AN ELASTICITY-BASED MESH SCHEME APPLIED TO THE COMPUTATION OF UNSTEADY THREE-DIMENSIONAL SPOILER AND AEREOELASTIC PROBLEMS

Robert E. Bartels
Aerelasticity Branch
NASA Langley Research Center
Hampton, VA 23681

Abstract

This paper presents a modification of the spring analogy scheme which uses axial linear spring stiffness with selective spring stiffening/relaxation. An alternate approach to solving the geometric conservation law is taken which eliminates the need for storage of metric Jacobians at previous time steps. Efficiency and verification are illustrated with several unsteady 2-D airfoil Euler computations. The method is next applied to the computation of the turbulent flow about a 2-D airfoil and wing with two and three-dimensional moving spoiler surfaces, and the results compared with Benchmark Active Controls Technology (BACT) experimental data. The aeroelastic response at low dynamic pressure of an airfoil to a single large scale oscillation of a spoiler surface is computed. This study confirms that it is possible to achieve accurate solutions with a very large time step for aeroelastic problems using the fluid solver and aeroelastic integrator as discussed in this paper.

I. Introduction

Aerelasticity is the study of the interaction of inertial, elastic and aerodynamic forces acting on a structure. Aeroviscoplasticity includes the additional mutual interaction of a control system and the aeroelastic structural response. When constructing computational algorithms to model problems in these disciplines, accuracy and robustness must receive due attention. This is especially true in view of recent interest in the accurate computation of fluid-structure interaction in the presence of a strongly nonlinear flow field. This includes examples such as the computation of limit cycle oscillation and the aeroelastic response of a structure due to large-scale control surface motion. To address these issues of robustness and accuracy, recent aeroelastic research has focused on the development of new algorithms that integrate the structural equations of motion in synchronization with the aerodynamic equations of motion. Among the methods used or promoted are the closely coupled lagged approach,\(^1\)\(^2\) to updating the structure equations and the implicit iterative coupling of both structure and fluid.\(^3\) Other approaches recently discussed include the Arbitrary Lagrangian-Eulerian (ALE)\(^4\)\(^5\) and the Implicit Continuous-fluid Eulerian (ICED-ALE) methods.\(^6\) Recent mesh deformation algorithm development has emphasized the robustness and efficiency necessary for coupled structure-fluid time marching computations.\(^7\)

The purpose of the present paper is to revisit old approaches to mesh deformation and the integration of the structural equations of motion. In particular the spring analogy mesh scheme is revisited with modifications that enhance its value for structured grids with complex geometry. The finite dimensional state-space predictor-corrector method\(^8\) also merits renewed consideration in view of recent advances in computational fluid solvers. The performance of the modified spring analogy and the predictor-corrector integrator will be assessed in this paper.

In the development of the present method, several modifications of the typical approach to CFD with a deforming mesh have been made. For unsteady problems, a self-consistent approach to the Jacobian of the coordinate transformation is to compute it by solving an independent equation called the geometric conservation law (GCL).\(^9\)\(^10\) In reference 3, the GCL fluxes are retained in the Navier-Stokes equations as source terms, replacing the time rate of change of the volume. Not only does this result in self-consistency, it also eliminates the need for storage of either the discrete time derivative of the Jacobian or of the Jacobian itself at several previous time steps. A variation of that approach is used here. The metric fluxes are retained in the present scheme on both the left and right hand sides of the discrete approximate factorized (AF) equations in a cell centered finite volume discretization of the equations. In reference 3, the flow equations were solved in finite difference form using central differencing of the flux terms with explicit and implicit artificial dissipation. Several options are also available in the present code for construction of fluxes and time accuracy. Third order upwind Roe's flux differencing and second order backward time differencing are used here. Subiterations either include the physical time step

