Delayed Detached Eddy Simulation of Wing Flutter Boundary Using High Order Schemes

Jia-ye Gan ^{*}, Hong-Sik, Im [†]

Xiang-ying, Chen [‡], Ge-Cheng Zha[§]

Dept. of Mechanical and Aerospace Engineering

University of Miami

Coral Gables, Florida 33124

Crystal L. Pasiliao [¶]

AFRL/RWWV, Munition Aerodynamic Sciences Branch

101 West Eglin Blvd, Suite 330

Eglin AFB FL 32542

Abstract

This paper conducts Delayed Detached Eddy Simulation (DDES) of a 3D transonic wing flutter using a fully coupled fluid/structure interaction (FSI) with high order shock capturing schemes. Unsteady 3D compressible Navier-Stokes equations are solved with a system of 5 decoupled structure modal equations in a fully coupled manner. The low diffusion E-CUSP scheme with a 5th order WENO reconstruction for the inviscid flux and a set of 2nd order central differencing for the viscous terms are used to accurately capture the shock wave/turbulent boundary layer interaction of the vibrating wing. The predicted flutter boundary at different free stream Mach numbers achieves very good agreement with experiment, including the supersonic flutter boundary point, which is often substantially over-predicted by most of the other

^{*}PhD Student, AIAA Member

[†]PhD., Currently a Project Engineer at Doosan AST America, LLC

[‡]PhD., Research Associate Professor

[§]Professor, AIAA Associate Fellow, E-mail: gza@miami.edu

[¶]PhD., Basic Research Lead

simulations based on RANS models. Some new observations and explanation are given for the sonic dip mechanism. The transonic dip phenomenon is related to the anticlimax contribution of the second mode, which is caused by the complicated shock oscillation that decreases the wing pitching moment sharply. At the flutter boundary including the sonic dip, no flow separation due to shock/boundary layer interaction is observed.

Nomenclature

d	=	distance from the closest wall
f_d	=	delayed LES function in DDES
J	=	Jacobian of the coordinate transformation
I	=	identity matrix
ξ_t, η_t, ζ_t	=	ξ, η, ζ component of grid moving velocity
U, V, W	=	ξ, η, ζ component of contravariant velocity
$U_{i,j}$	=	velocity gradient
x, y, z	=	Cartesian coordinates
δ_{ij}	=	Kronecker delta
Pr	=	Prandtl number
Pr_t	=	turbulent Prandtl number
M_{is}	=	isentropic Mach number, $\sqrt{((\frac{p_0}{p})^{\frac{\gamma-1}{\gamma}} - 1) \frac{2}{\gamma-1}}$
ν	=	kinematic viscosity
δ_x	=	x-component of displacements of the structure surface
δ_y	=	y-component of displacements of the structure surface
δ_z	=	z-component of displacements of the structure surface
ξ, η, ζ	=	generalized coordinates

Subscripts

i, j, k	=	indices
∞	=	reference variable at free stream

Flutter parameters

b_s	=	wing root semi-chord
$\bar{m}$	=	wing panel mass
$\bar{V}$	=	frustum volume
V^*	=	reduced velocity, $\frac{U_\infty}{b_s \omega_\alpha}$
V_f	=	flutter speed index, $\frac{U_\infty}{b_s \omega_\alpha \sqrt{\bar{\mu}}}$
ω_α	=	1st torsional mode natural frequency
$\bar{\mu}$	=	mass ratio, $\frac{\bar{m}}{\bar{V} \rho_\infty}$
ω_j	=	natural frequency for j th mode
ζ_j	=	modal damping ratio for j th mode

Abbreviations

<i>CUSP</i>	=	convective upwind and split pressure
<i>DES</i>	=	detached eddy simulation
<i>DDES</i>	=	delayed detached eddy simulation
<i>FSI</i>	=	fluid/structure interaction
<i>WENO</i>	=	weighted essentially non-oscillatory

1 Introduction

Flutter is a self-excited aeroelastic instability that occurs when the aerodynamic damping is negative and could lead to structural failure. Flutter is a crucial problem in aircraft reliability and should be considered in the early phase of aircraft aeroelasticity design. Experimental testing of aircraft aeroelasticity is in general very expensive. Therefore, the tools for aeroelastic flutter prediction are very important to reduce cost in aircraft design.

Flutter may occur at any Mach number from subsonic to supersonic. There may be complicated flow phenomena such as flow separation and shock wave/turbulent boundary layer interaction(STBLI) during flutter. Various flow models have been developed to predict the flutter boundary of aircraft. The classical linear models[1] including piston theory and full potential flow models are widely used in aeroelastic design for their advantage of computational efficiency. The disadvantage of linear flow models is that they fail to accurately capture the location and strength

of local shock motion in transonic and supersonic regime.

To predict the transonic or supersonic flutter boundary with better accuracy, tools based on nonlinear Euler equations have been developed. Bendiksen et al.[2] study the transonic flutter of typical section wing models by solving the Euler equation coupled with a two degrees of freedom of structural dynamic equations. Reasonable flutter results are obtained in the regions with no strong shock. Rausch et al.[3] predict flutter boundary of a set of configurations by solving unsteady Euler aerodynamic equations on unstructured grid. Good agreement with experiment is obtained in the predicted flutter boundary for subsonic flow. However the predicted flutter characteristics in transonic regime are not satisfactory. Since the viscous effect on these flutter boundary prediction is neglected, neither potential flow model nor Euler equations are suitable for understanding the STBLI and the mechanism that causes flutter.

Prananta et al.[4] report the results of aeroelastic simulations using the Euler and Navier-Stokes solvers. Their calculations show that the viscous effect plays a very important role in determining the transonic dip and flutter boundary for a NACA64A010 airfoil. Lee-Rausch et al.[5] also study the effect of viscosity on the flutter boundary of AGARD wing 445.6 by comparing the Euler and Navier-Stokes results over a range of Mach numbers from 0.499 to 1.14. Their calculation shows a significant viscous effect on the supersonic flutter boundary. Viscous effects and the STBLI may induce flow separation and shock wave oscillation that affect flutter speed. Isogai[6] considers that the drop of transonic flutter boundary is due to the presence of part chord shock and its oscillation that is not in phase with the airfoil motion. Bendiksen[7] attributed the sonic dip to the shock motion transition from Tijdeman type A to type B[8], which throttles the aerodynamic work done on the wing and forms single degree of freedom flutter occurring at transonic speed.

To consider the viscous effect of transonic flutter prediction, Reynolds averaged Navier-Stokes(RANS) equations are often solved for the flow solver. However, RANS models in general can not accurately resolve the flow separation caused by shock boundary layer interaction since RANS models are based on flawed assumption that large vortex structures are isotropic. Morgan et al.[9] introduce shock unsteadiness correction to improve RANS model's accuracy on shock boundary layer interaction. Large Eddy Simulation(LES) is more plausible than RANS models since LES directly simulates the large eddy structures and models the small eddies, which are more isotropic. However, LES is much more CPU expensive due to the excessive mesh size required.

The hybrid RANS/LES approach, Detached eddy simulation (DES) suggested by Spalart[10],

is developed for complex turbulence problems as a compromise between prediction accuracy and CPU efficiency. In DES, a RANS model is used in the near wall regions to reduce the grid density, and the LES is used outside the near wall regions to deal with massively separated flows. Spalart et al[10] suggested delayed DES(DDES) in 2006 to overcome the modeled stress depletion(MSD) problem of the original DES when the mesh is thin compared with boundary layer thickness. In DDES, a blending function similar to the one used by Mentor and Kuntz[11] for the SST model is introduced to limit the length scale of DES97 to ensure the transition of RANS to LES to be independent of grid spacing.

