JOURNAL OF AIRCRAFT
Vol. 30, No. 5, Sept.–Oct. 1993

Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Transonic Navier-Stokes Flow Computations over Wing-Fuselage Geometries

S. Agrawal*
McDonnell Douglas Corporation, St. Louis, Missouri 63166
V. N. Vatsa†
NASA Langley Research Center, Hampton, Virginia 23665
and
T. A. Kinard‡
Lockheed Corporation, Marietta, Georgia 30063

Introduction

ALTHOUGH significant progress has been made in recent years toward the computation of transonic viscous flows over aircraft configurations,[1,2] most numerical schemes still require prohibitive amounts of computer time to obtain accurate solutions for the Navier-Stokes equations, even on isolated components. Such computer time requirement makes it difficult to obtain grid-converged solutions, and therefore, numerical accuracy cannot be easily assessed. The multigrid-based Navier-Stokes solver recently developed by Vatsa and Wedan,[3] known as TLNS3D, appears to overcome such shortcomings to a large degree. For transonic flows over isolated wings, grid-converged solutions that correlate well with experimental data have been obtained with TLNS3D in approximately 2 h on a Cray-2 computer. Due to its efficiency, accuracy, and robustness, this code is in routine use in the aircraft industry to aid the wing design process. In the present study, the numerical algorithm TLNS3D is employed to solve the flowfields over a transport wing-fuselage configuration (RAE[4]) in transonic flow. Extremely fine grids, containing over 1.52 million grid points are used to minimize the truncation errors in the numerical solutions. The solutions are shown to be grid converged. Computed pressure distributions on both the wing and fuselage are compared with the experimental data.

Numerical Scheme

The thin-layer Navier-Stokes equations specialized to a body-fitted coordinate system are used for representing high Reynolds number viscous flow over aerodynamic configurations of interest. A finite-volume scheme is used for spatial discretization of the governing equations in conjunction with a Runge-Kutta type of pseudo-time-stepping scheme for integrating the equations to a steady state.[5] A controlled amount of artificial dissipation is added to the governing equations in order to suppress odd-even decoupling, and to prevent oscillations in the vicinity of shock waves and stagnation points. The Baldwin-Lomax turbulence model is used for closure.

In the present scheme, two different types of artificial dissipation models are employed. The first one is patterned after the work of Jameson et al.,[5] where a combination of second- and fourth-difference terms is used to construct the dissipative terms. The modifications for high aspect ratio cells[3,6] are also applied, and the resulting dissipation model is designated as the scalar dissipation model (SDM) in this study. SDM has been shown to produce good convergence properties and accurate solutions for transonic flows over isolated wings, provided fine mesh densities are used.[3] Recently, Turkel and Vatsa[7] have developed a dissipation model by using concepts from upwind-difference schemes, which lowers the artificial dissipation levels by individual scaling of the dissipation contribution to each equation. In essence, the coefficients used in the scalar dissipation model are replaced by the modulus of the flux Jacobian matrices, and this model is known as the matrix dissipation model (MDM). Although, the computational cost of the MDM is somewhat higher compared to the SDM, it has been shown to result in net savings in computational cost due to the improved accuracy of the resulting solutions.[7]

Grid Topology

In contrast with the *C-H* grids used by several researchers[2,8] for solving viscous flows over wing-fuselage geometries, a *C-O* grid topology is used in the present investigation. With *C-O* topology, the grid is clustered in only the normal direction, thereby requiring a smaller number of overall grid points. The viscous layer developing on both the wing and fuselage surfaces, including the juncture region, can be calculated with a standard, single-surface turbulence model without the need for any more complexity in turbulence modeling, because both of these surfaces form the family of J = constant lines. The *C-O* topology is also well-suited for resolving the flowfields in the vicinity of the wing tip.

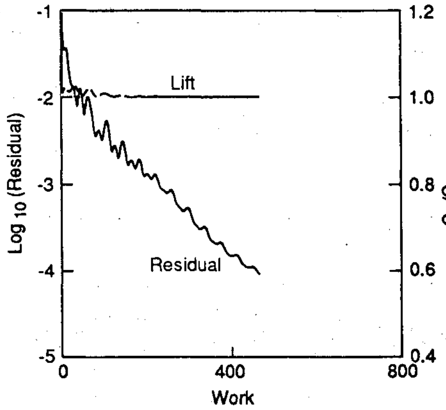


Fig. 1 Convergence histories for residual and lift (grid: 289 × 65 × 81), MDM. M_∞ = 0.8, α = 2 deg, Re = 3 million.

