Forward Flight Effects on Counterflow Thrust Vector Control of a Supersonic Jet

F. S. Alvi*
Florida A&M University and Florida State University, Tallahassee, Florida 32310
and
P. J. Strykowski†
University of Minnesota, Minneapolis, Minnesota 55455

Introduction

THE benefits of active thrust vector control in high-performance fighter aircraft, such as improved agility and maneuverability, are well established. Unfortunately, conventional thrust vectoring schemes, which rely on mechanical means to vector the jet thrust, have some associated drawbacks. They generally require complicated hardware, which can add to the aircraft weight; the dynamic performance of such systems is less than ideal; and there are heat transfer problems due to the hot exhaust jet impinging on the vectoring surface. Although recent advances have been made in improving mechanical systems,[1] some of the mentioned issues remain a concern. In recent years, a fluidic-based thrust vectoring technique, counterflow thrust vector control (CFTVC), has been explored with promising results.[2–4] These investigations have convincingly demonstrated that CFTVC can be used to achieve single-axis pitch vectoring and multiaxis thrust vectoring for supersonic nozzles of various geometries and over a wide range of operating conditions. One of the primary advantages of CFTVC is the simplicity of hardware and excellent dynamic response. A comprehensive summary of these studies can be found in Refs. 2 and 4.

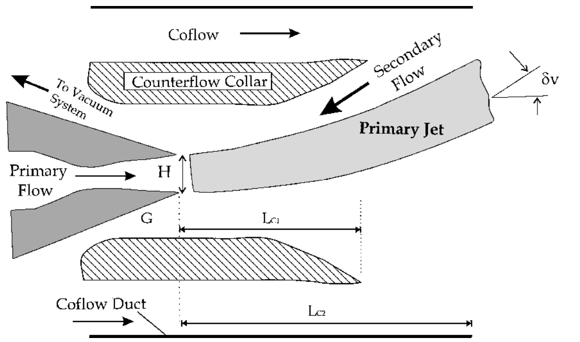
The ultimate goal of our research program is to develop a CFTVC system that can be implemented in aircraft propulsion systems of the future. To this end one needs to consider real flight effects, such as high jet exhaust temperatures and the influence of forward flight. Although high-temperature effects have been addressed to a limited extent in a previous investigation,[4] almost nothing is known about forward flight effects on CFTVC performance. In the present study, forward flight effects are simulated by generating a coflowing stream around the periphery of the CFTVC system consisting of a rectangular Mach 1.4 jet. The influence of the coflowing stream on the system performance in terms of jet vector angle response is described in this Note.

System Hardware and Facilities

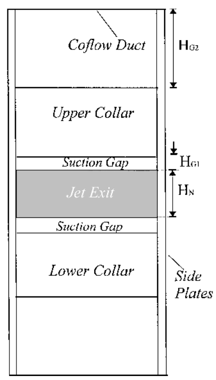
The experiments were conducted in the blowdown compressed-air facility of the Fluid Mechanics Research Laboratory, located at the Florida State University, capable of supplying hot and cold air to drive the primary jet; details of the facility may be found in Ref. 5. A schematic of the CFTVC system and the coflow hardware is shown in Fig. 1, where Fig. 1a shows the side view and Fig. 1b the end-on view of the hardware. A Mach 1.4 rectangular converging–diverging nozzle with an aspect ratio of 4 (height $H_N = 0.42$ in.) was used to generate the primary jet. The counterflow collar consists of curved surfaces placed downstream of the nozzle exit such that gaps are created in the nozzle exit plane between the collar surfaces and the upper and lower edges of the nozzle (Fig. 1b). Coflow was established by placing a rectangular duct around and extending downstream of the nozzle-collar geometry. The coflow duct was equipped with glass windows to allow optical access for flow visualization. The coflow duct and the upper and lower surfaces of the counterflow collar were also instrumented with rows of surface pressure taps along the duct and collar centerlines. Pertinent nominal dimensions of this geometry are provided in Table 1.