The other very important issue of FSI simulation is the artificial dissipation from the CFD schemes, which affects the prediction of aerodynamic damping to the structure. FSI simulation includes the resolution of acoustic waves and high order mode of structure vibration with high reduced frequency, which is a small scale wave phenomenon. Most of the current CFD simulations for FSI use 2nd or 3rd order schemes for the convective fluxes, which have high numerical dissipation due to the upwinding requirement to capture shock waves. The authors' group has developed high order shock capturing schemes for FSI with 5th order weighted essentially non-oscillatory (WENO) scheme for inviscid fluxes[12, 13, 14, 15, 16].

The advantage of high order low diffusion schemes is demonstrated in the simulation of the transonic NLR 7301 airfoil limited cycle oscillation (LCO). Since the LCO has bifurcation, some other research groups using 2nd order schemes only capture the amplitude about 10 times higher than the measured one in wind tunnel. Their schemes tend to be too diffusive to capture the LCO with the amplitude as small as 2/1000 of the airfoil chord. The simulation of Wang et al [14, 15] of the present author's group is the only one, and the first one, that accurately captures the small LCO amplitude measured in the experiment.

Recently, Chen et al. in the author's group[17] predict the transonic flutter boundary of AGARD wing 445.6 using delayed-detached-eddy simulation(DDES)[10]. In their study, time accurate Navier-Stokes equations are solved with a system of 5-decoupled structure modal equations in a fully coupled manner[18, 19]. The computed transonic flutter boundary agrees very well with the experiment. The same method was utilized by Im et al. in the author's group[20] to improve the prediction of the supersonic flutter boundary of the AGARD wing 445.6 model. Their results appear to be the first time that a numerical prediction of supersonic flutter boundary matches with experiment accurately. However, the flutter mechanisms in transonic and supersonic condition are

not investigated in [17, 20]. Furthermore, the complete AGARD wing 445.6 flutter boundary at different Mach numbers from subsonic to supersonic is not simulated in their work.

The purpose of this paper is to calculate the full AGARD wing flutter boundary based on the previous work in our group [17, 20, 21] to investigate the flutter mechanism using high order schemes and DDES with a fully coupled FSI. In particular, the focus is to understand the sonic dip phenomenon in the transonic regime. The work appears to be the first effort using DDES with high order shock capturing schemes to simulate transonic wing flutter. Benefiting from this high fidelity transonic flutter wing simulation, some new observations and explanation are given for the sonic dip mechanism, including the anticlimax of torsional mode and decrease of pitching moment at sonic dip due to shock oscillation.

2 Numerical Model for Flows

The FASIP (Flow-Acoustics-Structure Interaction Package) code is employed in this study for the fluid structural interaction simulation. The following is the numerical algorithms used in FASIP.

2.1 Navier-Stokes Equations

The spatially filtered unsteady compressible Navier-Stokes equations governing the viscous flows can be written in a normalized form in the generalized coordinates (ξ, η, ζ) .

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E}}{\partial \xi} + \frac{\partial \mathbf{F}}{\partial \eta} + \frac{\partial \mathbf{G}}{\partial \zeta} = \frac{1}{Re} \left(\frac{\partial \mathbf{E}_v}{\partial \xi} + \frac{\partial \mathbf{F}_v}{\partial \eta} + \frac{\partial \mathbf{G}_v}{\partial \zeta} \right) \quad (1)$$

where Re is the Reynolds number. The equations are non-dimensionalized based on airfoil chord L_∞ , free stream density ρ_∞ , velocity U_∞ , and viscosity μ_∞ . The conservative variable vector $\mathbf{Q}$, the inviscid flux vectors $\mathbf{E}$, $\mathbf{F}$, $\mathbf{G}$, and the viscous flux $\mathbf{E}_v$, $\mathbf{F}_v$, $\mathbf{G}_v$ are expressed as

$$\mathbf{Q} = \frac{1}{J} \begin{pmatrix} \bar{\rho} \\ \bar{\rho}\tilde{u} \\ \bar{\rho}\tilde{v} \\ \bar{\rho}\tilde{w} \\ \bar{\rho}\tilde{e} \end{pmatrix}, \mathbf{E} = \begin{pmatrix} \bar{\rho}U \\ \bar{\rho}\tilde{u}U + l_x\bar{p} \\ \bar{\rho}\tilde{v}U + l_y\bar{p} \\ \bar{\rho}\tilde{w}U + l_z\bar{p} \\ (\bar{\rho}\tilde{e} + \bar{p})U - l_t\bar{p} \end{pmatrix}, \mathbf{F} = \begin{pmatrix} \bar{\rho}V \\ \bar{\rho}\tilde{u}V + m_x\bar{p} \\ \bar{\rho}\tilde{v}V + m_y\bar{p} \\ \bar{\rho}\tilde{w}V + m_z\bar{p} \\ (\bar{\rho}\tilde{e} + \bar{p})V - m_t\bar{p} \end{pmatrix}, \mathbf{G} = \begin{pmatrix} \bar{\rho}W \\ \bar{\rho}\tilde{u}W + n_x\bar{p} \\ \bar{\rho}\tilde{v}W + n_y\bar{p} \\ \bar{\rho}\tilde{w}W + n_z\bar{p} \\ (\bar{\rho}\tilde{e} + \bar{p})W - n_t\bar{p} \end{pmatrix} \quad (2)$$

$$\mathbf{E}_{\mathbf{v}} = \begin{pmatrix} 0 \\ l_k \bar{\tau}_{xk} \\ l_k \bar{\tau}_{yk} \\ l_k \bar{\tau}_{zk} \\ l_k (\tilde{u}_i \bar{\tau}_{ki} - \bar{q}_k) \end{pmatrix}, \mathbf{F}_{\mathbf{v}} = \begin{pmatrix} 0 \\ m_k \bar{\tau} u_{xk} \\ m_k \bar{\tau} u_{yk} \\ m_k \bar{\tau} u_{zk} \\ m_k (\tilde{u}_i \bar{\tau}_{ki} - \bar{q}_k) \end{pmatrix}, \mathbf{G}_{\mathbf{v}} = \begin{pmatrix} 0 \\ n_k \bar{\tau}_{xk} \\ n_k \bar{\tau}_{yk} \\ n_k \bar{\tau}_{zk} \\ n_k (\tilde{u}_i \bar{\tau}_{ki} - \bar{q}_k) \end{pmatrix} \quad (3)$$

where where J is the Jacobian of the coordinate transformation, ρ is the density, p is the static pressure, and e is the total energy per unit mass. The overbar denotes a regular filtered variable, and the tilde is used to denote the Favre filtered variable. U , V and W are the contravariant velocities in ξ , η , ζ directions, and are defined as

$$U = l_t + \mathbf{l} \bullet \mathbf{V} = l_t + l_x \tilde{u} + l_y \tilde{v} + l_z \tilde{w} \quad (4)$$

$$V = m_t + \mathbf{m} \bullet \mathbf{V} = m_t + m_x \tilde{u} + m_y \tilde{v} + m_z \tilde{w} \quad (5)$$

$$W = n_t + \mathbf{n} \bullet \mathbf{V} = n_t + n_x \tilde{u} + n_y \tilde{v} + n_z \tilde{w} \quad (6)$$

where l_t , m_t and n_t are the grid velocity in ξ , η and ζ directions respectively. $\mathbf{l}$, $\mathbf{m}$ and $\mathbf{n}$ denote the normal vectors located at the centers of ξ , η and ζ interfaces of the control volume with their magnitudes equal to the surface areas and pointing to the directions of increasing ξ , η and ζ .

