ACCELERATING CFD-BASED AEROELASTIC PREDICTIONS USING SYSTEM IDENTIFICATION

Timothy J. Cowan* and Andrew S. Arena, Jr.†
Mechanical and Aerospace Engineering Department
Oklahoma State University
Stillwater, OK 74078

Kajal K. Gupta‡
NASA Dryden Flight Research Center
Edwards, CA

Abstract

System identification is evaluated as an efficient and accurate technique for modeling unsteady aerodynamic forces for use in time-domain aeroelastic analysis. In the system identification methodology, the constant coefficients of a linear system model are fit to the computed response time histories from a 3-D, unsteady CFD solver. The resulting model of the unsteady CFD solution is independent of both dynamic pressure and structural parameters. Hence, this methodology has the advantage that only one CFD flow-field computation for each Mach number must be completed to determine the aeroelastic instability boundary. Results show that system identification can accurately model the unsteady aerodynamic forces for complex aerospace structures of practical interest. The methodology results in a substantial savings in computational time when predicting aeroelastic instabilities.

Nomenclature

CFD = Computational Fluid Dynamics
[**C**] = generalized damping matrix
$\mathbf{f_a}$ = generalized aerodynamic force vector
[**K**] = generalized stiffness matrix
[**M**] = generalized mass matrix
nr = number of roots or mode shapes
q = free stream dynamic pressure
q = generalized displacement vector
u = vector of system inputs
y = vector of system outputs

* Graduate Research Assistant, Student Member AIAA.
† Assistant Professor, Senior Member AIAA.
‡ Aerospace Engineer, Member AIAA.

Introduction

Predicting instabilities in the aeroelastic behavior of aerospace structures is important in the design of modern aircraft which operate over a wide envelope. With recent advances in CPU speeds, current research has turned toward the application of CFD models to the solution of these aeroelastic problems. Using an unsteady Euler or Navier-Stokes CFD algorithm coupled with a structural dynamics solver, the complete aeroelastic response of the structure can be predicted. However, the major limitation in applying such a CFD solution is the computational time required to run a full aeroelastic simulation due to the high dimensionality of even the simplest geometry.

Compounding the problem, an aeroelastic instability cannot be predicted by just one such simulation. Rather, several simulations are required over the flight regime in order to predict the crossover from stable to divergent time histories. When running these coupled simulations, it is the unsteady CFD solution at each time step which requires the greatest amount of CPU time. The faster structural dynamics solver is essentially left waiting on the unsteady CFD solution at each time step. Hence, if an accurate and efficient replacement for the CFD solver could be developed, aeroelastic instability predictions would be much more computationally efficient.

Such a replacement might be found by developing a mathematical model for the unsteady CFD solution using system theory. Conceptually, the unsteady CFD solution implemented in an aeroelastic analysis is simply a dynamic system which computes an aerodynamic response based on a prescribed motion for the structure. Further more, the unsteady CFD solution can be assumed to be a dynamically linear system if only small perturbations about a nonlinear mean flow are considered.[1] Knowing this, a variety of efficient system modeling techniques can be applied which have been developed for linear systems.

One such technique is system identification. As it is defined, system identification is a process for obtaining a mathematical model of a dynamic system based on a set of measured data from the system.[2] This methodology is used to fit the parameters of a model structure to a set of recorded data from the dynamic system. The result is an algebraic model that is a mathematical map between the input and the output of the system. The success of this technique is then dependent on the initial choice of the model structure and the amount and quality of data used to train the model.

The emphasis of the present work is to determine the efficacy of using system identification techniques to accurately map the input-output relationship for an unsteady CFD solution to an arbitrary three-dimensional structure. This algebraic relationship then reduces the total computational time required for a CFD aeroelastic analysis by replacing the unsteady CFD solver in the coupled solution. Results are presented which address the extent to which the system identification methodology is applicable to geometries and flows of practical interest in aerospace applications.

Methodology

Computational analysis for this study was performed using the aeroelastic capabilities of the STARS codes developed at NASA Dryden Flight Research Center. STARS[3] is an highly integrated, finite element based code for multidisciplinary analysis of flight vehicles including static and dynamic structural analysis, computational fluid dynamics, heat transfer, and aeroservoelastic capabilities.

Structural analysis in STARS is accomplished using the finite element method to compute the eigenvectors and eigenvalues which describe the elastic modes for a structure. Any arbitrary motion of the structure can then be represented by multiplying each eigenvector by a generalized displacement and applying modal superposition. A complete aeroelastic analysis is accomplished by coupling a dynamics solver, using the modal vectors, with an unsteady CFD solver which computes the generalized aerodynamic forces acting on the structure. The basic aeroelastic equation of motion solved by STARS is given below in generalized coordinates.

$$[\mathbf{M}]\ddot{\mathbf{q}} + [\mathbf{C}]\dot{\mathbf{q}} + [\mathbf{K}]\mathbf{q} = \mathbf{f_a}(t) \qquad (1)$$

The unsteady aerodynamics forces on the right hand side of Eq. (1) are computed using a time-marched, finite element approach to solving the unsteady Euler equations. This CFD solution is performed on a mesh consisting of unstructured tetrahedra using the transpiration method to simulate structural deformations.