* Aerospace Engineer, Senior Member AIAA
Copyright © 1999 by the American Institute of Aeronautics and Astronautics, Inc. No copyright is asserted in the United States under Title 17, U.S. Code. The U.S. Government has a royalty-free license to exercise all rights under the copyright claimed herein for Governmental Purposes. All other rights are reserved by the copyright owner.
capability to dynamically oscillate a grid, a feature that will be used in the code comparisons that follow. Since the form of the thin layer Navier-Stokes equations as used here and the solution procedures are not typical, the equations and details of the solution will be presented. In a generalized deforming coordinate system the differential form of the equations can be written

\[
\frac{1}{J} \frac{\partial Q}{\partial t} = -R(Q)
\]  \hspace{1cm} (2.1)

where

\[
R(Q) = \frac{\partial (\hat{F} - \hat{F})}{\partial \zeta} + \frac{\partial (\hat{G} - \hat{G})}{\partial \eta} + \frac{\partial (\hat{H} - \hat{H})}{\partial \zeta} - \frac{\partial (\hat{G})}{\partial \zeta} \frac{\partial (\hat{H})}{\partial \eta} + \frac{\partial (\hat{G})}{\partial \zeta} \frac{\partial (\hat{H})}{\partial \eta}
\]  \hspace{1cm} (2.2)

and \( Q = (\rho, \rho u, \rho v, p, \omega, e) \). The inviscid flux and viscous diffusive terms are \( (\hat{F}, \hat{G}, \hat{H}) \) and \( (\hat{F}, \hat{G}, \hat{H}) \) respectively. Note that the time derivative on the left-hand side of equation 2.1 is not written in strong conservation-law form, and that the last bracketed term in the residual is composed of the grid speed fluxes arising from the CCL. Letting

\[
\hat{F} = \begin{bmatrix}
\rho U \\
\rho U + p \frac{\partial \theta}{\partial \zeta} \\
\rho U + p \frac{\partial \theta}{\partial \eta} \\
(\epsilon + p)U - p \frac{\partial \theta}{\partial \zeta}
\end{bmatrix}
\]

and constructing the remaining inviscid and diffusive fluxes, the complete equation set is obtained. In integral form, the equations 2.1 can be written

\[
\int_V \int_0^{J'} Q \, dV \, dt = \int_V \int_0^{J'} \hat{\dot{E}} \cdot (\hat{F} - \hat{F}) \, dV \, dt \\
- \int_V \int_0^{J'} \nabla \cdot \hat{\dot{E}} \, dV \\
- \int_V \int_0^{J'} \dot{Q} (\nabla - \bar{\nabla}) \, dV \, dt
\]  \hspace{1cm} (2.3)

where \( \bar{\nabla} \) is the grid speed, \( \hat{\dot{E}} = (\hat{\dot{F}}, \hat{\dot{G}}, \hat{\dot{H}}, \hat{\dot{I}}, \hat{\dot{J}}) \) and \( J' = \frac{\partial J}{\partial \tau} \) is the ratio of the physical to computational volumes. The fluxes are evaluated at the cell face centers, while volume integrals over the computational cell volumes \( V \) are evaluated discretely at control volume centers, making this a cell centered finite volume scheme.

The residual composed of the right hand side of equation 2.3 is linearized about the most recent subiteration \( m \) and the resulting equations are approximately factored. The resulting discrete equations take on the form

\[
\begin{bmatrix}
\left( \frac{1}{J \Delta t} + \frac{\phi}{J \Delta t} \right) & \left( \frac{1+\phi}{J \Delta t} \right) \\
\frac{J \Delta t}{\phi} & \frac{J \Delta t}{\phi}
\end{bmatrix} \frac{\Delta \theta}{\Delta \tau} + \frac{\phi \Delta \theta^{m-1} + (1+\phi) \epsilon^{m-1} - \epsilon^{m}}{J \Delta t} \Delta \theta^{m} = 0
\]  \hspace{1cm} (2.4a)
\[
\left[ \frac{1+\phi}{J_0 \tau} \right] + \left[ \frac{1+\phi}{J_0 \Delta t} \right] \tilde{M} + \delta_s B^* + \Omega \right] \Delta q^* = \\
\left[ \frac{1+\phi}{J_0 \tau} \right] + \left[ \frac{1+\phi}{J_0 \Delta t} \right] \tilde{M}_\Delta q^* \\
\left[ \frac{1+\phi}{J_0 \tau} \right] + \left[ \frac{1+\phi}{J_0 \Delta t} \right] + \Omega \right] \tilde{M}_\Delta q^* \\
(2.4b)
\]