$$\mathbf{l} = \frac{\nabla \xi}{J}, \mathbf{m} = \frac{\nabla \eta}{J}, \mathbf{n} = \frac{\nabla \zeta}{J} \quad (7)$$

$$l_t = \frac{\xi_t}{J}, m_t = \frac{\eta_t}{J}, n_t = \frac{\zeta_t}{J} \quad (8)$$

Let the subscripts i, j, k represent the coordinates x, y, z and use the Einstein summation convention. By introducing the concept of eddy viscosity and sub-grid stress of LES to close the system of equations, the shear stress $\bar{\tau}_{ik}$ and total heat flux $\bar{q}_k$ in Cartesian coordinates can be expressed in tensor form as

$$\bar{\tau}_{ik} = (\mu + \mu_{DES}) \left[\left(\frac{\partial \tilde{u}_i}{\partial x_k} + \frac{\partial \tilde{u}_k}{\partial x_i} \right) - \frac{2}{3} \delta_{ik} \frac{\partial \tilde{u}_j}{\partial x_j} \right] \quad (9)$$

$$\bar{q}_k = - \left(\frac{\mu}{Pr} + \frac{\mu_{DES}}{Pr_t} \right) \frac{\partial \tilde{T}}{\partial x_k} \quad (10)$$

where the molecular viscosity $\mu = \mu(\tilde{T})$ is determined by Sutherland's law, and $\mu_{DES}(= \bar{\rho}\tilde{\nu}f_{v1})$ is determined by the DDES of turbulence[10] as explained in the following section. Eq.(9) and (10) are also transformed to the generalized coordinate system in computation. The equation of state as a constitutive equation relating density to pressure and temperature is given as follows;

$$\bar{\rho}\tilde{e} = \frac{\bar{p}}{(\gamma - 1)} + \frac{1}{2}\bar{\rho}(\tilde{u}^2 + \tilde{v}^2 + \tilde{w}^2) \quad (11)$$

For simplicity, all the bar and tilde in above equations will be dropped in the rest of this paper.

2.2 Delayed Detached Eddy Simulation of Turbulence[10]

In 2006 Spalart et al.[10] suggested an advanced hybrid RANS/LES turbulence, the delayed detached eddy simulation based on the Spalart-Allmaras(S-A) one equation model[22] which solves a transport equation for the working variable $\tilde{\nu}$. The turbulent eddy viscosity μ_{DES} related to working variable $\tilde{\nu}$ is given by

$$\mu_{DES} = \rho\nu_t = \rho\tilde{\nu}f_{v1} \quad (12)$$

where

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3} \quad (13)$$

$$\chi = \frac{\tilde{\nu}}{\nu} \quad (14)$$

In the S-A model, the transport equation for the working variable $\tilde{\nu}$ yields in generalized coordinate system as

$$\begin{aligned} \frac{\partial \frac{1}{J}\rho\tilde{\nu}}{\partial t} + \frac{\partial \rho\tilde{\nu}U}{\partial \xi} + \frac{\partial \rho\tilde{\nu}V}{\partial \eta} + \frac{\partial \rho\tilde{\nu}W}{\partial \zeta} = \frac{1}{Re} \left(\frac{\partial \frac{\rho}{\sigma}(\nu + \tilde{\nu})(\mathbf{l} \bullet \nabla \tilde{\nu})}{\partial \xi} \right. \\ \left. + \frac{\partial \frac{\rho}{\sigma}(\nu + \tilde{\nu})(\mathbf{m} \bullet \nabla \tilde{\nu})}{\partial \eta} + \frac{\partial \frac{\rho}{\sigma}(\nu + \tilde{\nu})(\mathbf{n} \bullet \nabla \tilde{\nu})}{\partial \zeta} + \frac{1}{J}S_\nu \right) \end{aligned} \quad (15)$$

where

$$\begin{aligned} S_\nu = \rho c_{b1} (1 - f_{t2}) \tilde{S}\tilde{\nu} + \frac{1}{Re} \left[-\rho \left(c_{w1}f_w - \frac{c_{b1}}{\kappa^2}f_{t2} \right) \left(\frac{\tilde{\nu}}{d} \right)^2 \right. \\ \left. + \frac{\rho}{\sigma}c_{b2}(\nabla \tilde{\nu})^2 - \frac{1}{\sigma}(\nu + \tilde{\nu})\nabla \tilde{\nu} \bullet \nabla \rho \right] + Re \left[\rho f_{t1}(\Delta q)^2 \right] \end{aligned} \quad (16)$$

$$\tilde{S} = S + \frac{\tilde{\nu}}{Rek^2d^2}f_{v2} \quad (17)$$

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \quad (18)$$

$$S = \sqrt{2\Omega_{ij}\Omega_{ij}} \quad (19)$$

$$f_w = g\left(\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6}\right)^{1/6} \quad (20)$$

$$g = r + c_{w2}(r^6 - r) \quad (21)$$

$$r = \frac{\tilde{\nu}}{Re\tilde{S}k^2d^2} \quad (22)$$

$$f_{t2} = c_{t3}exp\left(-c_{t4}\chi^2\right) \quad (23)$$

$$f_{t1} = c_{t1}g_texp\left[-c_{t2}\frac{\omega_t^2}{\Delta U^2}\left(d^2 + g_t^2d_t^2\right)\right] \quad (24)$$

$$g_t = min\left(0.1, \frac{\Delta q}{\omega_t\Delta x_t}\right) \quad (25)$$

Where $\Omega_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right)$ is the rotation tensor. ω_t is the wall vorticity at the wall boundary layer trip location, d is the distance to the closest wall, d_t is the distance of the field point to the trip location, Δq is the difference of the velocities between the field point and the trip location, Δx_t is the grid spacing along the wall at the trip location. The values of the coefficients by reference[22] are: $c_{b1} = 0.1355, c_{b2} = 0.622, \sigma = \frac{2}{3}, c_{w1} = \frac{c_{b1}}{k^2} + (1 + c_{b2})/\sigma, c_{w2} = 0.3, c_{w3} = 2, k = 0.41, c_{v1} = 7.1, c_{t1} = 1.0, c_{t2} = 2.0, c_{t3} = 1.1, c_{t4} = 2.0$.

To overcome the modeled stress depletion (MSD) and avoid grid induced separation in DES97[23], the DDES suggested by Spalart et al.[10] switches the subgrid scale formulation in S-A model by redefining the distance to the nearest wall $\tilde{d}$ as

$$\tilde{d} = d - f_d max(0, d - C_{DES}\Delta) \quad (26)$$

where

$$f_d = 1 - tanh([8r_d]^3) \quad (27)$$

$$r_d = \frac{\nu_t + \nu}{(U_{i,j}U_{i,j})^{0.5}k^2d^2Re} \quad (28)$$

where $U_{i,j}$ is the velocity gradients, k is the Karman constant, and d is the distance to the nearest wall. Within the boundary layer close to walls, $\tilde{d} = d$, and away from the boundary layer, $\tilde{d} = d - f_d max(0, d - C_{DES}\Delta)$ is most of the cases. f_d is designed to be 1 in the LES region, where

$r_d \ll 1$, and 0 elsewhere. Unlike DES97, this modification can fully preserve RANS mode inside boundary layer, therefore the attached boundary layers are safe from MSD. This mechanism enables DDES to behave as a RANS model in the near-wall regions and LES away from the wall coupled with the Navier-Stokes equations, Eq. (1). All the 6 equations are solved simultaneously.

