Aircraft Thrust Vectoring Using Flexible Nonaxisymmetric Nozzles

Kevin W. Whitaker* and Noshir S. Gowadia†
University of Alabama, Tuscaloosa, Alabama 35487

Introduction

THE application of thrust vectoring techniques, while relatively common in rocket and missile propulsion, is in its infancy with air-breathing propulsion. Current design trends indicate that future fighter aircraft will rely heavily on two- and three-dimensional thrust vectoring engines to boost their maneuverability. New technology engines use postexit vanes or large movable surfaces to redirect engine exhaust to yield the desired thrust vectoring.[1-3] Although this method has proven to be effective, penalties must be paid. As typified by the recent U.S. Air Force STOL/SMTD program,[4] most thrust vectoring devices are heavy, primarily due to structural requirements involving the impinging exhaust flow. Control of the vectoring apparatus is complex and adds even more weight to the aircraft. Furthermore, the installation of typical thrust vectoring devices tends to mandate large clearance gaps to allow nozzle movement and allows little opportunity for aerodynamic fairing.

A novel concept of vectoring engine exhaust that addresses these concerns is certainly needed. One possible concept is where engine exhaust gases are not turned by some postexit flow deflection apparatus. Instead, a vehicle pitching moment is induced fluid dynamically by subtle asymmetric changes in the contour of the nozzle. With the advent of smart, flexible structures, it was hypothesized that flexible nozzles could be produced which would allow the nozzle wall contour to be altered in real-time by an onboard control system. The contour changes would produce a complex shock and expansion wave pattern in the exhaust flow, changing the pressure distribution on the nozzle walls. A small pitch variation will certainly occur due to the shock and expansion waves turning the velocity vector, but most of the vehicle pitch variation will come from pressure distribution (lift-force) changes on the walls of the nozzle. The concept is similar to liquid injection thrust vector control implemented on rockets.[5]

A feasibility study was undertaken by the authors to prove the validity of this novel concept. Of primary interest was the amount of wall modification necessary to achieve the desired thrust vectoring. A baseline nozzle was designed and then the influence modified wall contours had on thrust vectoring was investigated. A design goal of the study was to develop nozzles that produce ±20 deg of pitch vectoring (i.e., vehicle nose-up or nose-down). This range was selected as being representative of what can be achieved with current thrust vectoring systems.[4]

Procedure

The feasibility study undertaken consisted of two primary tasks. The first task was to design, as a baseline, a nonaxisymmetric nozzle possessing no thrust vectoring (i.e., a force angle of zero) with as high a gross thrust coefficient C_{fg} as possible. For this investigation, industry standard definitions[6] were used for all force coefficients and the force angle. The thrust coefficient C_f was defined using the nondimensional axial force F coincident with the nozzle centerline at the exit plane. The lift coefficient C_l used the nondimensional force L normal to the nozzle centerline. The force angle, defined as the angle the resultant thrust vector makes with the horizontal axis, is thus given by

$$\Theta = \tan^{-1}(C_l/C_f) = \tan^{-1}(L/F) \tag{1}$$

The gross thrust coefficient was defined as the ratio of the thrust coefficient to the ideal thrust coefficient. The ideal thrust coefficient is defined using the isentropic gross thrust

$$F_i = \omega_p \sqrt{RT_0\left(\frac{2\gamma}{\gamma-1}\right)\left[1-\left(\frac{p_\infty}{p_0}\right)^{(\gamma-1)/\gamma}\right]} \tag{2}$$

where ω_p is defined as the mass flow rate at the nozzle throat.

The nozzle type selected for study was an asymmetric free-expansion type with the upper wall shorter than the lower deck. The baseline nozzle had a nonuniform flowfield with the lip internal pressure matched to the external pressure for reduced external flow disturbance. Advantages of a free expansion nozzle (as compared to a nozzle with all internal expansion) are reduced plume detection from "look-up" threats due to shielding from the lower deck, greater low-frequency radar cross-section attenuation, and the gross thrust coefficient is less sensitive to nozzle pressure ratio.