Table 1 Nozzle–collar dimensions

Dimension	Value, in. (mm)
Nozzle height H_N	0.42 (10.7)
Counterflow gap height H_{G1}	0.22 (5.6)
Counterflow collar length L_{C1}	1.5 (38.1)
Coflow gap height H_{G2}	0.5 (12.7)
Coflow collar length L_{C2}	2.1 (53.3)



a) Side view



b) End view

Fig. 1 Schematic of the counterflow thrust vector nozzle and collar geometry with the coflow duct (not to scale).

Jet thrust is vectored by establishing a counterflowing secondary airstream along the outer surface of one of the primary jet shear layers. For the present rectangular jet, this counterflowing stream is created by connecting a vacuum source to the counterflow gap on the upper or lower side, thus vectoring the jet in the upward or downward direction, respectively. For a more complete description of the CFTVC technique, the reader is referred to Alvi et al.[2] and Strykowski et al.[4] The jet stagnation pressure was fixed to produce isentropically expanded Mach 1.4 flow for a jet exiting into an ambient environment at atmospheric pressure. However, as the jet is vectored, a vacuum pressure is established in the nozzle exit plane due to the presence of the counterflowing stream and the collar surface, thus producing an underexpanded jet with the accompanying expansion–compression wave train. The CFTVC performance was evaluated at three different levels of coflow, Mach 0.3, 0.5, and 0.7, measured in the coflow duct. For this study, the primary jet and the coflow stream were at a nominal stagnation temperature of 300 K.

Received Jan. 5, 1998; revision received Oct. 2, 1998; accepted for publication Oct. 28, 1998.

*Assistant Professor, Fluid Mechanics Research Laboratory, Department of Mechanical Engineering. Senior Member AIAA.

†Professor, Department of Mechanical Engineering. Member AIAA.

Results

The system performance was qualitatively examined via schlieren photography; a representative example, shown in Fig. 2, corresponds to a moderate level of counterflow with Mach 0.3 coflow. In this image, the coflowing stream is evident by the presence of the braided structures visible in the upper shear layer generated by the upper coflowing stream as it exits the coflow duct into quiescent surroundings. As discussed in the preceding section, the schlieren image also shows the presence of the expansion and compression waves generated in the jet column due to the asymmetric pressure mismatch at the nozzle exit. Also visible in Fig. 2 is the array of surface pressure taps on the top surface of the coflow duct.

Previous studies that examined CFTVC systems without coflow for various nozzle geometries[2–4] have shown that, for a fixed CFTVC geometry, the jet deflection angle, δ_v in Fig. 1a, is proportional to the vacuum pressure established in the counterflowing stream. Similar behavior was observed in the present study, both qualitatively and from quantitative jet vector angles derived from surface pressure surveys. The vector angles were obtained through a control volume analysis, which uses the integrated pressure profiles on the counterflow collar and coflow duct surfaces. Past studies[2,4] have demonstrated that estimates of δ_v using this approach provide values consistent with those obtained using other techniques, such as direct force measurements and/or flow visualization.

A summary of the CFTVC performance for various counterflow and coflow levels is provided in Fig. 3, where δ_v is plotted as a function of counterflow level, represented by the pressure parameter $\Delta P_{\text{exit}} A_{\text{side}} / \rho U_{\text{exit}}^2 A_{\text{jet}}$. The pressure parameter is the leading-order term obtained from a control volume analysis of the nozzle–collar system; it is simply a nondimensional ratio of the side force acting on the jet, $\Delta P_{\text{exit}} A_{\text{side}}$, and the axial force imposed by the jet, $\rho U_{\text{exit}}^2 A_{\text{jet}}$. The pressure ΔP_{exit} is the vacuum pressure due to the secondary counterflowing stream as measured in the jet exit plane. All of the filled symbols correspond to the Mach 1.4 primary jet deflection angles at three levels of coflow (Fig. 3). For comparison, vectoring measurements previously obtained for a Mach 2 rectangular jet without coflow[4] are also shown as open circles.