Considering the described solution scheme, the input-output relationship for the unsteady CFD solution can be represented by the simple block diagram shown in Figure 1.

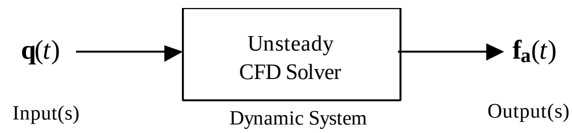

Figure 1: Block Diagram Representation of STARS Unsteady CFD Solver

As discussed previously, it is the unsteady CFD solution for the generalized aerodynamic force vector which requires the greatest proportion of CPU time. Hence, system identification will be applied to this system in order to develop an efficient algebraic model for the generalized aerodynamic forces. The basic system identification procedure will involve fitting the constant coefficients of a model structure to a set of actual time history data taken from the unsteady CFD solver.

There are several different types of model structures that could be used, but for the case of a multi-input, multi-output (MIMO) system, the most common is the autoregressive moving average (ARMA) model. The ARMA model describes the response of a dynamic system as a sum of scaled previous outputs and inputs to the system.[5] The basic ARMA model structure for a single-input, single-output (SISO) system is easily vectorized to yield the following model structure for a MIMO system:

$$\mathbf{y}(k) = \sum_{i=1}^{na} [\mathbf{A}_i]\,\mathbf{y}(k-i) + \sum_{i=0}^{nb-1} [\mathbf{B}_i]\,\mathbf{u}(k-i) \qquad (2)$$

Notice that the system response at any time step, k, is simply a linear combination of past inputs and outputs, making this model very easy to implement mathematically.

With the model structure of Eq. (2) in mind, the task is then to identify the matrices of constant coefficients, $[\mathbf{A}_i]$ and $[\mathbf{B}_i]$, for an assumed model order consisting of na past outputs and nb inputs. This is accomplished by fitting the model coefficients to a set of time history data from the unsteady CFD solver. In order to obtain time history data suitable for system identification, the unsteady CFD solution for a prescribed motion of the structure is completed rather than allowing the structure to move freely in the flow as in a typical aeroelastic simulation. The prescribed motion of the structure is a known input which is chosen because it excites a broad spectrum of response frequencies which contain the primary flow physics. The only limitation on choosing the prescribed input signal is that the generalized displacements and velocities must be mathematically consistent.

Historically, system identification has been used extensively in flight testing to estimate stability derivatives and modal damping parameters from flight test data. Thanks to this application, a great deal of research has been done on the optimal input required for successful parameter identification for flight vehicles. One of the most widely accepted inputs is the 3211 multistep due to its ease of implementation and broad frequency content.[8] Figure 2 shows the basic format of this input signal.

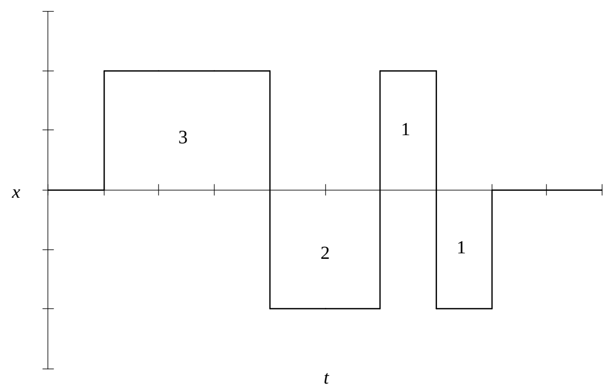

Figure 2: 3211 Multistep Input Signal

Although the multistep input signal is easy to implement experimentally, the sharp corners cause some numerical problems if this input is applied to displacements. As discussed previously, the displacements and velocities for the prescribed input signal must be mathematically consistent so that both boundary conditions for the unsteady CFD solution are satisfied. Hence, a multistep input applied to displacements must be differentiated to get the mathematically consistent velocity input. Doing so, one finds that the velocity input would be made up of five infinite spikes. Obviously, such an input cannot be numerically realized as a CFD boundary condition.

To avoid this numerical problem, the multistep input could be implemented on the velocity boundary condition and then integrated to get the mathematically consistent displacement boundary condition. This type of velocity multistep was tested along with several other inputs in the unsteady CFD solution, and the multistep was found to work best for identification.[10] Figure 3 shows the prescribed generalized displacement and velocity input for a two mode system which was found to work best for modeling.