Received April 20, 1992; revision received July 15, 1992; accepted for publication Oct. 30, 1992.

*Technical Specialist, Aerodynamics, CFD Project; currently at McDonnell Douglas Aerospace, Long Beach, CA 90846. Associate Fellow AIAA.

†Senior Scientist, Computational Aerodynamics Branch. Member AIAA.

‡Senior Engineer, Aerodynamics. Member AIAA.

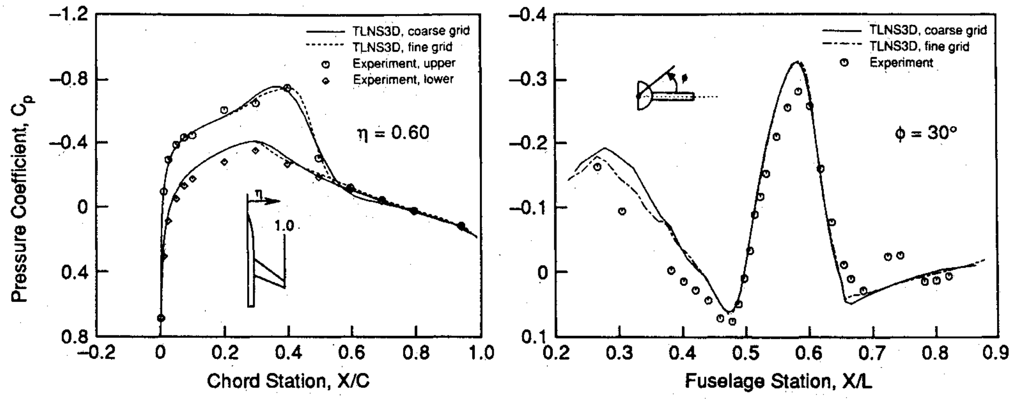


Fig. 2 Effect of grid refinement on wing and fuselage pressure distributions using SDM. $M_\infty = 0.9$, $\alpha = 1$ deg, $Re = 3$ million.

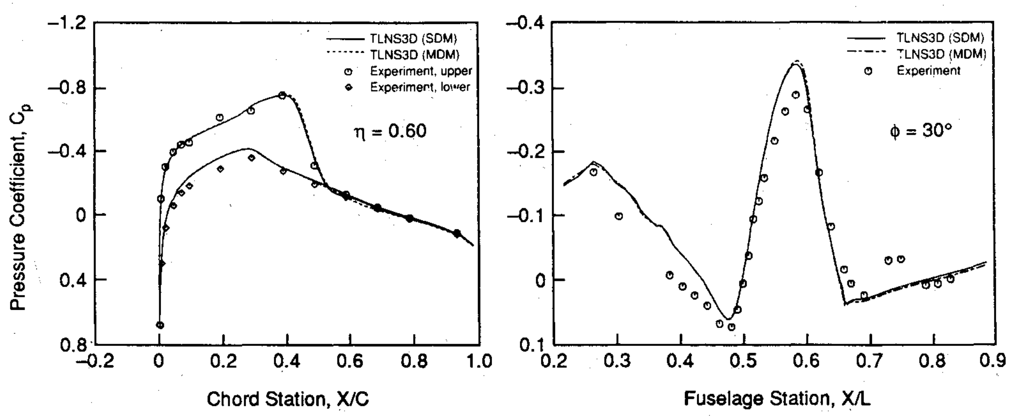


Fig. 3 Effect of dissipation model on wing and fuselage pressure distributions for fine grid (289 × 65 × 81). $M_\infty = 0.9$, $\alpha = 1$ deg, $Re = 3$ million.

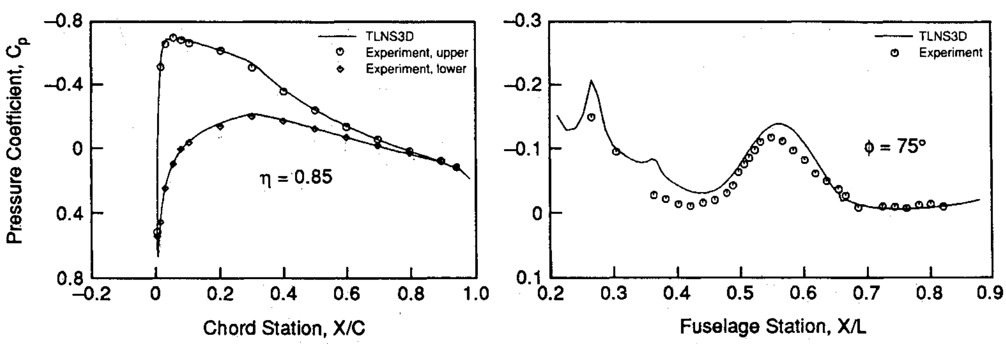


Fig. 4 Wing and fuselage pressure distributions for fine grid (289 × 65 × 81), MDM. $M_\infty = 0.8$, $\alpha = 2$ deg, $Re = 3$ million.

