AIAA 2003–1347
Parallel Computation of Wing Flutter with a Coupled Navier-Stokes/CSD Method

M. Sadeghi, S. Yang, F. Liu
Department of Mechanical and Aerospace Engineering
University of California, Irvine, CA 92697-3975

H. M. Tsai
Temasek Laboratories
National University of Singapore
Kent Ridge Crescent, Singapore 119260

41th AIAA Aerospace Sciences Meeting and Exhibit
January 6–9, 2003/Reno, NV
Parallel Computation of Wing Flutter with a Coupled Navier-Stokes/CSD Method

M. Sadeghi*, S. Yang†, F. Liu‡
Department of Mechanical and Aerospace Engineering
University of California, Irvine, CA 92697-3975

H. M. Tsai§
Temasek Laboratories
National University of Singapore
Kent Ridge Crescent, Singapore 119260

A code is developed for the computation of three-dimensional aeroelastic problems such as wing flutter. The unsteady Navier-Stokes flow solver is based on a finite-volume approach with centered flux discretization and artificial diffusion. For the structural displacements a modal approach is applied. The temporal discretization is implicit for both the flow equations and the structural equations. An explicit dual-time method is used to integrate the coupled governing equations. A multigrid method is applied to advance the flow solution, and the computation is performed in parallel with a multiblock approach. A supercritical 2-D wing and the AGARD 445.6 wing serve as test cases for flutter investigations. Results for inviscid flow are compared with results obtained by solving the Navier-Stokes equations with the Baldwin-Lomax and $k$-$\omega$ turbulence models, respectively. Inclusion of viscous effects is critical for the 2-D wing. LCO of the 2-D wing is predicted, but with larger amplitude compared to experimental measurements. Predicted flutter boundary for the AGARD wing agrees well with experimental data in subsonic and transonic range but deviates significantly from experimental data in the supersonic range. Inclusion of viscous effects only slightly improves the result for this case.

I. Introduction

Numerical investigations of aeroelastic phenomena such as wing flutter require considerations of both aerodynamics and structural dynamics. The structural deformation of interest is usually small enough to assume linear structural equations which can be solved by linear methods. On the other hand, the flow system is complex owing to complex boundary conditions, compressibility, existence of shock waves, and effects of viscosity and turbulence. Thus, a large variety of methods have been applied to solve for the unsteady flow around moving or deforming structures.

Early approaches to solve for the unsteady flow over oscillating airfoils applied linear potential theory to both the mean flow and the unsteady perturbations. Assuming the flow is linear, it can be viewed as the superposition of a time independent mean solution and a harmonic unsteady perturbation. The unsteady perturbation potential is solved by a small disturbance theory for each frequency and mode of interest, regarding the flow equations as uncoupled from the structural equations.

However, the most severe limitations on the stability of wings are usually caused by shock motion under transonic flow conditions. Linear theory has been applied for the unsteady perturbations, even where the steady mean flow is nonlinear. The superposition holds under the assumption that the shock motion caused by the wing oscillation is so small that nonlinear higher order terms can be neglected. Such methods are not capable of capturing nonlinear phenomena such as limit-cycle oscillations.

Ballhaus and Goorjian present results for airfoil flutter obtained with three different methods: the linear harmonic approach, the indicial method, and a coupled time-domain approach using their LTRAN2 code and the structural equation for a pitching airfoil. The authors find that the indicial response gives comparable results to the coupled approach only if the shock waves are sufficiently weak. LTRAN2 solves the nonlinear transonic small disturbance (TSD) equation in 2-D with an implicit finite-difference scheme in conservative form.

Since flutter is a phenomenon that arises from the
interaction between aerodynamics and structure dynamics, the flow equations and structural equations should be solved as a coupled system of equations.

By solving the transonic small disturbance equation coupled to a spring-mass model for typical wing sections, Isogai finds that the phase delay of the shock wave motion plays the dominant role in the transonic dip phenomenon of swept-back wings.

With their XTRAN3S code, Borland and Rizzetta\textsuperscript{3} solve the three-dimensional TSD equation by an implicit finite-difference scheme using ADI, coupled with the modal structural equations. The authors predict the transonic dip for a thin rectangular wing with a parabolic-arc profile.

Edwards\textsuperscript{4,5} developed a novel interactive boundary-layer method coupled with the CAP-TSD (Computational Aeroelasticity Program – Transonic Small Disturbance) code developed at NASA Langley.\textsuperscript{6}

These methods are limited to potential flow with small disturbances. In order to fully account for nonlinear flow behavior, the Euler or Navier-Stokes equations have to be applied. Shock motion plays a major role in transonic wing flutter. Viscosity and turbulence become important where shock-boundary-layer interference has an effect on the shock motion, or where the flutter behavior is dominated by separation (stall flutter).

A method for computing unsteady flows and aeroelastic response for wings using the 3-D Euler equations is presented by Guruswamy.\textsuperscript{7} An implicit finite difference scheme is used to solve the Euler equations simultaneously to the modal structural equations. In each time step, the aerodynamic grid is regenerated by an analytical method. Results obtained for a semi-infinite wing compare better to experimental results than results obtained by solving the TSD equation. The code ENSAERO is extended by implementing a solver for the thin-layer Navier-Stokes (TLNS) equations.\textsuperscript{8} The viscous terms are treated by an explicit formulation.

