AIAA JOURNAL
Vol. 31, No. 1, January 1993

Direct Solution of Two-Dimensional Navier-Stokes Equations for Static Aeroelasticity Problems

Fort F. Felker*
NASA Ames Research Center, Moffett Field, California 94035

A new method has been developed to calculate the steady flow and structural deformations for fluid/structure interaction problems. The discretized fluid dynamic and structural equations are regarded as a single set of coupled, nonlinear, algebraic equations. The equilibrium solution is directly obtained using Newton's method. The governing equations used for the fluid flow are the two-dimensional Navier-Stokes equations, and a finite element model is used to represent the structure. This paper describes the analytical method and presents sample calculations demonstrating the technique. The results show rapid convergence and good agreement with experimental data.

Nomenclature

BW = half bandwidth of tangent matrix
D = set of structural displacement variables
E = set of governing equations for the aeroelastic system
F = flux vector in x direction
G = flux vector in y direction
h = distance from nozzle throat to centerline
I = number of fluid dynamic control volumes in streamwise direction
J = number of fluid dynamic control volumes in cross-stream direction
L = reference length, total length of nozzle
N = total number of variables in system
p = pressure
p_e = exit static pressure
p_t = total pressure
$\mathbf{R}$ = system residual
R = magnitude of residual
r = radius at nozzle throat
T = tangent matrix (system Jacobian)
t = wall thickness
U = set of fluid dynamic variables
X = set of variables for the aeroelastic system
ΔX = change in system variables during a Newton iteration
x = streamwise coordinate
y = cross-stream coordinate

Introduction

FOR many problems of interest in aeronautics, coupling exists between fluid flow and structural deformations. An example is the flow over a high-aspect-ratio wing (as might be found on a transport aircraft or helicopter rotor). The flow over the wing depends on the spanwise angle-of-attack distribution. The spanwise angle-of-attack distribution is affected by structural twist deformations. These deformations are determined by the pressure distribution on the wing, thereby coupling the structural and aerodynamic problems. It is impossible to solve one without solving the other.

Problems with strong coupling between steady aerodynamics and steady structural deformations are generally referred to as static aeroelasticity problems.[1] Some classical examples include high-speed torsional divergence of a wing and roll control reversal. A static aeroelasticity problem of current interest is the optimization of the shape and structural configuration of a flexible wing.[2]

Linear methods can be used for static aeroelasticity problems in the low subsonic or high supersonic speed ranges. However, for the transonic Mach numbers that are often of interest, linear aerodynamic models are inadequate, and computational fluid dynamics (CFD) must be used to reliably calculate the aerodynamic forces.

Existing techniques for transonic static aeroelasticity problems generally use a time-accurate fluid dynamics model and a time-accurate structural model.[3,4] The calculation is advanced in time until a steady state is achieved. Sometimes strict time accuracy is sacrificed to speed convergence, but an unsteady set of equations is still used as the basis for the solution method. This approach is straightforward, since unsteady CFD and structural models are well developed and are readily available. However, there are some limitations associated with this procedure: A fundamentally unsteady method may not be the most efficient way to find a steady-state solution, and convergence may be very slow if the dynamic aeroelastic problem is lightly damped.

A method was recently developed for directly obtaining the equilibrium solution of transonic, static aeroelasticity problems.[5] The direct solution method does not involve an advance in time until all transients have decayed. In fact, time does not even appear in the governing equations as a variable. Instead, the equilibrium solution is found by an iterative process, with each iteration of the method providing a better approximation of the equilibrium solution than the previous iteration. For the sample calculations presented in Ref. 5, the two-dimensional Euler equations were used as the governing equations for the fluid dynamics, with a simple discrete spring model for the structural deformations.

The analysis of Ref. 5 has been extended to use the two-dimensional Navier-Stokes equations, with an algebraic turbulence model[6] as the governing equations for the fluid flow. A finite element model is used for the structural deformations. A new data structure has been developed that reduces computer memory and CPU time requirements for a given computation.

This paper reviews the direct solution technique, which uses Newton's method to solve the governing equations and describes the developments listed earlier. Sample calculations are provided for a range of two-dimensional, transonic, convergent-divergent nozzles, both with and without wall flexibility.

Received Feb. 15, 1992; presented as Paper 92-2123 at the AIAA Dynamic Specialist Conference, Dallas, TX, April 16–17, 1992; revision received May 27, 1992; accepted for publication June 7, 1992.

