2E8 Flutter Simulation of a Transonic Wing

Takaaki Sato, Shigeru Obayashi and Kazuhiro Nakahashi
Tohoku University
Department of Aeronautics and Space Engineering,
Sendai 980-8579-01, Japan
Email: sato@ad.mech.tohoku.ac.jp, :obayashi@ieee.org

Key Word: Unsteady Flows

Abstract

Flutter simulation of a transonic wing has been presented using a moving grid system. At first, a Navier-Stokes code has been validated by comparing computed solutions with experimental data for the oscillatory motion of rectangular wing. Then, flutter simulation of a high-aspect-ratio swept back wing has been presented. Aeroelastic responses are computed using the modal analysis based on the finite-element method. The computed flutter boundaries are obtained and compared with NAL (National Aerospace Laboratory) flutter tunnel test.

Introduction

To advance the safety of aircraft, the capability to predict unsteady loads such as maneuver loads and gust loads on the aircraft will be needed with greater accuracy. Because of the high cost and risk involved, however, it is not practical to conduct a large number of aeroelastic wind-tunnel tests. By complementing such expensive experiments with computational methods, the overall cost of the development of an aircraft can be considerably reduced.

To estimate unsteady loads, the rigid body assumption of the aircraft may not be good enough. The aircraft should be treated as a flexible body. Several studies have been reported by coupling CFD analysis with Computational Structural Dynamics. For example, Guruswamy developed a Naveir-Stokes code for aeroelaststic simulations [1][2]. Then to reduce the computational time, Byun and Guruswamy developed a parallel version of the code[3]. The flutter calculations based on a parallel, multiblock, multigrid flow solver by Liu, et al[4]. Kheirandish et al. also presented flutter simulation[5]. However, these codes are not in public domain, and the access to such codes is limited. Also, they concentrated on calculating the flutter boundary. There are not many reports estimating maneuver loads or gust loads.

Our final goal is to simulate various unsteady aeroelastic phenomena. In this paper, the computational aeroelastic method is developed and validated with experiment as a milestone.

To verify the present code at first, unsteady flows over rectangular wing undergoing prescribed oscillatory motions[6] are computed. The unsteady code solves the Navier-Stokes equations using the moving grid systems. Then, flutter simulation of a high-aspect-ratio swept back wing is presented. Structural responses are loosely coupled with CFD analysis. The modal data is generated by the finite-element method[5].

Numerical Algorithms

The thin-layer Navier-Stokes equations used in this study can be written in conservation-law form in a generalized body-conforming curvilinear coordinate system for three dimensions as follows:

$$\partial_{\hat{o}}\hat{\mathbf{Q}} + \partial_{\hat{i}}\hat{\mathbf{E}} + \partial_{ç}\hat{\mathbf{F}} + \partial_{æ}\hat{\mathbf{G}} = \frac{\mathbf{1}}{\mathbf{R_e}}\partial_{æ}\hat{\mathbf{G}}^{n} \tag{1}$$

where $t = t$, $x = x(t,x,y,z)$, $h = h(t,x,y,z)$, and $z = z(t,x,y,z)$. The turbulent viscosity is evaluated by the Baldwin-Lomax algebraic eddy-viscosity model.

The governing structural equations of motion of a flexible wing are using the Rayleigh-Ritz method[6]. In this method, the resulting aeroelastic displacements at any time are expressed as a function of a finite set of assumed modes.

It is assumed that the deformed shape of the wing can be represented by a set of discrete displacement vector $\{\mathbf{d}\}$ can be expressed as

$$\{\mathbf{d}\} = [\mathbf{ö}]\{\mathbf{q}\} \tag{2}$$

where $[\mathbf{ö}]$ is the modal matrix and $\{\mathbf{q}\}$ is the generalized displacements vector. The matrix form of the structural equations of motion is

$$[\mathbf{M}]\{\ddot{\mathbf{d}}\} + [\mathbf{C}]\{\dot{\mathbf{d}}\} + [\mathbf{K}]\{\mathbf{d}\} = \{\mathbf{F}\} \tag{3}$$

where $[\mathbf{M}]$, $[\mathbf{C}]$, and $[\mathbf{K}]$ are modal mass, damping, and stiffness matrices, respectively. Each matrices and modal data are generated from a finite-element analysis. $\{\mathbf{F}\}$ is aerodynamic force vector and it is obtained from integrating aerodynamic forces acting on the wing surface.