Sisto et al.\textsuperscript{9,10} apply a coupled method to study stall flutter in a linear cascade. The authors use a vortex and boundary layer method for incompressible flow coupled with a spring model for the blade motion. A similar torsional spring and linear spring model for rigid profiles is used by Bakhle et al.,\textsuperscript{11} by Hwang and Fang,\textsuperscript{12} and by Alonso and Jameson.\textsuperscript{13} Bakhle et al.\textsuperscript{13} investigate potential flow through a cascade. A case with linear flow behavior was chosen in order to validate the coupled method by comparison with uncoupled results. Hwang and Fang solve the Euler and Navier-Stokes equations through a cascade on unstructured grids including a transonic test case and a case of stall flutter. Alonso and Jameson\textsuperscript{13} apply an implicit finite-volume method with Jameson’s artificial diffusion to solve the two-dimensional Euler equations, coupled to the modal equations for Isogai’s pitching airfoil.

Investigations of mistuning effects on cascade flutter are performed by Sadeghi and Liu\textsuperscript{14} solving the quasi-three-dimensional Euler equations on multiple oscillating blades. The influence of fluid-structure coupling on mistuning effects is studied in Ref. 15. A nonlinear type of cascade flutter is observed in transonic flow, using the quasi-three-dimensional Navier-Stokes equations coupled with a structural model in Ref. 16.

Three-dimensional Euler and thin-layer Navier-Stokes computations are performed by Lee-Rausch and Batina.\textsuperscript{17–19} Results by time-marching coupled computations, using the modal structural equations, are compared to results by a linear stability analysis (V-g method).

A closely coupled approach is developed by Bendiksen and Hwang,\textsuperscript{20} using an Eulerian-Lagrangian method to solve the fluid and structural dynamics as single system. A finite-element method is applied for both fluid and structure, marched in time simultaneously by a Runge-Kutta scheme. The efficiency of this rather time consuming approach is improved by parallel computation.

With time-marching grid-based computations, the flow grid has to be adjusted in each time step in order to match the boundaries of the deformed structure. Tsi et al.\textsuperscript{21} present a method for the deformation of a multiblock grid. A spring-analogy method is used to determine displacements of block corners. Transfinite interpolation is then applied to interpolate boundary displacements in the interior domain. The original grid angles are approximately preserved near wall surfaces by blending the deformed angles with the original angles in near-wall regions. For a pitching airfoil, the unsteady results by the grid deformation method are shown to be identical to results by rigid grid motion.

Liu et al.\textsuperscript{22} apply a coupled approach as well as the indicial response method on a two-dimensional airfoil and on a wing. The three-dimensional Euler equations are solved on deforming multiblock grids by parallel computation. The flutter boundary of Isogai’s airfoil is obtained by time-marching calculations and compares well to Alonso’s results.\textsuperscript{13} Results for the AGARD 445.6 wing, the results by the indicial response method agree with those obtained by coupled computations.

This work presents a numerical method for parallel computation of aeroelasticity (PARCAE). The code solves the unsteady three-dimensional RANS equations on structured multi-block grids with a finite-volume method coupled to the modal structural equations. The basic numerical scheme and grid deformation method are the same as those in Refs. 21 and 22 but they are implemented in a different computer code and extended to solve the Reynolds-averaged Navier-Stokes equations with either the Baldwin-Lomax turbulence model or the k-ω two-equation turbulence model. Pseudo-time subiterations on flow and struc-
tecture ensure a strongly coupled solution in the time-domain. Flow computations on multiple grid blocks are performed in parallel.

II. Flow Solver

Fluid motion is governed by the fundamental conservation laws for mass, momentum and energy. For a flow without internal heat or mass sources, and neglecting effects of body forces, the governing equations can be written in integral form as

$$\frac{\partial}{\partial t} \int \int \int_V \mathbf{W} \, dV + \int \int_S [F] \cdot \mathbf{n} \, dS = 0$$  \hspace{1cm} (1)

where $V$ is an arbitrary control volume with closed boundary surface $S$, and $\mathbf{n}$ is the unit normal vector in outward direction. The vector of state variables $\mathbf{W}$ in Eq. (1) is defined as follows:

$$\mathbf{W} = \left\{ \begin{array}{c} \rho \\ \rho \mathbf{u} \\ \rho E \end{array} \right\}$$  \hspace{1cm} (2)

where $\rho$ is the density, $\mathbf{u} = \{u, v, w\}^T$ is the velocity vector, and $E$ is the total energy of the flow. The flux tensor $[F]$ in Eq. (1) consists of a convective (inviscid) part $[F_c]$ and a diffusive (viscous and thermal) part $[F_d]$, with

$$[F] = [F_c] - [F_d]$$  \hspace{1cm} (3)

