addition to 16 points from the actual calculations. The dashed lines and filled circles in the figure indicate conditions outside the range of data used to obtain the correlation. However, the agreement between the correlation and actual calculations at these conditions is very good. In Fig. 4, we show the correlation of $(Re_x)_{N=9}$ with ν_w at two values of the freestream Mach number and $T_w/T_{ad} = 1$ with 8 points from the actual calculations. The agreement between the actual calculation and the correlation at $\nu_w = -3 \times 10^{-6}$ (which is outside the range of data used to obtain the correlation) is satisfactory.

Mack[5] correlated low-speed experimental transition data with wind-tunnel turbulence level and suggested the N factor correlation

$$N_T = -8.43 - 2.4 \ln Tu \tag{8}$$

where Tu is the turbulence level.

Bushnell et al.[6] reported some experimental evidence that suggests the previous correlation (8) can be used even at higher subsonic Mach numbers. Hence, Eqs. (4) and (8) together can be used to estimate the effect of suction, wall heat transfer, and freestream turbulence on transition location, whereby, with $N = 9$ as the reference case, Eq. (8) can provide information about the forward movement of transition caused by freestream turbulence.

Acknowledgment

This research is supported by the Theoretical Flow Physics Branch, Fluids Mechanics Division, NASA Langley Research Center, Hampton, Virginia, under Contract NAS1-19299.

References

[1]Smith, A. M. O., and Gamberoni, N., *Transition, Pressure Gradient, and Stability Theory,* Douglas Aircraft Co., Inc., Rept. ES 26388, El Segundo, CA, Aug. 1956.

[2]Jaffe, N. A., Okamura, T. T., Smith, A. M. O., "Determination of Spatial Amplification Factors and their Application to Predicting Transition," *AIAA Journal,* Vol. 8, No. 2, 1970, pp. 301–308.

[3]Schlichting, H., *Boundary-Layer Theory,* 7th ed., McGraw-Hill, New York, 1979, p. 334.

[4]Mack, L. M., *Boundary-Layer Stability Theory,* Jet Propulsion Lab. Document 900-277, Rev. A, California Inst. of Technology, Pasadena, CA, May 1969.

[5]Mack, L. M., *Transition Prediction and Linear Stability Theory,* AGARD CP-224, May 1977.

[6]Bushnell, D. M., Malik, M. R., and Harvey, W. D., *Transition Prediction in External Flows Via Linear Stability Theory,* presented at the International Union of Theoretical and Applied Mechanics Symposium Transsonicum III, Göttingen, West Germany, May 1988.

Application of a Generalized Minimal Residual Method to Two-Dimensional Unsteady Flows

R. Hixon* and L. N. Sankar†
Georgia Institute of Technology, Atlanta, Georgia 30332

Introduction

DURING the past two decades, there has been significant progress in the field of numerical simulation of unsteady compressible flowfields (e.g., transonic full potential equation-based methods, unsteady Euler solvers, and unsteady Navier-Stokes solvers). Despite the advances in algorithms and computer hardware, the numerical simulation of unsteady viscous flows remains a computationally intensive problem, even in two dimensions. Because of the excessive CPU requirements of the existing unsteady viscous flow solvers, dramatic improvements in algorithms and/or computer architecture are necessary before unsteady viscous flow analyses become practical day-to-day tools.

One scheme that has been of recent interest is the generalized minimal residual (GMRES) method originally proposed by Saad and Schultz.[1] This procedure uses a conjugate-gradient-like method to accelerate the convergence of existing solvers and was used by Wigton et al.[2] to accelerate computational fluid dynamics codes to steady solutions. In this work, the GMRES scheme has been considered as a candidate for acceleration of a Newton-iterative time-marching scheme for unsteady two-dimensional compressible viscous flow calculations. GMRES has provided significant reductions in the computer time requirements over the existing class of explicit and implicit time marching schemes.

Numerical Formulation

The starting point for the GMRES method is an existing flow solver, which is used as a preconditioner for the flow equations. A Newton iterative time marching scheme that solves the two-dimensional compressible Navier-Stokes equations in a curvilinear body-fitted coordinate system has been used as a preconditioner.[3] The solver employs approximate factorization of the spatial inviscid flux terms, and uses the Baldwin-Lomax turbulence model.

