AIAA 97-0733

An Efficient Method For Time-Marching Supersonic Flutter Predictions Using CFD

John P. Hunter and Andrew S. Arena, Jr.

School of Mechanical and Aerospace Engineering
Oklahoma State University
Stillwater, Oklahoma

35th Aerospace Sciences Meeting & Exhibit
January 6-10, 1997 / Reno, NV
AN EFFICIENT METHOD FOR TIME-MARCHING
SUPersonic FLUTTER Predictions USING CFD

J. P. Hunter\textsuperscript{a}, A.S. Arena, Jr\textsuperscript{b},
School of Mechanical & Aerospace Engineering
Oklahoma State University
Stillwater, OK

SUMMARY

A time-marching aeroelastic analysis is enhanced by combining aerodynamic modeling
techniques with CFD. This enhancement significantly reduces the computation time
required to accurately predict critical flight conditions. Furthermore, this method is tested
over a wide range of mach numbers and proven valid in all reasonable cases. It is shown that
estimated flutter boundaries obtained by this method are sufficiently close to the flutter
boundaries obtained by the Euler method, that the enhanced CFD solution may then be used for
refinement which requires significantly fewer transient calculations.

NOMENCLATURE

- $a_m$, $P_m$, $\rho_m$, $M_m$ = velocity of sound, pressure, density, and
  mach number respectively, in free stream conditions
- $a_o$, $P_o$, $\rho_o$, $M_o$ = velocity of sound, pressure, density, and
  mach number respectively, in mean flow
- $C_p$ = coefficient of pressure
- $C_L$ = coefficient of lift
- $\gamma = 1.4$, ratio of specific heats
- $\delta = \text{impact angle with free-stream velocity}$
- $\theta_o$ = mean flow angle
- $\theta^*$ = perturbed angle
- $M_{ns}$ = mach number normal to shock
- $n^\prime$ = outward normal
- $n^\prime = \text{outward normal perturbed from mean flow}$
- $P$ = Pressure
- $P^\prime$ = pressure perturbed from mean flow
- $q = \text{generalized displacement}$
- $w(t) = \text{piston velocity as a function of time}$
- $V_b = \text{velocity of body}$
- $V_s = \text{local steady velocity}$

INTRODUCTION

A prediction of aircraft flight dynamics and aeroelastic characteristics such as flutter\textsuperscript{1} are
crucial to the design of modern aircraft as well as to flight test operations. Recently, there has been
resurgent interest in the development of hypersonic vehicles which operate in speeds
where flutter may occur during at least part of the vehicle's operation. Examples of such vehicles
include the Space Shuttle, Pegasus, the National Aerospace Plane (NASP), and the X-34. Also
presently under development is the X-33, which is a joint effort between NASA and major aerospace
companies, and will be the world's first single stage to orbit vehicle.

Using a recently developed STARS\textsuperscript{2} capability for aeroelastic analysis, a time-
marching approach based on the unsteady Euler equations may be utilized to predict flutter
boundaries over a wide Mach number range for complex three-dimensional geometries.
Determination of the flutter boundaries is presently achieved by searching over the flight
regime for potential crossovers between stable and divergent time history oscillations based on modal
damping terms. This analysis is followed by interpolation of these results to determine the
point at which the system is neutrally stable.

STARS which stands for "Structural Analysis RoutineS" was developed by Gupta\textsuperscript{2} at
the NASA Dryden Flight Research Center. STARS is a highly integrated code for
multidisciplinary analysis of flight vehicles including static and dynamic structural analysis,
computational fluid dynamics, heat transfer, and aeroelastodynamic capabilities. With the recently
developed capability for Aeroelastic analysis using a time-marching approach based on the
unsteady Euler equations, the aforementioned prediction of flutter boundaries may be obtained

\textsuperscript{a}Graduate Research Assistant, Student Member AIAA
\textsuperscript{b}Assistant Professor, Senior Member AIAA
for a wide variety of flight conditions and geometries.