The convective fluxes are given by

$$[F_c] = \left[ \begin{array}{c} \rho \mathbf{u} \cdot \mathbf{u}^T \\ \rho [\mathbf{u} \cdot \nabla \mathbf{u} + p I] \\ (\rho E \mathbf{u} + \rho \mathbf{u})^T \end{array} \right]$$  \hspace{1cm} (4)

Since the control volume and its surface in Eq. (1) may generally move in the fixed coordinate system, the fluxes through the surface are expressed in terms of the contravariant velocity $\mathbf{\bar{u}} = \{\bar{u}, \bar{v}, \bar{w}\}^T$:

$$\mathbf{\bar{u}} = \mathbf{u} - \mathbf{u}_g$$  \hspace{1cm} (5)

where $\mathbf{u}_g = \{u_g, v_g, w_g\}^T$ is the grid velocity vector. The fluxes arising from viscous shear stresses and thermal diffusion are

$$[F_d] = \left[ \begin{array}{c} 0 \\ [\tau] \\ ([\nabla \cdot \mathbf{u}] \cdot \mathbf{u} - \mathbf{q})^T \end{array} \right]$$  \hspace{1cm} (6)

where the shear stress tensor and the heat flux vector are defined as

$$\tau_{ij} = (\mu + \mu_t) \left[ \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\nabla \cdot \mathbf{u}) \delta_{ij} \right]$$

$$\mathbf{q} = -(k + k_t) \nabla T$$  \hspace{1cm} (7)

with the laminar viscosity $\mu$, the turbulent eddy viscosity $\mu_t$, the laminar thermal conductivity $k$, the turbulent eddy thermal conductivity $k_t$, and the temperature $T$.

The basic numerical algorithm for solving the Navier-Stokes equations with the $k$-$\omega$ turbulence model follows that presented by Liu & Zheng\textsuperscript{23} and Liu & Ji.\textsuperscript{24} A cell-centered finite-volume method with artificial dissipation as proposed by Jameson et al.\textsuperscript{25} is used. In semi-discrete form the governing equations can be written for each cell as

$$\frac{d}{dt} (\mathbf{W} \Delta V) = \mathbf{R} (\mathbf{W})$$  \hspace{1cm} (8)

where the residual $\mathbf{R} (\mathbf{W})$ is given by the discretized convective and viscous fluxes and artificial dissipation.

The time derivative is discretized by an implicit backward-difference scheme of second-order accuracy to obtain

$$D_t (\mathbf{W} \Delta V)^{n+1} = \mathbf{R} (\mathbf{W}^{n+1})$$  \hspace{1cm} (9)

with

$$D_t (\mathbf{W} \Delta V)^{n+1} = \frac{1}{2 \Delta t} \left[ 3(\mathbf{W} \Delta V)^{n+1} - 4(\mathbf{W} \Delta V)^n + (\mathbf{W} \Delta V)^{n-1} \right]$$  \hspace{1cm} (10)

where $n + 1$ denotes the current time level, and the two previous time levels are denoted by superscripts $n$ and $n - 1$.

We reformulate the problem at each time step as the following steady-state problem in a pseudo-time $t^*$

$$\frac{d}{dt^*} \mathbf{W}^n = \frac{1}{\Delta V^{n+1}} \mathbf{R}^* (\mathbf{W}^{n+1})$$  \hspace{1cm} (11)

where

$$\mathbf{R}^* (\mathbf{W}^{n+1}) = \mathbf{R} (\mathbf{W}^{n+1}) - D_t (\mathbf{W} \Delta V)^{n+1}$$  \hspace{1cm} (12)

For each real-time level, the solution to Eqs. (9) is found by iterating Eqs. (11) to steady state in pseudo-time.

A hybrid multistage Runge-Kutta scheme is used to integrate the semi-discrete Equation (11). For $m$ stages, the integration is carried out as follows:

$$\mathbf{W}^{(0)}_{i,j,k} = \mathbf{W}^n_{i,j,k}$$

$$\mathbf{W}^{(1)}_{i,j,k} = \mathbf{W}^{(0)}_{i,j,k} + \alpha_1 \frac{\Delta t}{\Delta V^{n+1}} \mathbf{R}^* (\mathbf{W}^{(0)}_{i,j,k})$$

$$\vdots$$

$$\mathbf{W}^{(m)}_{i,j,k} = \mathbf{W}^{(0)}_{i,j,k} + \alpha_m \frac{\Delta t}{\Delta V^{n+1}} \mathbf{R}^* (\mathbf{W}^{(m-1)}_{i,j,k})$$

$$\mathbf{W}^{n+1}_{i,j,k} = \mathbf{W}^{(m)}_{i,j,k}$$  \hspace{1cm} (13)

A 5-stage scheme with the following coefficients is used:

$$\alpha_1 = \frac{1}{4}, \alpha_2 = \frac{1}{6}, \alpha_3 = \frac{3}{8}, \alpha_4 = \frac{1}{2}, \alpha_5 = 1$$

\begin{flushright}
AMERICAN INSTITUTE OF AERONAUTICS AND ASTRONAUTICS PAPER 2003-1347
\end{flushright}
The artificial dissipation is updated at stages 1, 3 and 5. Local pseudo-time stepping is used in order to advance the flow solution at the local maximum speed. Residual smoothing is applied at stages 1, 3 and 5 in order to increase the stability limit. A multigrid method is adopted to accelerate the convergence of the solution.