The structural equation of motion (3) is solved by a numerical integration technique based on the Runge-Kutta sheme. Using the resulting displacements, all computational grids are moved using the grids generation system described in the following section. Then the flow field is calculated on the new grid and the resulting aerodynamic forces are integrated again. Iterating this cycle, aeroelastic simulation is performed.

Grids Deformation Systems

The present CFD grid uses the C-H grid topology. The C-H grid is deformed every time based on the wing deformation as follows.

1) Obtain a camber surface of the initial configuration of wing.
2) Integrate the structural equation of motion and obtain the generalized displacement for each mode.
3) Deform the camber surface using the generalized displacements. Interpolate the surface with a spline curve in the chordwise direction and with a liner interpolation in the spanwise direction.
4) Add wing thickness to the deformed camber surface and determine the new wing surface grid points.
5) Generate the new computational grid based on the new surface configuration algebraically.

Results

Oscillating Rectangular Wing

To verify the present code, unsteady flows over rectangular wing undergoing prescribed oscillatory motions are computed. It has NACA64A010 airfoil section and an aspect ratio of 4. The unsteady data are given when a rigid wing is oscillating in the pitching motion, $\alpha(t) = \alpha_m - \bar{\alpha}\sin(w t)$ about the axis at *x/c* = 0.5,

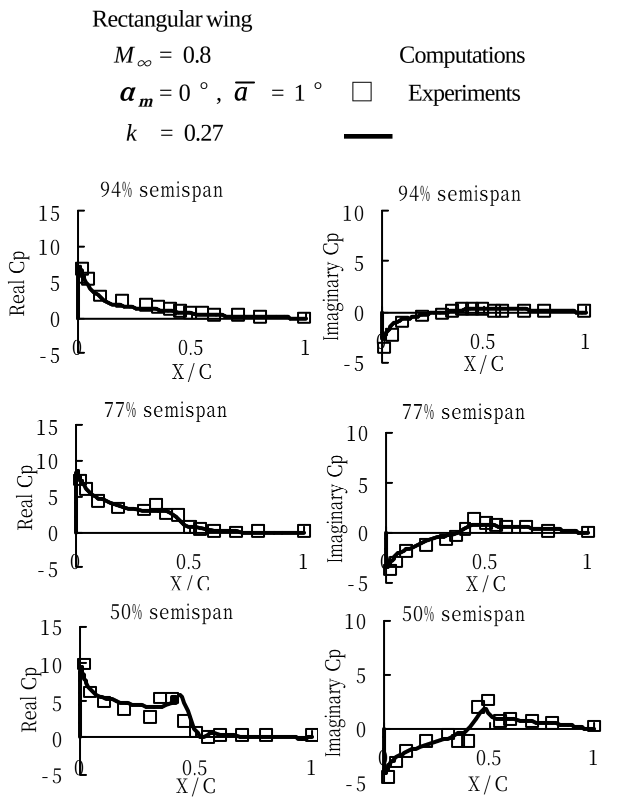

Fig. 1 Comparison of computed upper surface unsteady pressures with experiment over the rectangular wing.

where *c* is the chord length and ω is the pitching frequency in radian per second. The flow is computed at M_∞ = 0.8 with a mean angle of attack α_m = 0 deg, a pitch amplitude $\bar{\alpha}$ = 1 deg, and a reduced frequency k = 0.27 ($k = w\, c / U_\infty$). Unsteady computations are started from the corresponding steady-state solution.