The baseline nozzle developed for use in this study is shown in Fig. 1. Arbitrary length units were used and the origin of the coordinate system was placed at the end of the upper wall. The force angle of the baseline nozzle was −0.85 deg with a thrust coefficient of 0.988. In this study, a positive force angle corresponded to a vehicle nose-up pitching moment, and a negative force angle corresponded to a vehicle nose-down pitching moment. The upper and lower wall local pressure coefficient C_p distributions for the baseline nozzle also can be seen in Fig. 1. It was assumed that on-design operating conditions for the nozzle would be a nozzle pressure ratio (NPR) of 7, a flight Mach number of 1.60, and a fluid specific heat ratio of 1.35. It should be noted that the flight Mach number along with the NPR are used in the calculation of C_p.

The second task of the study was to analyze the performance of the baseline nozzle when the wall contours were subtly altered. A variety of cases were studied in order to determine how changes in certain wall regions influenced vec-

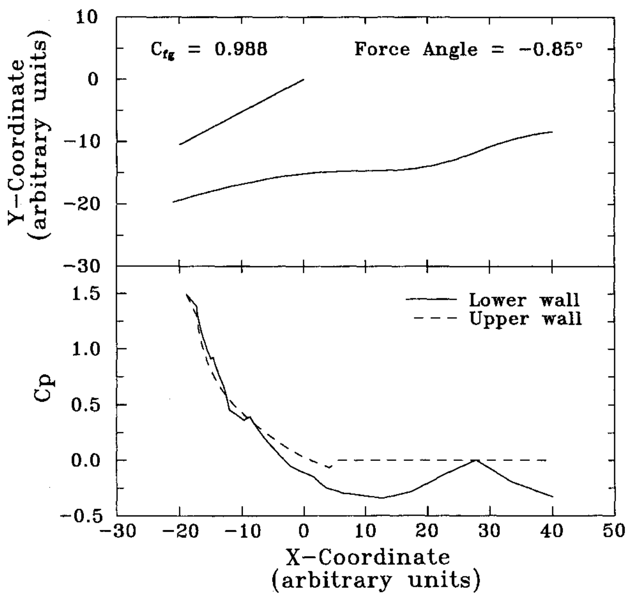

Fig. 1 Baseline nozzle geometry.

Received July 13, 1992; revision received May 27, 1993; accepted for publication May 27, 1993.

*Associate Professor, Department of Aerospace Engineering. Senior Member AIAA.

†Senior Research Engineer, Department of Aerospace Engineering. Member AIAA.

toring. It was important not only to investigate modifying one wall, but also the interaction resulting from modifying two walls simultaneously. To this end, a systematic investigation procedure was developed where the nozzle walls were divided into regions, and then each wall region was modified separately, making sure no discontinuities existed in the entire wall contour.

The flowfield solver used in this study was a computer program based on the traditional inverse method of characteristics. This code allows for the analysis of supersonic flowfields internal to a nozzle as well as the supersonic exhaust plume. The code has been extensively validated with experimental data from NASA and industry, and has a unique user-friendly interface which allowed many nozzle geometries to be analyzed quickly.[7] To initiate a solution, the code requires the nozzle on-design conditions and an initial value line located slightly downstream of the nozzle throat. The initial value line for this study used a Mach number distribution of 1.05 with all flow starting in the streamwise direction.

Results

A total of 150 modified nozzle geometries were investigated. Many modified nozzle contours produced results that were unremarkable in the sense that they produced little change in the thrust vector angle, and typically only caused a degradation in nozzle performance. These nozzles will not be discussed further. Also, some geometries did not allow complete flowfield solutions due to the formation of shocks. The method of characteristics program used in this study requires that the flowfield be supersonic, and if a shock caused subsonic flow to occur, the program would stop calculating. These cases also will not be discussed further, but they are important since they dictated the limits to which a contour could be changed.

