FLOW CONTROL TECHNIQUES FOR TRANSPORT AIRCRAFT

Arvin Shmilovich, Yoram Yadlin
The Boeing Company

Keywords: *flow-control, vortex, engine ingestion, wing-wake, high-lift*

Abstract

A set of flow control approaches have been recently investigated for improved aerodynamic performance in a range of transport aircraft applications. Active Flow Control concepts are employed for reducing operational hazards during airplane ground maneuvering, alleviating trailing wakes for higher air traffic efficiency and improving takeoff and landing performance. Computational Fluid Dynamics has been extensively used to gain insight into complex flows subject to various modes of actuation and to develop promising techniques of flow control. The use of numerical simulations is especially important in scenarios involving powered systems, intricate vortex structures, separated flows, and active flow control devices, where alternative testing is extremely expensive and often impractical.

1 Introduction

In recent years a variety of flow control techniques have been developed for a wide range of applications [1-5]. Reduced flow separation is an example of a targeted application where significant breakthroughs have been achieved using Active Flow Control (AFC) for improved aerodynamic performance. A review of flow control applications for several body shapes and simple airfoils is given by Wygnanski [2]. Several studies were geared towards demonstrating the effectiveness of pulsed actuation in controlling the leading edge flow separation which occurs for airfoils at high angles of attack. Other examples include airfoil sections with relatively simplified high-lift devices, i.e., drooped leading edges and simple hinge flap elements. While significant inroads have been made in the context of relatively simple airfoil shapes, the question of whether AFC could be integrated into a practical airplane design is still outstanding. This is crucial since one of the principal design objectives of commercial and military airplanes is to achieve mission requirements for takeoff and landing. Because superior high lift capability is a key objective it is important to assess the merit of AFC and to establish performance increments. The current study is aimed at identifying potential implementations of flow control for a set of high-lift wing sections and providing guidelines on practical modes of actuation.

Potential extensions of AFC for control of vortex systems in order to enhance airplane operational capabilities are also explored. One application addresses the engine ingestion of ground vortices which occurs at certain conditions around airplanes operating in ground proximity. Engine vortex ingestion poses a serious operational hazard due to the risk of Foreign Object Damage (FOD) and engine surge. Another application of AFC is aimed at control of trailing wakes. The flow control objectives are (a) to permit safer and shorter airplane separation distances, thereby mitigate airport congestion, and (b) to reduce Blade Vortex Interaction to suppress noise from rotorcraft blades.

The paper will describe the flow control approaches for the respective applications. The numerical tool will be reviewed in the context of time-dependent modeling for a set of actuation modes. A set of diagnostics tools will be employed in order to identify promising AFC implementations for the set of aerodynamic problems.

2 Numerical procedure

The numerical tool used for the simulation of active flow control is a modified version of the OVERFLOW code originally developed by NASA [6]. OVERFLOW uses the unsteady Reynolds Averaged Navier-Stokes formulation for overset grid systems. The numerical procedure has been modified in order to facilitate the development of a family of flow control techniques for vortex alleviation and reduced flow separation. Special modules have been developed for the modeling of time-varying boundary conditions to simulate flow excitation due to control devices. Jet actuation is described by the mass flow rate, cross sectional area and the stagnation pressure and temperature in order to define the velocity at the ejection nozzle. The numerical algorithm uses the characteristics approach for consistent application of the boundary conditions.

The current flow control techniques can be generally grouped in two basic actuation modes. One group employs pulsed actuation which is used to model synthetic jets and non zero-mass-flow actuation. In the case of pulsed jets, the flux vector is aligned with the nozzle axis and jet pulsation is determined by the forcing frequency. Various signal shapes ranging from sinusoidal to step-function (i.e., jet velocity as a function of time) are defined by sets of analytical functions with continuous first and second derivatives.

The second family of flow control actuators is based on periodic motion of the ejection nozzle, much like a water sprinkler system. Jet flow is continuous but the fluid is discharged through a swiveling nozzle. A direct approach for modeling jet ejection from moving nozzles requires moving grid systems. In order to avoid this complexity and to simplify the computational procedure, two alternative implementations of boundary conditions were employed to approximate moving jets. In one implementation the nozzle is assumed to be stationary, but the jet flux vector at the exit plane is prescribed in a time-varying fashion to mimic the swiveling motion of the nozzle. For a fixed jet velocity at the nozzle exit this treatment produces non-constant jet mass flow rate as the jet vector cycles through. In an alternative version which preserves constant jet mass flow, the jet velocity is periodically modified according to the instantaneous angle between the jet vector and the nozzle exit plane. Further details on the boundary conditions are described in Ref. [7]. The current study indicates that the two optional sets of boundary conditions produce similar global flows, although the time-varying flow characteristics are slightly different. The constant mass flow condition is used in the simulations presented here.

The calculations were obtained using a second order upwind differencing scheme. Depending on the application, either the SA or the SST turbulence models have been used. The flow control computations use a second order time-accurate scheme. Time dependent simulations are initiated from a steady-state solution obtained for the flow in the absence of any actuation. A minimum of 800 time steps per actuation cycle is used, depending on the application and frequency of actuation.

3 Alleviation of engine ground vortex

Aircraft with turbo-jet engines mounted relatively close to the ground develop vortex activity during high-power, low speed and static ground operation. The suction generated by the engine results in the formation of a vortex on the ground. Usually, the ambient flow contains significant amounts of vorticity (turbulence) due to gusts, ground turbulence, wake flow of neighboring aircraft components (i.e., wing, fuselage), and mixing of engine reverser plumes when thrust-reversers are deployed. The mechanism of ground vortex formation is the amplification of the seed vorticity in the ambient flow due to the contracting streamlines approaching the inlet. This interaction results in a concentrated vortex originating at the ground plane and terminating inside the engine. The rotational flow field induced by the ground vortex is the cause for kicked up dust and dirt, which can become entrained in the airflow drawn into the engine inlet. The tornado-like flow is capable of dislodging sizable foreign objects off the ground (for example, rocks, chunks of ice or asphalt), causing FOD which may lead to engine failure. The vexing problem of ground vortex ingestion hinders the ability to land in austere fields and to perform essential ground maneuvers on unimproved terrain. Furthermore, the engine ingestion problem is exacerbated by the advent of larger and more powerful high by-pass turbo jet engines.