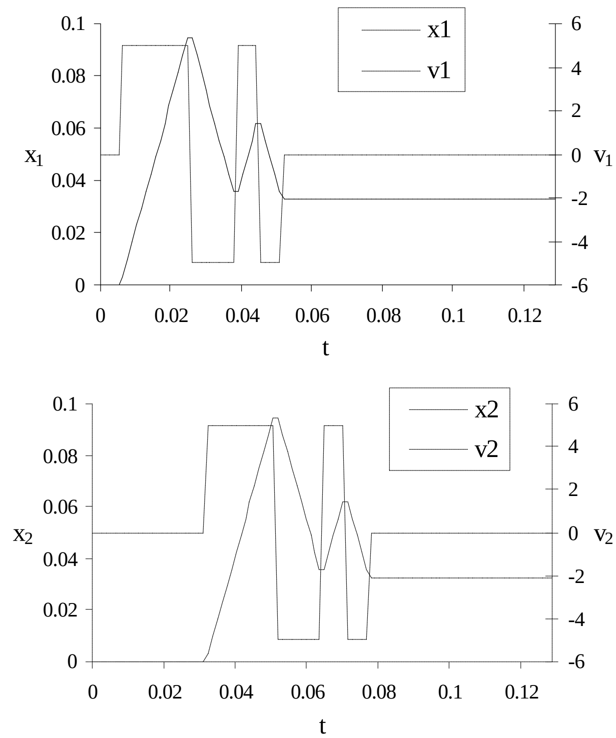

Figure 3: Prescribed Input Signal for Unsteady CFD Flow Solver

Notice in Figure 3 that the input signals for each mode are out of phase. This is necessary to allow the identification procedure to distinguish between the effects of different inputs in the system response. Results presented here will show that the time history data obtained using this input signal is sufficient for modeling the aerodynamic response of structures with as many as nine modes.

Once this input signal has been run through the unsteady CFD solution, singular value decomposition (SVD) is used to fit the parameters of the ARMA model structure to the time history data. SVD[9] produces a solution to the system of overdetermined equations that is the best approximation in a least-squares sense. This identification procedure requires an initial guess for the order of the ARMA model structure (*na* and *nb*). By then varying the order of the model, one searches for the model which minimizes the error between the CFD time history and the predicted model time history.

Once completed, the model is used in place of the unsteady CFD solution in the coupled simulation to predict the full aeroelastic response of the structure. Figure 4 illustrates conceptually how the ARMA model is implemented in a STARS aeroelastic simulation. The coupled aeroelastic solution with the discrete-time model in place can be executed at almost no computational cost relative to the unsteady CFD solution. Furthermore, the model is only dependent on the Mach number for which it was derived. It is not dependent on the free stream dynamic pressure or any of the structural parameters, such as generalized mass and stiffness. This allows one to vary these parameters and use the fast model solution to observe their effect on the aeroelastic response of the structure. Hence, a substantial amount of computational time is saved in the search for aeroelastic instabilities at each Mach number by running the coupled model solution rather than the time-marched CFD solution.

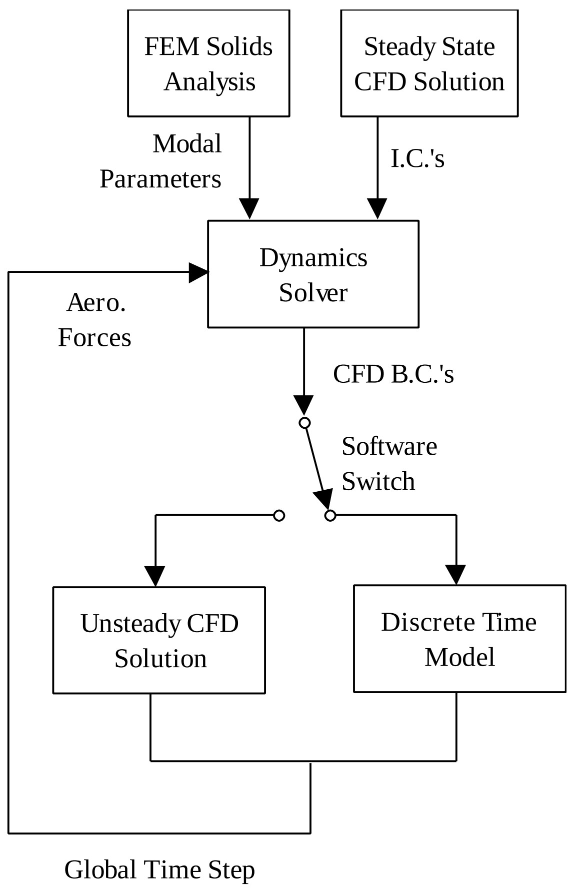

Figure 4: Implementation of System Model in STARS Coupled Aeroelastic Simulation

Results

Unsteady CFD solutions for the prescribed structural motion were run for several geometries over a wide range of Mach numbers. One such geometry is the AGARD 445.6 wing configuration which is a standard aeroelastic test case that has been investigated experimentally in the Langley Transonic Dynamics tunnel. A planform view of the configuration is shown in Figure 5.