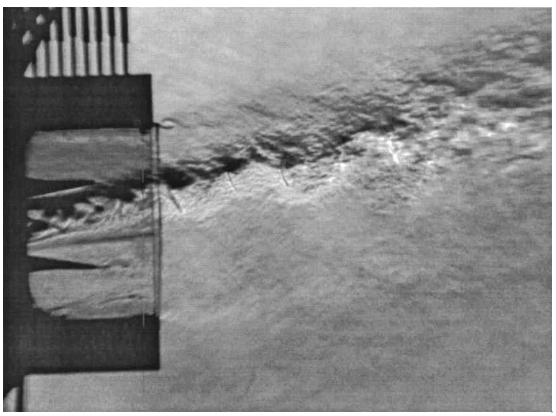


Fig. 2 Schlieren image of the vectored rectangular jet with moderate counterflow and Mach 0.3 coflow.

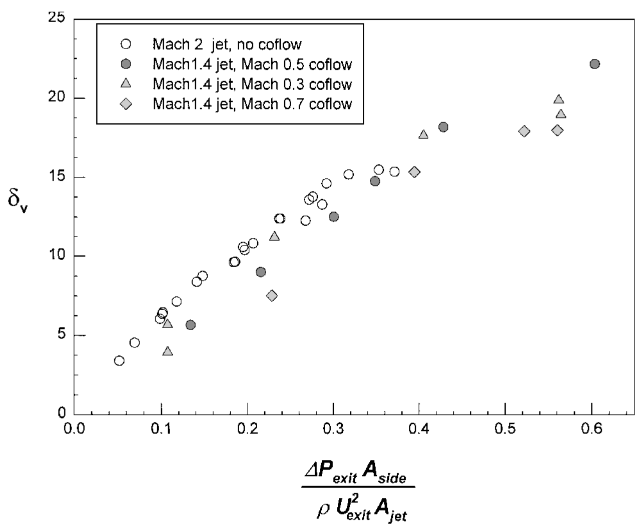


Fig. 3 Thrust vector performance for CFTVC system with coflow.

At first glance it appears that the performance of the CFTVC system with coflow is very similar to the behavior observed without coflow, in that the jet vector angle increases almost linearly with increasing vacuum pressures. However, a closer examination reveals that, albeit very small, there is a systematic degradation in the system performance due to coflow. For a given level of counterflow, the jet vector angle decreases as the coflow Mach number is increased. The reason for this can be appreciated if one considers the effect of vectoring in one direction, for example, up, on the flow regime on the opposite side, in this case the region below the curved primary jet. In the present tests as the jet is vectored up by applying counterflow along the upper collar surface, there is no flow through the lower gap. However, the lower shear layer of the primary jet and the upper shear layer of the lower coflowing stream, which originates at the end of the lower counterflow collar, both entrain fluid from the region bounded by these two shear layers. This creates a low-pressure zone, essentially a wake, on the lower side of the primary jet, which effectively reduces the side force on the jet, thus reducing the vectoring efficiency of the system. As the entrainment rates increase so does the vacuum pressure in this zone, further reducing the vectoring angles. Support for this argument comes from the trend observed in Fig. 3, as well as from pressure distributions obtained on the lower collar and coflow surfaces (not shown here). We believe that there are two primary mechanisms responsible for this behavior, as follows: First, for a fixed coflow Mach number, as δ_v increases, the vacuum pressure in the lower region decreases (considering the situation where the jet is vectored up) because of an increased distance between the lower shear layer of the primary jet and the lower collar surface. This would suggest that, as δ_v increases, CFTVC system performance should approach the efficiencies observed without coflow. This is the trend observed in Fig. 3. Second, as the Mach number of the coflowing stream increases so does the entrainment rate, a well-known behavior of shear layers, producing a higher vacuum pressure in the wake region. Therefore, as seen in Fig. 3, for a fixed pressure parameter the vectoring efficiency decreases with increasing coflow Mach number.