One of the salient features of flows with increased propensity to ground vortex activity is the unsteady characteristic of the air motion. Flow visualization in full-scale experiments indicates that the flow field is highly unsteady and the movement of the vortex origination point on the ground is sporadic. Consequently, the vortex filament fluctuates incessantly over a significant portion of the lower inlet sector. The apparent randomness of the flow in realistic and an uncontrolled environments is due to gusts with varying strengths and direction (time-variation of ambient flow), ensuing structural response of aircraft components (i.e., the flexing of the entire wing/engine assembly) and the unsteady turbulent mixing of the thrust reverser plumes.

There have been several attempts to solve the engine vortex ingestion but none have proven to be satisfactory in an operational sense. Johns [8] has reviewed these approaches from a practical point. Prior techniques target specific flow situations and do not provide effective solutions for real life applications. Specifically, they constitute ‘point design’ solutions since they provide localized treatment of a ground vortex without consideration of the inherent problem of sporadic vortex motion.

A couple of vortex disruption methods that address the unsteady characteristics of realistic inlet vortex flows have been recently developed. The pulse jets device developed by Smith and Dorris [9] uses high-pressure air to alternatively eject fluid from two nozzles mounted underneath the engine nacelle close to the nacelle lips. The intermittent high frequency ejection provides turbulent mixing to prevent the formation of a coherent vortex. The sprinkler jet actuator proposed by Shmilovich et al [10] uses continuous ejection through a moving nozzle mounted on the nacelle lip in order to provide wide area coverage, thereby reducing the risk of vortex ingestion even when the vortex moves rapidly. The effectiveness of these inlet vortex alleviation methods has been demonstrated for isolated engines in proximity to the ground plane [7]. The current numerical simulations focus on the evaluation of the control systems for full airplane configurations.

3.1 Methods of vortex control

The flow control techniques utilize fluidic injection in critical regions close to the engine inlet. The flow actuation is accomplished by high-pressure bleed air from the compressor, which is supplied to a valve located inside the engine cowl and close to the nacelle lip. The amount of air required to affect the inlet vortex is less than one percent of total inlet flow, well within the bleed limit of the engine. Two modes of actuation have been considered for the treatment of vortex ingestion.

3.1.1 Pulsed actuation

The pulsed jets system [9] uses high-pressure air to alternately eject fluid from two nozzles mounted underneath the engine nacelle close to the nacelle lips. The nozzles would be deployed during low aircraft speed and high engine power setting. The injection nozzles are directed upstream and towards the ground plane. The intermittent high frequency ejection provides sufficient turbulent mixing to avoid the formation of a coherent vortex.

The effect of pulsed actuation is demonstrated on a transport airplane which includes all relevant components for adequate representation during ground operations. A pair of actuators is included at each engine as shown in Figure 1. The nozzles are pointing 15° inward and 30° below the horizon. A tail wind of M_∞ = 0.007 is considered in the simulation. High power setting is used at each engine to simulate realistic operational conditions. The flow control signal consists of a pulsed jet at a frequency of 140 H_z , defined by a signal resembling a step function and is shown in the inset. The short time-scale of the ejection patterns is illustrated by releasing particles from the nozzles during the first five actuation cycles. The particles are colored by the local Mach number where red represents high velocity. In this snapshot one nozzle blows at maximum velocity, close to sonic, while the other nozzle has zero jet velocity. The global flow structure is described in a series of snapshots in Figure 2, where the actuation starts at t = 0. Traces of particles released at select locations on the ground plane and on the fuselage are used to examine the vortex system in the transient flow field. The ground plane is described by the pressure field. In the baseline flow (t < 0) the outboard engine is largely exposed to the oncoming tail wind and therefore it does not develop a vortex off the ground plane. In contrast, the inboard engine experiences flow blockage due to the fuselage and the outboard engine, resulting in high suction power in order to satisfy inlet airflow requirement. The suction results in the formation of the ground vortex leading to inboard engine ingestion. Moreover, due to the proximity of the inboard engine to the fuselage, an additional vortex element is formed off the fuselage surface. The fuselage vortex is also ingested by the inboard engine, but it does not pose risk of FOD. A more thorough description of the vortical structure around airplanes in ground operations is given in Ref. [11]. The excitations introduced by the jets affect the flow in front of the engines. The pressure waves created by the periodic jets impinge on the ground and disperse in a radial direction in the region underneath the engines. According to the simulation, as time progresses, the vortex filament is pushed away from the engine in the forward direction, toward the front of the airplane. Note that the vortex originating at the side of the fuselage is hardly affected by the actuation.

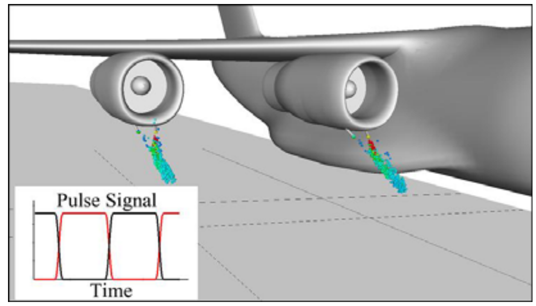

Fig. 1: Pulsed jet ejection described by particles released from actuator nozzles

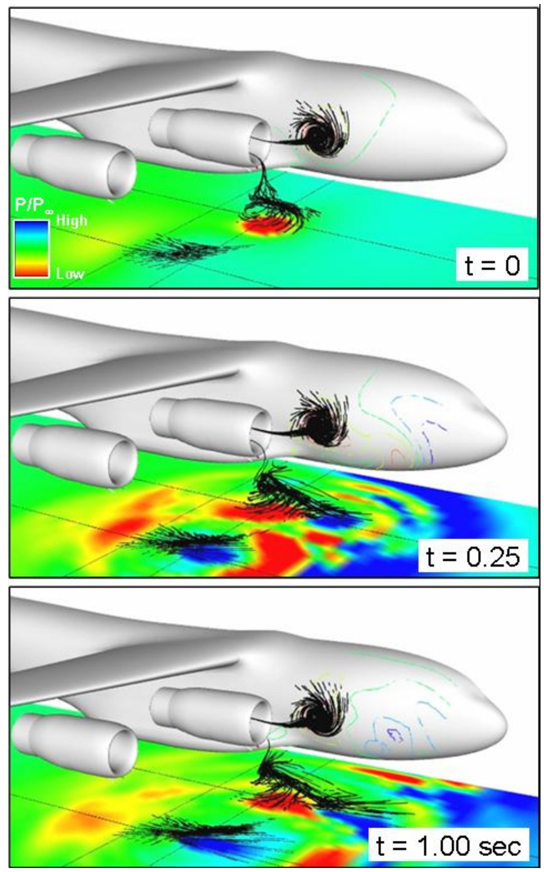

Fig. 2: Flow development due to pulsed jet actuation (pressure contours on the ground plane and particle traces)