This wing geometry is often used in the literature as a validation case for computational aeroelastic codes in the transonic flow regime. Recent work has shown that the STARS aeroelastic analysis module is capable of predicting the experimental data for this wing geometry including the transonic dip in the flutter boundary around Mach 1.0.[9]

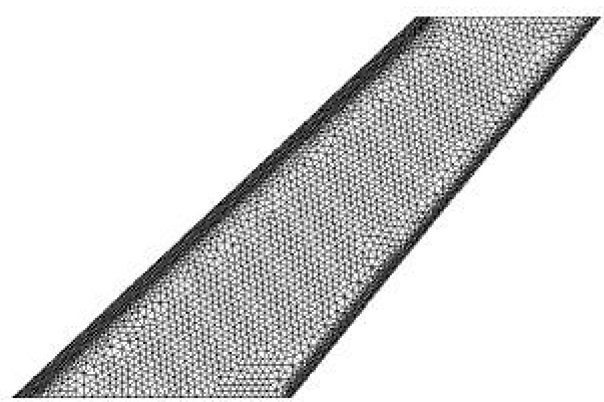

Figure 5: AGARD 445.6 Test Wing Geometry and Surface Discretization

In STARS, the AGARD is modeled structurally using the two dominant eigenvectors representing the first two natural vibration modes of the structure. These mode shapes physically represent wing first bending and torsion. The CFD mesh for the AGARD consists of 70,036 nodes and 376,125 tetrahedral elements.

Before beginning the system identification procedure on this geometry, it is important to remember that the modeling procedure assumes that the system is only *dynamically* linear. In order to make this assumption, the complete nonlinear mean flow, or steady CFD solution, must be computed and used as the initial condition for the unsteady CFD solution. This guarantees time accuracy for the unsteady solution and allows one to assume that the system is dynamically linear for small perturbations about the nonlinear mean flow.

Following the steady solution, the multistep input signal show in Figure 3 was analyzed using the unsteady Euler solver for the AGARD geometry at Mach 0.96 and a free stream density of 6.04×10^{-9} slinch/in^3, or a dynamic pressure of 0.44 psi. Using the resulting generalized aerodynamic force time history, the coefficients of the ARMA model structure were computed using the SVD algorithm. The generalized aerodynamic force time history from both the unsteady Euler solution and the discrete-time model solution found to best fit this data is shown in Figure 6.

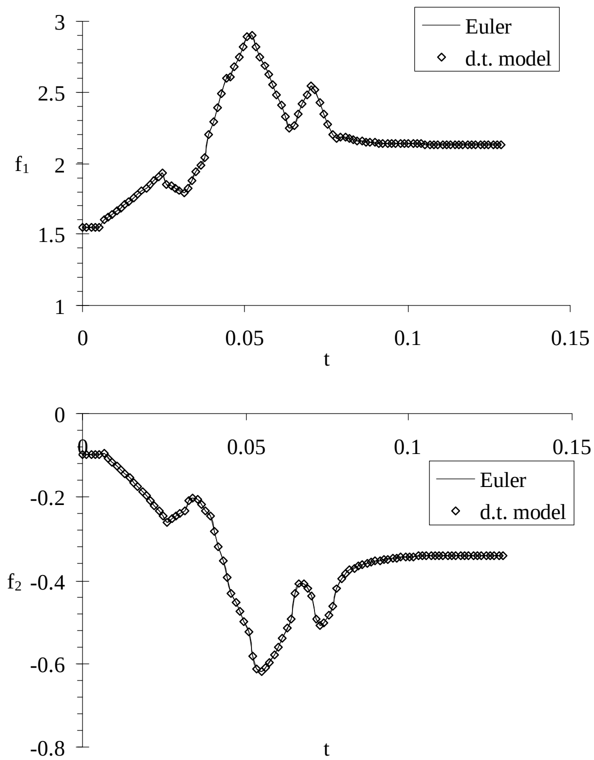

Figure 6: Comparison of Unsteady Euler and Model Solutions for the AGARD Multistep Input Signal of Figure 3.

The new discrete-time model is then used to search for instabilities at this Mach number by repeatedly varying the free stream density. Once the point of aeroelastic instability is found, the coupled Euler solution can then be run once to verify the accuracy of the coupled model solution. For Mach 0.96, a sample comparison between the aeroelastic time history predicted by the discrete-time model solution and the unsteady Euler solution is shown in Figure 7 for a free stream density near the instability boundary, $r = 3.2\times10^{-9}$ slinch/in^3. Notice that the two responses are in good agreement.

