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

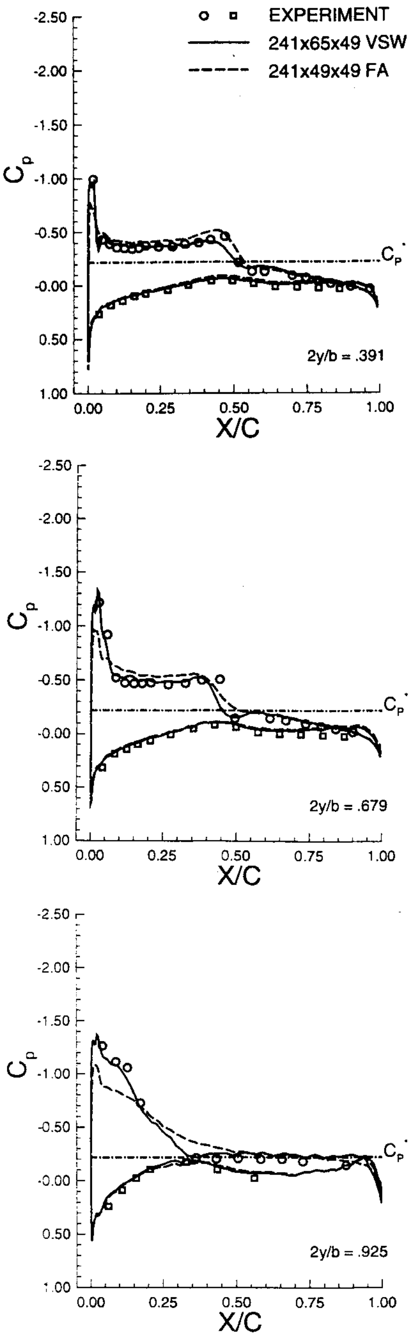

Fig. 1 Influence of sidewall boundary layer on wing surface pressure distribution (M_∞ = 0.8860, α = 5.46 deg).

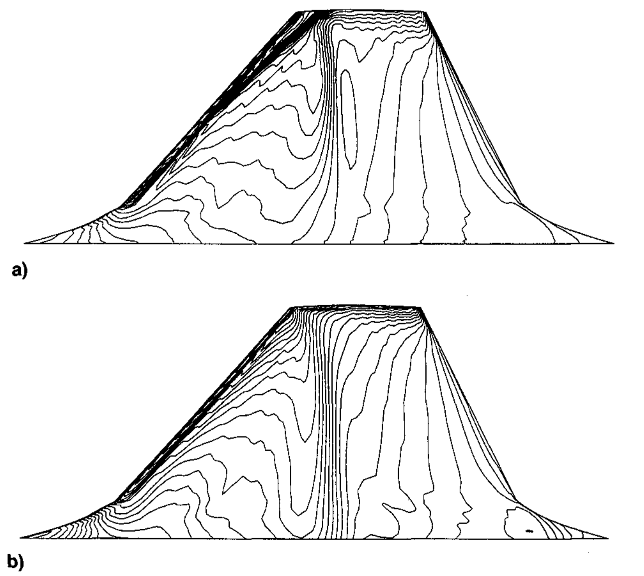

Fig. 2 Influence of sidewall boundary layer on computed surface pressure contours (M_∞ = 0.8860, α = 5.46 deg): a) viscous sidewall computation and b) free-air computation.

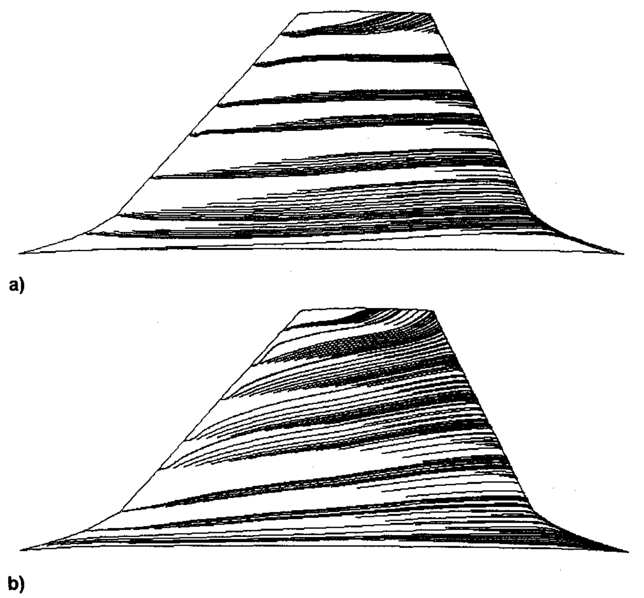

Fig. 3 Influence of sidewall boundary layer on wing streamline pattern (M_∞ = 0.8860, α = 5.46 deg): a) viscous sidewall computation and b) free-air computation.

