Navier-Stokes Simulations of Transonic Flows over a Practical Wing Configuration

Kozo Fujii*
National Aerospace Laboratory, Tokyo, Japan
and
Shigeru Obayashi†
University of Tokyo, Japan

Abstract

COMPUTATIONS of transonic flows over a practical wing shape are carried out using three-dimensional Reynolds-averaged "thin-layer" Navier-Stokes equations. Computations are done for several angles of attack and the results reveal the flow pattern for each case. The comparison of the computed surface pressure with the experiment shows good agreement. This indicates the importance of the Navier-Stokes computations. The computation which uses about 200,000 grid points for each case requires about 1.5 h on a Japanese supercomputer.

Contents

Recent advances in computer performance and numerical methods have made it possible to simulate a wide variety of complicated flowfields not only in two dimensions, but also in three dimensions. The purpose of this work is to show the capability of the current "thin-layer" Navier-Stokes code to simulate the flowfield over practical body geometries. The LU-ADI factorization used here is one of those schemes that simplify inversions for the implicit operators of the Beam and Warming scheme.[1] Each ADI operator written in the diagonal form is decomposed into two one-sided differencing operators by using the flux vector splitting technique. In general, the LU factorization based on flux vector splitting reduces stability of the code, compared to the original tridiagonal solver. Thus, the diagonally dominant factorization is adopted, which improves the stability. Details about the LU-ADI scheme are contained in Ref. 2.

The body geometry used is a supercritical-type wing designed for the transonic transport aircraft and is shown in Fig. 1. The C-H topology for the grid system is adopted and the grid generation is done two-dimensionally for each spanwise location using an algebraic procedure. The total number of grid points is approximately 200,000. A freestream boundary condition is specified at the inflow and the outer boundary. The pressure is fixed to be the freestream value and an extrapolation of the remaining physical variables is used at the outflow boundary. Flow symmetry is assumed at the root section of the wing.

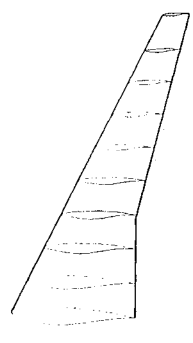
The flow was assumed to be fully turbulent and the so called Baldwin-Lomax model was used. Computations were carried out for five angles of attack. Only two of these are presented here. More results are shown in the original paper.[2]

Figure 2 shows the pressure contour plots over the upper surface and two typical spanwise stations for an angle of attack of 2.46 deg. The existence of the shock wave is clear. Compression waves can be observed emanating from the leading edge near the root. These compression waves and the shock wave in the rear portion coalesce and create a strong shock wave in the outboard region.

The pressure contour plots for an angle of attack of 7.50 deg are presented in Fig. 3. Surface contour plots indicate that compression waves from the leading edge near the root section are strong in this case and that the surface pressure after the shock wave is no longer two-dimensional. A strong curved shock wave is observed in the chordwise plots. Figure 4 shows the corresponding surface oil flow pattern. At this relatively high angle of attack, the shock wave has a spanwise curvature. The computed particle path traces, although not shown here (see Ref. 2), indicate that there is a spiral vortical flow in the separated region. This angle of attack is beyond the buffet boundary and the question arises about the unsteadiness of the flow. The computed result showed no unsteadiness, while the experiment showed flow unsteadiness. It should be remembered that buffet is a phenomenon associated with the wind tunnel model structure. The computation for a rigid model will not predict this phenomenon. Thus, the computed result is understood to correspond to the steady-state component of the flow. It is believed that the flow separation pattern and the trailing-edge pressure, which can be predicted by the computation, are good indications of buffet onset.

The computations are validated by comparing them with experimental results. The comparison of the surface C_p distribution for the first case is presented in Fig. 5 for several spanwise stations. After an empirical angle-of-attack correction is made, the computational result shows the same tendency as the experiment. The data agreement for both upper and lower surfaces is good at every station and, in addition, the pressure at the trailing edge is also well predicted, which is important for the design process of a new wing. A lift force comparison is shown in Fig. 6. As compared to experiment, the nonlinear effect due to the large angles of attack is well predicted by the Navier-Stokes computations.

Fig. 1 Wing geometry.



Received Dec. 26, 1985; presented as Paper 86-0513 at the AIAA 24th Aerospace Sciences Meeting, Reno, NV, Jan. 6-9, 1986; synoptic received Aug. 25, 1986. Full paper available from AIAA Library, 555 W. 57th St., New York, NY 10019. Price: microfiche, $4.00; hard copy, $9.00. **Remittance must accompany order.**

*Principal Research Scientist. Member AIAA.

†Graduate Student, Department of Aeronautics. Student Member AIAA.

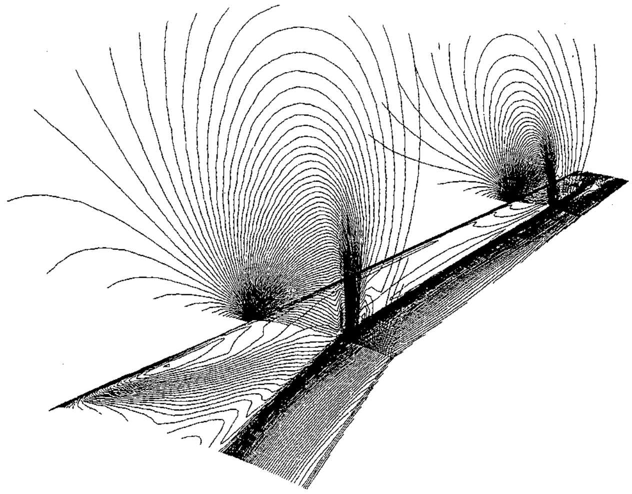


Fig. 2 Pressure contour plots on the upper surface and at two spanwise stations ($M_\infty = 0.82$, $Re = 2.0 \times 10^6$, and $\alpha = 2.46$ deg).