*Branch Scientist, Rotorcraft Aeromechanics Branch, Mail Stop T-042. Member AIAA.

The results show rapid convergence and good agreement with experimental data.

Direct Solution Technique

In the present analysis the discretized fluid dynamic and structural equations are regarded as a single, coupled set of nonlinear algebraic equations. The equilibrium solution (steady state) is directly found using Newton's method (also called the Newton-Raphson method).[7] This approach allows a variety of fluid dynamic and structural models to be considered, with a consistent procedure used to find the solution of the coupled set of equations.

Newton's method has been used in the past for a variety of fluid dynamic applications.[5,8-12] Newton's method is the only technique for solving a system of nonlinear equations that provides second-order convergence. With quadratic convergence the number of digits in the solution that are correct can be doubled in a single iteration. This property ensures rapid convergence of the solution.

Newton's method proceeds as follows: The system of governing equations is written as

$$E(X) = 0 \tag{1}$$

For the present analysis, E consists of both structural and fluid dynamic equations, and X contains both structural and fluid dynamic variables.

R is the negative of E evaluated using the current values of the variables X_n:

$$R = -E(X_n) \tag{2}$$

The Jacobian of the system is the matrix whose elements are the partial derivatives of each of the equations with respect to each of the variables:

$$T = [T_{ij}] \qquad \text{where} \qquad T_{ij} = \frac{\partial E_i}{\partial X_j} \tag{3}$$

Throughout this paper, T will be referred to as the "tangent matrix." This convention avoids confusion between the system Jacobian and flux Jacobians or grid Jacobians, and the name tangent matrix is suggestive of the information provided by the Jacobian: a multidimensional local tangent to E.

ΔX is found by solving the linear system:

$$T\Delta X = R \tag{4}$$

and the new solution vector is simply

$$X_{n+1} = X_n + \Delta X \tag{5}$$

If the system were linear, only one iteration would be required. For transonic static aeroelasticity problems, the nonlinearities associated with transonic flow, the interaction between the structure and the fluid, and the algebraic turbulence model all act to slow convergence. For most of the problems examined to date, 10–20 iterations were required to reduce the residual to machine zero. Note that the iterations of the solution process do not in any way correspond to an advance in time. They are merely successive approximations to the steady-state solution.

Implementation of the Method

Newton's method is very simple and elegant. However, many challenges arise when one attempts to use this method for the solution of a transonic static aeroelasticity problem. Some of the most important details will be reviewed here, and the reader is referred to Ref. 13 for complete details.

Fluid Dynamic Model

The steady, two-dimensional, thin-layer, Reynolds-averaged Navier-Stokes equations were used as the governing equations for the fluid flow:

$$\frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0 \tag{6}$$

Note the absence of time or time derivatives in the steady equations. F and G are flux vectors in the x and y directions, respectively, written in strong conservation law form. These equations were nondimensionalized using a reference length, velocity, density, and temperature (see Ref. 14 for details of the nondimensionalization and for complete expressions for F and G). Next the equations were transformed to an arbitrary nonorthogonal coordinate system. In the interest of brevity, this transformation is not presented here, and the reader is referred to Ref. 15 for details of the coordinate transformation.

A first-order accurate finite volume method was used to discretize these equations in space,[15] with Roe's dimension-by-dimension flux difference vector splitting.[16] The finite volume method provides a discretization scheme for the fluid dynamics that is consistent with the finite element structural model. Flux splitting eliminates the need for nonphysical artificial dissipation models with their tuning coefficients, and Roe's flux difference vector splitting method provides good shock-capturing capabilities.

The thin-layer approximation was used, which means that viscous fluxes parallel to the walls of the body of interest were neglected. Even if this approximation were not formally invoked, the lack of streamwise grid resolution in a typical stretched, body-fitted Navier-Stokes grid would effectively prevent the resolution of viscous fluxes parallel to the body surface.[6] An eddy viscosity model was used to represent the effects of turbulence in the boundary layer, with an algebraic turbulence model.[6]

The sample calculations presented here considered two-dimensional, transonic, convergent-divergent nozzles. The nozzle inlet and outlet boundary conditions were set using the one-dimensional theory of characteristics. Total pressure, total temperature, and the flow angle were specified at the nozzle inlet. Static pressure was specified at the nozzle exit (for subsonic flow). The wall boundary conditions were the usual no-slip conditions, with adiabatic walls. The nozzle centerline was regarded as a plane of symmetry, with appropriate boundary conditions imposed.