III. Parallel Multiblock Method

The application of multiblock grids allows for complex geometries and provides domain decomposition for parallel computation. Two ghost-cell layers around each block serve for the implementation of boundary conditions as well as for the exchange of information at block-to-block interfaces. In case of an interface, the ghost-cell geometry coincides with the geometry of the corresponding cells in the neighboring block. The values of the flow variables in these cells are exchanged after each multigrid step. If both blocks are treated by the same processor, the flow variables are simply copied from one array to the ghost-elements of the other array. On the other hand, if the blocks are calculated on different processors in parallel computations, MPI is used to exchange the information.

IV. Structural Solver

The general form of the structural equations for a mechanical system with a finite number of degrees of freedom is given by

\[
[M]\ddot{q} + [C]\dot{q} + [K]q = F
\]

where \([M]\) is the mass matrix, \([C]\) the damping matrix, \([K]\) the stiffness matrix, \(q\) the vector of displacements, and \(F\) the forcing vector.

The linear structural equations can be solved using a modal approach, composing the solution with the eigenvectors of the free vibration problem. With the first \(N\) modes, the approximate description of the displacement vector is given by

\[
q = \sum_{i=1}^{N} \eta_i \phi_i
\]

where \(\phi_i\) is the \(i\)-th eigenvector of the generalized eigenvalue problem, and \(\eta_i\) is the corresponding generalized coordinate.

The eigenvectors are orthogonal with respect to both the mass and stiffness matrices. If we assume classical damping (e.g. Rayleigh damping) the eigenvectors are also orthogonal with respect to the damping matrix \([C]\). Thus, pre-multiplying Eq. (14) by \(\phi_i^T\) (normalized such that the eigenvectors are orthonormal with respect to the mass matrix) yields a set of uncoupled equations in generalized coordinates of the form

\[
\ddot{\eta}_i + 2\zeta_i \omega_i \dot{\eta}_i + \omega_i^2 \eta_i = Q_i, \quad i = 1, ..., N
\]

where

\[
Q_i = \phi_i^T F, \quad \omega_i^2 = \phi_i^T [K] \phi_i, \quad \phi_i^T [M] \phi_i = 1
\]

and \(\zeta_i\) are the modal damping parameters.

For each mode \(i\), the second-order differential equation (16) is transformed into two first-order equations

\[
\begin{align*}
x_{1i} &= \eta_i \\
x_{2i} &= \eta_i \\
\dot{x}_{2i} &= Q_i - 2\zeta_i \omega_i x_{2i} - \omega_i^2 x_{1i}
\end{align*}
\]

which can be written in matrix form as

\[
\mathbf{X}_i = [A_i] \mathbf{X}_i + \mathbf{Q}_i, \quad i = 1, 2
\]

where

\[
[A_i] = \begin{bmatrix} 0 & 1 \\ -\omega_i^2 & -2\zeta_i \omega_i \end{bmatrix}, \quad \mathbf{Q}_i = \begin{bmatrix} 0 \\ \eta_i \end{bmatrix}
\]

Decoupled by the transformation \(Z_i = [P_i]^{-1} X_i\), with

\[
[P_i] = \begin{bmatrix} -\zeta_i + \sqrt{\zeta_i^2 - 1} & -\zeta_i + \sqrt{\zeta_i^2 - 1} \\ 1 & 1 \end{bmatrix}
\]

the equations of motion take the form

\[
\dot{Z}_i = ([P_i]^{-1} [A_i] [P_i]) Z_i + [P_i]^{-1} \dot{Q}_i
\]

or, in component notation,

\[
\frac{dz_{(1,2)i}}{dt} = \omega_i \left(-\zeta_i \pm \sqrt{\zeta_i^2 - 1}\right) z_{(1,2)i} + \frac{\sqrt{\zeta_i^2 - 1} \mp \zeta_i Q_i}{2\sqrt{\zeta_i^2 - 1}}
\]

with \(Z_i = \{ z_{1i}, z_{2i} \}^T \).

The time derivative operator is now discretized with the same second-order scheme that is used to discretize the Navier-Stokes equations in Eq. (10), and the result is the following set of two finite-difference equations for each mode

\[
R_{z,i}^*(Z_i^{n+1}) = \frac{3z_{(1,2)i}^{n+1} - 4z_{(1,2)i}^n + z_{(1,2)i}^{n-1}}{2\Delta t}
\]

\[
- \omega_i \left(-\zeta_i \pm \sqrt{\zeta_i^2 - 1}\right) \frac{z_{(1,2)i}^{n+1}}{2\sqrt{\zeta_i^2 - 1}} + \frac{\sqrt{\zeta_i^2 - 1} \mp \zeta_i Q_i^{n+1}}{2\sqrt{\zeta_i^2 - 1}}
\]

\[
= 0
\]

which can be integrated to steady state in pseudo-time \(t^*\)

\[
\frac{dZ_i^{n+1}}{dt^*} + R_{z,i}^*(Z_i^{n+1}) = 0
\]
V. Fluid-Structure Coupling

The aeroelastic Eqs. (22) are coupled with the flow Eqs. (11) since the aerodynamic residual $R^s(W)$ depends implicitly on the blade motion $Z$ through the flow boundary conditions, and the structural residual $R_p(Z)$ depends on the flow variables $W$ through aerodynamic forcing.