reduced the shock wave strength on the inboard portion of the wing. This is in contrast to the free-air result, Fig. 2b, where the predicted shock wave has a uniform strength in the root region. The oblique shock wave predicted by the viscous sidewall computation is clearly seen, and merges with the normal shock wave in the wingtip region. The influence of the sidewall boundary layer on the flow over the wing was examined in detail from velocity profiles and upper surface streamline patterns.[8] Only the surface streamline plots will be presented here. The predicted surface streamline patterns for both cases are shown in Fig. 3. Over the inboard portion of the wing, the viscous sidewall computation, Fig. 3a, shows that the flow is predominantly in the streamwise direction. This is in sharp contrast to the free-air result, Fig. 3b, which shows an appreciable spanwise flow over the entire wing. As discussed above, the presence of the sidewall boundary layer decreases the favorable spanwise pressure gradient, and hence, the spanwise flow over the wing. The viscous sidewall computation predicted that the sidewall boundary layer remained attached.

Figure 4 compares the computed streamline patterns on the root plane for both computations. The free-air computation, Fig. 4b, predicts that the streamlines flow around the root section of the wing in a smooth manner. In sharp contrast, the viscous sidewall computation, Fig. 4a, predicts that the streamlines are skewed in the root plane. The viscous sidewall computation predicted weak juncture vortices in both the upper and lower surface juncture regions for all of the test cases investigated. The weak juncture vortices induce a downwash on the root plane, which accounts for the observed skewing of the streamlines.[6] Since the sidewall boundary layer did not separate, these vortices are not termed horseshoe vortices. The primary vortex generation mechanism is believed to be lateral skewing of the approaching streamlines in the sidewall boundary layer.

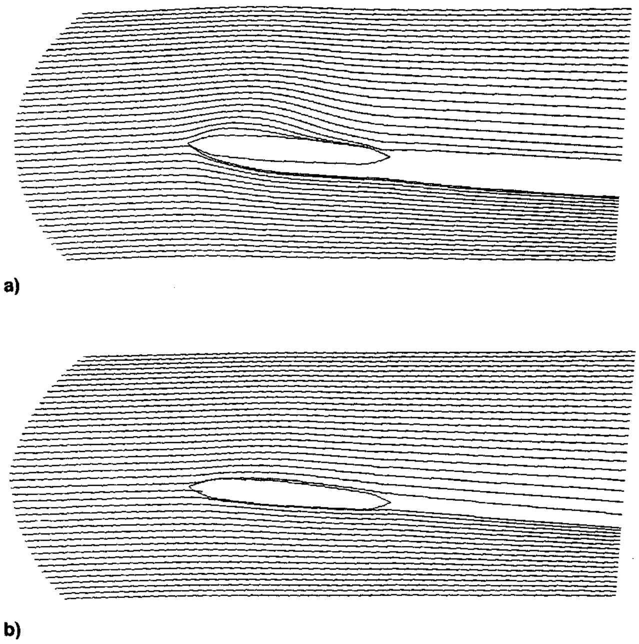

Fig. 4 Influence of sidewall boundary layer on root plane streamline pattern (M_∞ = 0.8860, α = 8.24 deg): a) viscous sidewall computation and b) free-air computation.

Conclusions

A numerical investigation of the interaction between a wind-tunnel sidewall boundary layer and a thin low-aspect-ratio wing has been performed for transonic speeds and flight Reynolds numbers. A three-dimensional Navier-Stokes code was applied to calculate the flowfields. The results indicated that the sidewall boundary layer had a strong influence on the flowfield around the wing. The low momentum of the sidewall boundary layer resulted in higher pressures in the juncture region, which decreased the favorable spanwise pressure gradient. This significantly decreased the spanwise migration of the wing boundary layer. The computations which modeled the sidewall boundary layer were found to be in better agreement with experimental data.