Further validations of the system identification procedure where run for the AGARD at five additional Mach numbers, resulting in a complete transonic flutter boundary prediction. As with Mach 0.96, a new multistep solution was computed with the unsteady Euler solver at each different Mach number. An ARMA model was then fit to each set of multistep time history data, and the resulting discrete-time model was then implemented in an aeroelastic solution to search for the instability boundary at that Mach number. Once the point of instability was found, the Euler solution was run once to verify that the model solution was valid. For each Mach number tested, the unsteady Euler and model time histories at the flutter boundary were in excellent agreement. The complete transonic flutter boundary for the AGARD wing configuration is given in Figure 8.

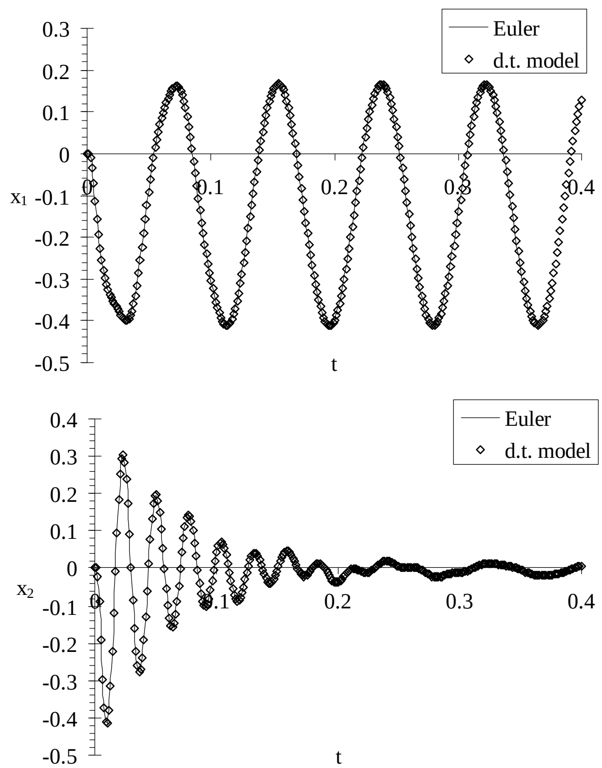

Figure 7: Comparison of Unsteady Euler and Model Solutions for the AGARD Aeroelastic Response at Mach 0.96

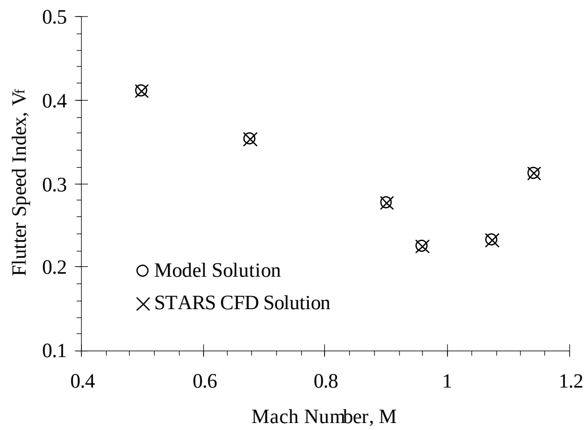

Figure 8: Comparison of AGARD Flutter Boundary Predicted by STARS CFD and Model Solutions

As it will be discussed, the time savings associated with the use of the model is significant. First consider the current method for applying CFD to aeroelastic analysis in STARS. For a given Mach number, the full unsteady CFD solution is run at least four times at different densities in a search for the crossover point from stable to divergent time histories. The results from these time histories are then interpolated to determine the approximate point at which the system is unstable. The total computational time to run just one unsteady CFD solution of sufficient length to be qualitatively useful is 120 CPU hours on an IBM RS600 3BT for the AGARD. Multiply that time by four and it requires 20 days to determine the approximate stability boundary for the AGARD at one Mach number.

The system identification technique requires only one run of the unsteady CFD solution for each Mach number. The length of the required multistep time history is about one fourth of the length required for a full aeroelastic run, so it runs in just under 30 CPU hours. Following the multistep solution, the entire procedure for computing the best parameters of the discrete-time model takes less than 30 minutes, and then the discrete-time model can be run repeatedly to predict complete aeroelastic time histories in less than 60 CPU seconds. Hence, the total savings in computational time for each Mach number is over 400 CPU hours, or 94% reduction in total CPU time. A comparison of the total time required to compute the neutral point of the AGARD at each Mach number is shown graphically in Figure 9.