3.1.2 Sprinkler actuation

The sprinkler system uses a single nozzle located close to the nacelle lip [10]. During actuation the nozzle swivels according to a prescribed motion in order to inject flow into a large domain in front of the engine inlet in the general upstream direction. The slew motion of the ejecting fluid disrupts the global flow field in front of the engine and prevents the formation of vortices. Since realistic full-scale engine vortex ingestion is a highly unsteady phenomenon, with the vortex meandering underneath the engine, this method is very effective in breaking up of the non-stationary ground vortex.

Wind conditions and engine power setting as in the pulsed jets case are used for the sprinkler actuation. The sprinkler jet actuation is applied at the lower part of each engine, close to the inlet lips. The nozzle is pointed at an angle of 20º below the horizontal plane and the jet moves side-to-side within a ±30º range at a frequency of 140 H_Z. The short time-scale flow development in the vicinity of the engine highlight regions is presented in Figure 3, where particles are released from the nozzles during five cycles from start of actuation.

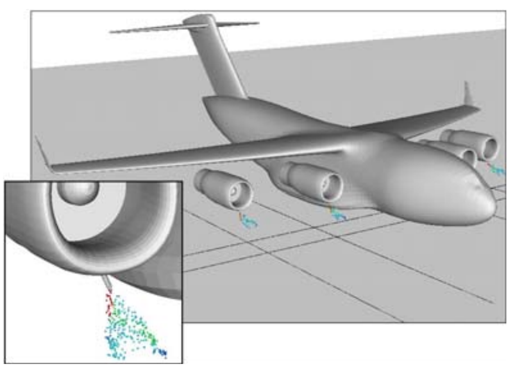

Fig. 3: Sprinkler jet ejection described by particles released during the first five cycles (0.036 sec) from start of actuation

The long time-scale flow development is examined in Figure 4. The intermittent motion provided by the periodic excitation at the bottom side of each of the engine cowls perturbs the flow in front of the engines. After 0.14 seconds the perturbations reach the ground surface and pressure waves are generated in the near field underneath the respective engines. The ripple effects propagate radially with decreasing intensity in pressure fluctuations until the flow reaches a limit-cycle behavior at t > 1.00 seconds. The ground vortex is disrupted close to the inboard engine inlet at t = 0.14 second. The subsequent time frames show that the vortex filament is altered by the ejecting flow and it is expelled away from the engine. Engine vortex ingestion from the bottom side has been curbed by the sprinkler actuation while the vortex off the fuselage is only slightly affected.

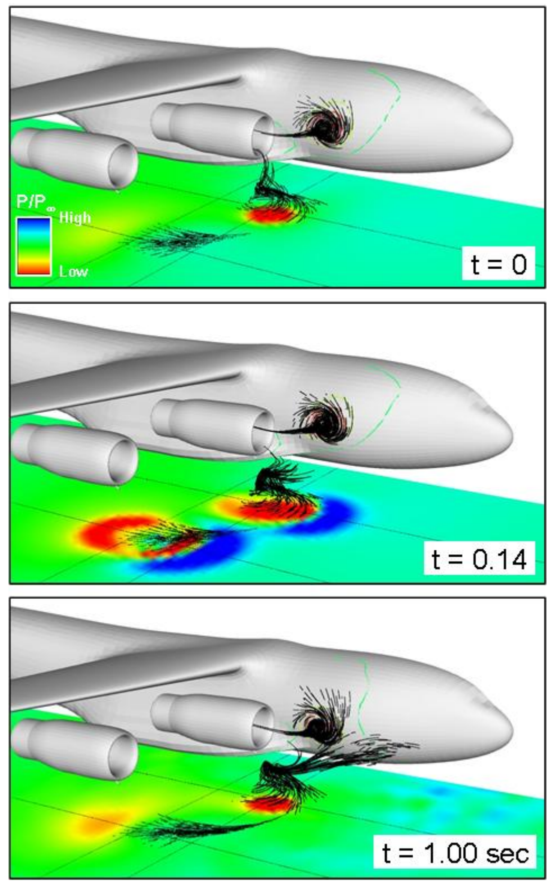

Fig. 4: Flow development due to sprinkler actuation

4 Wing wake alleviation

An area of high priority in the aerospace transport industry is solving the looming problem of airport congestion. The capacity of many airports is close to saturation, yet the number of aircraft in commercial aviation is projected to double in the next 12 years. Current airport capacity is largely controlled by the frequency with which planes can be brought in and out of the airport. A pacing item in landing and takeoff frequency is the time necessary for the dissipation of wake vortices produced by planes in motion since trailing vortices pose particularly dangerous conditions for following airplanes. The eventual breakup of the vortices is the result of instabilities generated by perturbations due to ambient turbulence [12]. Unfortunately, these instabilities evolve very slowly and do not result in flow conditions that allow meaningful reductions in airplane separation. Trailing vortices have caused numerous airplane crashes and have been a main concern for the Federal Aviation Administration and similar organizations around the world. Currently, the only way to solve the problem of this invisible hazard is to avoid the flight path of large aircraft. Air traffic regulations require aircraft separation to be maintained to assure that severe vortex encounters are avoided. The minimum separation distance represents a key limiting factor in productivity at a growing number of airports, with ripple effects on the entire air traffic system. The problem of vortex encounters is exacerbated with the advent of very large transports requiring increased separation and offsetting some of the economic advantages of the larger vehicles. Operating by current standards presents a bleak prospect for the future of air traffic. Consequently, there exists a keen practical interest in developing techniques for wake flow control and alleviation.