Structural Model

A displacement method finite element model was used to represent the flexible structure. The walls of the nozzle were modeled using two-dimensional beam elements. The element stiffness matrices, global coordinate transformation matrices, and element force vectors that were used were standard for two-dimensional beam elements and were taken from Ref. 17. The "consistent" approach was used for the calculation of the element force vector, with the fluid pressure assumed to be constant within a given fluid dynamic control volume.

The nozzle walls were assumed to have constant properties (thickness, modulus of elasticity, etc.) along the length of the nozzle. Variable properties could be included if that were desired. The nozzle walls were assumed to be cantilevered from the inlet of the nozzle, although other structural boundary conditions could be used.

Structural degrees of freedom were included at the edges of each fluid dynamic control volume along the nozzle wall. Thus, a beam finite element was associated with each cross-stream row of control volumes. There were three structural degrees of freedom at the end of each beam element: an x displacement, a y displacement, and a rotation. Therefore, if there were, for example, 40 fluid dynamic control volumes in the streamwise direction, there would be a total of 120 structural degrees of freedom (structural boundary conditions eliminate 3 degrees of freedom).

The resulting number of structural degrees of freedom was far in excess of what is required to adequately represent a cantilever beam. However, it is important that each cross-stream row of fluid dynamic control volumes have its own structural degrees of freedom in order to minimize the bandwidth of the tangent matrix. This important issue will be covered in detail in the section on data structure.

Evaluation of Elements of the Tangent Matrix

Although Newton's method is very simple in concept, in practice it can be very challenging to compute all of the partial derivatives required for the tangent matrix. The partial derivatives of every equation with respect to every variable are needed. These partial derivatives are generally complicated nonlinear functions of the system variables. Two approaches have been explored for the calculation of these partial derivatives: evaluation of exact analytical expressions and approximate evaluation by numerical finite differences.

The use of exact analytical expressions for the partial derivatives is appealing because of the unquestionable accuracy of the results, and this approach was initially used by the author.[5] However, the required analytical expressions can be very difficult to derive, particularly if Roe's method is used to model the Euler terms in the Navier-Stokes equations. A symbolic processor (Mathematica) was used to assist with the derivations and to generate FORTRAN code for the computation of the partial derivatives.

The alternative method is to calculate the partial derivatives using numerical finite differences. There are a number of advantages associated with this approach. First, no complicated analytical derivations are required, with the associated opportunities for mistakes in the analysis or the resulting computer code. Second, the required computations are straightforward, since the governing equation of interest need only be evaluated with the variable in question incremented slightly from its current value, and all others are held constant. No significant disadvantages associated with the use of numerically computed partial derivatives have been encountered. If the partial derivatives are computed using second-order accurate central finite differences, then the second-order convergence of Newton's method is preserved.

For the present investigation, both analytical and numerically computed partial derivatives were used. The partial derivatives of the fluid dynamic equations with respect to the fluid dynamic variables were evaluated as follows: The derivatives associated with the Euler terms were computed using exact analytical expressions, and the derivatives associated with the viscous terms were computed numerically. The algebraic complexity of the turbulence model makes analytical evaluation of the derivatives associated with the viscous terms extremely difficult, and the present approach was used successfully in Ref. 12 as well. The partial derivatives of the fluid dynamic equations with respect to the structural variables were computed numerically. This provides great flexibility in the specification of how the CFD grid changes in response to structural deformations. Finally, the partial derivatives of the structural equations with respect to all variables were computed using exact analytical expressions. These derivatives were the easiest to compute, since the finite element structural model was linear.

Frozen Tangent Matrix

The bulk of the computational effort lies in the factorization of the tangent matrix required to solve the linear system in Eq. (4). In Newton's method the tangent matrix must be recomputed and refactored with each iteration, since the tangent matrix is a nonlinear function of the changing solution vector. However, large gains in efficiency can be obtained by only computing and factoring the tangent matrix once and storing the result. This "frozen" tangent matrix is then used for all subsequent iterations.