2.3 The 5th Order WENO Scheme

For reconstruction of the interface flux, $E_{i+\frac{1}{2}} = E(Q_L, Q_R)$, the conservative variables Q_L and Q_R are evaluated by using the 5th order WENO scheme[24, 25]. For example,

$$(Q_L)_{i+\frac{1}{2}} = \omega_0 q_0 + \omega_1 q_1 + \omega_2 q_2 \quad (29)$$

where

$$q_0 = \frac{1}{3}Q_{i-2} - \frac{7}{6}Q_{i-1} + \frac{11}{6}Q_i \quad (30)$$

$$q_1 = -\frac{1}{6}Q_{i-1} + \frac{5}{6}Q_i + \frac{1}{3}Q_{i+1} \quad (31)$$

$$q_2 = \frac{1}{3}Q_i + \frac{5}{6}Q_{i+1} - \frac{1}{6}Q_{i+2} \quad (32)$$

$$\omega_k = \frac{\alpha_k}{\alpha_0 + \dots + \alpha_{r-1}} \quad (33)$$

$$\alpha_k = \frac{C_k}{\epsilon + IS_k}, \quad k = 0, \dots, r-1 \quad (34)$$

$$C_0 = 0.1, \quad C_1 = 0.6, \quad C_2 = 0.3 \quad (35)$$

$$IS_0 = \frac{13}{12} (Q_{i-2} - 2Q_{i-1} + Q_i)^2 + \frac{1}{4} (Q_{i-2} - 4Q_{i-1} + 3Q_i)^2 \quad (36)$$

$$IS_1 = \frac{13}{12} (Q_{i-1} - 2Q_i + Q_{i+1})^2 + \frac{1}{4} (Q_{i-1} - Q_{i+1})^2 \quad (37)$$

$$IS_2 = \frac{13}{12} (Q_i - 2Q_{i+1} + Q_{i+2})^2 + \frac{1}{4} (3Q_i - 4Q_{i+1} + Q_{i+2})^2 \quad (38)$$

ϵ is originally introduced to avoid the denominator becoming zero and is supposed to be a very small number. In reference [25], it is observed that IS_k will oscillate if ϵ is too small and also shift the weights away from the optimal values in the smooth region. The higher the ϵ values, the closer the weights approach the optimal values, C_k , which will give the symmetric evaluation of the interface flux with minimum numerical dissipation. When there are shocks in the flow field, ϵ can not be too large to maintain the sensitivity to shocks. In reference [25], $\epsilon = 10^{-2}$ is recommended

for the transonic flow with shock waves. After the reconstruction of the conservative variables, the interface flux is evaluated using the approximate Riemann solver of the E-CUSP scheme suggested by Zha et al[26, 27]. The viscous terms are discretized by 2nd order central differencing scheme.

2.4 Time Marching Scheme

The time dependent governing equation (1) is solved using dual time stepping method suggested by Jameson[28]. A pseudo temporal term $\frac{\partial Q}{\partial \tau}$ is added to the governing Eq. (1). This term vanishes at the end of each physical time step and has no influence on the accuracy of the solution. An implicit time marching scheme using line Gauss-Seidel line relaxation is employed to achieve high convergence rate[29]. The pseudo temporal term is discretized with first order Euler scheme. Let m stand for the iteration index within a physical time step, the semi-discretized governing equation can be expressed as

$$\begin{aligned} & \left[\left(\frac{1}{\Delta \tau} + \frac{1.5}{\Delta t} \right) I - \left(\frac{\partial R}{\partial \mathbf{Q}} \right)^{n+1,m} \right] \delta \mathbf{Q}^{n+1,m+1} \\ & = R^{n+1,m} - \frac{3\mathbf{Q}^{n+1,m} - 4\mathbf{Q}^n + \mathbf{Q}^{n-1}}{2\Delta t} \end{aligned} \quad (39)$$

where $\Delta \tau$ is the pseudo time step, and R is the net flux of the Navier-Stokes equations discretized in space using the schemes described in section 2.3.

3 Structural Model

3.1 Modal Approach

The equation of motion of an N-DOF(degree of freedom) system with mechanical damping and the aerodynamic loading as the excitation force can be expressed in matrix form:

$$[\mathbf{M}] \{\ddot{\mathbf{X}}\} + [\mathbf{C}] \{\dot{\mathbf{X}}\} + [\mathbf{K}] \{\mathbf{X}\} = \{\mathbf{F}\} \quad (40)$$

where, $\mathbf{M}$, $\mathbf{C}$, $\mathbf{K}$ are the mass, structural damping and stiffness matrices. $\mathbf{F}$ is total aerodynamic force acting on the wing surface. Total aerodynamic force can be defined as follows:

$$\mathbf{F} = - \oint p \cdot \hat{n} dA + \oint \tau_w \cdot \hat{t} dA \quad (41)$$

where, $\hat{n}$ is the unit normal vector to the wing surface and $\hat{t}$ is the unit tangent vector to the wing

surface. And p is the fluid static pressure and $\tau_{\mathbf{w}}$ is the fluid wall shear stress acting on the wing surface. Eq. (11) is used to compute static pressure of the wing surface by setting flow velocity components to zero.

To decouple the equations of motion for the damped systems(40), we use the mass normalized mode shape($\tilde{\phi}$) defined as the normal modes divided by square root of the the generalized mass($\sqrt{\phi^T m \phi}$), where m is the mass of the elastic body. Let $\{\mathbf{X}\} = [\tilde{\Phi}]\{\mathbf{q}\}$ and premultiply Eq. (40) by the transpose $[\tilde{\Phi}]^T$.

$$\begin{aligned} [\tilde{\Phi}]^T[\mathbf{M}][\tilde{\Phi}]\{\ddot{\mathbf{q}}\} + [\tilde{\Phi}]^T[\mathbf{C}][\tilde{\Phi}]\{\dot{\mathbf{q}}\} + [\tilde{\Phi}]^T[\mathbf{K}][\tilde{\Phi}]\{\mathbf{q}\} \\ = [\tilde{\Phi}]^T\{\mathbf{F}\} \end{aligned} \quad (42)$$

where $\mathbf{q}$ is the vector of the principal coordinates. Using the orthogonality of the system matrices and assuming damping matrix to be a linear combination of the mass and stiffness matrices, Eq. (42) is then completely decoupled and the j th equation will have the form

$$\ddot{q}_j + 2\zeta_j\omega_j\dot{q}_j + \omega_j^2q_j = \frac{\tilde{\phi}_j^T}{m_j}\mathbf{F} \quad (43)$$

where $[\tilde{\Phi}]^T = [\tilde{\phi}_1, \dots, \tilde{\phi}_j, \dots, \tilde{\phi}_N]^T$. N is the number of modal coordinates. ω_j and ζ_j are natural frequency and modal damping ratio for mode j . m_j denotes the j th diagonal element of modal mass matrix which will be unity. In the current study, the structural system is reduced to only five mode shapes, since a few bending and torsional frequencies are usually sufficient to determine flutter. The normalized modal equation can be given as[19]