\[
\left[ \frac{1+\phi}{J_0 \tau} \right] + \left[ \frac{1+\phi}{J_0 \Delta t} \right] \tilde{M} + \delta_s C^* + \Omega \right] \Delta q'' = \\
\left[ \frac{1+\phi}{J_0 \tau} \right] + \left[ \frac{1+\phi}{J_0 \Delta t} \right] + \Omega \right] \tilde{M}_\Delta q^* \\
(2.4c)
\]

for the residual as defined above and

\[
\Omega = \left[ \frac{\partial}{\partial \xi^*} \left( \frac{\xi^*}{J_0} \right) \right] + \left[ \frac{\partial}{\partial \eta^*} \left( \frac{\eta^*}{J_0} \right) \right] + \left[ \frac{\partial}{\partial \zeta^*} \left( \frac{\zeta^*}{J_0} \right) \right] \tilde{M}. \!
(2.5)
\]

In the equations above, \( \tilde{M} = \partial Q / \partial q \).

The mesh scheme is a modification of the spring analogy using axial spring stiffness. The approach of Batina can be written

\[
M \ddot{\delta} + C \dot{\delta} + K \delta = 0 \!
(3.1)
\]

where \( K \) is the spring stiffness matrix, \( M \) and \( C \) are the mass and damping matrices and the mesh displacement is given by \( \delta \). In this formulation \( M = C = 0 \). The stiffness would be defined by \( k_i = 1/l_i \) where \( l_i = \left\| \vec{r}_i - \vec{r}_{i+1} \right\| _2^{1/2} \) and \( i \) and \( j \) represent two adjacent nodes. For an unstructured grid, storage of stiffness values based on the current locations of each node pair is required. The spring stiffness would be updated as the mesh deforms. For a structured grid, the problem is somewhat simplified. Spring stiffness in the mesh interior can be controlled by the spacing of the appropriate boundary grid points. For instance, if the nodes at \((i,j,k)\) and \((i,j,k+1)\) are considered and if boundary spacing at boundaries \( i = 1 \) and \( i = \text{max} \) are used

\[
k_m = 1/l_{i+1,k}
\]

where \( m = k+1 \) designates in this case the volume edge between the \( k \) and \( k+1 \) grid points, and

\[
l_{i+1,k} = f \left( \left\| \vec{r}_{i+1,k} - \vec{r}_{i,k} \right\| _2 \right)_m^{1/2} +
(1 - f) \left( \left\| \vec{r}_{i+1,k} - \vec{r}_{i,k} \right\| _2 \right)_m^{1/2}
\]

\[
f = (i_{\text{max}} - i) / (i_{\text{max}} - 1).
\]

Since these stiffness values are set at the start of the computations they do not vary in time. They also require storage only for each of the six computational boundaries. Linear spring stiffness allows the possibility of grid point crossings, but in view of the fact that this is a restricted case of the general problem of grid lines crossing, the better solution is the addition of nonlinear torsion stiffness. This can be added to the present spring stiffness at a future time. Finally, it is commented that the axial stiffness approach used here results in smoothing of the mesh and also allows adaptation based on the flow solution.
The problem of grid collapse around convex surfaces is handled by selectively increasing/decreasing stiffness based on surface curvature. Stiffness values in two coordinate planes normal to a surface are varied based on surface curvature in the coordinate projection of those planes onto the surface. The final mesh is the weighted combination of the two planar solutions. Take for example a $\xi\eta$-plane solid surface at $k=1$ from which curvature information is required to construct the grid. Mathematically the expression for interior grid point location can be written

$$\bar{r}_{ik} = (1 - \epsilon)[f_1 \sum k_{im} r_{im} + f_2 \sum k_{ms} r_{ms}] / [f_1 \sum k_{m1} + f_2 \sum k_{ms}] + \epsilon p$$