This degradation in CFTVC performance can be alleviated if atmospheric pressure is maintained in the regions that do not have a counterflowing stream. In principle, this can be achieved in an aircraft by ducting some of the freestream flow into the counterflow nozzle–collar system through the gaps (see Fig. 1) not being actively used to draw counterflow. Although the actual hardware used to accomplish this will depend on the overall vehicle configuration, we believe that the effect of coflow on CFTV performance can be minimized using this approach.

Conclusions

The influence of forward flight on counterflow thrust vectoring was examined. The CFTVC system for a Mach 1.4 rectangular jet was used, and forward flight effects were simulated by establishing a coflowing stream at the periphery of the CFTVC system. Results demonstrate that at all three coflow Mach numbers investigated the jet could be easily and continuously vectored to angles as high as 22 deg in the present system. The system response (in terms of jet vector angles) with coflow was nearly comparable to that without coflow, with a slight decline in performance with increasing coflow Mach number. Based on the entrainment characteristics of the primary jet and the coflowing jet shear layers, a possible mechanism responsible for this degradation is proposed and a solution has been suggested. In light of the well-established advantages of CFTVC,[2–4] i.e., simple hardware, robustness, and superior dynamic response, we believe that this study further demonstrates the many potential benefits of this technique for propulsion applications.

Acknowledgments

Support for this research provided by the Office of Naval Research and NASA is gratefully acknowledged. The authors would like to thank Donnell Washington, Martin Garcia, and E. Rajakuperan for their help in conducting these tests and A. Krothapalli for his advice and support.

References

[1]Snow, B. H., "Thrust Vectoring Control Concepts and Issues," Society of Automotive Engineers, TP Series 901848, Oct. 1990.

[2]Alvi, F. S., Strykowski, P. J., Washington, D. M., and Krothapalli, A., "Multiaxis Fluidic Thrust Vectoring of Supersonic Jet via Counterflow," AIAA Paper 97-0393, Jan. 1997.

[3]Washington, D. M., Alvi, F. S., Strykowski, P. J., and Krothapalli, A., "Multiaxis Fluidic Thrust Vector Control of a Supersonic Jet Using Counterflow," *AIAA Journal*, Vol. 34, No. 8, 1996, pp. 1734–1736.

[4]Strykowski, P. J., Krothapalli, A., and Forliti, D. J., "Counterflow Thrust Vectoring of Supersonic Jets," *AIAA Journal*, Vol. 34, No. 11, 1996, pp. 2306–2314.

[5]Wishart, D. P., "The Structure of a Heated Supersonic Jet Operating at Design and Off-Design Conditions," Ph.D. Dissertation, Dept. of Mechanical Engineering, Florida State Univ., Tallahassee, FL, April 1995.

D. S. McRae
Associate Editor

Streamwise Vortices in the Outer Layer of Wall Jets with Convex Curvature

P. Pajayakrit* and R. J. Kind†
Carleton University, Ottawa, Ontario K1S 5B6, Canada

Introduction

WALL jets appear where tangential blowing from a slot is used to delay or prevent flow separation in diffusers or on trailing-edge flaps. They also appear on circulation-control airfoils[1] where tangential blowing is used to move the upper-surface stagnation point around the well-rounded trailing edge to produce high values of lift coefficient. In circulation control and often in other applications, the wall jets have substantial convex curvature.

A definition sketch of a wall jet appears in Fig. 1. It can be seen that when there is convex curvature the radial gradient of angular momentum, $\partial(ur)/\partial r$, is negative in the outer layer ($y > y_m$), corresponding to centrifugal instability.[2] Such instability is well known to give rise to Goertler vortices[2] in conventional boundary layers with concave curvature. One of the authors has long suspected that similar streamwise vortices are present in the outer layer of wall jets on convex surfaces.