$$\ddot{q}_j + 2\zeta_j(\frac{\omega_j}{\omega_\alpha})\dot{q}_j + (\frac{\omega_j}{\omega_\alpha})^2q_j = \frac{\tilde{\phi}_j^{*T}}{m_j^*} \cdot \mathbf{F}^* \cdot V_f^2 \cdot \frac{b_s^2 L}{\bar{V}} \cdot \bar{m} \quad (44)$$

where the dimensionless quantities are denoted by an asterisk. $V_f(= \frac{U_\infty}{b_s\omega_\alpha\sqrt{\bar{\mu}}})$ is the flutter speed index which is an input flutter control parameter. $\bar{m}$ is the measure wing panel mass, $\bar{V}$ represents the conical frustum volume. A conical frustum volume is defined as a truncated cone with lower and upper based diameters being the stream-wise chords at the root and the tip of the wing and with height equal to the semi-span of the wing. And b_s is the streamwise root semi chord. L is the reference length and ω_α is the angular frequency of the first torsional mode in units radians/sec. $\bar{\mu}(= \frac{\bar{m}}{V\rho_\infty})$ stands for the mass ratio, i.e. the ratio between the structural mass and the mass of the equivalent volume of fluid at reference density. It is noticed that m_j^* should be equal to one when

the mass normalized mode shapes are used. For example, the mode shapes of AGARD wing 445.6 Weakened model 3 reported by Yates[30] are normalized using the generalized mass such that m_j^* equals one in units $\text{lbf} \cdot \text{in} \cdot \text{s}^2$. This corresponds to $0.112979 \text{ kg} \cdot \text{m}^2$ in SI units.

The equations are then transformed to a state form as follows:

$$[\mathbf{M}] \frac{\partial \mathbf{S}}{\partial t} + [\mathbf{K}] \{\mathbf{S}\} = \mathbf{q} \quad (45)$$

where

$$\mathbf{S} = \begin{pmatrix} q_j \\ \dot{q}_j \end{pmatrix}, \mathbf{M} = [I], \mathbf{K} = \begin{pmatrix} 0 & -1 \\ (\frac{\omega_j}{\omega_\alpha})^2 & 2\zeta_j(\frac{\omega_j}{\omega_\alpha}) \end{pmatrix}$$

$$\mathbf{q} = \begin{pmatrix} 0 \\ \phi_j^{*T} \cdot \mathbf{F}^* \cdot V_f \cdot \frac{b_s^2 L}{V} \cdot \bar{m} \end{pmatrix}$$

3.2 Implicit Structural Solver

To solve the structural equations with CFD solver[18, 19] in a fully coupled manner, the decoupled structural equations are integrated using the same method as the flow governing equations(39) within each physical time step:

$$\begin{aligned} & \left(\frac{1}{\Delta\tau} I + \frac{1.5}{\Delta t} M + K \right) \delta S^{n+1,m+1} \\ & = q^{n+1,m+1} - M \frac{3S^{n+1,m} - 4S^n + S^{n-1}}{2\Delta t} - K S^{n+1,m} \end{aligned} \quad (46)$$

The fluid/structural interaction is implemented in a fully coupled manner[19]. Within each physical time step, the flow equations and structural equations are solved iteratively via every successive pseudo time step until the prescribed convergence criteria are satisfied for both flow and structural solver. After the convergence criteria are reached, the fluid-structural coupling goes to next physical time step.

3.3 Parameters for Flutter Control

There are in general three wing flutter control(or input) parameters: mass ratio $\bar{\mu} = \frac{\bar{m}}{V \rho_\infty}$, reduced velocity $V^* = \frac{U_\infty}{b_s \omega_\alpha}$, and flutter speed index $V_f = \frac{V^*}{\sqrt{\bar{\mu}}}$. The mass ratio $\bar{\mu}$ takes into account the effect of stiffness in flutter. It represents the ratio between the structural mass and the mass of

the equivalent volume of fluid at reference density. Typically flutter speed index V_f is selected as the main parameter in flutter boundary prediction because V_f reflects the effects of both dynamic pressure of the surrounding flow and stiffness of the structure. The effect of aircraft altitude in wind tunnel tests[30, 31] is obtained based on variation of dynamic pressure(varying the density) at constant Mach number. Liu et al.[32] and Chen et al.[19] used V_f , whereas Bakhle et al.[33] used V^* to search the flutter boundary at a given Mach number. In this study, either V_f or V^* can be used explicitly as shown in Eq. (44). The V_f is selected for the wing flutter simulation.

Several iterations are usually needed for a given freestream Mach number to search the neutrally stable point. Most of the computations only need to calculate a few periods to see whether the responses are divergent or damped with time. The flutter velocity index V_f is iterated to find the flutter boundary, all other variables such as the normalized inlet total pressure, inlet total temperature, and the static pressure at outlet are not varied. However, the absolute inlet total pressure, temperature and outlet static pressure are allowed to be varied. This will let the freestream velocity be changed while keeping the same Mach number due to the varied speed of sound. The Reynolds number, $Re = \frac{\rho_\infty U_\infty L_\infty}{\mu_\infty}$ varies with the freestream velocity. In the wing flutter experiment[31], dynamic pressure $q = \frac{1}{2}\rho_\infty V_\infty^2$ is the main fluid quantity to adjust flutter level. The way to control the dynamic pressure is either by changing the free stream density using the real gas such as freon-12 or by changing the velocity instead of density. In this paper, we change the freestream velocity, U_∞ which has direct relation with the reduced velocity V^* , and hence the flutter velocity index V_f .

4 Boundary Conditions

For the subsonic flutter calculations, steady state freestream conditions are used at the upstream portion of the outer boundary. At downstream boundary, the static pressure is specified as freestream value, and the streamwise gradients of other variables are forced to vanish. In spanwise direction, the symmetric boundary condition is used at the root of the wing. For the supersonic flutter case, all the variables are extrapolated at downstream boundary. The supersonic inflow condition with specified inlet Mach number is applied at the upstream portion of the outer boundary. The rest of the boundary conditions for the supersonic case are the same as those for the subsonic cases. The wall treatment suggested in [29] to achieve flux conservation by shifting half interval of the mesh on the wall is employed. During the calculation, the Mach number is kept constant by

Table 1: AGARD Wing 44.5 Weakened model 3 [30]

Airfoil section	NACA 65A004
Measured panel mass($\bar{m}$) [kg]	1.8627
Panel span(H) [m]	0.762
Sweep angle at half chord [deg]	43.15
Root chord(b_s) [m]	0.559
Tip chord(b_t) [m]	0.3682
Aspect ratio	1.65

changing the Reynolds number that is varied with the free stream velocity.

5 Computational Model

5.1 The geometry of AGARD wing 445.6

A limited number of AGARD standard wing configurations were tested[30] in order to promote the evaluation of existing and emerging unsteady aerodynamics codes and methods for flutter from subsonic to supersonic regime. In this study, the AGARD Wing 445.6 Weakened 3 is used for flutter simulation. This wing model has the symmetric NACA65A004 airfoil with a 4% thickness, and the wing structural details are listed in Table 1.

5.2 Mesh

The O-mesh topology is used as shown in Fig. 1. The outer span boundary away from the wing tip is about 10 span length of the wing. Total 18 partitioned blocks are used for parallel computation. The 1st grid spacing away from the the wing surface is set to yield y^+ less than unity. The far field inlet and far field outlet boundary is located 50 root chords away from the wing. The symmetric boundary condition is utilized at the root plane with spanwise velocity component forced to be zero.