Figure 1 shows the comparison of real and imaginary parts of the first Fourier component between the computed and measured unsteady upper surface pressure coefficients of the wing at various spanwise locations with a time step size of 3600 steps/cycle. The results show a good agreement with experimental data[6] and the inboard shock wave motion is captured clealy. Throughout the test case presented here, the accuracy of the present unsteady code is confirmed favorably.

Flutter Simulation

Aeroelastic-response analyses are conducted for a high-aspect-ratio swept back wing shown in Fig 2. This configuration is based on the preliminary design of the YXX transport project and a computational model is taken from the NAL (National Aerospace Laboratory) flutter tunnel model with 1/45 of a full scale aircraft. Aspect ratio, taper ratio and thickness-to-chord are 10, 0.324 and 16%, respectively. It has supercritical airfoil section made of metal spar and urethane panel[5].

The wing is modeled by the plate. Figure 3 shows the mode shapes and frequencies of the first six normal modes for the wing. Using the normal modal data shown in Fig. 3, aeroelastic responses were computed by integrating the flow equation and the aeroelastic equation.

The flow conditions are M_∞ = 0.70, R_e = 2.4×10^6, angle of attack α = 2 deg and several dynamic pressure are picked up around the flutter boundary obtained by the NAL flutter tunnel test. Aeroelastic computations are started from corresponding steady state solutions of the rigid wing.