Due to the potentially large domain required to ensure sufficient grid resolution of a given geometry, however, there lies a critical drawback of the time-marching approach. Dowell\(^3\) points out that the computational time required will be on the order of \(P \times T_F\) where \(P\) is the number of parameter combinations required and \(T_F\) is the time required for a simultaneous fluid-structure time marching calculation to complete a transient. In order to ensure time accuracy and sufficient grid resolution, the use of time-marching solutions to the Euler equations on a three-dimensional configuration requires a significant amount of computation time. As an illustration, on a present day high speed workstation, utilization of the unsteady Euler equations to calculate a single fluid structure transient on a three-dimensional system may demand well in excess of one-hundred CPU hours. Hence, identification of the flutter boundaries over the full domain will require many times this number.

**FOCUS OF THE PRESENT INVESTIGATION**

The focus of the present investigation is to enhance the practicality of the time-marching aeroelastic analysis by combining aerodynamic modeling techniques and Computational Fluid Dynamics in order to significantly reduce the computational time required for an aerodynamic solution of a given geometry in the supersonic flight regime. Moreover, the error of such a method must be well within acceptable limits as compared with experimental data or the predictions of the unsteady Euler equations previously discussed.

**METHODS CONSIDERED**

Since it is the CFD which requires the overwhelming proportion of the computation time, a method which gives an accurate estimate to later be refined is researched. Different areas of importance in the determination of such a method are as follows: Since this procedure will be implemented into STARS, ease of implementation and compatibility must be considered. Also, this method must be accurate over a wide range of geometric shapes and flow regimes, and not just limited to special cases. Simplicity is also a plus, hence a basic method would augments ease of rigorous testing and validation over the widest range possible. Different methods with these goals in mind are discussed.

**Piston Method**

The piston method\(^4\) is a popular technique for supersonic and hypersonic flutter prediction. In the form of the unsteady wave equation, this method requires only the outward normal vector on a given surface to determine the pressure at any point. Figure 1 illustrates the unsteady wave equation as follows:

![Figure 1. Piston motion in a one-dimensional channel.](image)

where:

\[
p = p_\infty \left[ 1 + \frac{\gamma - 1}{2} \frac{w}{a_\infty} \right]^{\frac{2\gamma}{\gamma - 1}}.
\]

Due to its simplicity and ease of use, it is an attractive technique for approximating the surface pressure in a supersonic flow. The unsteady wave equation, however, does not take into account the losses across a shock nor does it accurately predict pressure in a shock interaction area. Furthermore, since this theory is based on a point function, i.e., the pressures are only dependent upon local conditions, it over predicts the pressure on a three-dimensional geometry such as a cone. This is due to the three-dimensional relaxing effect for which the piston theory cannot account. Also, for a relatively blunt surface with respect to the flow, the piston method again over
predicts the pressure. As an example, in figure 2, the unsteady wave equation is used to show the pressure coefficient versus mach number for a wedge and a cone at half angles of ten degrees. Since the piston theory does not differentiate between the two geometries and is based on the assumption that the flow is a point function, only one curve represents both geometries. This curve is compared with data taken from literature for a cone and wedge at the noted half angle.

Due to the limited application range and accurate prediction of pressure via the Piston theory, a method which will better predict surface pressure about a more three-dimensional flow is considered.

**Tangent-Cone Method**

The Tangent-Cone method is an approximate technique for the prediction of three dimensional pressure effects on a cone surface analogous to the nose of a fuselage for example. The equations for the surface pressure are as follows:

\[
C_p = \frac{48 \cdot M_{ns}^2 \cdot \sin^2 \delta}{23 \cdot M_{ns}^2 - 5}
\]

where:

\[
M_{ns} = (0.87 \cdot M_{in} - 0.544) \cdot \sin \delta + 0.53
\]

\(C_p\) is based on the Modified Newtonian theory which yields a result as a function only of the impact angle \(\delta\):

\[
C_p = K \cdot \sin^2 \delta
\]

Here \(K\) is equal to the stagnation pressure coefficient and \(M_{ns}\) is the mach number normal to the shock which is empirically determined by Edwards. Results and further explanation of this method may be found in reference 8.