Given a known flowfield at the nth time level, the Newton iterative solver takes an initial guess for the flowfield at the next time level $(n+1)$, and iterates in order to improve the answer at the new time level. The advantage of the Newton iteration in this application is that the errors associated with the approximate factorization can be reduced or removed.

The Newton iterative solver may be written as

$$[A]^{n+1,k}\{\Delta q\}^{n+1,k+1} = \{R\}^{n+1,k} \tag{1}$$

where $n+1$ and k are the time level and iteration level, respectively, A is the preconditioning matrix, Δq is the correction vector for the flow quantities, and R is the residual of the discretized equation at the last iteration level.

Generalized Minimal Residual Formulation

The Newton iterative solver may also be written as

$$M(q^{n+1,k}) = \{\Delta q\}^{n+1,k+1} = [A]^{-1}\{R\}^{n+1,k} = 0 \tag{2}$$

When Eq. (2) is satisfied, the flowfield is converged at the new time level $n+1$.

The GMRES formulation that is used in this investigation is documented in Refs. 1 and 2. The GMRES solver uses the Newton solver as a function evaluator and computes the vector of flow properties $q^{n+1,k}$ that will satisfy Eq. (2).

It should be noted here that the GMRES routine uses the Newton iterative solver as a "black box" to determine the effect of changing the input flow properties on the correction vector Δq. Because of this, the GMRES solver is very portable and can be easily implemented in a wide variety of codes regardless of the original code's solution procedure [as long as a residual $M(q)$ can be defined]. This is a major advantage of the GMRES acceleration method over schemes that are tied closely to the details of the algorithm (e.g., multigrid methods).

The GMRES solver starts by assuming that the Δq required to set the residual given by Eq. (2) to zero lies in a vector space defined by a set of orthogonal direction vectors. In a two-dimensional flow problem, there are a total of 4*$i$max*$k$max possible direction vectors. The GMRES method uses the original code as a preconditioner to the problem to define J (usually < 20) orthogonal unit direction vectors which hopefully contain a majority of the error components. After computing the slope of the residual in

Presented as Paper 92-0422 at the AIAA 30th Aerospace Sciences Meeting, Reno, NV, Jan. 6–9, 1992; received April 19, 1992; revision received Oct. 15, 1992; accepted for publication March 11, 1993.

*Graduate Research Assistant, School of Aerospace Engineering. Member AIAA.

†Professor, School of Aerospace Engineering. Senior Member AIAA.

each direction, the GMRES routine solves a least squares problem to reduce the residual as much as possible by using a linear combination of the directions.

Obviously, the success and speed of the GMRES solution depends greatly on the original flow solver's ability to help define useful direction vectors and hence a subspace that contains many of the important error components.

The objective, of course, is to reduce the amount of CPU time necessary to perform unsteady flow calculations. In the original noniterative alternating direction implicit (ADI) version of the code, the allowable time step was small in order to keep the errors due to the approximate factorization small. With the GMRES version, the time step is only limited by the ability of the code to capture the physics of the flow. Thus, the time step may be significantly larger than that of the noniterative code. Since the GMRES solver requires the equivalent of one ADI step to compute each direction, the number of GMRES directions used must be smaller than the time step multiplier to obtain a net speedup [e.g., GMRES (10/20)—10 directions at 20 times the time step—is roughly a 2x speedup]. The drawback to the GMRES scheme is that all of the direction vectors (each the equivalent of a complete flowfield) must be stored.

Results

All of the calculations presented here were done on an algebraic 157×41 grid. All CPU times quoted are on a NASA-Langley Cray Y/MP system.

After validation of the GMRES code in the steady-state mode,[4] several two-dimensional unsteady calculations were performed using the GMRES solver to determine if significant savings in CPU time may be obtained compared to the original noniterative ADI scheme.