5.3 Mode shape

The first five mass normalized mode shapes in the report[30] are used, which are displayed in Fig. 2. In the plot, the mode shapes are interpolated from structural nodes to CFD mesh nodes by using radial basis function method so that the grids in the interface between fluid and structure are

one to one connected. Among the five modes, the 1st, 3rd and 5th mode are bending mode, and 2nd and 4th mode are torsion mode. As the initial conditions to start the structure vibration, 1st mode initial normalized velocity of the structure in the modal coordinates is prescribed with a very small value of 0.0005, whereas other modal displacement velocities are set to zero. The uniform modal damping ratio(ζ) of 0.0 is applied for all flutter computations to isolate the aerodynamic damping. The unsteady flutter computation is started using the initialized flow field obtained by the unsteady CFD simulation without FSI. The residual in each physical time step is reduced by three orders of magnitude, which is achieved usually within 30 iterations.

6 Results and Discussion

6.1 Mesh Refinement Test

Mesh convergence test is done for Mach number of 1.072. Since it is very time consuming for FSI simulation, three mesh sizes were tested with the total mesh size doubled consecutively with mesh A=129 (around airfoil) $\times$ 49(normal to the surface) $\times$ 49(span), mesh B=137 $\times$ 90 $\times$ 60, mesh C=289 $\times$ 64 $\times$ 77. Such mesh refinement is informative to show if the solution is independent of the mesh size without being preventively expensive to double the mesh size in each direction of the 3D mesh. When the mesh is refined, the mode shapes corresponding to the surface mesh coordinates are interpolated by a radial basis function interpolation. Fig. 3 shows the modal displacements of mode 1 for different meshes. The flutter velocity index V_f used for mesh test is 0.30. The predicted responses for mode 1 using mesh B are well converged with mesh C. Therefore, mesh C is chosen for flutter simulations in this study.

6.2 Flutter Boundary

The computed flutter velocity index and frequency compared with experimental data at flutter boundary for six free-stream Mach numbers are shown in Fig. 4. Overall, the computed flutter boundary points are in good agreement with the experiment. In particular, the predicted flutter boundaries at the two supersonic conditions match the experiment accurately. The sonic dip near $M = 1.0$ in the flutter map is very well captured for both the speed index and frequency by the computation. In the frequency plot in Fig. 4, the frequency ratio is defined by the ratio of the first mode frequency over the first natural torsional frequency(second mode) during the neutral

vibration. The simulation over-predicts the frequency ratio with maximum deviation about 3.0% for Mach number less than 1.141.

The predicted modal displacements for Mach number of 0.499, 0.901, 0.960 and 1.072 with different flutter velocity index(V_f) are displayed in Fig. 5, 6, 7 and 8 respectively. Three different responses, including damped, neutral, diverging are shown in each of the plots on the left, middle, and on the right. Take the transonic dip $M=0.96$ as an example at $V_f = 0.2961$, the response decays in time, whereas at $V_f = 0.3021$ the response is divergent. A neutrally stable point, the flutter boundary, is captured at $V_f = 0.2991$. The damped oscillation with $V_f = 0.2961$ is 1.0% below the measured flutter boundary. It indicates that the present FSI approach has very good accuracy and is sensitive to a small change of the flutter speed index.

The predicted aerodynamic damping coefficients at $M = 1.072$ are plotted in Fig. 9. The aerodynamic damping can be derived from the logarithmic decrement: $\delta = \frac{1}{n} \ln \frac{x(t)}{x(t+nT)}$, where $x(t)$ is the amplitude at time t , $x(t+nT)$ is the amplitude of the peak n periods away with n as any integer number of successive and positive peaks. Then the damping ratio can be calculated by $\zeta = \frac{1}{\sqrt{1+(\frac{2\pi}{\delta})^2}}$. The structural damping is set to zero. The coefficient hence represents the net aerodynamic damping. The flutter occurs when the aerodynamic damping is negative. The damping coefficient is plotted in Fig. 9 as a function of the dynamic pressure, which corresponds to the flutter speed index. The predicted aerodynamic damping shows that the flutter boundary is at P3 with the dynamic pressure of about 3100 Pa, which is close to the experimental value 3166 Pa at the flutter boundary.

As shown in Fig. 5, 6, 7 and 8 the generalized amplitudes of the first mode at the neutral points do not vary much with the Mach number except at the sonic dip point with a large drop. However, it can be also seen from those plots that the amplitudes of the second mode decrease with the increased Mach number.

The ratio of the maximum amplitude of the first mode(bending mode) to the second mode versus Mach number at neutral vibration is shown in Fig. 10, which almost has a shape of the reversed flutter boundary shown in Fig. 4. It is clear that the weight of the torsional mode(second mode) contributing to the flutter of the wing decreases sharply at transonic and supersonic regime. At the sonic dip Mach number of 0.96, even though the first mode amplitude has a large drop, the amplitude of the second mode drops even more. It gives the ratio a sonic rise. The contribution of the bending mode to structural flutter increases abruptly at transonic Mach number 0.96. It

appears that the transonic dip is related to the anticlimax of the second mode in transonic regime.

The lift and drag coefficients neutral oscillation across the sonic dip at $M=0.901$, $M=0.96$ and $M=1.072$ are shown in Fig. 11. The lift coefficient at the sonic dip Mach number of 0.96 nearly drops by an order of magnitude. This is also due to the decrease of the torsional vibration that determines the angle of attack. The averaged amplitudes of C_d increase from subsonic to supersonic due to the wave drag. The drag coefficients at Mach 0.901, 0.906 and 1.072 are 0.0104, 0.0116 and 0.0145 respectively. The pitching moment coefficients(C_m) during the wing neutral vibration are shown in Fig. 12. The moment is calculated around z axis, which is in the direction of the wing span. The left plot is the time history of C_m and the right plot is the amplitude of C_m versus of Mach numbers. There is also a dip for C_m at the sonic dip Mach number as shown in Fig. 12. Since the pitching moment is the primary cause of torsional vibration, the dip of C_m at sonic point is consistent with the anticlimax of the torsional mode.

Fig. 13 shows the modal force defined as $\frac{\tilde{\phi}_j^{*T}}{m_j^*} \cdot \mathbf{F}^* \cdot V_f^2 \cdot \frac{b_s^2 L}{V} \cdot \bar{m}$ and modal displacement for $M=0.901$, $M=0.96$ and $M=1.072$. For the three Mach numbers, all the modal forces and displacements have reversed phase, which are also the case for the subsonic Mach numbers(not shown). This means that the reversed phase is not the cause for sonic dip. The phase of the first modal force lags half period of that of the first modal displacement at all free stream conditions. And there is no phase lag between the second modal force and the second modal displacement.

Fig. 14 illustrates the first modal displacement and the wing physical displacement during half a period at the neutral vibration across the sonic dip at $M=0.901$, $M=0.96$ and $M=1.072$. The neutral vibration is used as the flutter boundary with the amplitude not increasing nor decreasing. The amplitude of the trailing edge is larger than that of the leading edge, which indicates that both the pitching(torsion) and plunging(bending) are associated with the wing flutter. The pitching makes the wing experience a variation of angle of attack(AOA) that generates lift for wing with symmetric airfoil.