Prior attempts to control airplane trailing wakes can be grouped in two categories. One set seeks to alter the vortex structure while the other group targets vortex instabilities which lead to vortex disintegration. An early technique for break-up of wake vortex was investigated by Crow and Bate [13], whereby airplane control surfaces were used to periodically alter the wing loading in order to introduce perturbations into the vortex system which ultimately lead to vortex instability. Crouch and Spalart [14] extended this method to airplanes with flap deployed, where quicker vortex break-up was accomplished by periodic movement of control surfaces in order to trigger instabilities associated with multiple trailing vortices. An undesirable by-product of this mode of control surface movement is the continual change in wing load, with implications on dynamic loads, airplane controllability and ride quality. Recently Greenblatt [15] demonstrated the feasibility of near-field vortex management via control of flow separation using pulse zero net mass flux actuation on wing flaps.

4.1 Method of wake control

The current study explores a technique based on fluidic actuation for control of tip vortex as proposed in Ref. [16]. The flow actuation introduces time-varying perturbations into vortex elements in order to hasten vortex break-up. The control devices can be installed at the tips of lifting surfaces, including wing tips and edges of flap elements. The current approach can be used for the control of either airplane wing wakes or rotorcraft blade wakes since it does not require usage of control surfaces. A schematic layout of the wake alleviation system for a typical transport is depicted in Figure 5, which shows the flow control actuator at the tip of the wing. The system utilizes high-pressure bleed air from the engine compressor (or from other air sources), which is supplied to valves located close to the tips of the wing and/or the flap elements. Individual valves are connected to respective movable nozzles that control the fluid ejection. During actuation the jet off each of the individual nozzles is blown in the general lateral direction. The continuous jet is discharged through a nozzle which swivels intermittently at given frequency and within predefined azimuth bounds.

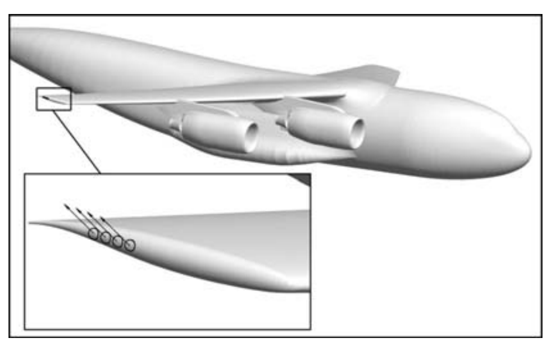

Fig. 5: System for the control of wing vortex

The controlled motion of each nozzle is prescribed in order to achieve maximum effect on vortex formation and its subsequent development. Various scanning patterns can be obtained by using select combinations of sinusoidal horizontal and vertical motions (i.e., sets of amplitude, frequency and phase of the planar motions). Depending upon the frequency, the flow control can be effective in either accelerating vortex decay or in introducing instabilities leading to vortex disintegration.

The method for controlling wing tip vortex development has been evaluated for a wing mounted on a vertical wall. No high lift devices are included in the current analysis. A free stream Mach number of 0.25, an angle of attack of 8º and Reynolds number of 38.5 million (based on C_{mac}) are used to represent final approach conditions. These conditions result in a lift coefficient of 0.68.

Special considerations must be given to grid resolution to ensure adequate representation of wing wake development, particularly in the context of active flow control. Proper modeling of the bounded wing viscous layers, vortex rollup and downstream flow development is assured only if the level of numerical dissipation is small in comparison with the viscous terms. Excessive numerical dissipation results in unrealistic vortex decay, and therefore demands high grid resolution. Moreover, the diameter of the vortical element is quite small and it occupies a small portion of the computational domain. Clearly, without a priori knowledge of vortex location a conventional grid system with adequate fineness presents an unacceptable option since it is both wasteful and impractical. Therefore, the current study employs a grid adaptation technique for efficient modeling of high resolution vortex flows [17].

The adaptation process is performed for the baseline flow (without flow control) by keying off minimum total pressure. When convergence is reached and grid adaptation is complete, the vortex core is usually represented by stencils of about 16x16 grid points on each of the cross sectional grid surfaces. This procedure ensures that the rollup is well captured and the vortex core is preserved. The adapted grid obtained from the baseline flowfield is used for flow control computations. The vortex fitted grid occupies a sufficiently large volume such that the excursions of vortex core due to flow actuation are contained within this grid block.

Actuation is applied at the edge of the wing tip. The boundary condition for the actuation is prescribed on the surface of the wing tip cap. The exit plane of the ejection nozzle is defined by an elongated rectangular shape. In the following simulations the actuation nozzle effectively moves up and down in the lateral direction in the range of ±30º off the horizontal plane.

Actuation at a frequency of 10.7 H_z is first considered for describing the flow excitation in the very near field. Figure 6 shows the flow structure in the front views, where the vortex is represented by the streakline traces off the wing tip during time interval of one actuation cycle (0.093 seconds). A distinct vortex originates at the wing tip before active control is being applied. The resulting flow after jet activation demonstrates that the intermittent mixing provided by the periodic motion of the jet perturbs the flow in the tip region and alters the development of the trailing vortex by reducing its strength and diffusing it in the cross plane.

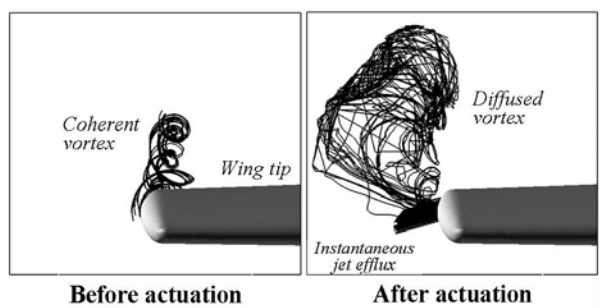

Fig. 6: Near-field wing tip flow structure before and after actuation (front view)

The flow fields obtained with high and low actuation frequencies of 10.7 H_z and 1.07 H_z , respectively, are described next. The wake structure for the baseline and the controlled flows are depicted in Figure 7 where the tip vortex is tracked by the total-pressure loss. The vortex core is represented by an iso-surface of total pressure. Cross sectional contours are also shown at select streamwise stations. The computed baseline flow indicates that after the initial rollup phase the vortex is asymptotically preserved, producing a coherent wake with strong tip vortex. The trailing wake flows due to active control are represented by flowfield snapshots of the respective actuation frequencies. In the high frequency case the strength of the vortex is significantly reduced and the flow is nearly constant, with very small variations in wake structure. The interaction between the oscillatory jet and the cross flow is virtually invariant on the global time scale and therefore the global impact on wake structure is effectively decoupled from the operating frequency. On the other hand, the low frequency oscillating jet introduces periodic disturbances along the tip vortex which propagate downstream. Here, remnants of the vortex core are represented by blobs defined by iso-surfaces of total pressure. The motion of these fragmented vortical elements resembles a corkscrew pattern, formed by a combination of a wobbling motion in the cross plane and streamwise advance. Contrasted with the natural instabilities which evolve slowly, the current technique introduces perturbations in the region where the vortex originates, potentially triggering instabilities much sooner.