Certain nozzle geometries did reveal interesting results, however, and the best overall performing nozzles for nose-up and nose-down vectoring were selected for complete presentation. "Best" was determined by the magnitude of the resultant force angle for both nose-up and nose-down vehicle pitching moments. All modified nozzle geometries which yielded a complete flowfield solution produced some degree of nose-down pitching moment. A nose-up pitching moment was found to be more difficult to obtain, a feature attributed to the baseline nozzle configuration selected for study.

The best nose-up (or positive Θ) vectoring nozzle was produced by altering the lower wall in the region centered around $X = 0.0$, as shown in Fig. 2. This location seemed to be well suited for allowing the shock and expansion wave pattern to establish itself and propagate downstream. Looking at the pressure coefficient distribution on the lower wall it can be seen how the nose-up pitching moment is derived. In the region from $10 \leq X \leq 25$ there is a positive C_p and this pressure distribution causes a net downward force on the nozzle. Nose-down (or negative Θ) vectoring was obtained by modifying the contour of the entire upper wall. As can be seen in Fig. 3, this modification caused the C_p distribution on the lower wall to be almost entirely negative, which resulted in a net upward force on the nozzle.

In both cases C_{fg} decreased, signifying a drop in nozzle performance. (This was also the trend exhibited in all other nozzles studied.) This is obviously due to the total pressure losses incurred when the flow crosses the oblique shocks generated in the plume. However, thrust vectoring is a transient operation, and a momentary drop in nozzle performance can be traded for increased vehicle maneuverability.

An interesting characteristic of both nozzles shown is that thrust vectoring was achieved with very slight changes in wall geometry. In fact, in observing the wall contours shown in Figs. 2 and 3, it is very difficult to distinguish between the baseline and modification. However as hypothesized, the performance of the nozzle was significantly altered by these subtle changes. This is an important design concern indicating that the resulting plume flowfield is very sensitive to wall contour modifications in the regions indicated.

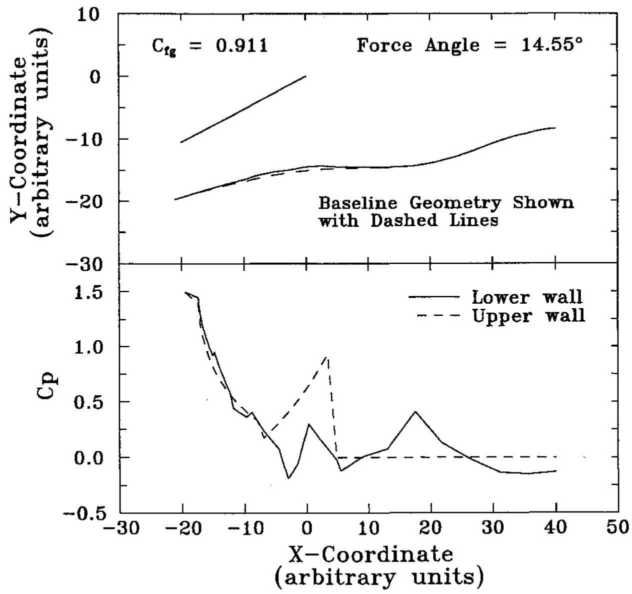

Fig. 2 Best vehicle nose-up nozzle geometry.