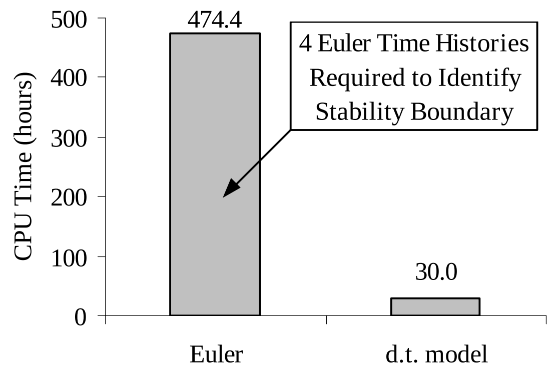

Figure 9: Comparison Between Required Computational Time For Euler Solution and Discrete-Time Model Solution

Another interesting geometry to study is that of the Generic Hypersonic Vehicle (GHV). The GHV is a testcase developed by NASA to test the aeroelastic effects that might be observed on a hypersonic vehicle. Figure 10 shows the CFD surface mesh used to model the GHV. The CFD mesh for the GHV consists of 58,786 nodes and 323,417 tetrahedral elements.

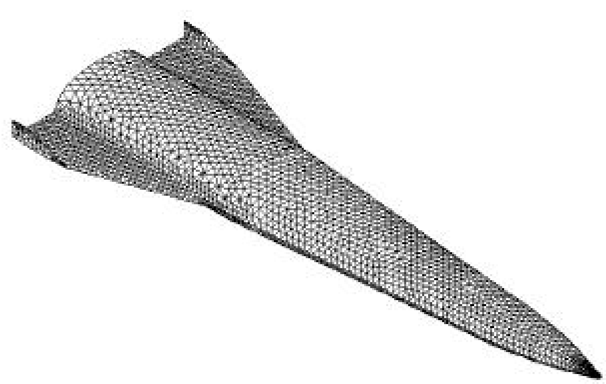

Figure 10: GHV Geometry and Surface Discretization

Structurally, the GHV is much more complicated than the AGARD as it is modeled using nine eigenvectors which represent various bending and torsional modes for the wings and the body itself. This geometry was first analyzed using STARS at Mach 2.2 and a free stream density of 2.8658×10^{-7} slinch/in^3, which corresponds to a dynamic pressure of 114.5 psi.

As with the AGARD, the steady flow field was first computed, followed by an unsteady CFD solution to a sequence of staggered multistep inputs on all nine modes. An aerodynamic model for the GHV is then computed by fitting the coefficients of the ARMA model structure to the multistep time history data. Figure 11 presents a comparison between the unsteady Euler solution and discrete-time model solution to the multistep input for the GHV. Only six out of the nine modes are shown here for brevity, however results are consistent for all nine.

The computed aerodynamic model for the GHV can then be used in the coupled solution to compute the aeroelastic response of the GHV. A sample comparison between the aeroelastic time history predicted by the discrete-time model and the unsteady Euler solution is shown in Figure 12 for modes one through six of the GHV. Again, results are consistent for all nine modes and can be found in reference 10.