The actuation at the edge of lifting surfaces affects the load distribution in the vicinity of the tip. Figure 8 shows wing lift oscillations over the respective actuation periods during the limit cycles. Also, families of instantaneous wing load distributions are presented for the respective actuation frequencies. The flow control produces very mild periodic variations of wing load in the outboard 15% segment of the wing span. The time-averaged lift is about 0.7% higher than the lift of the baseline wing, with only 0.17% and 0.44% lift oscillations for the high and low frequencies, respectively. Relative to active methods that utilize control surfaces for wake alleviation through wing load management, the current technique has an important advantage with respect to dynamic loads, flight control and ride quality.

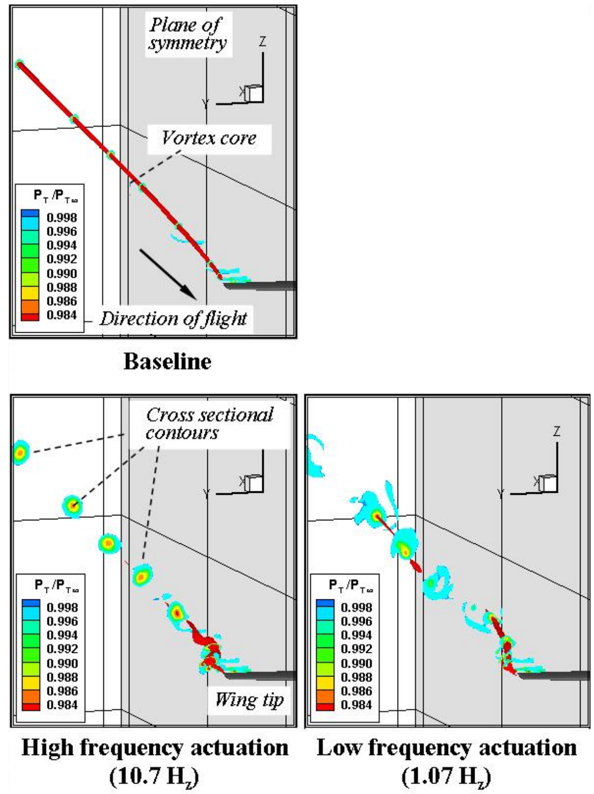

Fig. 7: Trailing vortex before and after application of flow control (total pressure)

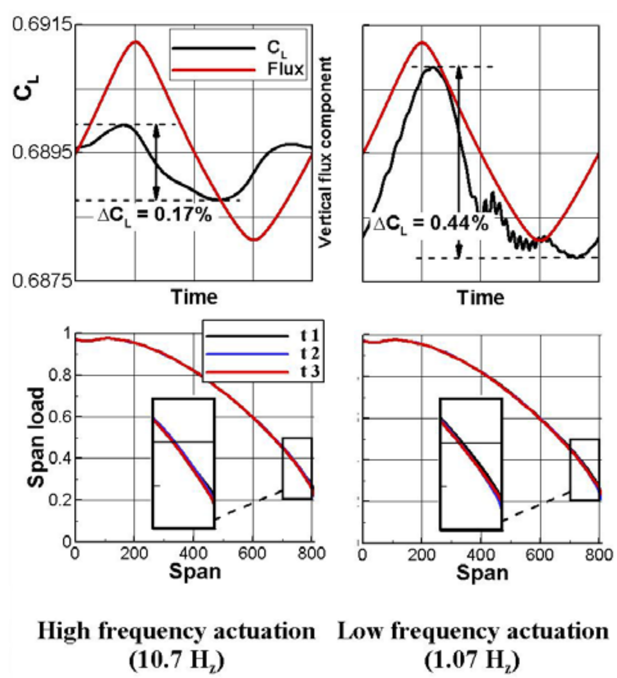

Fig. 8: Lift response and instantaneous wing load distributions due to actuation

5 Enhanced high-lift for takeoff and landing

Takeoff and landing performance are two of the principal design objectives of transport airplanes. Whether the requirement is for a higher airplane weight or for shorter runways, superior high-lift capability is a key requirement of the airplane manufacturers. Techniques for altering the viscous flow structures at high-lift conditions are highly desirable due to the potential for improved efficiency. Although previous investigations of AFC demonstrate various degrees of performance improvements, it is not clear whether the gains are substantial enough to warrant the development of a whole new airplane based on this technology. It is therefore important to explore ways of AFC implementation for providing very high performance levels while addressing issues of practical airframe integration.

In the application of AFC to high-lift the numerical tool was first validated against experimental data for several wing sections and various actuation modes [18, 19]. Numerical simulations were then used to develop flow control approaches for a set of high-lift systems. Actuation is prescribed at ports represented by nozzles embedded within the airfoil. Details of the numerical implementation are described in Ref. [18]. Sinusoidal pulsed zero mass-flux is generally used, unless otherwise noted. In the following analyses the inviscid simulations are also used for reference. Inviscid lift determines the theoretical upper limit of an airfoil to produce lift in the absence of viscous effects and is conveniently used as a yardstick for AFC efficiency.

5.1 Single element section

High lift systems that consist of simple hinge moveable elements are very attractive since they are light and mechanically simpler than slotted multi-element systems, resulting in lower manufacturing cost. The aerodynamic characteristics of simple hinge systems are not as good as slotted wing sections. The objective is to augment their performance using AFC. A single element airfoil with a drooped leading edge and a simple hinge flap is considered for AFC implementation. Results of two-dimensional simulations for flap deflection of 25° are shown in Figures 9 and 10. The baseline flow consists of a very large flow separation bubble at the flap, resulting in maximum lift of about 2.4. Flow control using a momentum coefficient (C_μ) of 0.03 is first applied at port #1 at the hinge line where the separation originates in the baseline case. The computations indicate that AFC is effective, providing a lift increment of approximately 0.65 in the linear lift range. In a time-average sense the application of AFC at the hinge line is only partially effective, reducing flow separation in the front portion of the flap.