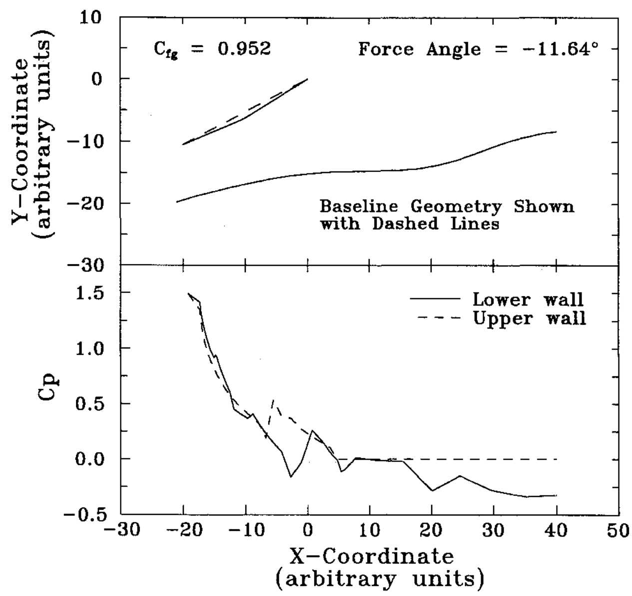

Fig. 3 Best vehicle nose-down nozzle geometry.

Conclusions

This feasibility study has shown that it is indeed possible for thrust vectoring to be achieved by altering the contour of the exhaust nozzle walls. Although the initial design goal of ± 20 deg of pitch vectoring was not achieved, a very respectable range of approximately -12 to $+15$ deg was obtained. The authors feel that with further detailed study utilizing inverse design techniques, this envelope could be expanded out to ± 20 deg. The fact that thrust vectoring is possible via this technique suggests that significant weight savings could be achieved utilizing this flexible nozzle wall approach with smart materials instead of some postexit flow deflection apparatus.

An important conclusion drawn from this study is that in selecting a wall region for modification, a subtle change upstream has much more of a pronounced effect than a large change near the end of the nozzle. This is primarily due to the reflection of shock and expansion waves emanating from an upstream modification. The reflections ensure that a subtle modification has an affect over a large portion of the plume. Therefore, when effecting thrust vectoring in this manner, a designer needs to make full use of wave reflections to achieve the design goal.

This study has certainly suggested areas for further investigation. First, there is the optimization issue. No attempt was made to find the optimum contour necessary to produce a required thrust vector angle. Obviously, the most efficient implementation of this technique would insure the largest thrust vector angle for the smallest wall contour modification. A study needs to be conducted into optimizing nozzle contours, initially for specified force angles, and then subsequently for entire force angle envelopes.

A second area where further investigation is needed is based on the fact that only a two-dimensional analysis was performed. A three-dimensional study must be performed to identify any side wall affects on pitch vectoring. Also, yaw vectoring could be accomplished in a very similar manner by flexing the side walls. A very high performance nozzle with both pitch and yaw vectoring could possibly be designed.

References

[1]Yetter, J. A., and Leavitt, L. D., "Effects of Sidewall Geometry on the Installed Performance of Non-Axisymmetric Convergent-Divergent Exhaust Nozzles," NASA TP-1771, Dec. 1980.

[2]Berrier, B. L., and Mason, M. L., "Static Performance of an Axisymmetric Nozzle with Post-Exit Vanes for Multi-Axis Thrust Vectoring," NASA TP-2800, May 1988.

[3]Berrier, B. L., and Re, R. J., "Effect of Several Geometric Parameters on the Static Performance of Three Non-Axisymmetric Nozzle Concepts," NASA TP-1468, July 1979.

[4]Dickinson, R., and Bursey, R., "Flight Test Results of the F-15 SMTD Thrust Vectoring/Thrust Reversing Exhaust Nozzle," AIAA Paper 90-1906, July 1990.

[5]Darwell, H. M., and Truebridge, F. P., "Design of Rocket Nozzles to Reduce Gas Misalignment," *Journal of Spacecraft and Rockets*, Vol. 5, No. 1, 1968, pp. 36–41.

[6]Re, R. J., and Leavitt, L. D., "Static Internal Performance of Single-Expansion-Ramp Nozzles with Various Combinations of Internal Geometric Parameters," NASA TM-86270, Dec. 1984.

[7]Whitaker, K. W., and Cates, J. E., "A User-Friendly Exhaust Nozzle Design Program Based on the Method of Characteristics," AIAA Paper 90-2029, July 1990.