The computational grid used in this study had the overall dimensions of 289 (streamwise), 65 (normal), and 81 (spanwise), referred to as the fine grid herein. A coarser grid was generated by eliminating every other point from the fine grid in the tangential directions. The grid spacing in the normal direction off the surface of the wing was of the order 10^{-5} times the chord length.

Results

The TLNS3D computed solutions for the RAE configuration are presented in Figs. 1–4. Approximately 400 multigrid iterations were required for solution convergence. The average residuals were reduced by at least four orders of magnitude. Figure 1 shows typical convergence histories for residual and lift, using MDM. Approximately 8 h of CPU time were required to obtain converged solutions on the fine grid (about 2.5 h on the coarse grid) on the NAS Cray-2 computer for each test case.

Computations were performed for two transonic attached flow conditions. These conditions were $M_\infty = 0.9$, $\alpha = 1$ deg, and $M_\infty = 0.8$, $\alpha = 2$ deg. The Reynolds number based on the semispan was 3 million. Figure 2 shows the results obtained with the fine and the coarse grids, for $M_\infty = 0.9$, $\alpha = 1$ deg. Typical pressure distributions at one semispan station (η) on the wing, and at one angular location (ϕ) on the fuselage are compared with the experimental data. In general, better shock location and strength on the wing, and better solutions near the fuselage nose, are obtained with the fine grid.

Since only two grid densities are used, an obvious question arises as to whether the fine grid solutions are grid converged. To answer such a question, computations were also performed with MDM, which has been demonstrated to produce a similar improvement as that obtained by grid refinement with SDM.[7]

Figure 3 shows the results obtained on the fine grid with the two dissipation models. The effect of reducing artificial dissipation levels by means of MDM is seen to be minimal on both the wing and the fuselage solutions, indicating that the solutions are nearly grid-converged, on the fine grid employed.

Figure 4 shows the solutions with MDM for $M_\infty = 0.80$, $\alpha = 2$ deg, on only the fine grid. Overall comparison with the experimental data is again found to be very good on the wing surface. The pressure distributions compare quite well with experimental data, except near the nose and the wing-fuselage juncture. There are perhaps two reasons: 1) grid skewness near the nose of the fuselage, and 2) inadequacy of the simple algebraic turbulence model (Baldwin-Lomax) for predicting complex flows that are encountered in the wing-fuselage juncture region. Further details can be found in Ref. 9.

Summary and Conclusions

Transonic Navier-Stokes solutions were obtained for a transport (RAE) wing-fuselage configuration. A *C-O* grid topology was used since it has the advantage of requiring strong clustering one coordinate direction for resolving the boundary layers developing on both wing and fuselage surfaces. Extremely fine grid density (1.52 million nodes) was employed to obtain accurate numerical solutions, and the resulting pressure distributions compared well with experimental data. Solution convergence for each case was found to be very fast, the CPU time being approximately 8 h.

A grid refinement study was also conducted to assess the effect of artificial dissipation models (SDM and MDM), and truncation errors on the numerical solutions. Based on this study, it was concluded that the fine grid solutions were grid-converged. Computed results using MDM on the fine grid compared best with the experimental data.

Acknowledgments

This study was supported by the McDonnell Douglas Independent Research and Development program. Computer resources for this work were provided by the Numerical Aerodynamic Simulation (NAS) facility at the NASA Ames Research Center. The authors are thankful to Timothy Gatzke for assisting in the grid generation, and Francois Creasman for assisting in plotting results.

References

[1]Flores, J., Reznick, S. G., Holst, T. L., and Gundy, K., "Transonic Navier-Stokes Solutions for a Fighter-Like Configuration," AIAA Paper 87-0032, Jan. 1987.

[2]Deese, J. E., and Agarwal, R. K., "Navier-Stokes Calculations of Transonic Viscous Flow About Wing/Body Configurations," *Journal of Aircraft*, Vol. 25, No. 12, pp. 1106–1112.