Equations (11) and (22) form a coupled system in pseudo-time which can be solved by the same explicit Runge-Kutta scheme (Eq. 13). In principle, Eqs. (11) and (22) can be marched in pseudo-time simultaneously, i.e. when Eq. (11) is marched one pseudo-time step, Eq. (22) is also marched by one pseudo-time step. In practice, it is found that this procedure may lead to divergence because the flow equations usually converge slower in pseudo-time than the structural step. Intermediate flow solutions may lead to inaccurate aerodynamic forcing, which in turn would cause a large deformation in the structures, resulting in potential divergence. View of the above, several pseudo-time iterations are performed on the flow equations before the structural equations are also marched by several pseudo-time steps, followed by an update of the grid coordinates, grid velocities and the forcing. In this way, both iterations, for the flow model and the structural model, are converging in a coupled manner within each real-time step.

The structural mode shapes are provided for a structural grid that does not necessarily coincide with the walls of the flow grid. Therefore, a spline interpolation method is applied in order to determine the structural forcing from the aerodynamic forcing:

$$F_s = [G]^T F_a$$  \hspace{1cm} (23)

where $F_s$, denotes the forcing to be applied on the structural grid, $F_a$ are the forces obtained from the flow solution on the flow grid. The matrix $[G]$ is the the spline matrix that is used to obtain the deformation of the flow grid $\Delta x_s$ from the structural displacements $\Delta x_a$:

$$\Delta x_s = [G] \Delta x_a$$  \hspace{1cm} (24)

VI. Grid Deformation

The solution of the Navier-Stokes flow around a moving and deforming structure requires an efficient algorithm for grid deformation. If the structural motion is prescribed and therefore known a priori, the grid has to be deformed only once per time step. However, if the structural motion itself is a part of the aeroelastic solution, the iterative coupled computation involves several grid updates per time step.

The grid deformation is performed in three steps, adopting a method by Tsai et al.:

1. Structural displacements are imposed on the boundaries between structure and fluid. The structural displacements are obtained by the structural solver and then interpolated onto the flow grid as described above.

2. The corner points of all grid blocks are displaced using a spring-analogy method.

3. When the displacements of all corner points have been determined, they are interpolated along the surface edges. Hermite polynomials are used in order to be able to specify the displacement derivatives and thereby control the angle of the edges. This is done only where an edge touches a wall surface in order to preserve the near-wall grid angles. Subsequently, the same technique is applied in two and three dimensions in order to obtain the surface displacements and the displacements of the interior grid points. This is done by 2-D and 3-D transfinite interpolation, using Hermite polynomials in order to specify the grid angles close to walls. These grid angles are specified as a blend between the angles of the original grid and the angles of neighboring edges or surfaces, respectively. In this way we avoid large angular discontinuities which may lead to overlapping grid lines. This is particularly important for Navier-Stokes grids.

In parallel computations, the displacement of the unstructured spring network is most efficiently calculated on the master node, which carries information about all blocks. Since the correct angle of the surface edges may require information from neighboring blocks, the 1-D interpolation along the edges is also performed on the master node. The edge displacements are then distributed to the other processors, and the following 2-D and 3-D interpolations are performed in parallel. Finally, all deformations are superimposed on the original grid.

For the flux calculation on a moving grid we need the grid velocity $u_g$ of each grid point. These velocities are not known exactly, therefore they have to be obtained in discrete form. Applying the same difference operator that is used for the time derivative of the flow variables, we obtain the grid velocities using the grids from the current time level and two previous time levels as

$$u_{g}^{n+1} = \frac{1}{2\Delta t} (3x^{n+1} - 4x^{n} + x^{n-1})$$  \hspace{1cm} (25)

where $x$ is the vector of grid coordinates.
VII. Results
A. Nonlinear Flutter of a Supercritical Airfoil

A test case for the investigation of nonlinear flutter phenomena is presented by Schewe et al.\textsuperscript{26} Attached to pitching and heaving springs, a two-dimensional wing with the supercritical NLR 7301 profile is studied experimentally and is shown to exhibit a sharp transonic dip at a Mach number of about 0.77. Limit-cycle flutter is observed in the region of the transonic dip. Numerical investigations on the transonic test case MP77 (nominal $M_\infty = 0.768$, $\alpha = 1.28^\circ$ and $Re = 1.7 \cdot 10^6$) have been performed in the time-domain by a number of authors.\textsuperscript{27–30} The common result in those investigations is the prediction of limit-cycle flutter at amplitudes one order of magnitude larger than those observed in the experiments.

By taking the wind-tunnel walls into account in their computation, Castro et al.\textsuperscript{28,29} are able to show some effects of the wall porosity on steady and unsteady flow. These effects do not, however, fully explain the large difference between the predicted and measured limit-cycle amplitudes.