Simultaneous actuation is then applied at 5 ports along the flap, with C_μ=0.03 at each port and phase shift of 180º between adjacent ports. A unique flow structure ensues in this case, where the original separation bubble is effectively altered to a predominantly attached flow. The cumulative effect provided by the multiple ports results in a lift gain of 1.33 relative to the baseline and it represents more than twice the gain realized with the single port. Further confirmation of the effectiveness of distributed actuation is demonstrated by the resemblance of this flow control mode with the inviscid case in Figure 10. Also, the lift is close to the level obtained in the inviscid case.

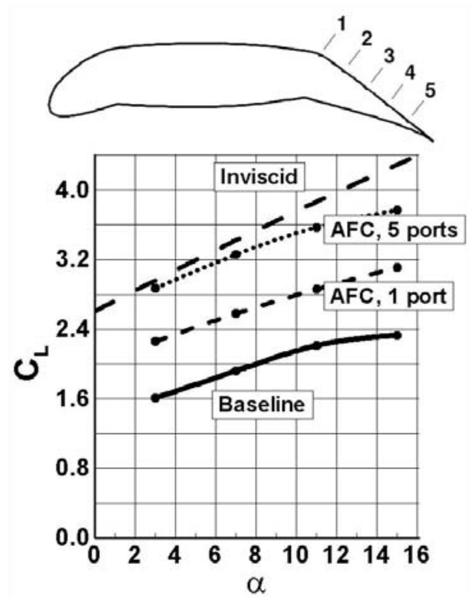

Fig. 9: Lift augmentation due to distributed flow control – simple hinge flap

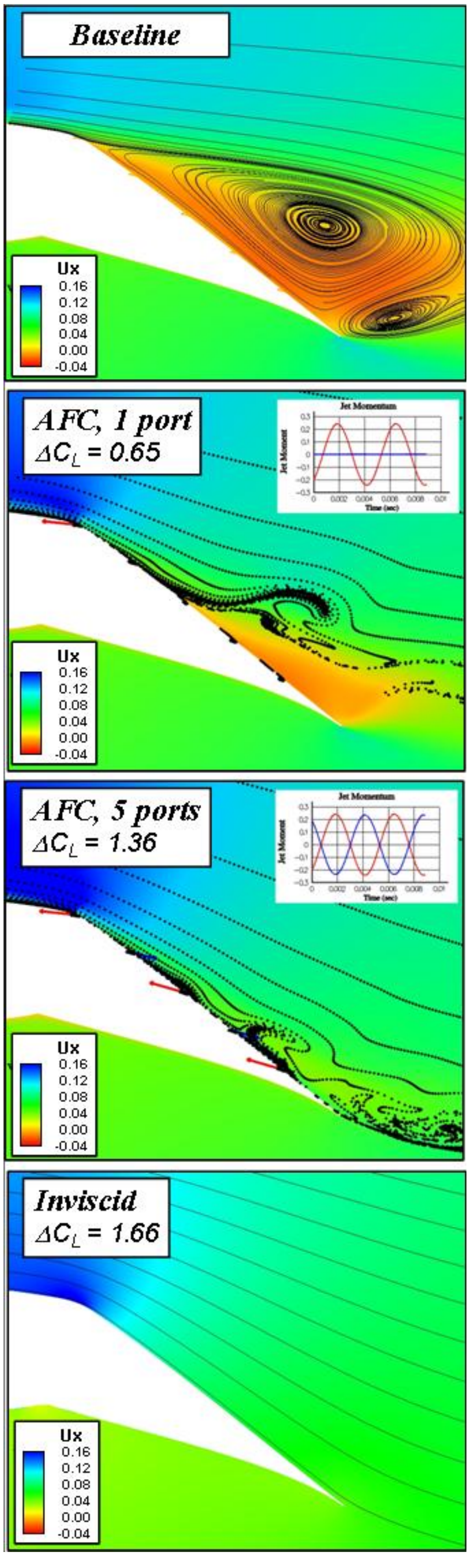

Fig. 10: Instantaneous flow field description for a simple hinge flap system (streamwise component of velocity, α=11°)

5.2 Multi-element wing sections

The next set of simulations focuses on AFC for multi-element systems where the flow is highly interactive. For instance, the trailing-edge flap is strongly influenced by the downwash generated by the lift on the main wing. Several factors can limit the maximum lift that can be achieved by a multi-element system. The maximum lift is limited by viscous effects resulting from the very strong pressure gradients introduced by the high suction levels. Or, it can be limited by boundary-layer separation in the vicinity of the slat and main wing leading edge. It can also be limited by boundary-layer thickening or separation on the trailing edge of the main wing or on the flap elements. Another limitation can occur due to bursting of the viscous wake from the slat or the main wing as it passes through the high pressure gradients developed by the flap. In this case, the boundary layers on each of the high-lift components may be attached, but the rapid spreading of the viscous wakes will limit the maximum lift that can be achieved [20]. An attempt to improve the flow over a high-lift system by addressing one of these limiting factors may improve the performance, but it will still be limited by the other factors. The goal is to identify AFC implementations with meaningful performance improvements by addressing all of these effects. This will be evaluated for configurations that are optimized for high-lift.

5.2.1 Conventional wing section

The conventional airfoil represents a typical transport and it is derived from the wing of the McDonnell Douglas MD80 airplane. The effect of distributed AFC at a representative takeoff condition with flap deflection of 24° is shown in Figures 11 and 12. The simulation indicates that at α=19º the baseline flow is on the verge of separation at the trailing edge of the slat and in the aft portion of the main element. The combined 10-port actuation at the slat, wing and flap elements according to the AFC layout described in Figure 11 produces near inviscid lift in the linear range and profound improvement in stall characteristics. The application of AFC at all wing elements is beneficial over the range of angles of attack. In particular, the actuation at the slat reduces the size of its wake considerably. Consequently, the slat wake traverses the adverse pressure gradient regions of the main element and flap without significant degradation in flow quality. This results in lesser tendency for off-surface flow reversal, streamlined flow around the flap and higher circulation. The streamlining effect is significant, resulting in larger turning angle in the aft airfoil portion. For practical implementations it is also important to explore ways of minimizing the required input to the flowfield while still achieving meaningful lift increments. Results obtained for a 6-port flow control pattern with two ports on each wing element are included in Figure 11. Compared to the full AFC actuation, the 6-port set results in relatively small degradation in lift.