This suspicion was triggered by measurements of the spanwise distribution of maximum total pressure in the wall jet on the trailing edge of a circulation-control airfoil.[3] Typical data are shown in Fig. 2. The distributions were very nonuniform, with a spanwise distance between successive peaks about equal to the local thickness of the wall jet. Blowing slot thickness and slot-exit total pressure were uniform within 0.5% and 0.25%, respectively, and the trailing edge was smooth to the touch. The particularly large perturbation about 50 mm (2 in.) from midspan corresponds to the spanwise position of a streamlined spacer in the plenum upstream of the slot. There was a contraction ratio of 17 : 1 between the spacer and the slot exit; slot-exit conditions downstream of the spacer were thus very nearly the same as elsewhere. All attempts to eliminate or reduce the spanwise perturbations seen in Fig. 2 were unsuccessful. They were present when freestream velocity over the airfoil was both zero and finite. Very small flow perturbations present at the slot exit were clearly being amplified with distance downstream of the slot. Similar observations were made years later on another apparatus.[4] There is also mention of difficulties with spanwise nonuniformities in the work of Guitton[5] and co-workers. The aforementioned instability mechanism appears to be a plausible cause.

It is well known that the growth rate of wall jets on convex surfaces is much higher than that for plane wall jets; it is generally accepted that this is due to the enhanced turbulent mixing that results from the aforementioned centrifugal instability. The existence of relatively steady streamwise flow structures has, on the other hand, not previously been confirmed and indeed has not been mentioned as a possibility by most authors. If they exist, such vortices could be an important mechanism for radial transport of momentum flux, and they could thus have an important influence on flow development.

This Note reports measurements of crossflow velocity components that support the existence of streamwise vortices in the outer layer ($y > y_m$) of turbulent wall jets flowing over convex surfaces.

Apparatus and Experimental Methods

The measurements were done on a wall jet flowing over an existing 127-mm-diam circular cylinder in still air. The cylinder span was 495 mm, and end plates were present. The blowing slot height b was 0.95 mm. Velocity at the slot exit, U_J, was about 90 m/s, dictated by the available blower. The Reynolds number $U_J b/\nu$ was thus about 4700. The upper lip of the slot was ground square, with a thickness of 0.25 mm. A somewhat more detailed description of the cylinder is available in Ref. 4.

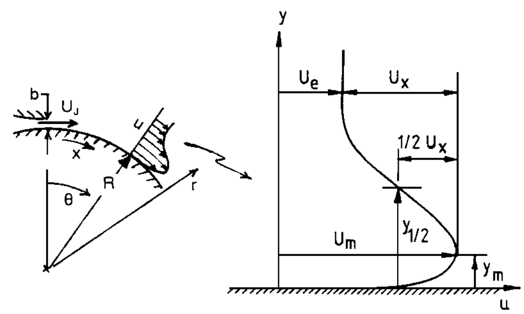


Fig. 1 Definition sketch of a wall jet.

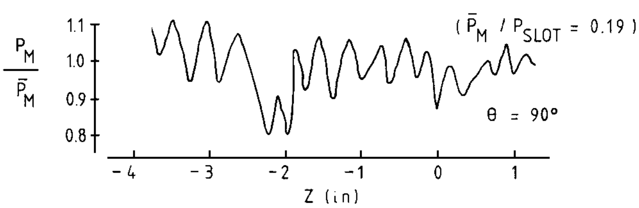


Fig. 2 Spanwise distribution of maximum total pressure P_M in the wall jet on the circular trailing edge of a circulation-control airfoil (still air, $R = 14.3$ mm, $b = 0.75$ mm, $U_J b/\nu = 3500$; $\bar{P}_M$ = spanwise average of P_M) (figure from Ref. 3).

Received July 25, 1998; revision received Oct. 16, 1998; accepted for publication Oct. 19, 1998.

*Graduate Student, Department of Mechanical and Aerospace Engineering; currently Wing Commander, Royal Thai Air Force, Air Technical Officer, First Aircraft Maintenance Division, Directorate of Aeronautical Engineering, Donmuang Air Force Base, Bangkok, Thailand. E-mail: palanunt@loxinfo.co.th.

†Professor, Department of Mechanical and Aerospace Engineering. E-mail: dkind@mae.carleton.ca. Associate Fellow AIAA.