The disadvantage of this procedure is that the second-order convergence property of Newton's method is lost. The frozen tangent matrix provides only an approximation to the true tangent for all iterations but the first, and it is therefore not possible to reduce the residual by two orders of magnitude with each iteration. However, this hardly matters, since dozens of iterations can be computed using the frozen tangent matrix in the same time required to recompute and factor a new tangent matrix. The significant gains in efficiency obtained by using a frozen tangent matrix will be shown in the section presenting results from the sample calculations.

Data Structure

The tangent matrix is square, with dimensions equal to the total number of variables in the aeroelastic system. For example, the sample calculations presented herein were performed with a 40×30 CFD grid and with 120 structural degrees of freedom. Since there were 4 fluid dynamic variables associated with each fluid dynamic control volume, there were a total of 4920 variables. Thus, the tangent matrix has $4920^2 \cong 24$ million elements. Fortunately, most of these elements are zero.

The ordering of the fluid dynamic and structural variables within the set of unknowns $\boldsymbol{X}$ determines where the zero and nonzero elements of the tangent matrix lie. For example, in Ref. 5 all of the fluid dynamic variables were followed by all of the structural variables:

$$\boldsymbol{X} = \left\{ \begin{array}{c} \boldsymbol{U} \\ \hdashline \boldsymbol{D} \end{array} \right\} \tag{7}$$

This data structure resulted in a tangent matrix that was sparse, but had a bandwidth that was comparable to its dimension. For example, if there were 12 fluid dynamic control volumes and 4 structural degrees of freedom, the tangent matrix would have the following structure (here only 1 fluid dynamic variable is used for each fluid dynamic control volume in the interest of clarity):

$$\boldsymbol{T} = \left[\begin{array}{cccccccccccc:cccc}
x & x & x & x & & & & & & & & & x & & & \\
x & x & x & & x & & & & & & & & x & & & \\
x & x & x & & & x & & & & & & & x & & & \\
x & & & x & x & x & x & & & & & & x & x & & \\
 & x & & x & x & x & & x & & & & & x & x & & \\
 & & x & x & x & x & & & x & & & & x & x & & \\
 & & & x & & & x & x & x & x & & & & x & x & \\
 & & & & x & & x & x & x & & x & & & x & x & \\
 & & & & & x & x & x & x & & & x & & x & x & \\
 & & & & & & x & & & x & x & x & & & x & x \\
 & & & & & & & x & & x & x & x & & & x & x \\
 & & & & & & & & x & x & x & x & & & x & x \\
\hdashline
x & & & x & & & & & & & & & x & & & \\
 & & & x & & & x & & & & & & & x & & \\
 & & & & & & x & & & x & & & & & x & \\
 & & & & & & & & & x & & & & & & x
\end{array} \right] \tag{8}$$

To store and factor this type of matrix structure as if it were full would be prohibitively expensive for large problems. Instead, Ref. 5 presents a matrix partitioning scheme that effectively embeds the structural equations in the fluid equations and reduces the bandwidth of the matrix that must be stored and factored.

A new data structure has been developed that maintains a narrow bandwith tangent matrix without the need for the matrix partitioning procedure. The structural variables are interleaved with the fluid dynamic variables. Let J be the number of fluid dynamic control volumes in the cross-stream direction. In the interest of clarity, assume for the moment that there is 1 (instead of 4) fluid dynamic variables per control volume and 1 (instead of 3) structural degrees of freedom between each cross-stream row. Then the present data structure is

$$X = \begin{Bmatrix} U_1 \\ \vdots \\ U_J \\ D_1 \\ U_{J+1} \\ \vdots \\ U_{2J} \\ D_2 \\ \vdots \end{Bmatrix} \tag{9}$$

This data structure results in a tangent matrix with the following structure, again for a problem with 12 fluid dynamic and 4 structural variables:

$$T = \begin{bmatrix}
X & X & X & X & X & & & & & & & & & & & \\
X & X & X & X & & X & & & & & & & & & & \\
X & X & X & X & & & X & & & & & & & & & \\
X & & & X & X & & & & & & & & & & & \\
X & & & X & X & X & X & X & X & & & & & & & \\
& X & & X & X & X & X & X & & X & & & & & & \\
& & X & X & X & X & X & X & & & X & & & & & \\
& & & & X & & & X & X & & & & & & & \\
& & & & X & & & X & X & X & X & X & X & & & \\
& & & & & X & & X & X & X & X & X & & X & & \\
& & & & & & X & X & X & X & X & X & & & X & \\
& & & & & & & & X & & & X & X & & & \\
& & & & & & & & X & & & X & X & X & X & X \\
& & & & & & & & & X & & X & X & X & X & X \\
& & & & & & & & & & X & X & X & X & X & X \\
& & & & & & & & & & & & X & & & X
\end{bmatrix} \tag{10}$$