The first unsteady case was a transonic, inviscid flow about a plunging NACA 64A010 airfoil, as in Ref. 5. In this case, the freestream Mach number was 0.8, and the reduced frequency based on half-chord was 0.2. The plunging motion was defined by

$$y_\tau = -M_\infty \sin(1\,\text{deg}) \sin(\omega t) \tag{3}$$

At first, a time step of 20 times the ADI time step was employed, but it soon became apparent that this was too large to resolve the shock motion properly. A time step factor of five was found to be small enough to adequately resolve the physics of the problem, but the GMRES was not stable using less than 10 directions (100% increase in computer time). This illustrated the tradeoff between having the large time step necessary for effective speedup with GMRES and the small enough time step needed to accurately model the physics of the problem. This may be peculiar to inviscid flows where a relatively coarse grid will allow large time steps. The results of these runs are given in Ref. 3.

The next unsteady case that was tested was a NACA 0012 airfoil in deep dynamic stall. This was a Navier-Stokes calculation,

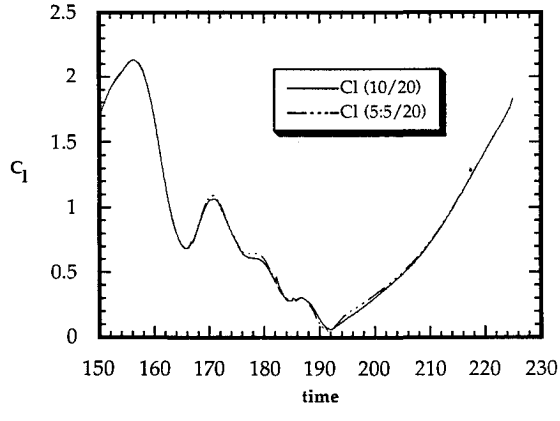

Fig. 1 Comparison of restart GMRES (5:5/20) with GMRES (10/20) results for the lift coefficient of a pitching NACA 0012 airfoil: $M = 0.283$; $Re = 3.45 \times 10^6$; $k = 0.151$.

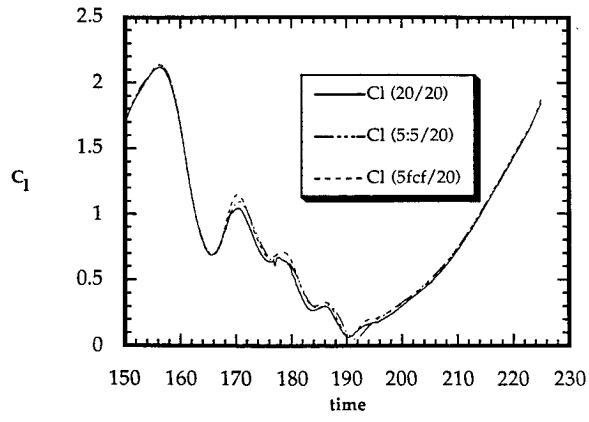

Fig. 2 Comparison of multigrid GMRES (5fcf/20) with restart GMRES (5:5/20) results for the global residual of a pitching NACA 0012 airfoil calculation: $M = 0.283$; $Re = 3.45 \times 10^6$; $k = 0.151$.

with a freestream Mach number of 0.283 and a Reynolds number of 3.45×10^6. The reduced frequency was 0.151. The airfoil motion was defined by

$$\alpha = 15\,\text{deg} - 10\,\text{deg} \cos(\omega t) \tag{4}$$

A time-step factor of 20 was tried initially, with 20 directions used to get a benchmark run that was comparable to the original ADI code. From this point, reductions in CPU time and computer memory requirements were sought, as the GMRES (20) code required 1.92 times the storage of the original ADI code.

At first, just reducing the number of directions to half of the time-step multiplier was tried (GMRES x/2x). Time-step multipliers of 10–80 were tried, but accuracy fell off after 20. After this, a multiplier of 20 was used for the rest of the runs. The GMRES 10/20 run was the most successful of the initial runs, with the memory increasing by 57% over the ADI code and the CPU time decreasing by 40%.