The Tangent-Cone method under predicts pressure for predominately two-dimensional flow and again over predicts pressure for flows about blunt surfaces such as the leading edge of a wing. Furthermore, as in the Piston Method, the Tangent-Cone method neglects the effects of the mean flow and any shock interactions.

**Modified Newtonian Impact Method**

In contrast to the unsteady wave equation and the Tangent-Cone procedure, the Modified Newtonian Impact method is used predominately for very blunt surfaces relative to the free-stream flow or at very high mach numbers in the hypersonic range. The equation for the Modified
Newtonian Impact theory is previously discussed with the Tangent-Cone method and is also listed as follows:

\[ C_p = K \cdot \sin^2 \delta \]

For true Newtonian flow, the parameter K is given as 2, and for the Modified theory, K is taken as the stagnation \( C_p \).

Since however, this method has a limited application of mainly blunt surfaces at high supersonic to hypersonic speeds, other methods are again considered.

**Other Methods**

There are a number of other methods for prediction of surface pressure in supersonic and hypersonic flow. The Modified Newtonian Plus Prandtl-Meyer Method for example is another blunt body technique based on the analysis presented by Kaufman\(^8\). Or even the Van Dyke Unified Method\(^8\) useful for thin profile shapes. These methods are usually applied to specific cases and tend to be too specific for the task at hand.

**METHODOLOGY**

After careful examination and analysis of several different methods to augment the efficacy of the STARS code in flutter boundary prediction, the Piston method is chosen not only to be implemented into STARS but also to be utilized as a perturbation to an already existing mean flow solution, hence the Piston Perturbation Method. A brief illustration of this idea, utilizing a cone similar to the one used for the generation of figure 2, is shown in figure 5 where the unsteady wave equation is modified to predict the pressure for a perturbations about the mean flow from 10° to 12.5°. The modified unsteady wave equation is as follows:

\[ \frac{P'}{P^*} = \frac{P_0}{P^*} \left[ 1 + \frac{\gamma - 1}{2} M_\infty \sin(\theta' - \theta_0) \right]^{2\gamma} \gamma^{-1} \]

where figure 4 illustrates this equation.

*Figure 4. An illustration depicting the modified unsteady wave equation for a cone.*

Previously in figure 2, Piston theory was applied to a cone without taking into account the mean flow conditions. Figure 6 shows the percent error, which comparatively shows better results than the previous situation.

*Figure 5. Pressure coefficient of a cone perturbed about the mean flow of 10° to 12.5°.*

Clearly, the percent error is less for the perturbed case than the unperturbed case in figure 2.

The Perturbation method has been implemented in STARS in the following manner: Initially, a supersonic steady-state solution for a three-dimensional flowfield is obtained with the
use of the finite-element Euler methodology. The flow variables, which take into account the nonlinearities due to three-dimensional effects such as shock interactions, are saved and used as a mean flow to the starting point of the aeroelastic simulation. Once these variables are obtained, they may be saved and used for future simulations given the structure is not significantly altered.

