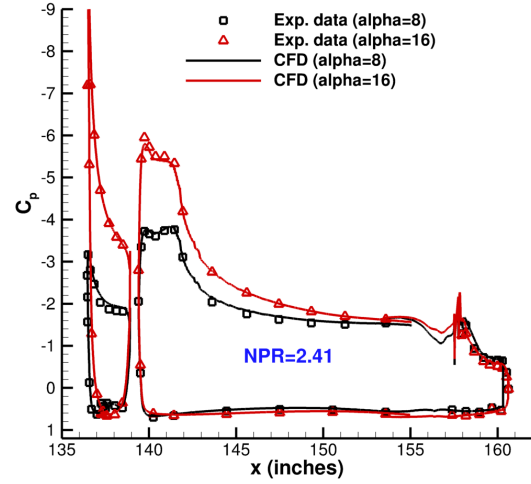
(a) AFC-OFF



(b) NPR=1.46

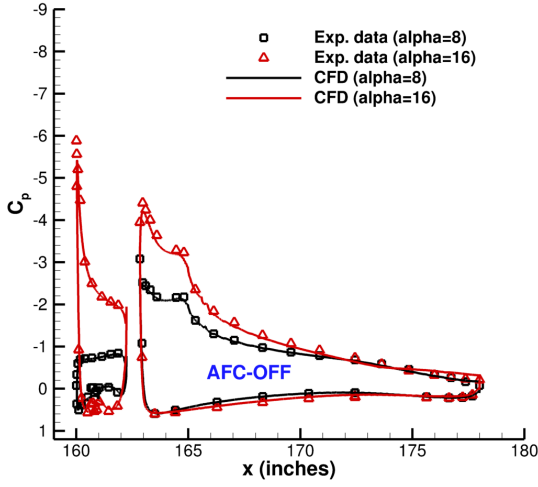


(c) NPR=2.0

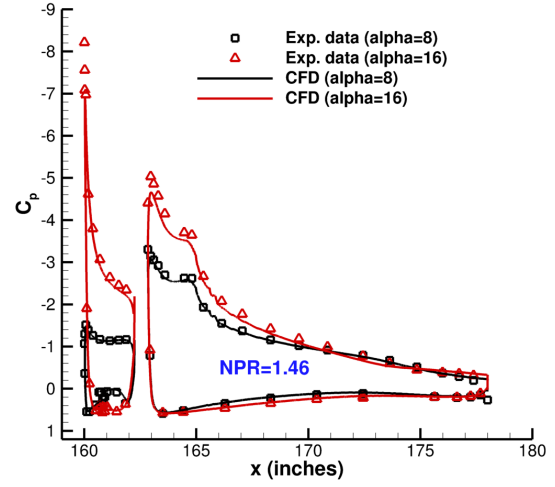


(d) NPR=2.41

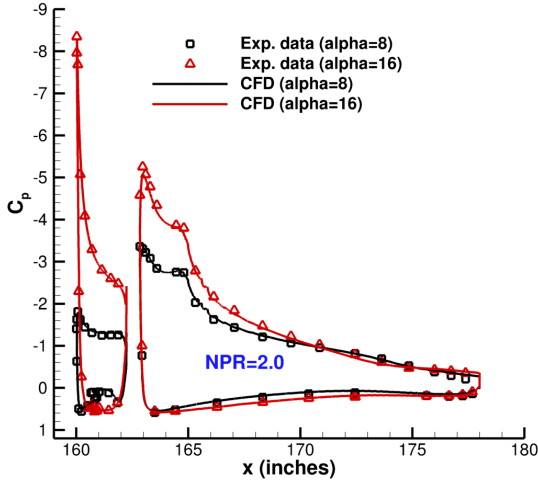
Fig. 12 Effect of pressure variation on sectional C_p at $y=63.8$ inches ($\eta = 0.552$)
Mach=0.2, $Re=3.27 \times 10^6$.