To reduce the storage required, a restart GMRES method was coded, in which more than one GMRES iteration was used in each time step (i.e., two five-direction iterations instead of one 10-direction iteration). This reduced memory by 11% and had the added benefit of making the code more stable (it had much less noise in the residual history). CPU time increased to 72% of the original ADI time, mainly due to the additional GMRES overhead. Figure 1 compares the lift coefficient history of the 10/20 single-iteration run with that of the 5:5/20 two-iteration run.

After looking at the contours of the directions, it was noticed that the first five directions tend to be smooth, while the higher directions contain more jagged, high-frequency components. At this point, a multigrid solver was implemented over the GMRES routine, which computes the low-frequency components of Δq on a coarser grid. With this, it was hoped that the low-frequency components of the error could be driven to zero more effectively. After steady-state validation, an unsteady run was performed, using five GMRES directions in a sawtooth pattern [fine-coarse-fine (fcf)] to see if any improvement was obtained.

Figure 2 shows the lift coefficient histories of a single-iteration 20/20 run compared to a 5:5/20 fine-grid-only run and a 5fcf/20 multigrid run. This comparison shows the effect of the coarse grid evaluation. From the residual history, no appreciable gain is apparent except when the flow is attached and the flowfield is relatively smooth. CPU time also increased to 82% of that of the original ADI code. Reference 3 gives additional results as well as the details of the GMRES formulation.

Concluding Remarks

The possibility of accelerating two-dimensional unsteady compressible flow calculations using a GMRES method has been investigated. A multigrid version of the code has also been evaluated. In

many instances a factor of 1.7 speedup was obtained. The solver is now being extended to three dimensions.

Acknowledgments

This work was supported by Grant NAG-1-1217 from NASA Langley Research Center. John B. Malone was the Technical Monitor.

References

[1]Saad, Y., and Schultz, M. H., "GMRES: A Generalized Minimal Residual Algorithm for Solving Nonsymmetric Linear Systems," *Journal of the Science of Statistical Computations,* Vol. 7, No. 3, 1986, pp. 856–869.

[2]Wigton, L. B., Yu, N. J., and Young, D. P., "GMRES Acceleration of Computational Fluid Dynamics Codes," AIAA Paper 84-1494-CP, 1984.

[3]Sankar, L. N., and Tang, W., "Numerical Solution of Unsteady Viscous Flow past Rotor Sections," AIAA Paper 85-0129, Jan. 1985.

[4]Hixon, R., and Sankar, L. N., "Application of a Generalized Minimal Residual Method to 2D Unsteady Flows," AIAA Paper 92-0422, Jan. 1992.

[5]Steger, J. L., "Implicit Finite Difference Simulation of Flow about Arbitrary Two-Dimensional Geometries," *AIAA Journal,* Vol. 18, No. 2, 1980, pp. 159–167.

Fast Three-Dimensional Vortex Method for Unsteady Wake Calculations

Kiat Chua* and Todd R. Quackenbush†
Continuum Dynamics, Inc., Princeton, New Jersey 08543

Introduction

RECENTLY, there has been much interest in the use of a Lagrangian vortex method for the simulation of unsteady vortical flows.[1] In this method, the moving packets of vorticity are discretized into a collection of vortex elements. The elements convect at the local velocity and automatically track the vorticity packets as they evolve in the flow. The velocity is computed from a direct summation of the Biot-Savart induction, and for N vortex elements, the calculation has an asymptotic time complexity of $O(N^2)$.

Results and Discussion

In the following, we describe a fast vortex method with enhanced computational efficiency. In this method, a grid of three-dimensional cubic boxes is superimposed on the computational domain, and the elements that reside in each box are clustered into a group. The velocity at a given observation point is computed from two components: a near-field vortex-induced velocity and a far-field group-induced velocity. The near-field velocity is computed by summing the Biot-Savart velocity due to all vortices that reside in the same box or immediate-neighboring boxes as the observation point. Vortices that reside in boxes that are well-separated from that of the observation point, i.e., more than one box length away, are considered to be in the far field, and their velocity inductions are computed using multipole expansions and Taylor series expansions.