This is a banded matrix that can be stored and factored efficiently. It is illustrative to compare the storage and factorization costs associated with the data structures represented by Eqs. (7) and (9). Even after the matrix partitioning scheme has been used to reduce the bandwidth of the tangent matrix associated with the data structure of Eq. (7), the tangent matrix associated with the new data structure uses 40% less memory and requires 60% fewer operations to factor.

To keep the bandwidth of the tangent matrix as small as possible, the governing equations of a given fluid dynamic control volume must only depend on "local" variables. Local variables are variables within the vector X that are close to the variables associated with that control volume. This is why structural degrees of freedom must be included for each cross-stream row of fluid dynamic control volumes. This ensures that the structural variables which define the grid metrics for a given control volume are local and the bandwidth of the tangent matrix is minimized. A modal structural model would require fewer structural degrees of freedom, but it would be impossible to maintain a narrow bandwidth tangent matrix.

Storage and Factorization Costs

The banded tangent matrix which results from the data structure of Eq. (9) has a half bandwidth of $BW = 4J + 6$ and a dimension of $N = 4IJ + 3I$. Thus, the total storage requirement is $N(2BW + 1) \cong 32IJ^2$ words of memory.

The number of floating-point operations required to factor the tangent matrix is approximately $2NBW^2$. It has been found that it takes about the same amount of time to compute all of the elements of the tangent matrix as to do the factorization; thus, the time per full Newton iteration can be estimated by doubling the factorization time.

It has been the author's experience that, for two-dimensional calculations, the memory requirement is far more imposing than the computer time. Even fairly modest computers are now capable of over one million floating-point operations per second, and fairly detailed two-dimensional calculations can be converged in a few minutes on such a computer. However, the available memory can easily be filled, even if many megawords are available. The memory limitation can be overcome by the use of out-of-core factorization routines.[12] These routines do not attempt to store the entire tangent matrix in central memory at any time, instead shuttling pieces of it back and forth between central memory and disk. The out-of-core factorization procedure eliminates machine memory as a fundamental limitation and allows solution of arbitrarily large problems.

Initialization of the Calculation

One of the limitations of Newton's method is that it requires a good initial guess for the solution, or it will diverge. For the present analysis the initial guess was provided by a conventional, time-accurate CFD method, with the structural deformations updated to remain consistent with the local pressure distribution at each iteration. The fluid dynamic equations were solved using a line Gauss-Seidel method.[15] Note that the conventional CFD method need not be run to convergence, only long enough to prevent divergence of the subsequent Newton iterations. Typically the line Gauss-Seidel was run until the residual was reduced by two to four orders of magnitude, and then the Newton method was run until the residual was reduced to machine zero.

Sample Calculations

The method has been used to compute the flow in a range of two-dimensional, transonic, convergent-divergent nozzles, both with rigid and flexible walls. The rigid wall results serve to validate the fluid dynamics model, and the flexible wall results illustrate the capability of the analysis for a typical static aeroelasticity problem. The geometry of the nozzles was selected to match that of Ref. 18, which provides the experimental data used for code validation. The geometry of these nozzles is summarized in Fig. 1 and Table 1.

Table 1 Nozzle geometry

Nozzle	θ_1, deg	θ_2, deg	r/L^a
A1	20.84	1.21	0.0588
A2	22.33	1.21	0.2370
B1	20.84	10.85	0.0588
B2	22.33	11.24	0.2370

$^a h/L = 0.1185$. $L = 0.1158$ m.

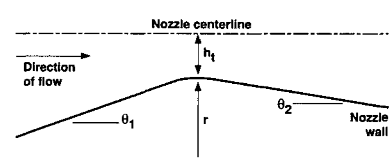

Fig. 1 Schematic of nozzle geometry.