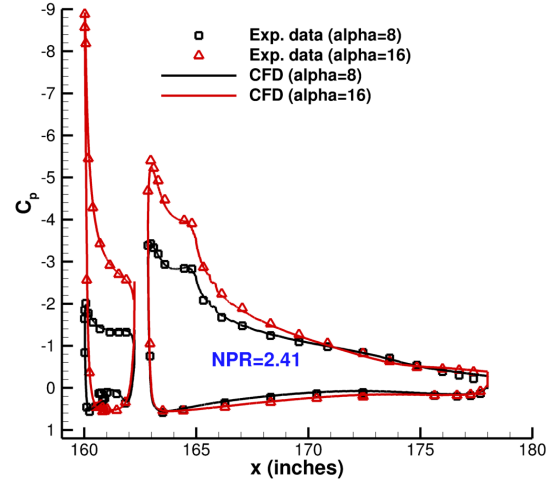
(a) AFC-OFF



(b) NPR=1.46

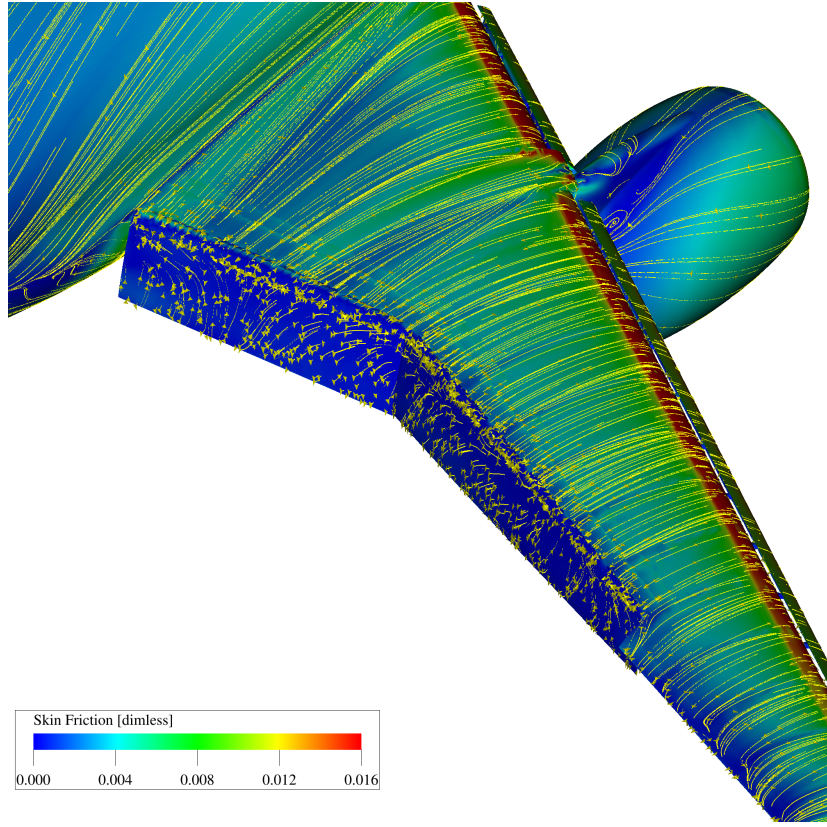


(c) NPR=2.0

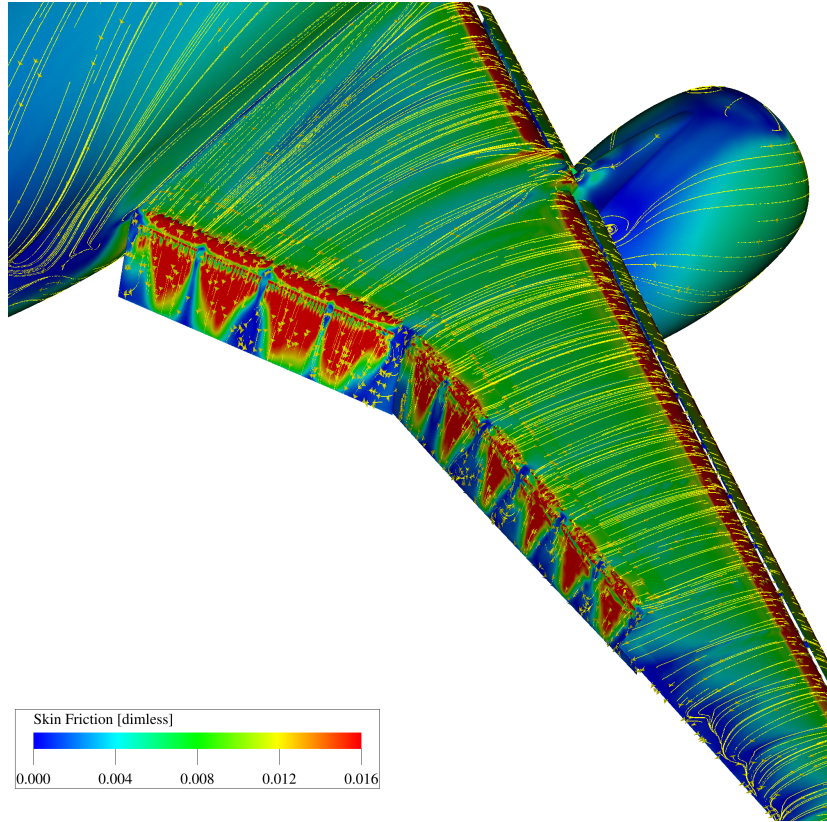


(d) NPR=2.41

Fig. 13 Effect of pressure variation on sectional C_p at $y=94.74$ inches ($\eta = 0.819$)
Mach=0.2, $Re=3.27 \times 10^6$.