The instantaneous isentropic Mach number(M_{is}) contours on the pressure and suction surface at 4 different free stream Mach numbers at three instants during half a period are shown in Fig. 15 and Fig. 16. M_{is} reflects both the local static pressure and the flow speed as it is an inviscid flow. At free stream Mach number of 0.901, the M_{is} does not have large variation on the wing surface. At $M=0.96$, M_{is} contours on the wing surface show a shock wave near the trailing edge. The shock strength is stronger near the wing tip and is mitigated toward the root. The peak isentropic Mach

number is high at the inner span. At the outer span the peak Mach number is reduced from 50% span to 70% span. The strength of shock wave on the wing surface varies during the vibration, but the shock patterns are not changed much. At supersonic conditions, $M=1.072$ and $M=1.141$, the shock strength becomes stronger near the trailing edge, which may be responsible for the torsion mode amplitude depletion in the transonic and supersonic regime shown in Fig. 10.

Fig. 17, 18 and 19 are the surface isentropic Mach number distributions of the flutter boundary at near root, mid-span and near tip at three different Mach number. At the sonic dip Mach number 0.96, the smaller pitching oscillation than at Mach number 0.901 creates less lift shown as the Mach number deviation on the suction surface and pressure surface. Fig. 19 shows that the unsteady shock waves near the trailing edge of the wing is well resolved by present method. It can be seen that the maximum lift coefficient is generated in the mid-span and the lift coefficient is reduced toward the inner span and tip. The same phenomenon is observed for the other two Mach numbers. The structural deformations affects the location and strength of the part-chord shock, which in turn results in a shift in the aeroelastic stability of the wing. The shocks is thus capable of turning bending-torsion flutter instability into mostly bending flutter at the sonic dip.

The shock motion is considered as playing an important role in the transonic and supersonic flutter problem. Three types of shock oscillations are identified in an transonic oscillating airfoil experiment, which are named as Tijdeman type A, B, and C[8]. Type A has small amplitude and sinusoidal motion. Type B has larger amplitude and discontinuous motion. And type C features of upstream propagating shock wave. Fig. 21, 21 and 21 show the Mach number contours during neutral vibration at three time instants, three different span locations, and different Mach numbers across the sonic dip. At $M=0.901$, the Mach number variation is more stable. At $M=0.96$, shock oscillation is large, in particular in the tip region, the shock appears on the upper and lower surface alternatively and forms the discontinuous motion similar to Tijdeman type B shock oscillation. At $M=1.072$, the shock wave patterns are quite different from those at transonic. And shock wave near trailing edge become more oblique and shock oscillation is also large between the upper and lower surface. At the flutter boundary including the sonic dip point, no flow separation due to shock/boundary layer interaction is observed.

7 Conclusion

In this paper, DDES of a 3D transonic wing flutter is conducted with free stream Mach number varied from subsonic to supersonic. Unsteady 3D compressible Navier-Stokes equations are solved with a system of 5 decoupled structure modal equations in a fully coupled manner. The low diffusion E-CUSP scheme with a 5th order WENO reconstruction for the inviscid flux and 2nd order central differencing for the viscous terms are used to resolve the shock wave/turbulent boundary layer interaction of the vibrating wing. The radial basis function is employed to interpolate the mode shapes from the coarse mesh to refined mesh.

The predicted flutter boundary at different free stream Mach number including the sonic dip achieves very good agreement with experiment. In particular, the predicted flutter boundaries at the two supersonic conditions match the experiment accurately.

The mechanism of sonic dip is studied with the following observation. The weight of the torsional mode that contributes to the flutter of the wing decreases at transonic and supersonic regime. The contribution of the bending mode to structural flutter increases abruptly at transonic Mach number 0.96 at which the sonic dip occurs. It appears that the transonic dip is related to the anticlimax of the second mode in transonic regime. The shock strength is enhanced more near the trailing edge at near sonic Mach number. The shock location and oscillation decrease the wing pitching moment sharply at the sonic dip Mach number. The decreased pitching moment induces lower torsional mode vibration. It creates an anticlimax of the amplitude ratio of the first mode to second mode at the sonic dip. No flow separation due to shock/boundary layer interaction is observed.

Acknowledgments

The simulation was performed under the support of High Performance Computing Challenge Project of DoD Supercomputing Resource Centers and the Center for Computational Sciences at University of Miami.

Figure 1: Computational mesh of $289 \times 64 \times 77$ for AGARD Wing 445.6 Weakened 3

Mode 1, $f = 9.58992$ Hz

Mode 2, $f = 38.165$ Hz

Mode 3, $f = 48.3482$ Hz

Mode 4, $f = 91.5448$ Hz

Mode 5, $f = 118.1132$ Hz

Figure 2: The first 5 mode shapes of AGARD Wing 445.6 Weakened 3[30]

Figure 3: Mesh convergence test for $M = 1.072$, $V_f = 0.30$

Figure 4: Flutter boundary predicted by fully coupled FSI methods and DDES for Wing 445.6. The present results are compared with other CFD results[34] and experimental data[30].

Figure 5: Modal displacements for $M = 0.499$

Figure 6: Modal displacements for $M = 0.901$

Figure 7: Modal displacements $M = 0.960$

Figure 8: Modal displacements for $M = 1.072$

Figure 9: Aerodynamic damping as a function of the dynamic pressure for $M = 1.072$

Figure 10: The amplitude ratio of the first mode to the second mode at neutral vibration flutter boundary

Figure 11: Comparison of predicted lift(C_l) and drag(C_d) coefficients during neutral vibration

Figure 12: Comparison of predicted momentum coefficients(C_m)(left) and maximum amplitude(right) during neutral vibration

Figure 13: Comparison of modal force and modal displacement during neutral vibration at different Mach numbers

Figure 14: Wing fluttering at $M = 0.901$, $M = 0.960$ and $M = 1.072$ during neutral vibration

Figure 15: Instantaneous isentropic Mach number contours on suction surface. Left: $M = 0.901$; Right: $M = 0.96$

Figure 16: Instantaneous isentropic Mach number contours on suction surface. Left: $M = 1.072$; Right: $M = 1.141$

Figure 17: Surface isentropic Mach number distributions at different span location for the flutter boundary at $M = 0.901$

Figure 18: Surface isentropic Mach number distributions at different span location for the flutter boundary at $M = 0.960$

Figure 19: Surface isentropic Mach number distributions at different span location for the flutter boundary at $M = 1.072$