(3.2)

where

$$f_1 = e^{C_A \xi}, \quad f_2 = e^{C_A \eta}.$$

In equation 3.2, $k_{m1} = \frac{1}{2} k_m e^{-C_A \xi}$ and $k_{m2} = \frac{1}{2} k_m e^{-C_A \eta}$ for the volume edges including the endpoints $k=1$ and $k=1$, and $k_{m1} = k_{m2} = k_m$ otherwise. The index $m$ in the summation ranges over the grid points adjacent to $ijk$ in the $\xi\eta$-plane (indices $j,k$) and index $m$ in the $n\zeta$-plane (indices $j,k$). The constant $C_A$ is a gain factor that adjusts sensitivity to surface curvature. A value of $C_A=20$ seems to work well for many geometries. The surface curvature parameters $C_A$ and $A_A$ can be arrived at by considering, for example, the surface grids of Figure 1 in the $\xi$ ($j$ index) direction. A measure that accounts, at grid $i,j,k$, for convex or concave curvature at the surface point $ij$ and $k=1$ is

$$\Lambda_j = \frac{\sin \theta}{1 - \cos \theta}.$$

In terms of grid geometry this is

$$\Lambda_j = \frac{\Delta r_{j1} \cdot (\Delta r_{j1} \times \Delta r_{j1})}{|\Delta r_{j1}|^2}$$

$$\Lambda_k = \frac{\Delta r_{k1} \cdot (\Delta r_{k1} \times \Delta r_{k1})}{|\Delta r_{k1}|^2}$$

where

$$\Delta r_{j1} = r_{j+1} - r_{j-1}$$

$$\Delta r_{k1} = r_{k+1} - r_{k-1}$$

In practice the values of $\Lambda_j$ and $\Lambda_k$ are limited in upper and lower bound, currently $\Lambda_j = \min(C_A, 0.5C)$ or $\Lambda_j = \max(C_A, -12C)$ and similarly for $\Lambda_k$. An update of $f_1(\Lambda_j)$ and $f_2(\Lambda_k)$ are required once per time step for each surface or grid edge plane. This leaves only the computation of equation 3.2 to solve for the interior of the mesh. This is a linear equation typically requiring 1-2 iterations of a Gauss-Seidel procedure for moderate motions of the grid. Finally, the function $P$ defines an outward normal to the surface. Setting $E = \exp(-C_2 k)$ ensures that at least several grid points near the surface remain orthogonal to the surface.

IV. Aeroelastic method

The time marching simulation of the aeroelastic responses is obtained using the finite dimensional space state predictor-corrector method, described in references 8 and 19-20, that solves the de-coupled modal structural equations of motion. Several versions of the finite dimensional state space variable method are presented in those references. Of those the predictor-corrector method offers the best performance. The predictor step marches the structure using the solution of the modal equations at step $n$ to get the surface deflection at time step $n+1$. This solution is based on a second order (trapezoidal) extrapolation of the generalized aerodynamic forces at $n$ and $n-1$. This provides the surface shape for a recomputation of the fluid mesh and the fluid domain solution at $n+1$. After the solution of the fluid domain, the corrector step then solves the modal equations at the time step $n+1$ using the averaged forces at step $n$ and $n+1$.

V. Results

The first computational results illustrate robustness and efficiency. A comparative study was made for a pitching airfoil using the present deforming mesh scheme in which only the inner boundary is oscillated and that in which the entire grid was dynamically oscillated. The capability to dynamically oscillate the (rigid) grid has been developed and verified in a separate paper. Both mesh movement schemes used the same grid having dimensions $297 \times 105$. Test cases were the AGARD NACA 0012 cases 3 and 5. In each, the number of subiterations and the t-TS local CFL number were adjusted to give a convergence of the L2-norm of at least $10^{-4}$ at each time step. The results are virtually indistinguishable, as seen in Figure 2 for the Euler computation of AGARD Case 5. This is true even at 16 t.s. per cycle, for which $\Delta t=3.2$ (non-dimensionalized by speed of sound), requiring between 20-30 subiterations and some tuning of the local CFL number to adequately converge. An additional test of the time step convergence using the deforming mesh capability is also shown in Figure 2. The lift coefficient shows no discernible variation, while the moment coefficient, although overall nicely convergent, reveals an anomalous lack of convergence around zero degrees (upward motion). At 128 time steps per cycle, any difference in the deforming mesh and rotating frame solutions is imperceptible with the exception of moment coefficient on the downward side of its motion at around-2 degrees. (Figure 3) In Figure 4 a comparison of Navier-Stokes solutions at 64 and 2048 time steps per cycle of airfoil oscillation for AGARD case 3 is presented. This case involves slight trailing edge separation. This, and all the computations to follow, are made with the Spalart-Allmaras turbulence model. As can be seen in the figure, very slight differences between the 64 t.s./cycle solution and that at 2048 t.s./cycle only show up during trailing edge separation just after the start of pitch down.