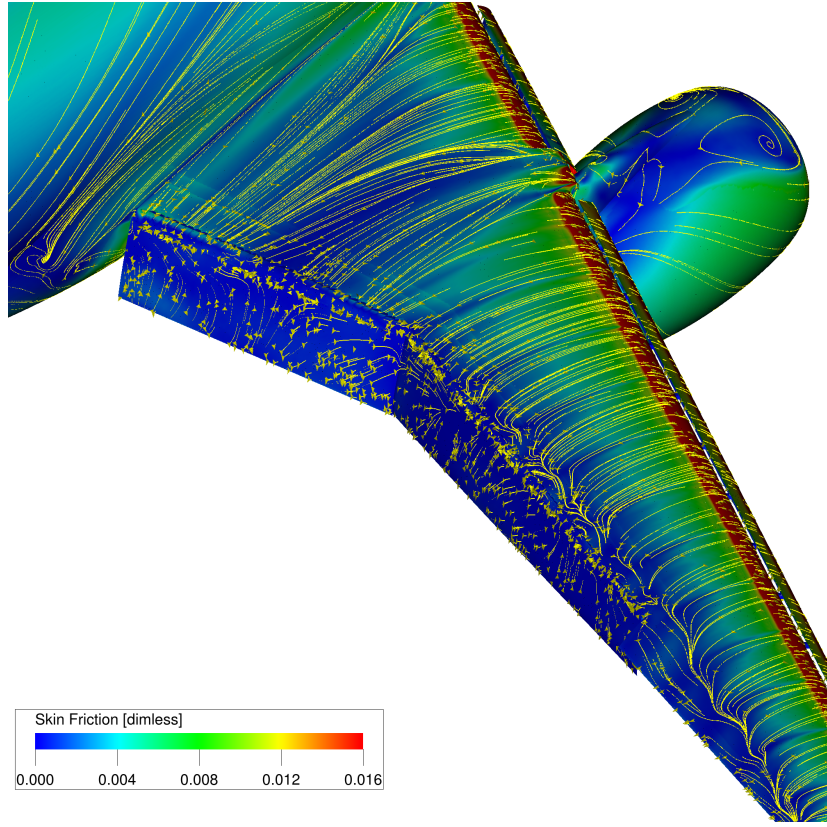


(a) AFC-OFF

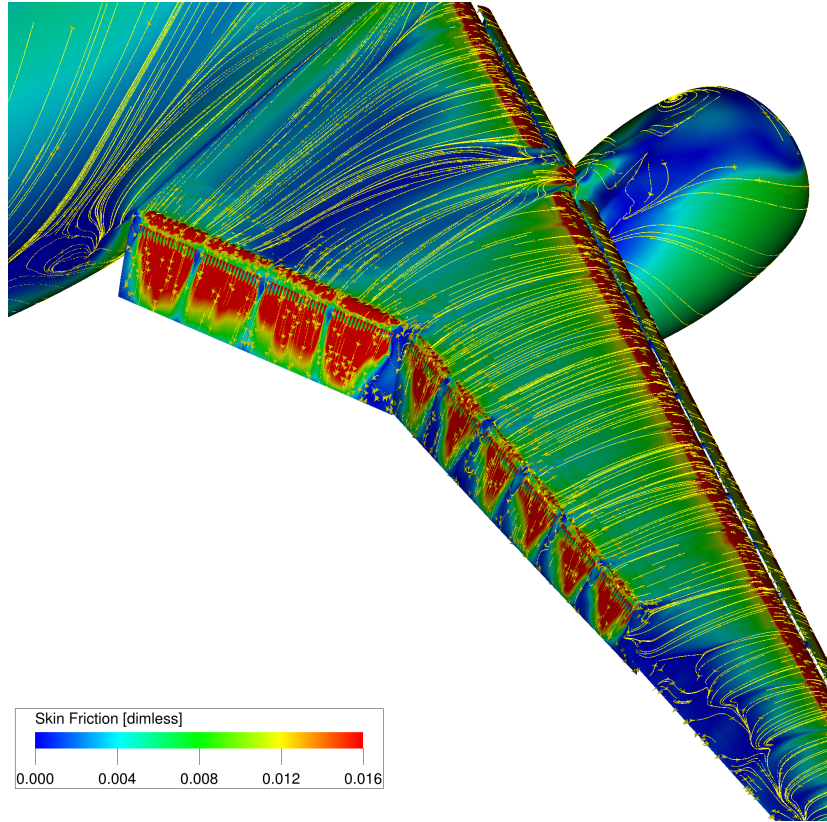


(b) AFC: NPR=2.41

Fig. 14 Effect of AFC on simulated surface streamline patterns, Mach=0.2, $\text{Re}=3.27 \times 10^6$, $\alpha = 8^\circ$.