Figure 20: Mach number contours for $M = 0.901$

Figure 21: Mach number contours for $M = 0.96$

Figure 22: Mach number contours for $M = 1.072$

References

- [1] E. H. Dowell, and K. C. Hall, “MODELING OF FLUID-STRUCTURE INTERACTION,” *Annual Review of Fluid Mechanics*, vol. 33, pp. 445–490, 2001.
- [2] O. Bendiksen and K. Kousen, “Transonic Flutter Analysis Using the Euler Equations.” AIAA Paper 87-0911, 1987.
- [3] J. Rausch, R.D. Batina and T. Yang, “Three-dimensional time-marching aeroelastic analyses using an unstructured-grid euler method,” *AIAA Journal*, vol. 33, pp. 1626–1633, 1993.
- [4] Prananta, B.B., Hounjet, M.H.L., and Zwaan, R.J., “Two-Dimensional Transonic Aeroelastic Analysis Using Thin-Layer Navier-Stokes Method,” *Journal of Fluid and Structures*, vol. 12, pp. 655–676, 1998.
- [5] E. Lee-Rausch and J. Batina, “Calculation of AGARD Wing 445.6 Flutter Using Navier-Stokes Aerodynamics.” AIAA Paper 93-3476, 1993.
- [6] K. Isogai, “Transonic Dip Mechanism of Flutter of a Sweptback Wing: Part II,” *AIAA Journal*, vol. 19, pp. 1240–1242, September 1981.
- [7] Bendiksen, O.O., “Influence of Shocks on Transonic Flutter of Flexible Wings.” 50th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference 4-7 May 2009, Palm Springs, California, 2009.
- [8] Tijdeman, H., “Investigations of the Transonic Flow Around Oscillating Airfoils.” Doctoral Dissertation, Technical University of Delft, The Netherlands, 1977.
- [9] Morgan, B., Kawai, S., Lele, S. K., “Large-Eddy Simulation of an Oblique Shock Impinging on a Turbulent Boundary Layer.” AIAA Paper 2010-4467, 40th Fluid Dynamics Conference and Exhibit, June 28 - July 1, Chicago, Illinois, 2010.
- [10] P.R. Spalart, S. Deck, M. Shur, and K.D. Squires, “A New Version of Detached-Eddy Simulation, Resistant to Ambiguous Grid Densities,” *Theoretical and Computational Fluid Dynamics*, vol. 20, pp. 181–195, 2006.
- [11] F.R. Menter, and M. Kuntz, “Adaptation of Eddy-Viscosity Turbulence Models to Unsteady Separated Flow Behind Vehicels, *The Aerodynamics of Heavy Vehicles: Trucks, Buses and*

Trains, Edited by McCallen, R. Browand, F. and Ross, J. ,” Springer, Berlin Heidelberg New York, 2004, 2-6 Dec. 2002.

- [12] Shen, Y.-Q. and Zha, G.-C. and Chen, X.-Y., “ High Order Conservative Differencing for Viscous Terms and the Application to Vortex-Induced Vibration Flows,” *Journal of Computational Physics*, vol. 228(2), pp. 8283–8300, 2009.
- [13] Shen, Y.-Q. and Zha, G.-C. and Chen, X.-Y., “ High Order Conservative Differencing for Viscous Terms and the Application to Vortex-Induced Vibration Flows,” *Journal of Computational Physics*, vol. 228(2), pp. 8283–8300, 2009.
- [14] Wang, B. Y and Zha, G.-C., “High Fidelity Simulation of Nonlinear Fluid-Structural Interaction with Transonic Airfoil Limit Cycle Oscillations,” *Journal of Fluids and Structures*, vol. doi:10.1016/j.jfluidstructs.2010.02.003, 2010.
- [15] Wang, B. Y and Zha, G.-C., “Detached-Eddy Simulation of Transonic Limit Cycle Oscillations Using High Order Schemes,” *Journal of Computer & Fluids*, vol. 52, pp. 58–68, 2011.
- [16] Im, H-S., Chen, X-Y and Zha, G-C., “ Detached Eddy Simulation of Rotating Stall Inception for a Full Annulus Transonic Rotor ,” *AIAA Journal of Propulsion and Power*, vol. 28, No. 4, pp. 782–798, 2012.
- [17] X.Y. Chen, B.Y. Wang, and G.C. Zha, “Delayed Detached Eddy Simulation of 3-D Wing Flutter with Fully Coupled Fluid-Structural Interaction.” AIAA Paper 2010-0053, 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, Jan. 4-7, 2010.
- [18] X.Y. Chen, and G.-C. Zha, “Fully Coupled Fluid-Structural Interactions Using an Efficient High Solution Upwind Scheme,” *Journal of Fluid and Structure*, vol. 20, pp. 1105–1125, 2005.
- [19] X.-Y. Chen, G.-C. Zha, and M.-T. Yang, “Numerical Simulation of 3-D Wing Flutter with Fully Coupled Fluid-Structural Interaction,” *Journal of Computers and Fluids*, vol. 35, pp. 856–867, 2007.
- [20] Im, H.-S. and Zha, G.-C., “Prediction of a Supersonic Wing Flutter Boundary Using a High Fidelity Detached Eddy Simulation.” AIAA Paper 2012-0039, 50th AIAA Aerospace Sciences Meeting, Jan. 9-12., Nashville, TN, USA, 2012.

- [21] Gan, J.Y., Im, H.S., Chen, X.Y., Pasiliao, C.L. and Zha, G.C., “Prediction of Wing Flutter Boundary Using High Fidelity Delayed Detached Eddy Simulation.” AIAA Paper 2015, Kissimmee, Florida, Jan. 2015.
- [22] P.R. Spalart, and S.R. Allmaras, “A One-equation Turbulence Model for Aerodynamic Flows.” AIAA-92-0439, 1992.
- [23] P.R. Spalart, W.H. Jou, M. Strelets, and S.R. Allmaras, “Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach.” Advances in DNS/LES, 1st AFOSR Int. Conf. on DNS/LES, Greyden Press, Columbus, H., Aug. 4-8, 1997.
- [24] Shen, Y.Q., and Zha, G.C., “Improvement of the WENO Scheme Smoothness Estimator,” *International Journal for Numerical Methods in Fluids*, vol. 64,, pp. 653–675, DOI:10.1002/fld.2186, 2009.
- [25] Shen, Y.Q., Zha, G.C., and Wang, B.Y., “Improvement of Stability and Accuracy of Implicit WENO Scheme,” *AIAA Journal*, vol. 47, pp. 331–334, DOI:10.2514/1.37697, 2009.
- [26] G.C. Zha, Y.Q. Shen, and B.Y. Wang, “An improved low diffusion E-CUSP upwind scheme ,” *Journal of Computer and Fluids*, vol. 48, pp. 214–220, 2011.
- [27] G.C. Zha, Y.Q. Shen, and B.Y. Wang, “Calculation of Transonic Flows Using WENO Method with a Low Diffusion E-CUSP Upwind Scheme.” AIAA Paper 2008-0745, 46th AIAA Aerospace Sciences Meeting, Reno, NV, Jan. 2008.
- [28] A. Jameson, “Time Dependent Calculations Using Multigrid with Applications to Unsteady Flows Past Airfoils and Wings.” AIAA Paper 91-1596, 1991.
- [29] Y.Q. Shen, B.Y. Wang, and G.C. Zha, “Implicit WENO Scheme and High Order Viscous Formulas for Compressible Flows .” AIAA Paper 2007-4431, 2007.
- [30] E.C. Yates Jr., “AGARD standard aeroelastic configurations for dynamic response. Candidate configuration I.-wing 445.6 .” NASA-TM-1000492, 1987.
- [31] R.V. Goggett, R.V. Rainey, and H.G. Morgan, “An experimental investigation of Aerodynamic Effects of Airfoil Thickness on Transonic Flutter Characteristics.” NASA TMX-79, 1959.

- [32] F. Liu, J. Cai, and Y. Zhu, “Calculation of Wing flutter by a Coupled CFD-CSD method.” AIAA-2000-0907, 2000.
- [33] M.A. Bakhle, T.S.R. Reddy, and T.G. Keith Jr., “Time Domain Flutter Analysis of Cascades Using a Full-Potnetial Solver,” *AIAA Journal*, vol. 30, pp. 163–169, 1992.
- [34] Silva, W.-A., Chwalowski, P., and Perry, B.-N., “Evaluation of Linear, Inviscid, Viscous, and Reduced-Order Modeling Aeroelastic Solutions of the AGARD 445.6 Wing Using Root Locus Analysis.” 55th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, AIAA SciTech Forum, 2014.