The preceding Euler computations afford the opportunity to assess performance. The grid deformation scheme, the added terms for volumetric change in the Euler equations and the recomputation of cell volumes combined add slightly less than 4 percent CPU time (128 t.s./cycle case, 6 subiterations) over that of the unsteady code with rotating frame. This is with the grid, volumes and metrics updated at each time step. The percent of CPU time is, of course, reduced if larger time
steps and more subiterations are used. Total code performance is approximately 11.83 µsec/cell/iteration C-90 for the two-dimensional Euler computations.

The final results of this paper are aimed at assessing what is required to compute steady and unsteady two and three dimensional spoiler flows. Specifically, the issues addressed are the geometric modeling of 2-D and 3-D spoilers and the convergence and time step behavior of the present fluid and aerelastic solvers. In each of the following cases the airfoil/wing-spoiler geometry are based on that used in the Benchmark Active Controls Technology (BACT) NACA 0012 test. This allows a comparison of the present results with the BACT experimental database of oscillating spoiler cases. The spoiler is modeled here as a ramp and backward facing step. A two dimensional version is shown in Figure 5. The three dimensional version, seen in Figure 7, is modeled as a backward facing wedge having finite spanwise width. As can be seen in both figures, the backward step has a slight slope modeled by spacing three grid points between the spoiler trailing edge and the wing surface. This clearly does not model the cavity beneath the spoiler, nor the gap between the spoiler and flap leading edge. As the spoiler moves the actual spacing over the backward step is constrained via arc length to remain more or less true to the original grid spacing over the wing. Spoiler motion is modeled with rigid body rotation about the spoiler hinge line. The typical approach to modeling the spoiler deflection is to shear the wing surface grids or, at best, to use a two coordinate modal control surface motion model, either of which result in surface warping. The present approach embodies true control surface motion.

Before comparing computed results with the steady and unsteady experimental data, unsteady Navier-Stokes solutions (and grids for a 2-D configuration) are presented in Figures 5 and 6. This computation investigates time step convergence, which is important before launching into the more costly 3-D oscillating spoiler computations. These computations are especially challenging since they involve the growth and collapse of a large region of recirculating flow. They require significantly more subiterations (50-60 at 64 time steps per cycle) and tuning of the T-S CFL (for extremely large physical time steps) to reach reasonable accuracy at each time step. Nevertheless, the convergence at each time step was carefully monitored. The L2-norm was maintained at approximately 10^-5 or better at each time step. As can be seen in Figure 6, there is no discernable difference in the lift and moment coefficients in going from 1024 to 64 time steps per cycle of spoiler oscillation. However, the out-of-phase part of the transform of the unsteady pressure coefficient shows some reduction in accuracy for the larger time steps. While all the computations appear to be sufficiently accurate for engineering purposes, it is clear that for high accuracy, 260-300 t.s./cycle or more are required. This is true, say, for simulation of response to control system input at a condition near flutter onset, which is typically sensitive to the phase angle of the aerodynamic forcing.

A comparison of steady and unsteady computed results will now be made with experimental data. The purpose here is to assess the computational model for a surface geometry having moderate complexity. The cases computed are for both a 3-D steady and oscillating spoiler on the rectangular NACA 0012 wing used in the Benchmark Active Controls Technology (BACT) Test. As discussed earlier, surface geometry and grids are shown in Figure 7. The deflection in the steady case is 0° = 15 degrees. In all other computations viscous terms were included only in the direction normal to the solid surface. It was found after the fact that the flow around the trailing edge of the spoiler is moderately altered when streamwise viscous terms are included. For this reason the steady 3-D spoiler calculation included viscous terms in all three index directions. Resolving the grid at the trailing and spanwise edges of the spoiler and the addition of the streamwise viscous terms results in a better match with experiment near the spoiler trailing edge.