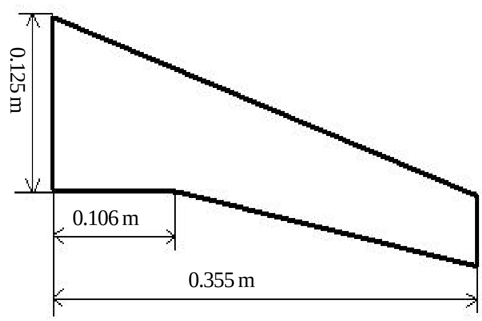

Fig. 2 YXX wing planform

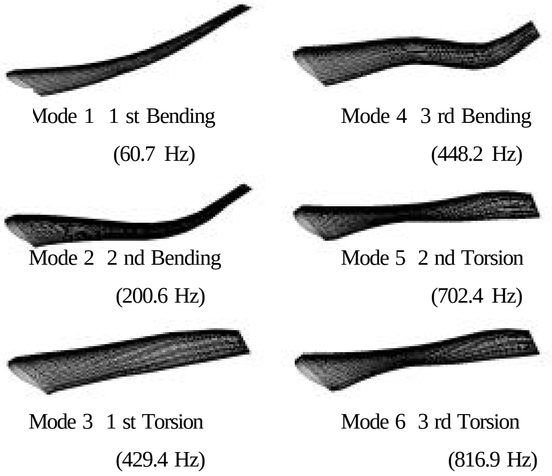

Fig. 3 Mode shapes and frequencies

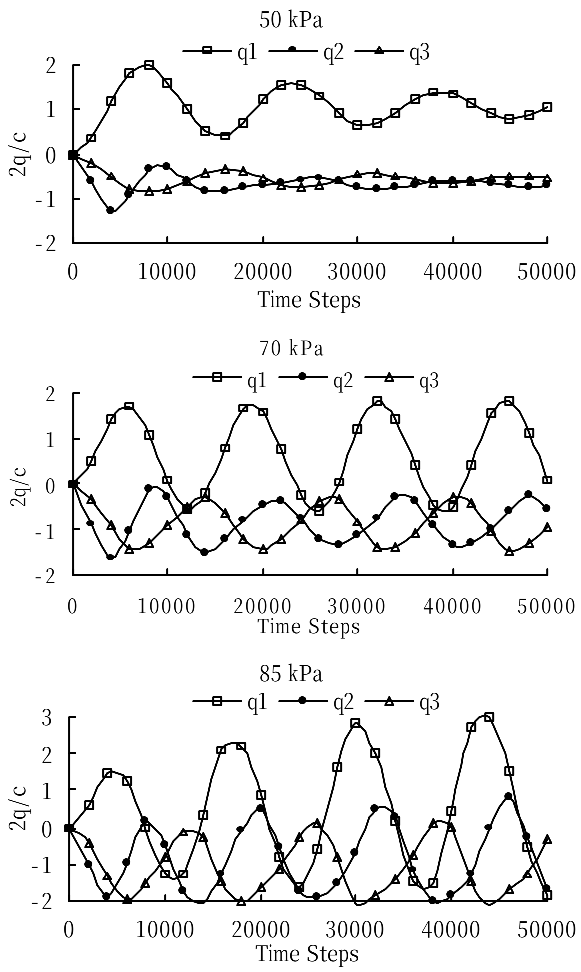

Fig. 4 The time history of generalized displacements of the fist three modes at the dynamic pressure 50kPa, 70kPa and 85kPa.

Figure 4 shows the time history of generalized displacements of the fist three modes at the dynamic pressures 50kPa, 70kPa and 85kPa. When the dynamic pressure is 50kPa, each generalized displacement decays with time, indicating that the aeroelastic system is stable at this condition. At higher dynamic pressures, the system becomes less and less stable until the displacements diverge as shown at 85kPa. When the dynamic pressure is 70kPa, all generalized displacements neither converge nor diverge. This is considered at the flutter boundary and it agrees well with the NAL flutter tunnel test.

Figure 5 shows time histories of actual displacements and aerodynamic forces at three spanwise locations at 70kPa. It shows that when the wing bends up, C_L reduces due to the wing twist. Aerodynamaic coefficients vary periodically along with the wing deformation at the flutter boundary.

Figure 6 shows the Mach contours of at 95% semi span at 70kPa during the oscillation. As the deformation becomes larger, the flow separates from the leading edge.

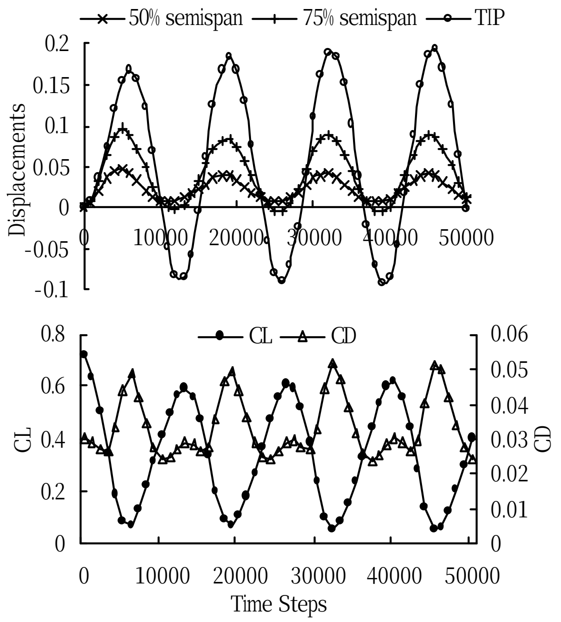

Fig. 5 Time histories of displacements, C_L and C_D at 70kPa.

Conclusions

Flutter Simulation of a transonic wing was performed associated with a moving grid system. At first, unsteady flows over rectangular wing undergoing prescribed oscillatory motions are computed. The results show good agreements with experimental data.

Then, flutter simulation of a high-aspect-ratio swept back wing is presented. Structural responses are computed using the modal analysis based on the finite-element method. The computed flutter boundary agrees well with the NAL flutter tunnel test.