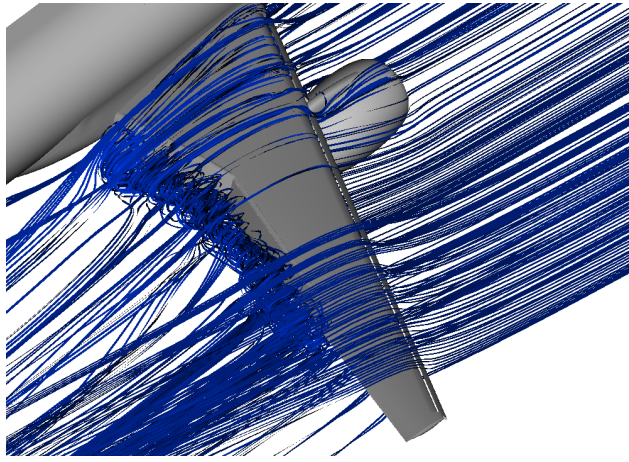


(a) AFC-OFF

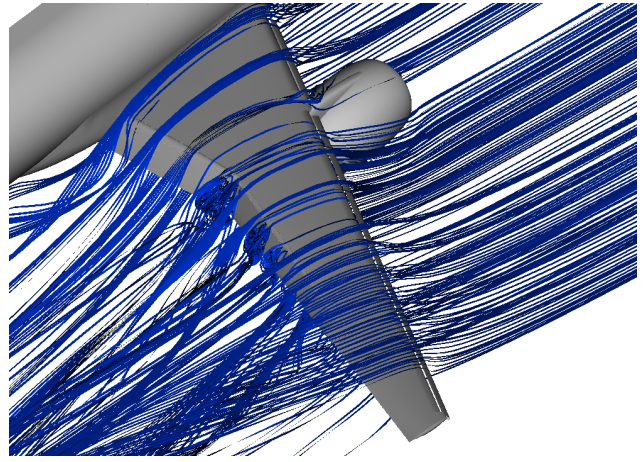


(b) AFC: NPR=2.41

Fig. 15 Effect of AFC simulated surface streamline patterns, $\text{Mach}=0.2$, $\text{Re}=3.27 \times 10^6$, $\alpha = 16^\circ$.

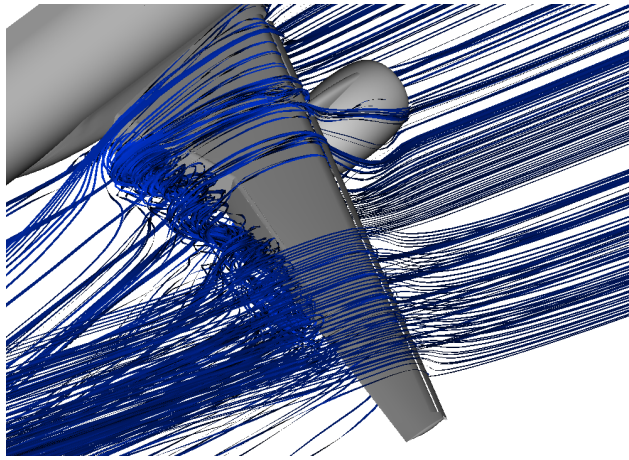


(a) AFC-OFF

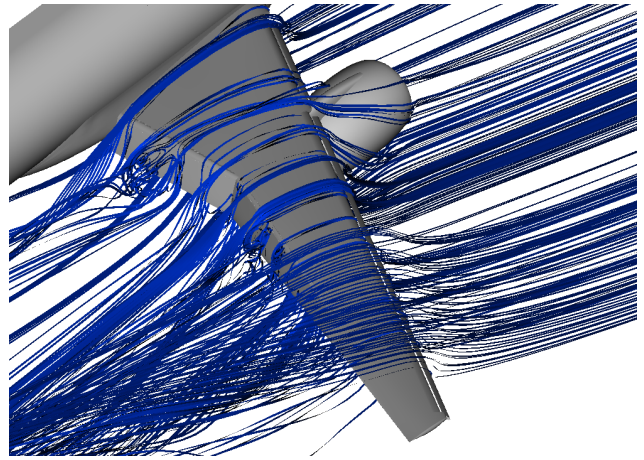


(b) AFC: NPR=2.41

Fig. 16 Effect of AFC on simulated 3-D streamline patterns, $\text{Mach}=0.2$, $\text{Re}=3.27 \times 10^6$, $\alpha = 8^\circ$.