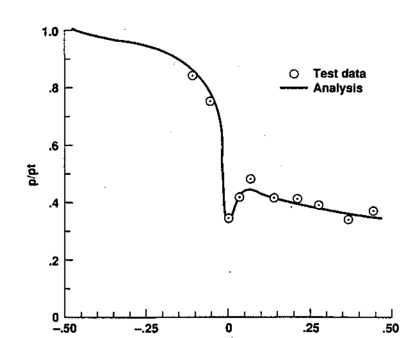

Fig. 2 Comparison of predicted and measured wall pressure ratio, nozzle A1, $p_t/p_e = 8$.

The transonic, convergent-divergent nozzles were used to validate the analysis because they provided a wide range of Mach numbers within the flow. Also, the internal flow configuration requires fewer grid points than an external flow configuration, which was beneficial during the code's development.

Rigid Wall Results

Wall Pressures

Comparisons of predicted and measured wall pressure ratios for the five nozzle configurations are shown in Figs. 2–5. For all of these cases the ratio of total pressure at the nozzle inlet to static pressure at the nozzle outlet p_t/p_e was 8.

The predictions of the wall pressures were generally very good. The principal discrepancy was that the analysis slightly underpredicted the pressure recovery just downstream of the nozzle throat, especially for nozzle A1 (the throat is at $x = 0$). The use of a spatially second-order-accurate finite volume method would improve the predictions, but the bandwidth of the resulting tangent matrix would be doubled. This doubles the storage requirement and quadruples the factorization time.

Figure 6 shows the convergence history obtained with several solution techniques. These data were computed for the B2 nozzle configuration at a pressure ratio of 8. The computer times are for an SGI Iris 4D/35 computer, which ran at about 3 megaflops for these problems. The solid line shows the magnitude of the residual (L2-norm) as a function of time using the line Gauss-Seidel technique. The circles show the results obtained with the full Newton method (tangent matrix recomputed and factored at each iteration). The Newton iterations were initiated when the line Gauss-Seidel method had reduced the residual by two orders of magnitude. Finally, the results obtained using the modified Newton method, with the tangent matrix frozen after the first Newton iteration, are marked with x's.

The full Newton method was able to reduce the residual to machine zero significantly faster than the line Gauss-Seidel method. No use was made of the techniques described in Refs. 15 and 19 for increasing the CFL number of line Gauss-Seidel methods. The Gauss-Seidel CFL number was held constant at the largest value that did not diverge during the initial iterations.

The most efficient method was the modified Newton method, in which the tangent matrix was frozen after the first iteration. This technique was able to reduce the residual to machine zero in approximately 20 iterations after the initial computation and factorization of the tangent matrix. Since each of these iterations required only 2 s, convergence was very rapid. It is difficult to imagine this level of performance using conventional time-marching CFD techniques.

It is common engineering practice to stop CFD calculations when the residual has been reduced by several orders of magnitude, and the results are converged to "plotting accuracy." Additional iterations produce very little change in the results, and since conventional CFD methods converge so slowly, this practice is difficult to criticize. However, with any numerical method that has not been converged to machine zero, the possibility always exists that the results obtained are spurious, and the next iteration will be the one in which the numerical method becomes unstable and the solution diverges.

The rapid convergence obtained with the modified Newton method makes routine convergence to machine zero practical. Indeed, the modified Newton method converges so rapidly and unambiguously that it is difficult to justify a failure to converge to machine zero.

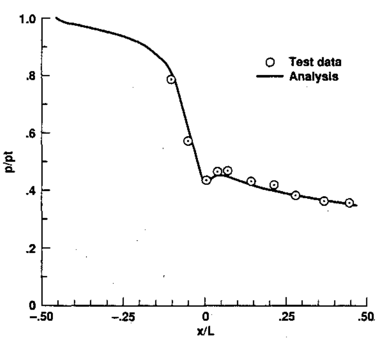

Fig. 3 Comparison of predicted and measured wall pressure ratio, nozzle A2, $p_t/p_e = 8$.

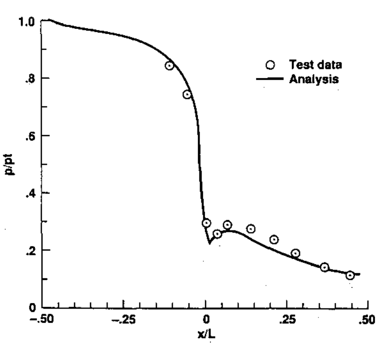

Fig. 4 Comparison of predicted and measured wall pressure ratio, nozzle B1, $p_t/p_e = 8$.