With the steady state Euler solution given as the mean flow, modal superposition is applied in the simulated aerodynamic structure to represent the aeroelastic effects. The modes of vibration are perturbed which represents a perturbation to the mean flow. Next, an application of the isentropic wave equation previously discussed is locally applied as a perturbation to the mean flow at every point. The equations governing this method are shown here with figure 6:

![Diagram](image)

**Figure 6. Simplistic illustration of a locally applied perturbation to the mean flow**

where:

\[
\frac{P'}{P_\infty} = \frac{P_o}{P_\infty} \left[ 1 + \gamma - 1 \frac{V_\infty}{2} \right]^{2\gamma} \frac{\Delta u^*}{a_o}
\]

and in non-dimensional form,

\[
\frac{P'}{P_\infty} = \frac{P_o}{P_\infty} \left[ 1 + \gamma - 1 \frac{M_\infty}{2} \frac{a_\infty}{a_o} \right]^{2\gamma} \frac{\Delta u^*}{a_o}
\]

with:

\[
a_\infty = \sqrt{\frac{P_\infty}{\rho_o}}
\]

and

\[
\Delta u^* = V_x \cdot n' + V_b \cdot n'.
\]

These pressures are then used in the coupled structural dynamic solutions which are numerically integrated to find a generalized displacement q shown as:

\[
[M] \cdot \ddot{q} + [K] \cdot q = [P]
\]

where \([M]\) and \([K]\) are the mass matrix, and the stiffness matrix respectively, obtained in STARS by a finite element structural analysis routine given the structural properties of the system and \([P]\) is the force matrix obtained by the piston perturbation method. Once the generalized displacement for every point on the aeroelastic surface is determined, the values are multiplied by the mode shapes to determine the actual displacement for calculation of a new set of transient data.

Given the new structural deflections based on the previous aerodynamic pressures, a new outward normal velocity is given for the next calculation of the pressures. This process repeats until enough transient data is acquired to determine the stability characteristics of the perturbed aerodynamic system. Finally, the flutter boundaries are then verified and refined with the non-linear time marching Euler method.

**RESULTS**

It will be shown that the Piston Perturbation method accurately predicts surface pressure of a number of different geometrical configurations in the supersonic flow region. It will also be shown that the estimated flutter boundaries of a Generic Hypersonic Vehicle (GHV) obtained by this method are sufficiently close to the flutter boundaries obtained by the Euler method, that the estimated CFD solution may then be used for refinement which requires significantly fewer transient calculations. This GHV will also be used to compare predicted flutter boundaries, and run time between the perturbation solution and the Euler solution.

To show the simplicity and accuracy of the piston perturbation solution, a simple perturbed wedge exemplifies the validity of this method over a wide range of mach numbers for
perturbations about the mean flow. Figure 7 shows these results.

Figure 7. A simple perturbed wedge in compression.

The Perturbation method also predicts pressure for an expansive perturbation as shown in figure 8.

Figure 8. A simple perturbed wedge in expansion.

The errors are minimal given a perturbation approximately less than 30% of the mean flow conditions and w/a₀ between -1 and +1 where negative and positive values represent expansion and compression respectively. The term w/a₀ is similar to w/a₀\textsuperscript{10} in the unsteady wave equation previously noted except it is perturbed about the mean flow not the free stream. Since the majority of flutter analyses deals mostly with small perturbations this criteria holds for a wide range of mach numbers and geometries.

The P.S. also predicts pressure about more three dimensional surfaces, such as cones, over a wide range of mach of mach numbers in compression as well as in expansion shown in figures 9, and 10 as follows.

Figure 9. A simple perturbed cone in compression.

Figure 10. A simple perturbed cone in expansion.

Again, the P.S gives accurate results, in the reasonable situations, of the surface pressure since it is applied as a small perturbation to the mean flow where the three-dimensional relaxation effects are accounted for.

Another advantage of the Perturbation method is it's ability to capture pressure predictions when nonlinearities such as shock interactions exist in the mean flow. As shown in the following image (figure 11), at M∞ = 2.2, a shock induced by a rigid wedge in a fixed rectangular duct interacts with an elastic clamped flat plate with differential pressure on the outer part of the duct.
Figure 11. A fixed rectangular duct with an elastically flexible clamped flat plate.

A side view of this geometry (figure 12) shows the pressure contours generated by steady Euler equations. Notice the heavy shock interactions due to the wedge and rigid boundaries of the geometry.