To account for wall interference, Schewe et al.\textsuperscript{26} suggest to correct the Mach number to 0.759 and the angle of attack to 0.74\textdegree for test case MP77. Figure 1 shows the steady-state distribution of the pressure coefficient computed by the present method running in 2-D mode on a $257 \times 33$ grid for the Euler computations and on a $257 \times 97$ grid for the Navier-Stokes computations, with the corrected Mach number and angle of attack suggested by Schewe et al.. The experimental steady-state results (test case MP2084) were obtained with a fixed wing.\textsuperscript{26} Even with the corrections, the numerical results do not agree well with the experimental data in Fig. 1 except for the rear loading and around the trailing edge. The pressure over the front half-chord of the pressure side is too large, and the shock on the pressure side is too weak. With inviscid flow the position and strength of the shock on the suction side is very different from the results of the viscous computation, which agree better with the experiment. The two-equation $k$-$\omega$ turbulence model yields slightly better results than the Baldwin-Lomax model. The lift is overpredicted in all cases.

In an attempt to obtain better steady-state results for this test case before proceeding to the aeroelastic computation, the Mach number and angle of attack are adjusted to yield the best possible match with the experimental data. For the Euler computation the best results were obtained with $M_\infty = 0.735$, $\alpha = -0.5^\circ$, and for the Navier-Stokes computations $M_\infty = 0.75$, $\alpha = 0^\circ$. The results are shown in Fig. 2. The fluid-structure coupled computations are performed with these adjusted values.

Figures 3 and 4 show the time histories of the heave and pitch displacements, obtained by Euler calculations and Navier-Stokes calculations with the Baldwin-Lomax and the $k$-$\omega$ turbulence models, respectively. Note that the angular displacement is defined nose-up positive, and the translational displacement is defined downward positive and is non-dimensionalized by the half-chord length $b$. By comparing Figs. 3 and 4 we can see that the pitch and plunge motions (as defined above) are almost in phase, which is in agreement with the experimental results.

Limit-cycle oscillations are observed with inviscid as well as viscous flow. However, the LCO amplitudes are overpredicted in all cases. Similar to the steady state, there are large discrepancies between inviscid flow and viscous flow in the unsteady behavior. The Euler computations predict very large displacements. Considerable improvement is achieved by using the Navier-Stokes equations with the algebraic turbulence model. The flutter amplitudes are further decreased when the $k$-$\omega$ model is used. However, even the lowest amplitudes shown in Figs. 3 and 4 are still about one order of magnitude larger than those found in the experiments. Furthermore, the flutter frequency is
underpredicted. Table 1 gives a summary of the limit-cycle amplitudes and reduced frequencies (defined as $\kappa_f = \omega_f \frac{c}{u_{\infty}}$).

<table>
<thead>
<tr>
<th>Model</th>
<th>$\Delta \alpha / ^\circ$</th>
<th>$h/b$</th>
<th>$\kappa_f$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Euler</td>
<td>15.6</td>
<td>0.323</td>
<td>0.199</td>
</tr>
<tr>
<td>B-L</td>
<td>4.8</td>
<td>0.088</td>
<td>0.206</td>
</tr>
<tr>
<td>$k-\omega$</td>
<td>3.1</td>
<td>0.055</td>
<td>0.205</td>
</tr>
<tr>
<td>Exp.</td>
<td>0.2</td>
<td>0.005</td>
<td>0.242</td>
</tr>
</tbody>
</table>

Table 1 LCO amplitudes and frequencies.

While the above comparisons between inviscid and viscous results suggest that this test case exhibits strong viscous effects and it is imperative that a Navier-Stokes code be used, the results also indicate that the current computations do not capture all significant flow features. Schewe et al. suggest that there might be a small separation region beneath the shock, which would have significant influence on the shock motion and might cause the flutter amplitudes to be limited to remarkably small values. However, the pressure distributions obtained by the experiments are not sufficient to prove the existence of such a small separation region and it is not revealed in our numerical simulations. Since there is no reliable information available about the transition point, the current calculations assume fully turbulent flow. In addition, it is possible (as noted in Ref. 26) that the thin rear of the airfoil is subject to some deformation, which might introduce unsteady effects on the shock or the trailing edge separation, not accounted for in the computations.

B. Flutter of the AGARD 445.6 Wing

The AGARD 445.6 wing has a quarter-chord sweep angle of 45 degrees and its cross-section is given by the NACA 65A004 airfoil. The flutter characteristics of this wing were investigated experimentally over a wide range of Mach numbers. The results were presented as an AGARD standard aeroelastic configuration and have since been widely used to test and validate flutter calculations.

Lee-Rausch and Batina find that at subsonic Mach numbers the flutter boundary of this wing can be predicted by coupled time-marching computations using the Euler equations and modal structural equations. However, in the supersonic regime, the flutter boundary is overpredicted by the inviscid computations. Using the thin-layer Navier-Stokes equations, the authors are able to achieve some improvement. However, the viscous results still overpredict the flutter boundary.