(a) AFC-OFF

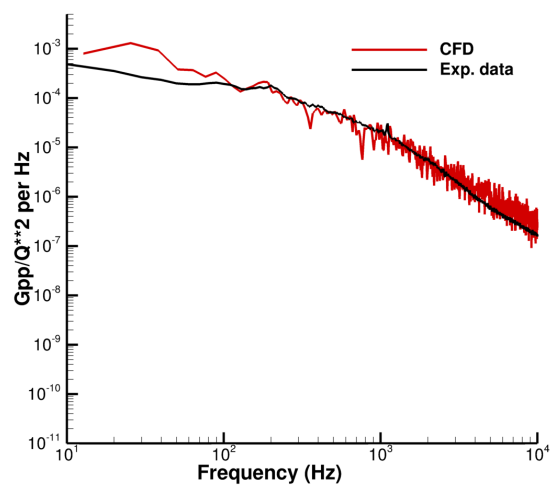


(b) AFC: NPR=2.41

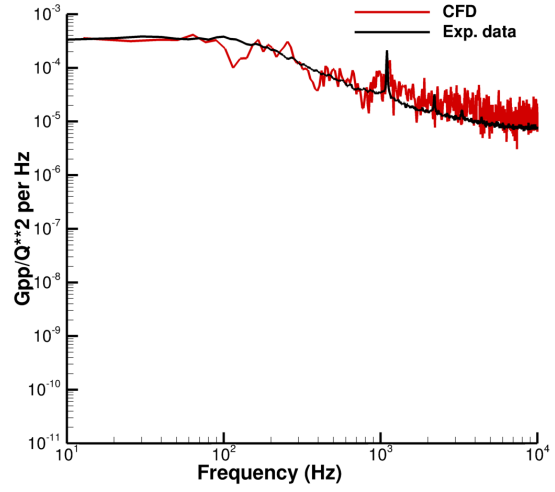
Fig. 17 Effect of AFC on simulated 3-D streamline patterns, $\text{Mach}=0.2$, $\text{Re}=3.27 \times 10^6$, $\alpha = 16^\circ$.



Fig. 18 Wind-tunnel model with Kulites® (covered by blue rectangles) installed on flaps.

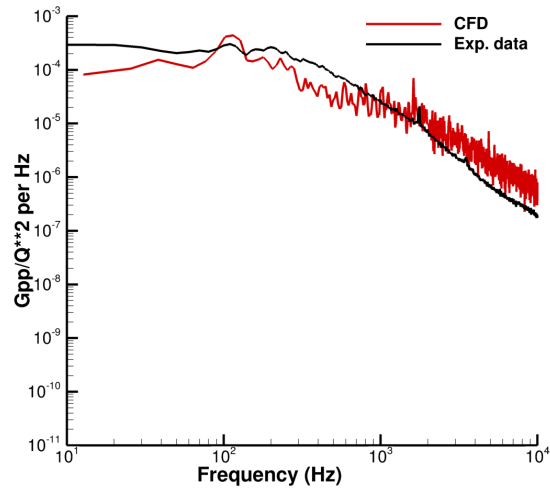


(a) trailing edge

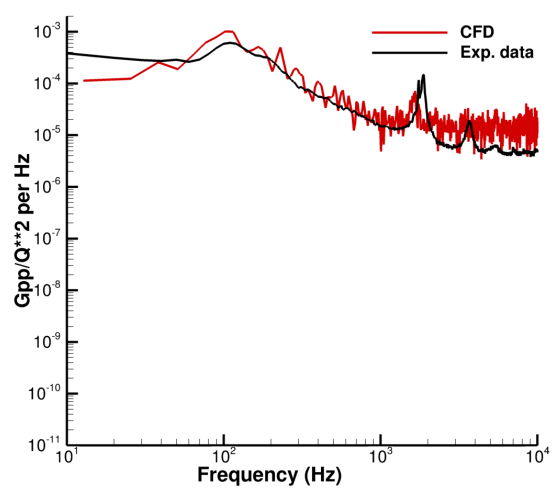


(b) shoulder

Fig. 19 Comparison of PSD distributions at the midspan inboard flap Kulite® locations, NPR=2.41, Mach=0.2, $Re=3.27 \times 10^6$, $\alpha = 8^\circ$.



(a) trailing edge



(b) shoulder

Fig. 20 Comparison of PSD distributions at the midspan outboard flap Kulite® locations, NPR=2.41, Mach=0.2, $Re=3.27 \times 10^6$, $\alpha = 8^\circ$.