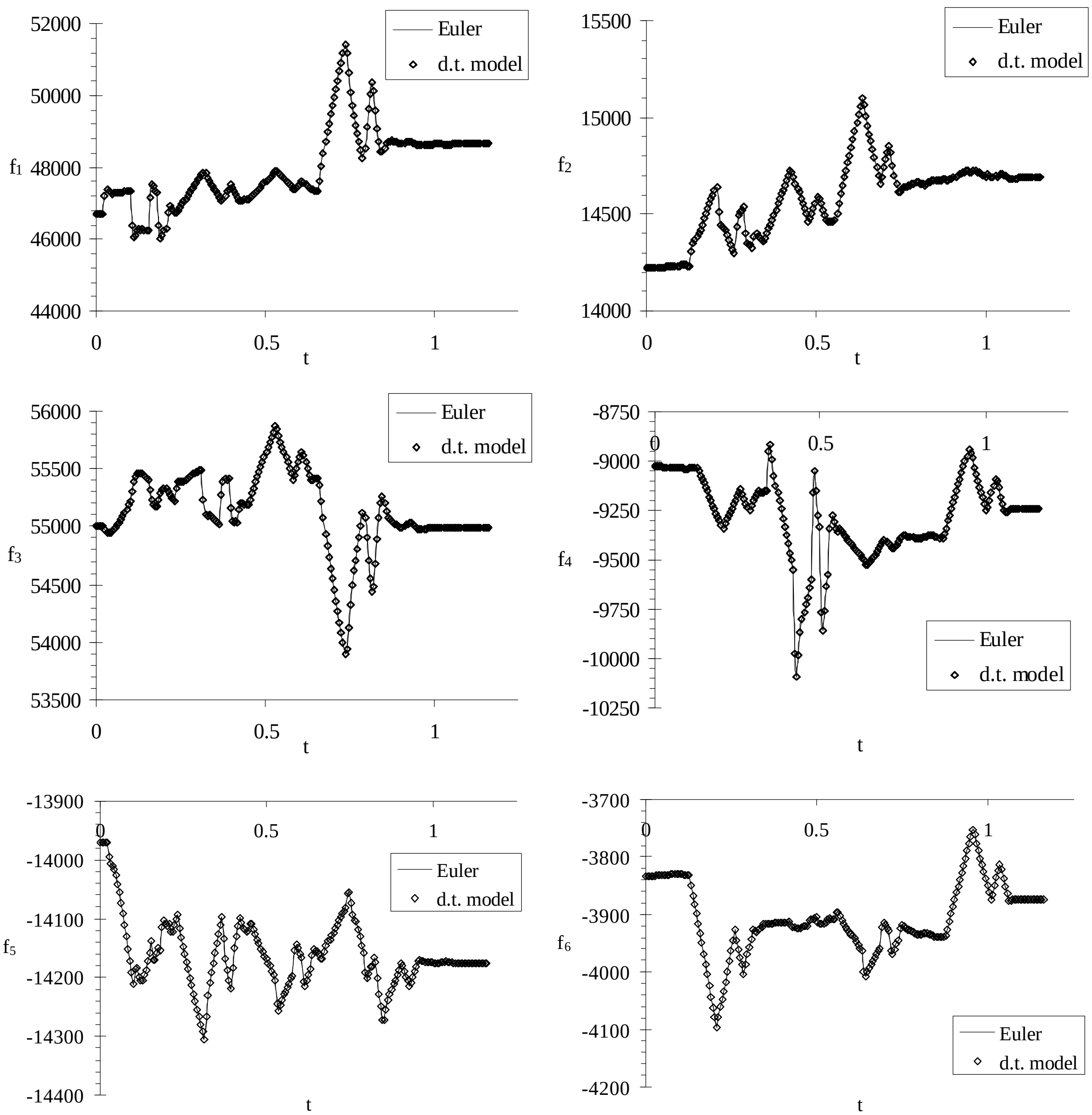

Figure 11: Comparison of Unsteady Euler and Model Solutions for the GHV Multistep Input Signal for Modes 1 through 6 at Mach 2.2