Liu et al. apply a time-marching method for solving the Euler equations coupled to the modal equations for the first four modes. Their results confirm the large overprediction in case of inviscid flow, while good agreement with the experiments is achieved in the subsonic domain.

In an attempt to demonstrate differences between the inviscid and viscous flow, this work presents results by using the Euler and Navier-Stokes computations. The Euler computations were performed on a $129 \times 33 \times 33$ grid. The Navier-Stokes computations were performed on a $129 \times 49 \times 33$ grid. Figure 5 shows the flutter speed index as a function of the freestream Mach number. Experimental results are plotted for comparison with those obtained by the coupled aeroelastic calculation. The first five structural modes are taken into account. Results are shown for inviscid flow and viscous flow using the Baldwin-Lomax and the $k-\omega$ turbulence models. The flutter speed index is defined as the value

$$V_f = \frac{u_{\infty}}{b \omega_{\alpha} \sqrt{\mu}}_{\text{neutral}}$$

at which an initial oscillation is neither amplified nor damped by aerodynamic forcing. If the speed index is above the flutter speed index, the motion is unstable, if the speed index $V$ is smaller than $V_f$, the motion is stable. Here, $u_{\infty}$ is the freestream velocity, $\delta$ is the half-chord length at the wing root, $\omega_{\alpha}$ is the eigenfrequency of the first torsional mode, and $\mu$ is the mass ratio of the wing.
In the subsonic and transonic range the results of the computation match the experimental data reasonably well in Fig. 5. Since this is true even for the inviscid results, it appears that in the subsonic to transonic range the aeroelastic behavior is not significantly influenced by viscous effects for this wing. However, for supersonic flow the calculations clearly overpredict the flutter speed. The overprediction is most apparent with inviscid flow. Part of the reason may be that the effect of viscosity on the shock motion is not accounted for by the inviscid flow calculation. The viscous calculations with both turbulence models show only small improvement over the inviscid results. It seems likely that with the given computational grid, neither turbulence model is able to predict all significant flow features.

Another possible cause for the discrepancy at supersonic speeds might be given by the nonlinear flutter behavior. If the Mach number is low, the oscillation amplitude increases monotonically in an unstable situation, as shown in Fig. 6. In the supersonic range, however, the amplitudes may initially increase but then decay to yield a stable solution after some time, as shown in Fig. 7. The wind-tunnel tests might have been terminated at an early stage, and the behavior misinterpreted as unstable.

Figure 8 shows the flutter frequency ratio, i.e. \( \omega_f/\omega_{ns} \), for various Mach numbers. Despite the good agreement on the flutter speed index in the subsonic range (Fig. 5), the frequency is overpredicted in all the computations. Considering some scatter in the experimental data, the computational results are reasonable in the transonic Mach number range, while the predicted frequencies in the supersonic region are again too large. It is interesting to note that the flutter speed index is accurately predicted despite the overprediction of the frequency in the subsonic regime. Since stability is directly related to the phase difference between structural motion and forcing, one might infer that in the supersonic regime the phase is more sensitive to the frequency than in subsonic flow.
inviscid solution. Figs. 10a-c show the magnitude and phase of the unsteady pressure coefficient at 95% span, for a) inviscid flow, b) viscous flow using the Baldwin-Lomax model, and c) the k-ω model. Here there seems to be little difference between the two viscous solutions, the oblique shock near the trailing edge is stronger in the inviscid flow solution.

For the same Mach number of $M_\infty = 1.141$, Figs. 10a-c show the magnitude and phase of the unsteady pressure coefficient over the wing for inviscid flow at three different speed indices. Here $\Delta C_p$ is the difference between the pressure coefficients on the lower and upper wing surfaces:

$$\Delta C_p(t) = C_{p,l}(t) - C_{p,u}(t) = \frac{p_l(t) - p_u(t)}{\frac{1}{2} \gamma p_\infty M^2_\infty} \tag{27}$$

The unsteady pressure has a destabilizing effect where the phase difference between $\Delta C_p$ and the displacement is positive. This phase difference is zero along the shock contour lines in Figs. 10a-c, therefore, the lines separate stabilizing regions (+) from destabilizing regions (-). The magnitude of the oscillating pressure difference is shown as underlying shades of gray.

Three different results are shown: a) a stable solution at a speed index slightly below the flutter limit, b) the neutral solution at $V = V_f$, and c) a slightly unstable solution. In all cases, the pressure magnitude is dominant in the region of the oblique shock near the wing tip. In that region the phase is positive (destabilizing). In Fig. 10a there is also an unstable region close to the wing root, which does not, however, contribute much to the bending forces, because the magnitude in that region is very small. Over a large part of the wing, a small region about the shock excluded, the phase angle is negative, so that overall the wing is stable at the speed index of $V = 0.67$. As the speed index is increased to 0.685, the unstable region about the shock extends all the way to the wing root as shown in Fig. 10b. In addition, there is a growing region of instability about the leading edge near the wing tip. At this speed index, the unstable and stable regions seem to eliminate each other to yield overall neutral stability for the wing. The two unstable regions at the leading edge and around the shock grow together to cover most of the wing area when the speed index is further increased (Fig. 10c). At the speed index of $V = 0.7$ the wing exhibits flutter.