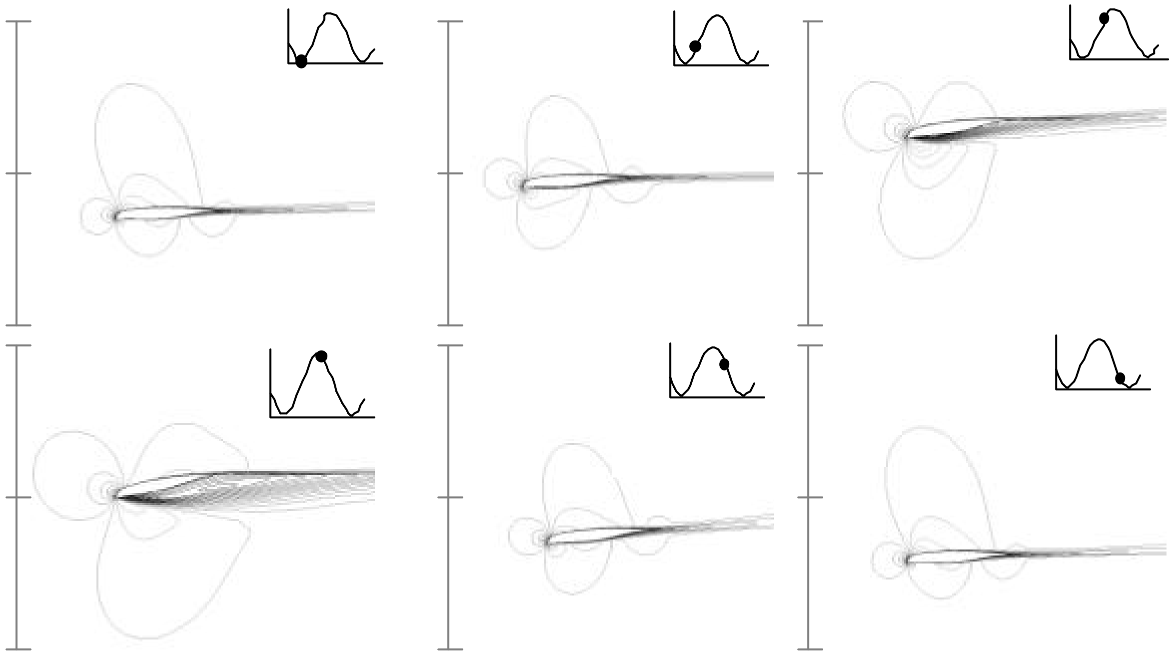

Fig. 6 The Mach contours of at 95% semi span at 70kPa during the oscillation.

Acknowledgements

This research was supported by Japan Aircraft Development Corporation. The authors would like to thank Jiro Nakamichi at National Aerospace Laboratory, Chofu, Japan, for his helpful discussions and for providing aeroelastic experimental data. This research was partly supported by Bombardier Aerospace, Toronto.

References

[1] G. P. Guruswamy, "Integrated Approach Coupling of Structures and Fluids," AIAA Jornal, Vol. 27, No. 6, 1989, pp. 788-793.

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

[3] C. Byun and G. P. Guruswamy, "Aeroelastic Computations on Wing-Body-Control Configurations on Parallel Computing," AIAA Paper 96-1389, April, 1996.

[4] F. Liu, J. Cai, Yhu, A. S. F. Wong, and H. M. Tsai, "Calculation of Wing Flutter by a Coupled CFD-CSD Method," AIAA Paper 2000-0907, January, 2000.

[5] H. R. Kheirandish, G. Beppu and J. Nakamichii, "Numerical Flutter Simulation of a High-Aspect-Ratio Swept-Back Wing Based on the Navier-Stokes Equations," Journal of the Japan Society for Aeronautical and Space Sciences, Vol 45, No. 525, Oct. 1997 (in Japanese).

[6] S. Obayashi, G. P. Guruswamy and P. M. Goorjian, "Streamwise Upwind Algorithm for Computing Unsteady Transonic Flows Past Oscillating Wings," AIAA Jornal, Vol. 29, No. 10, 1991, pp. 1668-1677.

[7] E. H. Dowell, "A Modern Course in Aeroelasticity," Kluwer Academic Publishers, Dordrecht, 1995.