5.2.2 Advanced wing section

The advanced wing section is derived from an experimental model that has been thoroughly tested and optimized for very high lift. It consists of a variable camber Krueger slat and a flap with extensive Fowler motion. Results for a 50° flap deflection are presented in Figures 13 and 14. This is representative of landing conditions in which the baseline flow is separated over most of the flap even at low angles of attack. The combined 12-port actuation results in smooth and attached flow on the upper surface of all elements. The circulation increases on all elements as indicated by the counter-clockwise movement of the stagnation points. Consequently, inviscid lift level is achieved in the linear lift range. The flow control at the slat boosts performance at high angles of attack by producing a wake that favorably interacts with the flow on the flap.

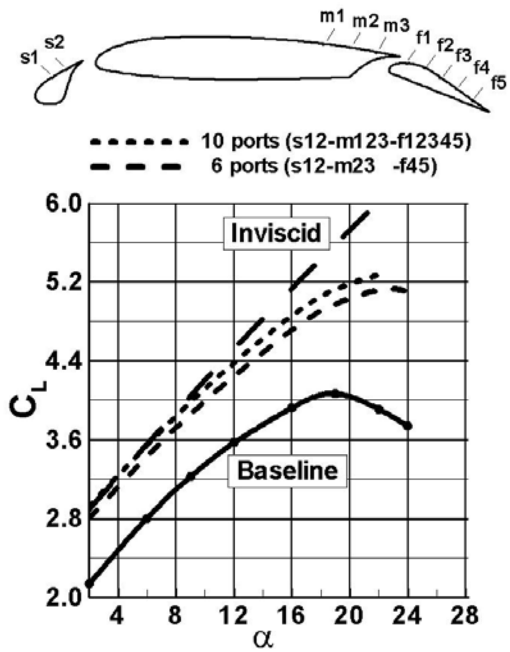

Fig. 11: Effect of AFC application at each element of the conventional high-lift wing section (representative takeoff configuration, δ_{flap} = 24°)

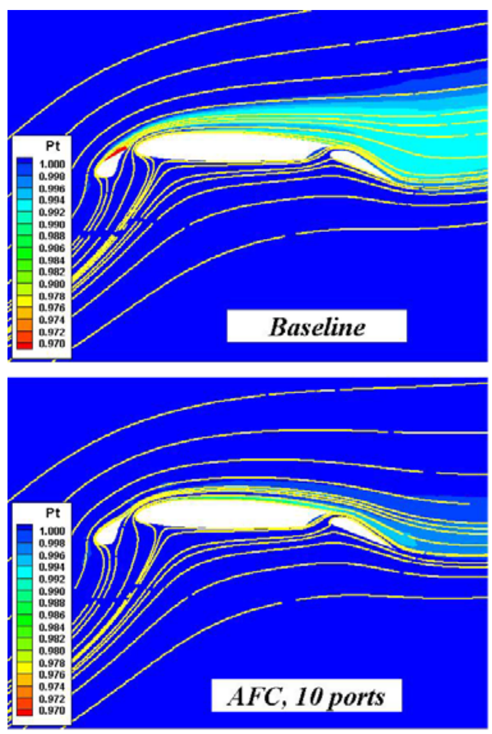

Fig. 12: Impact of AFC on flow structure of the conventional high-lift system (time-averaged total-pressure, α=19°)

A 4-ports set has also been investigated, where one port is employed at each of the slat and main elements, and two ports at the flap. This actuation mode reduces the lift in the linear range to about 70% of the level obtained with the full AFC application, with a commensurate drop in maximum lift.

The next set of flow control patterns includes constant suction at the slat in conjunction with zero-mass-flow actuation on the main element and the flap. The suction intensity is defined using mass flow rate equivalent to the uniform flow with a velocity equal to the maximum velocity of the pulsed jets. The results obtained with the AFC patterns employing suction are also shown in Figure 13. The suction at the slat removes a portion of the bounded viscous layer, rendering a smaller trailing wake. The slat wake influences the flow over the flap and helps improve C_{LMax}. In particular, the m2-f24 pattern with slat suction produces maximum lift equivalent to that obtained with the 12-port actuation.

The purpose of the high-lift applications was to provide insight into AFC flow mechanisms and to establish guidelines for port placement. The exercises in which fewer ports are used (Figures 11 and 13) demonstrate that it is possible to achieve meaningful lift increments using smaller AFC inputs. In this study there was no attempt to optimize AFC implementation for a given lift level. It is plausible that even smaller inputs to the flowfield could be achieved by regulating the jet intensity, frequency and signal phase at individual ports.

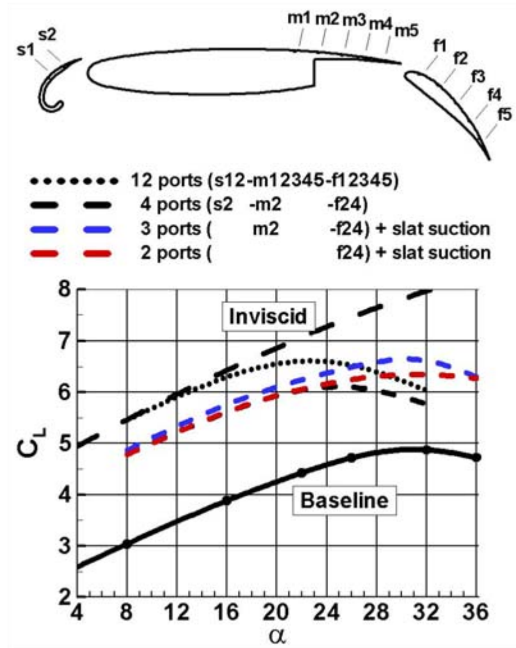

Fig. 13: AFC for advanced wing section (landing configuration, δ_{flap} = 50°)