For comparison with the inviscid results, Figs. 11a-c show the unsteady pressure difference obtained by the viscous computation using the k-ω turbulence model. There is no fundamental difference between the inviscid and the viscous result. The instability seems to arise from the shock motion (Fig. 11a) and at the leading edge near the tip (Fig. 11b). The destabilizing regions extend over a major part of the wing as the speed index is increased (Fig. 11c). Though qualitatively similar to the inviscid case, the viscous computation predicts flutter at a lower speed index, closer to but still higher than the experimental value for this supersonic Mach number.

In order to visualize the shock motion in relation to the wing deformation, Fig. 12 shows the time histories of the pressure coefficient around the shock as well as the time histories of the generalized displacements of the first three modes. For both inviscid and viscous flow, the pressure contours and displacements are obtained with $M_\infty = 1.141$ for the case of neutral stability, i.e. at $V_f = 0.685$ (inviscid) and $V_f = 0.63$ (viscous), respectively. The pressure coefficient is shown over a fraction of the chord at 95%
steady Euler or Navier-Stokes equations are solved by applied to test cases of airfoil and wing utter. The unstructured multigrid finite-volume method on structured multi-block grids. The Baldwin-Lomax model and a k- turbulence model are implemented. Using a modal approach, the structural equations are solved simultaneously with the flow equations. Coupled convergence is achieved by pseudo-time stepping with several updates of the forcing and deformation in each time step. The flow in multiple grid blocks is calculated in parallel.

Fig. 12 Shock motion at 95% span and modal structural deformations for $M_\infty = 1.141$ at neutral stability.

span, i.e. close to the wing tip, where the shock is most visible.

It is apparent in Fig. 12, that the inviscid flow yields a more confined and stronger shock than the viscous flow. This has already been observed with the steady state in Fig. 9. In viscous flow the shock appears to move sinusoidally, whereas in the inviscid case the time-history of the shock position exhibits various frequencies. The modal displacements are shown on the right hand side of Fig. 12. The deflections of the fourth and fifth modes are not shown here, because of their insignificant contribution to the total deformation.

The Euler computation yields a slightly higher frequency than the Navier-Stokes computation for the first-mode deflections. This was shown more clearly in Fig. 8. A difference between the inviscid and viscous cases can be seen in the third-mode deflection (second bending mode). While the third mode is almost suppressed in the viscous case, it is more clearly apparent in the inviscid result, where it exhibits a frequency which is about twice that of the dominant first mode. This harmonic frequency is reflected in the shock motion only in the inviscid case.

VIII. Conclusions

A numerical method for the computation of three-dimensional aeroelastic problems is presented and is applied to test cases of airfoil and wing flutter. The unsteady Euler or Navier-Stokes equations are solved by a multigrid finite-volume method on structured multi-block grids. The Baldwin-Lomax model and a k- turbulence model are implemented. Using a modal approach, the structural equations are solved simultaneously with the flow equations. Coupled convergence is achieved by pseudo-time stepping with several updates of the forcing and deformation in each time step. The flow in multiple grid blocks is calculated in parallel.

The flutter behavior of a pitching and plunging supercritical airfoil is investigated and compared to experimental results in Ref. 26. For a transonic test case, the calculated steady-state distribution of the pressure coefficient is in poor agreement with the experiments. Comparisons between inviscid and viscous computations in the current work confirm that viscous effects are significant. However, even with viscous computations the Mach number and angle of attack have to be adjusted in order to improve the agreement with the experimental data.

Aeroelastic computations are performed using the Mach number and angle of attack that yields the best match to the experimental steady-state results. Limit-cycle flutter is observed with both inviscid and viscous flow, qualitatively confirming the experiments. However, the oscillation amplitudes are largely overpredicted in all cases. The flutter frequency is underpredicted. As in the steady-state calculations, considerable improvement is achieved by taking viscosity into account. The best result is achieved by using the k-ω model.

The discrepancies between computation and experiments may be due to a number of physical factors that are not accounted for in the computations. Potentially, these include shock induced boundary-layer separation, deformation of the airfoil near the trailing edge, and wind-tunnel interference. Similar observations are made in Refs. 26–30. Further investigations are needed to resolve those issues, given that generally speaking flows over many other airfoils have been well predicted by modern Navier-Stokes computations, at least for steady-state solutions.

The AGARD 445.6 wing is used as a three-dimensional test case. Aeroelastic computations are performed over a large range of Mach numbers in order to obtain the flutter boundary. The five most dominant structural modes are taken into account. In the subsonic and transonic regimes, the flutter speed index for inviscid as well as viscous flow agrees well with the experimental data in Ref. 32, although the flutter frequency is slightly overpredicted. However, at supersonic Mach numbers the flutter speed index as well as the flutter frequency are overpredicted by all computations. The Navier-Stokes computations, assuming fully turbulent flow, achieve little improvement over the Euler results. The most visible effect of viscosity is to weaken the oblique shock on the wing. Even though the dynamics of this shock are shown to be somewhat different in the inviscid and viscous cases, the effect of viscosity on flutter stability seems to be small for this test case.

References