The grid for the steady case has somewhat finer resolution than that used in the unsteady 3-D computations. It has dimensions 201 x 73 x 73. Steady pressure coefficients at span locations of 40% and 60% are shown in Figure 8. These are just inboard and at mid span of the spoiler which has spanwise edges at 45% and 75% span. The pressure distribution in the region of the spoiler and its wake show quite good agreement with experiment. One minor exception is at 60% span where a slight divergence is observed just ahead of the spoiler trailing edge and just ahead of the shock. The solution at the 40% span location is off somewhat from 0.6c to 0.75c. This may be due to an inadequate geometric modeling of the spoiler side edges resulting in an improper side wall and vortex development. Figure 9 compares the computed real and imaginary unsteady pressure coefficients for an oscillating spoiler case with experiment. The grid used for this computation was 153 x 53 x 61 in dimension. The mean spoiler angle is 5 degrees and the amplitude of motion is 4.5 degrees. This was computed with 128 t.s./cycle for a reduced frequency based on semi-chord of k=0.1088 (9.56 Hz). An obvious notable difference between the computations and experiment is observed in the real part of the unsteady pressure coefficient at 60% span. While not verified, experience with steady cases indicates that some of this difference may be due to the thin-layer viscous approximation used. The other significant difference is in the imaginary part of the unsteady pressure coefficient around the trailing edge of the spoiler at 60% span.

The simulation of an aerelastic transient induced by a single oscillation of a 2-D spoiler for an airfoil subject to pitch and plunge is presented in Figures 10-12 and Tables 1 and 2. This case provides a nominal test of the performance of the complete fluid-mesh-structure solution technique. The aerelastic response is due to a single large-scale asymmetric oscillation of the spoiler. The spoiler was oscillated at approximately the free vibration frequency of the plunge mode. The dynamic pressure (q = 20 psf) is low enough to give a moderate aerelastic response for this size of spoiler motion. After an aerelastic response using the mesh shown in Figure 5 was computed, it was found that the aerelastic transient response was sensitive to grid spacing asymmetry between the upper and lower surfaces. A mesh that is symmetric and has the same spacing over both the upper and lower surface was used as the final grid. This grid had dimensions 319 x 105. Furthermore, the quality of the grid especially around the spoiler region was found to considerably impact the time step convergence of the aerelastic solutions. After some effort at improving grid quality, the described solutions were obtained.

The solutions of Figure 10 give transient pitch and plunge response calculated with four time step sizes. Note that the maximum CFL number for the largest time step (Δt = 5.859) is approximately 2x10^-2. The convergence of the plunge mode response with time step size reduction can be clearly seen in the detailed inset figure. Table 1 summarizes data from these runs. The fluid domain subiterations shown in this table are what were required to converge the solution (L2 norm of density) to around 10^-2 at each time step. Between 2048 and 8192 t.s/plunge cycle there is virtually no difference in the
damping ratios. Between the largest and smallest time steps the difference is less than 1%. The convergence of the aeroelastic response with number of subiterations at the largest time step can be seen in Figure 11. It appears from this figure that 32 subiterations of the fluid domain per time step represent a nearly converged solution. One can also see from this figure and from Table 2 that if less accuracy is required, fewer subiterations will also result in a solution. On the other hand, a computation that was performed using only a physical time step (t-TS subiteration) was unstable for time steps larger than \( \Delta t = 0.001 \). In fact, the solution at \( \Delta t = 0.0012 \), shown in Figure 12, diverged shortly after the final time step shown. Even if it had not diverged, this small time step makes this problem solution infeasible with the t-TS method, and necessitates CFL based local time step subiteration (identified as t-TS in the figure). The accuracy of the t-TS solution is clearly not established with only one computation. However, if the solutions with t-TS subiterations are converging to the correct solution, as appears from the figure, the t-TS solution even at \( \Delta t = 0.001 \) does not appear to be more accurate than the t-TS solution at a time step 100 times larger.