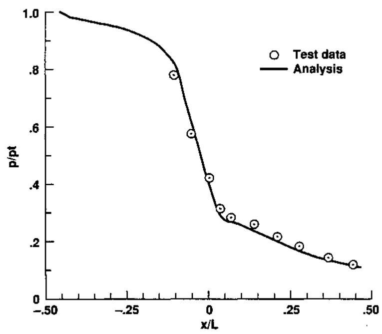

Fig. 5 Comparison of predicted and measured wall pressure ratio, nozzle B2, $p_t/p_e = 8$.

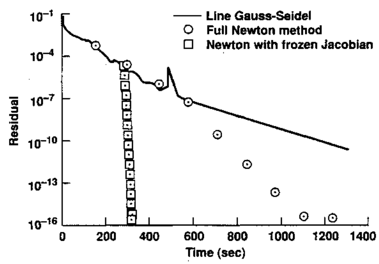

Fig. 6 Comparison of convergence history obtained using various numerical techniques, nozzle B2, $p_t/p_e = 8$.

Flexible Wall Results

The capability of the analysis to find the equilibrium solution for static aeroelasticity problems was demonstrated by considering cases in which the nozzle walls were flexible. Results obtained with flexible walls of varying stiffness are compared with rigid wall results. The value of Young's modulus was taken to be that of aluminum for all of the flexible wall calculations.

The effect of wall flexibility on the nozzle geometry is shown in Fig. 7. Since the pressure inside the nozzle was higher than the exit static pressure, the nozzle walls were deformed outward, with increasing deflections as the wall thickness was reduced. Since the wall stiffness was proportional to the cube

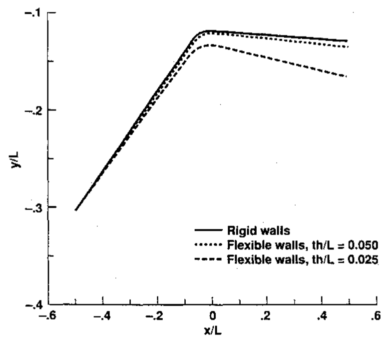

Fig. 7 Comparison of predicted nozzle wall geometry with rigid and flexible walls, nozzle A2, $p_t/p_e = 8$.

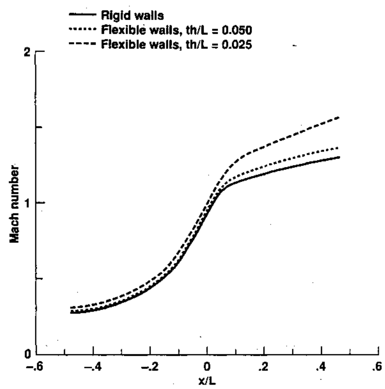

Fig. 8 Comparison of predicted centerline Mach number with rigid and flexible walls, nozzle A2, $p_t/p_e = 8$.

of the wall thickness, halving the wall thickness results in an eightfold increase in the wall deflections.

The change in nozzle geometry has a significant effect on the flow through the nozzle. Figure 8 shows the centerline Mach number for the same three cases which were shown in Fig. 7. Wall flexibility increased the nozzle expansion ratio, with a resulting increase in exit Mach number. This effect is favorable for nozzle performance. However, the assumption that inlet total pressure does not change in response to changes in nozzle throat area is probably incorrect for jet or rocket engines.

The presence of wall flexibility had an adverse influence on the convergence rate. With flexible walls, the line Gauss-Seidel initialization calculation required about twice as many iterations as the rigid wall case to provide an acceptable initial solution for the Newton method. More iterations were also required for the Newton method to converge. However, since each iteration of the modified Newton method with the frozen tangent matrix was so fast, the increase in time of the modified Newton method was insignificant compared with the total time required to obtain a solution.

Conclusions

An analysis has been developed to directly compute the equilibrium solution for two-dimensional, transonic, static aeroelasticity problems. The discretized fluid dynamic and structural equations are regarded as a single, coupled set of nonlinear algebraic equations and are solved using Newton's method. Many of the key issues that arose during the implementation of the method have been reviewed. Specific conclusions from this paper include the following:

1) The basic fluid dynamics module (no structural deformations) provides results that are in very good agreement with test data for the sample problems.