[3]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.

[4]Treasgold, D. A., Jones, A. F., and Wilson, K. H., "Pressure Distribution Measured in the RAE 8 ft × 6 ft Transonic Wind Tunnel on RAE Wing 'A' in Combination with an Axi-Symmetric Body at Mach Numbers of 0.4, 0.8, and 0.9," AGARD-AR-138, May 1989, Chap. B-4.

[5]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, June 1981.

[6]Martinelli, L., "Calculation of Viscous Flows with Multigrid Methods," Ph.D. Dissertation, MAE Dept., Princeton Univ., Princeton, NJ, 1987.

[7]Turkel, E., and Vatsa, V. N., "Effect of Artificial Viscosity on Three Dimensional Flow Solutions," AIAA Paper 90-1444, June 1990.

[8]Yu, N. J., Allmaras, S. R., and Moschetti, K. G., "Navier-Stokes Calculations for Attached and Separated Flows Using Different Turbulence Models," AIAA Paper 91-1791, June 1991.

[9]Agrawal, S., Vatsa, V. N., and Kinard, T. A., "Transonic Navier-Stokes Flow Computations over Wing-Fuselage Geometries," AIAA Paper 91-3205, Sept. 1991.

Effect of Leading-Edge Geometry on Delta Wing Unsteady Aerodynamics

L. E. Ericsson* and H. H. C. King†
Lockheed Missiles and Space Company, Inc., Sunnyvale, California 94086

Introduction

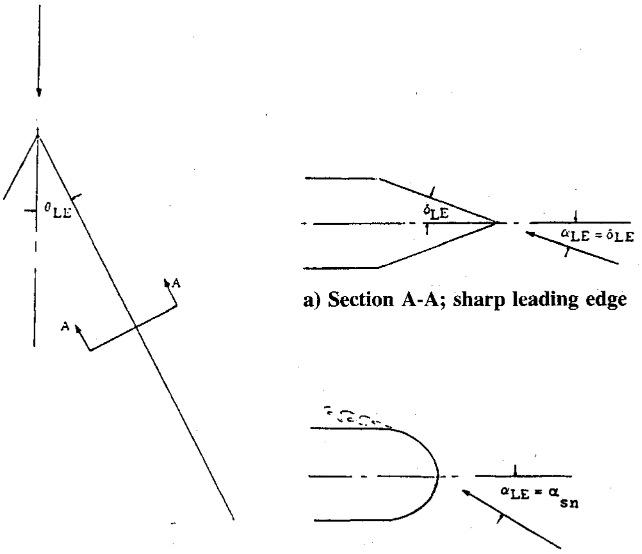
THE complexity of the flowfield on aircraft and aircraft-like configurations at high angles of attack prohibits the use of numerical computational methods for preliminary design. Also because of the continual changes in the early design, a purely experimental method cannot be used. One needs rapid computational methods to guide the early stages of preliminary design until a firmer design has evolved on which experimental and numerical methods can be applied.

The simple flow concept developed by Polhamus,[1] i.e., the leading-edge suction analogy, was used in Ref. 2 as a starting point in the development of a fast prediction method for the unsteady aerodynamics of sharp-edged delta wings. This note extends the prediction to include the effect of leading-edge cross-sectional shape.

Figure 1 shows how the delay of crossflow separation to $\alpha_{LE} > 0$, caused by the leading-edge geometry, results in a delay to $\alpha > \alpha_v$ before leading-edge vortices are generated, where α_v is

$$\alpha_v = \tan^{-1}(\tan \alpha_{LE} \sin \theta_{LE}) \qquad (1)$$

When the leading-edge geometry is of the type sketched in Fig. 1a, α_{LE} is determined directly by the geometry as $\alpha_{LE} = \delta_{LE}$. However, in the case of a rounded leading edge (see sketch in Fig. 1b), $\alpha_{Le} = \alpha_{sn}$, where α_{sn} is the crossflow sep-



a) Section A-A; sharp leading edge

b) Section A-A; round leading edge

Fig. 1 Delta wing leading-edge geometry.

Presented as Paper 92-0173 at the AIAA 30th Aerospace Sciences Meeting, Reno, NV, Jan. 6–9, 1992; received Feb. 11, 1992; revision received Nov. 23, 1992; accepted for publication Dec. 1, 1992.

*Retired; currently Engineering Consultant. Fellow AIAA.
†Currently with ETAK, Inc., Menlo Park, CA.