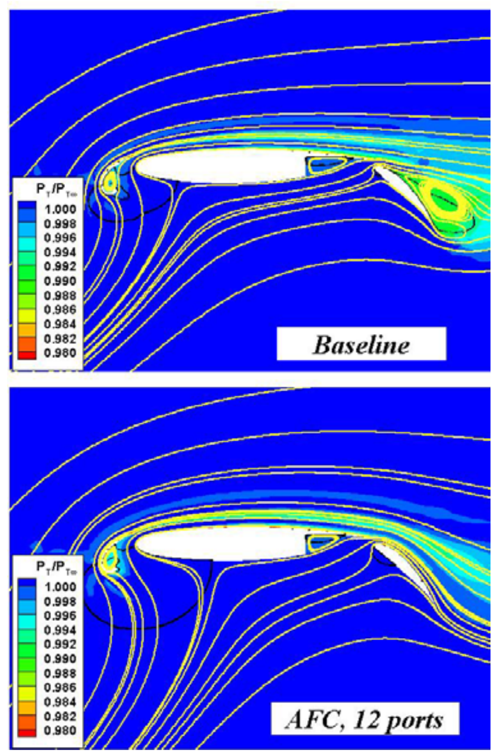

Fig. 14: Time-averaged total-pressure field showing impact of AFC on confluent viscous layers of the advanced wing section (time-averaged total-pressure, α = 22º)

6 Conclusions

AFC techniques have been computationally examined for a diverse set of practical aerodynamic problems. Flow control modes based on intermittent jet pulsation and on sprinkler jet motion were employed. Pulsed jet actuation is used to model either zero mass-flux devices or actuators which require fluidic sources (i.e., engine bleed).

In the engine vortex problem associated with ground operation, the pulsed jets and the sprinkler systems have resulted in the reduction of vortex ingestion and its concomitants, the risks of FOD and engine surge. Wide area coverage of actuation is a key attribute which makes the sprinkler system especially attractive for control of unsteady vortices.

The sprinkler jet has also been used for control of trailing wakes whereby flow actuation introduces time-varying perturbations into wing vortex elements in order to hasten vortex break-up. Depending upon the frequency of actuation, this actuation effectively disrupts the wake flow by either accelerating vortex decay or by encouraging instabilities along the vortex element. This effectively renders milder wakes and more benign conditions for following airplanes.

AFC for high-lift systems can potentially lead to performance enhancement over the practical range of angles of attack. Substantial lift increment can be realized by exploiting synergistic effects using actuation at individual wing elements. Depending on the flow conditions, full benefit of non-linear augmentation is realized when the actuation is simultaneously applied at select ports, producing lift levels approaching the inviscid limit. This has very important ramifications since there are high payoffs for vehicles that can operate from very short fields, from both economical and operational standpoints.

While the set of results from numerical simulations are very encouraging, we recognize that a practical implementation of the flow control concepts requires experimental validation at realistic operational conditions. Also, the range of optional implementations offered by AFC implies that a pragmatic approach to a practical design which encompasses other disciplines, such as structures, noise and systems integration, is indispensable.

References

[1] Sellers III W L, Jones G S and Moore M D. Flow control at NASA Langley in support of high-lift augmentation. AIAA Paper 2002-6006, 2002.

[2] Wygnanski I. The Variables affecting the control of separation by periodic excitation. AIAA Paper 2004-2505.

[3] Tilmann C P, Kimmel R L, Addington G A and Myatt J H. Flow control research and applications at the AFRL's Air Vehicles Directorate. AIAA Paper 2004-2622, 2004.

[4] Anders, S G, Sellers III W L, Washburn A E. Active flow control activities at NASA Langley. AIAA Paper 2004-2623, 2004.

[5] Kibens V and Bower W W. An overview of Active Flow Control applications at the Boeing Company. AIAA Paper 2004-2624, 2004.

[6] Buning P G, Chui I T, Obayashi S, Rizk Y M and Steger J L. Numerical simulation of the integrated space shuttle vehicle in ascent. AIAA Paper 1988-4359, 1988.

[7] Shmilovich A and Yadlin Y. Engine vortex flows and methods of ground vortex alleviation. *Proceedings of the 3rd International Conference on Vortex Flows and Vortex Models,* Yokohama, Japan, 2005.

[8] Johns C J. The aircraft engine inlet vortex problem. AIAA Paper 2002-5894, 2002.

[9] Smith D M and Dorris J. Aircraft engine apparatus with reduced inlet vortex. US Patent 6,129,309, 2002.

[10] Shmilovich A, Yadlin Y, Smith D M and Clark R W. Active system for wide area suppression of engine vortex. US Patent 6,763,651, 2004.

[11] Yadlin Y and Shmilovich A. Simulation of vortex flows for airplanes in ground operations. AIAA Paper 2006-0056, 2006.

[12] Crow S C. Stability theory for a pair of trailing vortices. *AIAA Journal*, December 1970, Vol. 8, No. 12, 2172-2179.

[13] Crow S C and Bate E R. Lifespan of trailing vortices in a turbulent atmosphere. *Journal of Aircraft*, 1976, Vol. 13, No. 7, 476-482.

[14] Crouch J D and Spalart P R. Active-control system for breakup of airplane trailing vortices. *AIAA Journal,* December 2001, Vol. 39, No. 12, 2374-2381.

[15] Greenblatt D. Managing flap vortices via separation control. *AIAA Journal*, November 2006, Vol. 44, No. 11, 2755-2764.

[16] Shmilovich A, Yadlin Y, Clark R W and Leopold D. Apparatus and method for the control of trailing wake flows. US Patent 7,100,875, 2006.

[17] Shmilovich A and Yadlin Y. Flow control of airplane trailing wakes. *Proceedings of the 4th International Conference on Vortex Flows and Vortex Models*, Daejeon, S. Korea, 2008.

[18] Shmilovich A and Yadlin Y. Flow Control for the Systematic Buildup of High Lift Systems. AIAA Paper 2006-2855, 2006.

[19] Khodadoust A and Shmilovich A. High Reynolds Number Simulations of Distributed Active Flow Control for a High-Lift System. AIAA Paper 2007-4423, 2007.

[20] Ying S X, Spaid F W, McGinley C B and Rumsey C L. Investigation of Confluent Boundary Layers in High-Lift Flows. AIAA Paper 1998-2622, 1998.

[21] Shmilovich A and Yadlin Y. Active Flow Control for Practical High-Lift Systems. AIAA Paper 2007-3971, 2007.

Copyright Statement