VI. Concluding remarks

A numerical scheme for the Euler and Navier-Stokes equations in a general dynamically deforming coordinate system that incorporates the geometric conservation law into the governing equations has been evaluated in this paper. By casting the equations in the form shown here, a somewhat more economical scheme is obtained for problems requiring deforming meshes. A structured mesh scheme, based on the spring analogy, has been applied to moderately complex surface geometry. Its performance has been assessed for time steps spanning a large range, applied to problems from oscillating two-dimensional airfoils and spoilers to a wing having a three-dimensional oscillating spoiler surface. The spoilers have been modeled as a ramp and backward step and the oscillation by true rigid body rotation. The mesh scheme does not appear to result in mesh entanglement for the geometry and mesh motion considered here even with extremely large time step (CFL=2×10^6). This is due both to the fact that the fluid mesh near the moving surface undergoes motion prescribed by the surface and to the selective relaxing/stiffening of the spring stiffness in the grid deformation scheme. Even at very large time steps the scheme appears to retain considerable accuracy if a sufficient number of t-TS subiterations are used. In the case of strongly nonlinear flow driven by spoiler oscillation, numerical results have shown that the error due to increasing time step that does arise is mainly in the out-of-phase component. This has implications for the accurate computation of flutter onset and other aeroelastic phenomena. The steady and unsteady computed results for the three-dimensional spoiler and wing qualitatively compare well with experiment, with the exception of the fairly small out of phase component of the unsteady pressure coefficient. Finally, aeroelastic capability has been demonstrated for the damped transients of a wing having pitch and plunge modes. The case computed was at low dynamic pressure (\( q_\infty = 20 \text{ psi} \)) in which the modes were initially perturbed by a single large-scale oscillation of the spoiler surface. The results show that accuracy at very large time step sizes, similar to that used in the oscillating spoiler computations, can be achieved here as well. This suggests that for these cases and solution method, the convergence of the fluid solution is the dominant factor in the accuracy of the structure-fluid solution. Again, this is possible because of the t-TS flow field solution. Because of stability limitations, the largest time step possible with t-TS subiteration was at least 600 times smaller than the largest time step with t-TS subiteration.

References


---

**Figure 1.** Orientation of surface and interior grids.

**Figure 2.** Time step study for deforming mesh scheme. AGARD NACA 0012 Case 5, NACA 0012, $M_w = 0.755$, $\alpha = 0.016 + 2.51 \sin(2ktM_w)$ degrees, $k = 0.081$
Figure 3. Comparison of results for rotating frame and deforming mesh.
AGARD NACA 0012 Case 5, NACA 0012, $M_\infty = 0.755$, 
$\alpha = 0.016 + 2.51 \sin(2ktM_\infty)$ degrees, $k = 0.081$

Figure 4. Time step study for deforming mesh viscous computation.
AGARD NACA 0012 Case 3, $M_\infty = 0.60$, $\alpha = 4.86 + 2.44 \sin(2ktM_\infty)$ degrees, $k = 0.081$

Figure 5. Two-dimensional mesh for oscillating spoiler study.

Figure 6. Time step study for 2D NACA 0012 airfoil with oscillating spoiler.
$M_\infty = 0.77$, $\alpha = 0$ degrees, $\delta_o = 5+4.5 \sin(2 M_\infty t)$ degrees.
Figure 7. BACT model upper surface and mesh.

Figure 8. Comparisons of BACT model steady pressure coefficients. $M_x = 0.77$, $\alpha = 0$ degrees $\delta_p = 15$ degrees.
Figure 9. Comparisons of BACT model unsteady pressure coefficients.
\( M_{\infty} = 0.77, \ \alpha = 0 \) degrees \( \delta_p = 5+4.5 \sin(2 M_{\infty}kt) \) degrees.

Figure 10. Computed aeroelastic response due to a single spoiler oscillation (9.5 degree excursion), 2D BACT model. \( M_{\infty} = 0.77, \ \alpha = 0 \) degrees, \( q_{\infty} = 20 \) psf, \( U_{\infty} = 373 \) fps. Figure 10.

Table 1. Computed damping ratios for two aeroelastic modes, (Same model and condition as Figure 10).
Figure 11. Computed aeroelastic response due to a single spoiler oscillation. (Same model and condition as Figure 10, $\Delta t=0.5859$.)

Table 2. Effect of number of subiterations on damping ratio, (Same model and condition as Figure 10)

<table>
<thead>
<tr>
<th>$\Delta t$</th>
<th>No. of fluid domain sub-iterations</th>
<th>$\xi_h$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5859</td>
<td>128</td>
<td>32</td>
</tr>
<tr>
<td>0.5859</td>
<td>128</td>
<td>20</td>
</tr>
<tr>
<td>0.5859</td>
<td>128</td>
<td>8</td>
</tr>
</tbody>
</table>

Figure 12. Comparison of t-TS and t-TS time stepping for aeroelastic response (same model and condition as Figure 10).