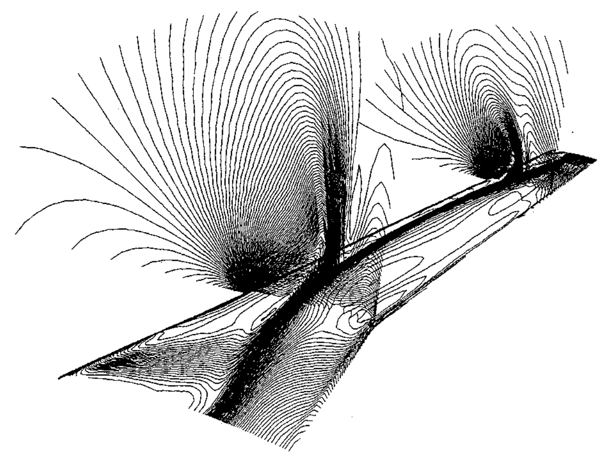
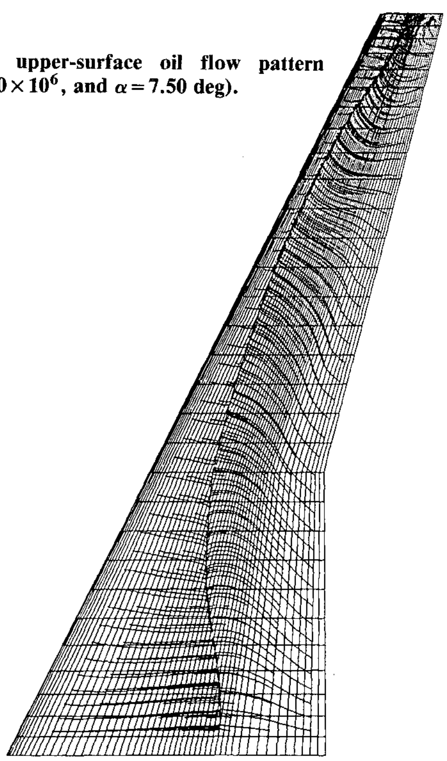


Fig. 3 Pressure contour plots on the upper surface and at two spanwise stations ($M_\infty = 0.82$, $Re = 2.0 \times 10^6$, and $\alpha = 7.50$ deg).

Fig. 4 Computed upper-surface oil flow pattern ($M_\infty = 0.82$, $Re = 2.0 \times 10^6$, and $\alpha = 7.50$ deg).



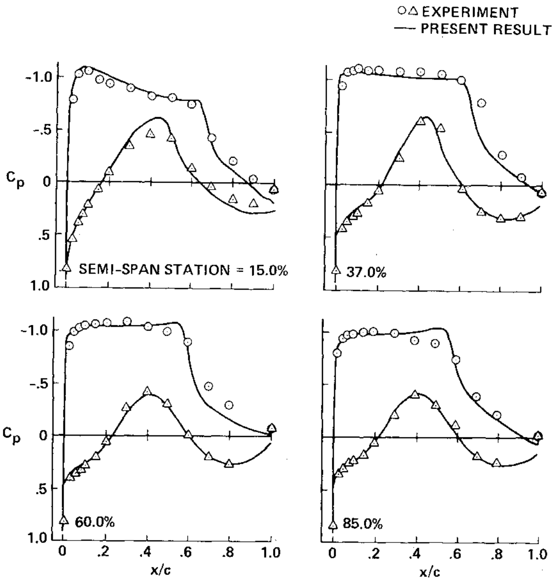


Fig. 5 Comparison of the computed C_p distributions with the experimental data at several spanwise stations ($M_\infty = 0.82$, $Re = 2.0 \times 10^6$, and $\alpha = 2.46$ deg).

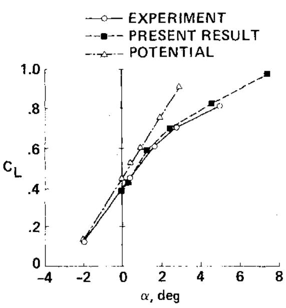


Fig. 6 Comparison of the C_L vs α curve between computational and experimental results ($M_\infty = 0.82$, $Re = 2.0 \times 10^6$).

One of the more important factors as these types of computations is the computational speed to obtain the converged solutions. Currently 1.5 h are required to obtain the steady-state solutions. The present code is well vectorized for the supercomputer used. However, only a general vectorization effort has been done and there is nothing special in the code for this particular machine. It took about 6.7 μs per iteration per grid point. The speed of the vector mode turned out to be more than 40 times faster than the scalar mode which had a speed of 8-9 MFLOP's. It should be mentioned that a special effort to utilize a longer vector length was done for the original LU-ADI code in Ref. 3, where the MFLOP ratio was larger.

Acknowledgment

The wing geometry data were offered by the JADC and MHI in Japan. The graphic programs used were developed by S. Shirayama, University of Tokyo.

References

[1]Pulliam, T.H. and Steger, J.L., "Implicit Finite Difference Simulations of Three-Dimensional Compressible Flow," *AIAA Journal*, Vol. 18, Feb. 1980, pp. 159-167.

[2]Fujii, K. and Obayashi, S., "Practical Applications of New LU-ADI Scheme for the Three-Dimensional Navier-Stokes Computation of Transonic Viscous Flows," AIAA Paper 86-0513, Jan. 1986.

[3]Fujii, K., "Viscous Compressible Flow Simulations Using Supercomputer," *Proceedings of 1st Nobeyama on High Reynolds Number Flow Computations, Lecture Notes in Engineering*, Springer-Verlag, New York, to be published.