2) The capability of the analysis to efficiently find the equilibrium solution of static aeroelasticity problems has been demonstrated.

3) In all cases, convergence was rapid and unambiguous, with the residual reduced to machine zero in approximately 20 iterations. Computer time requirements were dramatically reduced relative to the traditional time-marching approach.

References

[1]Dowell, E. H., Curtiss, H. C., Scanlan, R. H., and Sisto, F., *A Modern Course in Aeroelasticity,* 2nd ed., Kluwer Academic, Boston, MA, 1989, pp. 3–50.

[2]Grossman, B., Haftka, R. T., Kao, P. J., Polen, D. M., and Rais-Rohani, M., "Integrated Aerodynamic-Structural Design of a Transport Wing," *Journal of Aircraft,* Vol. 27, No. 12, 1990, pp. 1050–1056.

[3]Guruswamy, G. P., "Unsteady Aerodynamic and Aeroelastic Calculations for Wings Using Euler Equations," *AIAA Journal,* Vol. 28, No. 3, 1990, pp. 461–469.

[4]Schuster, D. M., Vadyak, J., and Atta, E., "Static Aeroelastic Analysis of Fighter Aircraft Using a Three-Dimensional Navier-Stokes Algorithm," *Journal of Aircraft,* Vol. 29, No. 9, 1990, pp. 820–825.

[5]Felker, F. F., "Fully-Coupled Structural Deformations and Computational Fluid Dynamics: Direct Solutions Using Newton's Method," *Proceedings of the 4th International Symposium on Computational Fluid Dynamics,* Davis, CA, Sept. 1991, pp. 323–328.

[6]Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-0257, Jan. 1978.

[7]Press, W. H., Flannery, B. P., Teukolsky, S. A., and Vetterling, W. T., *Numerical Recipes,* Cambridge Univ. Press, New York, 1986, pp. 254–259.

[8]Gustafsson, B., and Wahlund, P., "Finite Difference Methods for Computing the Steady Flow about Blunt Bodies," *Journal of Computational Physics,* Vol. 36, July 1980, pp. 327–346.

[9]Fornberg, B., "Steady Viscous Flow Past a Circular Cylinder up to Reynolds Number 600," *Journal of Computational Physics,* Vol. 61, Nov. 1985, pp. 297–320.

[10]Jackson, C. P., "A Finite Element Study of the Onset of Vortex Shedding in the Flow Past Variously Shaped Bodies," *Journal of Fluid Mechanics,* Vol. 182, Sept. 1987, pp. 23–45.

[11]Wigton, L. B., "Application of MACSYMA and Sparse Matrix Technology to Multielement Airfoil Calculations," AIAA Paper 87-1142, June 1987.

[12]Bailey, H. E., and Beam, R. M., "Newton's Method Applied to Finite-Difference Approximations for the Steady-State Compressible Navier-Stokes Equations," *Journal of Computational Physics,* Vol. 93, March 1991, pp. 108–127.

[13]Felker, F. F., "Direct Solutions of the Navier Stokes Equations with Application to Static Aeroelasticity," Ph.D. Dissertation, Stanford Univ., Stanford, CA, May 1992.

[14]Anderson, D. A., Tannehill, J. C., and Pletcher, R. H., *Computational Fluid Mechanics and Heat Transfer,* Hemisphere, New York, 1984, pp. 191–193.

[15]MacCormack, R. W., and Candler, G. V., "The Solution of the Navier-Stokes Equations Using Gauss-Seidel Line Relaxation," *Computers and Fluids,* Vol. 17, 1989, pp. 135–150.

[16]Roe, P. L., "Characteristics-Based Schemes for the Euler Equations," *Annual Review of Fluid Mechanics,* Vol. 18, 1986, pp. 337–365.

[17]Przemieniecki, J. S., *Theory of Matrix Structural Analysis,* Dover, New York, 1968, pp. 70–82.

[18]Mason, M. L., Putnam, L. E., and Re, R. J., "The Effect of Throat Contouring on Two-Dimensional Converging-Diverging Nozzles at Static Conditions," NASA TP 1704, Aug. 1980.

[19]MacCormack, R. W., "Solution of the Navier-Stokes Equations in Three Dimensions," AIAA Paper 90-1520, June 1990.