Figure 12. A side view showing the pressure contours of the heavy shock interactions with the elastic plate.

A magnified view for the steady state deformation of the elastic plate generated by the unsteady Euler equations is shown in figure 13.

Figure 13. Steady state deformation of the elastic plate generated by the unsteady Euler equations.

This deformation was generated by first running a steady Euler solution to obtain the aerodynamic properties throughout the duct. Next, the process is restarted, this time allowing the plate to deform due to the resulting aerodynamic forces just acquired. After the plate's oscillation damps out, steady state deflection is achieved. Notice the plate's outward deformation due to the shock induced by the rigid wedge.

As a comparison, pressure using the piston theory is calculated and shown below in figure 14.

Figure 14. Steady state deformation of the elastic plate generated by piston theory.

Since this theory only takes into account the local conditions, the pressure induced by the shock is totally neglected causing large error in the prediction of the steady state deformation. The Perturbation solution is now applied with the following results in figure 15.
Since the perturbation method perturbs about the mean flow, the pressure induced by the shock is accounted for giving more accurate results as compared with the Euler steady state deformation in figure 13 above.

As another example of the capability of the Perturbation method, the pressure at three sectional cuts on a Generic Hypersonic Vehicle (GHV) is calculated at a $5^\circ$ angle of attack using the P.S. The piston method, and the steady Euler equations are also calculated for comparison. For the perturbation method, the GHV is perturbed $1^\circ$ about a steady Euler solution at $4^\circ$ with the following results shown in figure 16.

As a final illustration of the perturbation method a surface mesh of the baseline configuration for the GHV is generated (fig. 17). Flutter analysis is performed in the following manner: Given the surface mesh, a finite-element structural model is then developed to obtain the structural mode shapes and frequencies. Details of this structural modeling may be found in references 2,11, and 12. Next, a steady solution to the flowfield at Mach 2.2 was obtained using the finite-element Euler methodology which is used as the mean flow condition about which the perturbation solution is applied. The aeroelastic simulation consists of a 9 mode solution for 705 time steps (which is approximately 7 cycles of mode 1). The Euler solver is run over a range of 4 dynamic pressures, and the flutter boundary is
estimated through polynomial interpolation. For the purpose of run-time comparison, the same procedure is used for the perturbation solution.

As seen in Figure 18, the difference among flutter boundary estimates between the two codes for this case is found to be less than 4%, however the difference in run times is extremely significant.

![Figure 6. Flutter boundary and run-time comparisons.](image)

**Figure 6. Flutter boundary and run-time comparisons.**

M=2.2, 705 time steps / transient

Run time for the perturbation solution (P.S.) estimate is 3 minutes for each transient or approximately 15 minutes to identify the flutter boundary. On the other hand, run time for the Euler solution used to define the flutter boundary is 117 hours for each transient and approximately 469 hours to identify the boundary. All simulations are run on an IBM RS6000 3BT workstation.

**CONCLUSION**

Obtained results reveal that the combination of aerodynamic modeling with CFD methodology enhance the practicality of the use of time-marching CFD for supersonic aerelastic calculations in an operational environment. This method was tested over a wide range of mach numbers and proven valid in all reasonable cases. Furthermore, the application of this method in areas of shock interactions was studied and also proven accurate. It was shown for the configurations tested that the time required to predict flutter by time marching CFD can significantly be reduced by first obtaining an accurate estimate using modeling techniques. Furthermore, present research also indicates significant time savings in the identification of critical flight conditions for aircraft-type configurations in addition to the ones discussed here.

An added advantage of the presented approach is that the same grid may be used for the steady CFD solution, the perturbation model, and the time marching CFD solution. In addition, the method used can be easily implemented in any CFD algorithm capable of simulating supersonic flowfields with relatively minimal increase in code size and complexity.

**REFERENCES**