The Biot-Savart velocity induction due to a group of vortex elements is given by

$$u(x,t) = -\frac{1}{4\pi}\sum_{i=1}^{N}\frac{[x-x_i(t)]\times\omega_i(t)\,\delta v}{|x-x_i(t)|^3} \tag{1}$$

where x_i, ω_i, and δv are the position, vorticity strength, and vorticity volume, respectively, of the ith vortex element. A multipole expansion of Eq. (1) can be written as

$$u(x) = -\frac{1}{4\pi r^3}\sum_{k=0}^{\infty}E_k\frac{1}{r^k} \tag{2}$$

where E_k are the moment coefficients of the expansion $r = x - x_{cm}$, and x_{cm} is the point of expansion. Provided that $r > D$, where D is the group diameter, the multipole expansion converges. In particular, the truncation error of the expansion is bounded by $c(D/r)^{p+1}$, where p is the number of terms in the expansion and c is a constant.[2] Using the multipole expansion, the computational speed of the vortex method can be accelerated. This acceleration is because the inducting effects of a large number of vortices are replaced by a single group induction computed using a small number of numerical terms.

A further improvement in the computational speed can be attained using a Taylor series expansion. Here, the far-field group-induced velocity is evaluated at the centroid x_{cm} of a group of observation points, and a Taylor series expansion is used to compute the velocity at individual points within the group:

$$u(x) = u(x_{cm}) + (x-x_{cm})\cdot\nabla u(x_{cm}) + \frac{1}{2!}[(x-x_{cm})\cdot\nabla]^2 u(x_{cm}) + \ldots \tag{3}$$

In the following, the accuracy of the multipole expansions is examined. The test problem involves a single thin vortex ring on the x-y plane. The ring has a radius $R=1$ and a circulation $\Gamma=4\pi$. It is represented as a single filament discretized into an array of 500 vortex elements. All of the elements are clustered into a group and the group-induced velocity is computed, using multipole expansions, at a set of observation points placed along an axial line intercepting the vortex ring. Three different cases with one, two, and three terms in the expansion are computed, and the velocities are compared to that obtained using a direct velocity summation. Figure 1 shows a plot of the relative velocity error $(U_a - U_e)/U_e$, where U_a is computed using a multipole expansion, and U_e is computed using a direct summation. A schematic illustrating the problem geometry is included as an insert in the figure. For points close to the vortex ring, all three cases give considerable error. This error is because the ratio $D/z > 1$, where $D=2$, is the diameter of the vortex group, and the multipole expansion fails to converge. For points more than one diameter away from the ring, the error falls off rapidly as the number of multipole terms is increased.

The propagation of the vortex ring is computed using the fast vortex method, and the computed propagation speed is compared

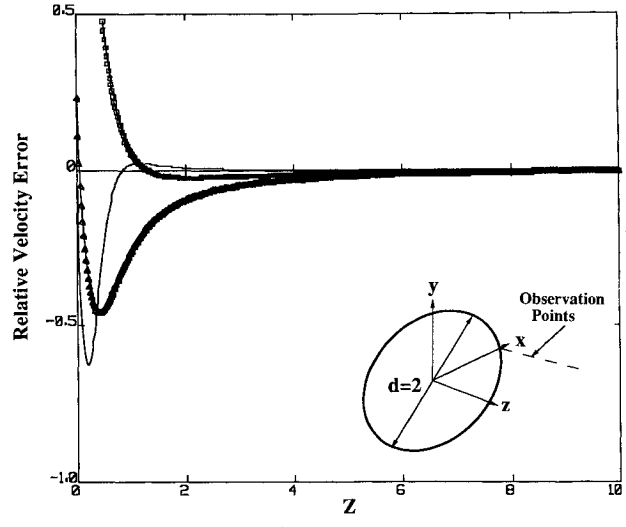

Fig. 1 Comparison of relative velocity error for multipole expansions, with one term—□; two terms—Δ; and three terms———, computed for the vortex ring problem (insert).

Presented as Paper 92-2624 at the AIAA 10th Applied Aerodynamics Conference, Palo Alto, CA, June 22–24, 1992; received Sept. 1, 1992; revision received March 29, 1993; accepted for publication March 31, 1993.

*Associate, P.O. Box 3073. Member AIAA.
†Senior Associate, P.O. Box 3073. Senior Member AIAA.