Effect of Sidewall Boundary Layer on Transonic Flow over a Wing

William E. Milholen II* and Ndaona Chokani†
North Carolina State University,
Raleigh, North Carolina 27695

Introduction

THE interaction between a sidewall boundary layer and the flow over a wing occurs in many practical applications of interest to a fluid dynamics engineer.[1] Of particular interest in this study is the juncture flow encountered in a wind-tunnel test of a sidewall-mounted wing at transonic speeds. Under such flow conditions, the sidewall boundary layer may significantly affect the quality of the data through strong nonlinear interactions.[2] The purpose of this investigation was to numerically model the interactions between a sidewall boundary layer and the flow over a low aspect ratio wing mounted on the sidewall. In order to meet this objective, a recently developed computational code for solving the three-dimensional Navier-Stokes equations was modified to include the presence of the sidewall boundary layer. The influence of the sidewall boundary layer was determined by making direct comparison with experimental data, and with previous free-air computations.[3]

Presented as Paper 92-4036 at the AIAA 17th Aerospace Ground Testing Conference, Nashville, TN, July 6–8, 1992; received Oct. 13, 1992; revision received May 27, 1993; accepted for publication May 27, 1993.

*Graduate Research Assistant. Student Member AIAA.

†Associate Professor. Member AIAA.

Numerical Procedure

The computational code, TLNS3D, which solves the time dependent, thin-layer Navier-Stokes equations for a body-fitted coordinate system was used in this study.[4] An explicit multistage Runge-Kutta time stepping scheme, which is second-order accurate, was employed to advance the solution to steady state. Closure of the governing set of equations was accomplished with the equilibrium turbulence model of Baldwin and Lomax.[5] The turbulence model was modified to account for the proximity of the wing and the sidewall following Ref. 2. For the present research, the boundary conditions at the far-field upstream boundary and the root plane were modified to simulate the sidewall boundary-layer flow.[6] These are termed here "viscous sidewall" calculations. For comparative purposes, calculations were also conducted with the root plane modeled as a symmetry plane; these are termed "free-air" computations. Fuller details of the modifications to the computational code, the grid generation procedure, and the grid refinement are presented in Ref. 6.

Results and Discussion

The planform of the wing model is similar to that of the canard on the X-29 experimental research aircraft. The experimental testing is described in detail by Chu and Lawing.[7] The root section of the model was offset 1.25 cm from the tunnel sidewall by the use of a fillet, in order to minimize the influence of the sidewall boundary layer on the model. Chordwise surface pressure data were obtained at three spanwise locations, which will be used below for validation of the computational results. Since the data has yet to be released for general publication, the test Reynolds number used in this study is referred to as medium.

Three test cases were investigated.[6] For the sake of brevity, only the moderate wing loading case, $M_\infty = 0.8860$, $\alpha = 5.46$ deg, is presented. Figure 1 shows a comparison between the experimental pressure distribution and two computational results. The result obtained with the viscous sidewall modeling is denoted by VSW, while FA denotes the free-air computation. At all three stations, the viscous sidewall computation is in excellent agreement with the data. The suction peaks in the leading-edge region are well predicted, along with their associated adverse pressure gradients. At the two inboard stations, the presence of the sidewall boundary layer has reduced the predicted shock wave strength. The pressure recovery downstream of the shock wave is well captured. At the outboard station, the viscous sidewall computation shows significant improvement over the free-air result, with the appearance of an oblique shock wave in the leading-edge region. A comparison of the computational results on the fillet revealed that the viscous sidewall computation predicted higher pressures, which are attributed to the low momentum of the sidewall boundary layer.[8] These higher pressures on the inboard portion of the wing effectively decrease the favorable spanwise pressure gradient.

Figure 2 compares the upper surface pressure contours for the two computational results. The viscous sidewall computation, Fig. 2a, clearly shows that the sidewall boundary has