Acknowledgments

This work was supported by Cooperative Agreement NCC1-98 between North Carolina State University and the High Reynolds Number Aerodynamics Branch at NASA Langley Research Center. The authors acknowledge many helpful and insightful discussions with Veer N. Vatsa and Bruce W. Wedan regarding the use of the Navier-Stokes solver and grid generation code. The authors thank Pierce Lawing and Julio Chu for providing the experimental data. The authors are grateful to L. Elwood Putnam and Blair B. Gloss for their support of this work. The computations were conducted on Cray-YMP supercomputers at the North Carolina Supercomputing Center, and at NASA Langley Research Center.

References

[1]Shabaka, I. M. M. A., and Bradshaw, P., "Turbulent Flow Measurements in an Idealized Wing/Body Junction," *AIAA Journal*, Vol. 19, No. 2, 1981, pp. 131, 132.

[2]Vatsa, V. N., and Wedan, B. W., "Navier-Stokes Solutions for Transonic Flow over a Wing Mounted in a Tunnel," AIAA Paper 88-0102, Jan. 1988.

[3]Milholen, W. E., II, Chokani, N., Lawing, P. L., and Chu, J., "Evaluation of a Three-Dimensional Navier-Stokes Solver for Transonic Flow over a Low Aspect Ratio Wing Mounted in a Wind Tunnel," AIAA Paper 91-1725, June 1991.

[4]Vatsa, V. N., and Wedan, B. W., "Development of an Efficient Multigrid Code for 3-D Navier-Stokes Equations," AIAA Paper 89-1791, June 1989.

[5]Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[6]Milholen, W. E., II, and Chokani, N., "Numerical Modeling of Transonic Juncture Flow," AIAA Paper 92-4036, July 1992.

[7]Chu, J., and Lawing, P. L., "High-Reynolds-Number Test of a 5-Percent-Thick Low-Aspect-Ratio Semispan Wing in the Langley 0.3-Meter Transonic Cryogenic Tunnel-Wing Pressure Distributions," NASA TM-4227, Dec. 1990.

[8]Milholen, W. E., II, "Numerical Modeling of Transonic Juncture Flow," M.S. Thesis, Mechanical and Aerospace Engineering Dept., North Carolina State Univ., Raleigh, NC, June 1992.

Experimental Investigation of Vortex Flaps on Thick Delta Wings

D. Levin* and A. Seginer†
Technion—Israel Institute of Technology, Haifa 32000, Israel

Introduction

MOST modern fighter airplanes and missiles use the "so called" nonlinear-lift. Swept sharp leading edges generate additional lift. The leading-edge separation rolls up the leading-edge vortex and rotates the leading-edge suction that normally points upstream, to a direction normal to the wing surface. However, the increased normal force adds a significant drag component.[1]

Vortex flaps have been introduced as a means to reduce the additional drag. It was postulated that a downward deflection of the leading edge to parallel the freestream direction will redirect the vector of the vortex suction.[2] The postulation was found in this work to be oversimplified. The deflected flap created cambered profiles of smaller geometrical and effective angles of attack, and transformed the flat delta wing into a cambered wing with a dihedral. The flowfield also became more complicated than that of a flat delta wing, with two additional vortices generated at the flap hinges.[3] These changes in the flowfield reduced the leading-edge suction, and resulted in both the lift and the drag forces being smaller. However, the aerodynamic efficiency of the wing, defined by the lift/drag ratio, has improved on occasion. This improvement has been the main goal of several works.[4–14] The aerodynamic coefficients of configurations with vortex flaps were measured experimentally,[3,15–17] and predicted theoretically,[18,19] Most of these works dealt with very thin wings. Although Rao[10] reports on the adverse effect of the thickness on the performance of vortex flaps, the current investigation is of delta wings with substantial thickness. This thickness, being more realistic, also made possible the design of an actual hinge geometry, rather than the virtual hinge used in prior experiments, as well as the observation of the effect of the leading-edge shape.

Received April 5, 1993; revision received April 21, 1993; accepted for publication June 23, 1993.

*Senior Research Associate, Faculty of Aerospace Engineering. Member AIAA.

†Professor, Faculty of Aerospace Engineering. Associate Fellow AIAA.