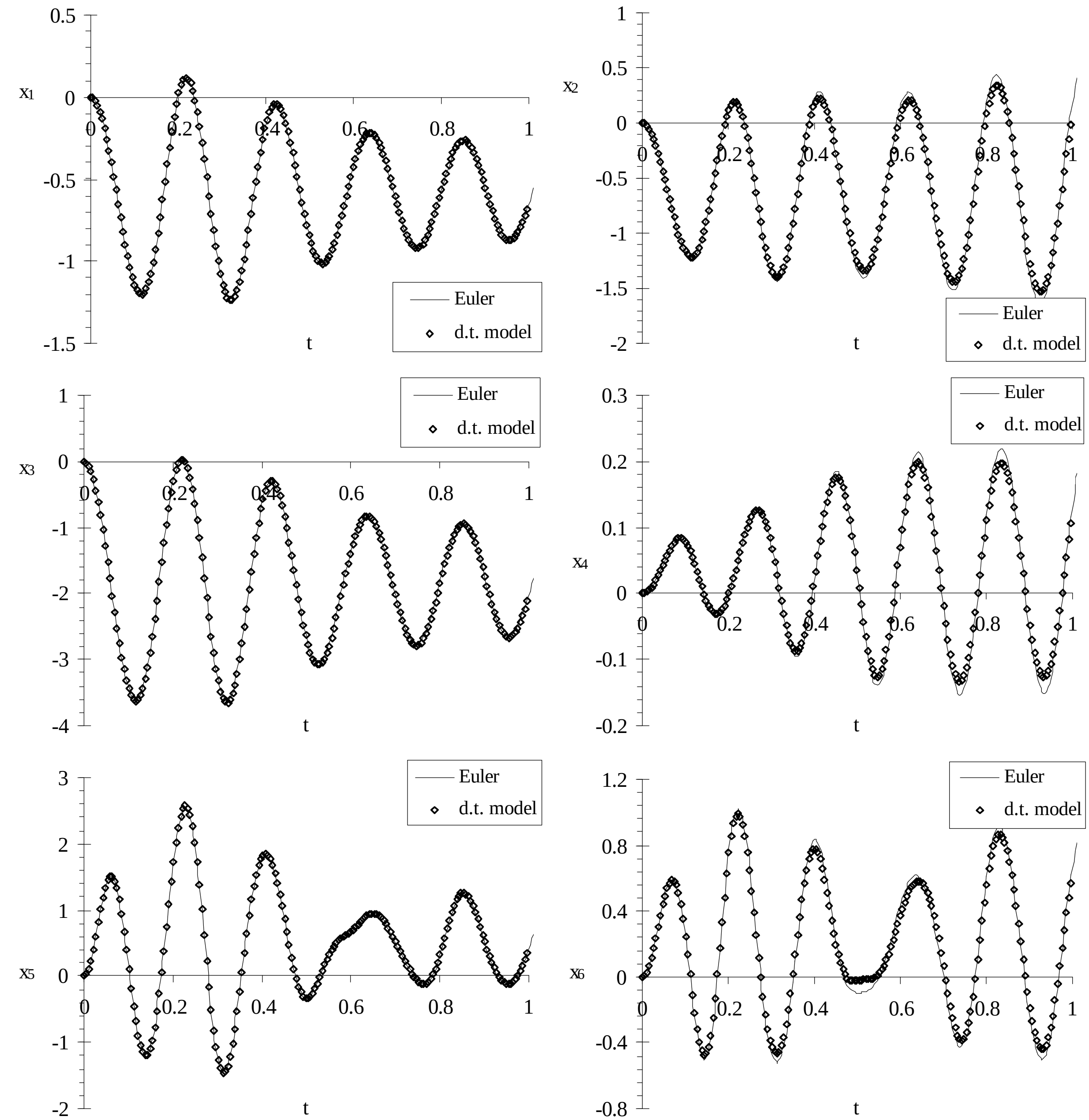

Figure 12: Comparison of Unsteady Euler and Discrete-Time Model Solution for the GHV Aeroelastic Response at Mach 2.2

Conclusions

The objective of this study was to develop an accurate and efficient method for developing a model of an unsteady CFD solution for use in computational aeroelastic analysis of complex aerospace structures. Research presented here demonstrates that system identification can be used to accurately map the input-output relationship of the unsteady CFD solution. The methodology assumes that the unsteady CFD solution is dynamically linear for small disturbances about a nonlinear mean flow. For such a system, an aerodynamic model can be developed by fitting the coefficients of a multi-input, multi-output ARMA model structure to a set of CFD time-history data. The resulting discrete-time aerodynamic model is then used in place of the unsteady CFD solution in the coupled aeroelastic analysis.

This technique can be extended to different structural geometries over a wide range of Mach numbers including the transonic regime. The methodology has the advantage that only one unsteady CFD solution is required for each Mach number. The aerodynamic model based on that solution is independent of both dynamic pressure and structural parameters. This allows one to use the same model while searching for aeroelastic instabilities by varying these parameters in the coupled solution. Computationally, the discrete-time model is extremely easy to apply and offers a large savings in total CPU time when used in an aeroelastic analysis. Hence, this approach may make the use of CFD simulations routine in the aeroelastic analysis of aerospace vehicles.

Acknowledgements

Funds for the support of this study have been allocated through the NASA-Ames University Consortium Office, under Interchange Number NCC2-5105, and Oklahoma State University.

References

1. Dowell, E.H., et al, *A Modern Course in Aeroelasticity*, 3rd Revised and Enlarged Edition, Klewer Academic Publishers, 1995.

2. Ljung, L., *System Identification: Theory For The User*, Prentice Hall, Inc., New Jersey, 1987.

3. Gupta, K.K., "STARS – An Integrated General-Purpose Finite Element Structural, Aeroelastic, and Aeroservoelastic Analysis Computer Program," NASA TM-4795, May 1997.

4. Hollkamp, J. J., and Batill, S. M., "Automated Parameter Identification and Order Reduction for Discrete Time Series Models", *AIAA Journal*, Vol. 29, No. 1, 1991, pp. 96-103.

5. Pinkelman, J. K., Batill, S. M., and Kehoe, M. W. "Total Least Squares Criteria in Parameter Identification for Flight Flutter Testing," *Journal of Aircraft*, Vol. 33, No. 4, 1996, pp. 784-792.

6. Hollkamp, J. J., and Batill, S. M., "A Recursive Algorithm For Discrete Time Domain Parameter Identification," AIAA Paper 90-26805.

7. Hamel, P. G., and Jategaonkar, R. V., "Evolution of Flight Vehicle System Identification," *Journal Of Aircraft*, Vol. 33, No. 1, 1996, pp. 9-28.

8. Press, W.H., Teukolsky, S.A., Vetterling, W.T., and Flannery, B.P., *Numerical Recipes in Fortran 77: The Art of Scientific Computing*, 2nd Edition, Cambridge University Press, 1992.

9. Gupta, K.K., "Development of a Finite Element Aeroelastic Analysis Capability," *Journal of Aircraft*, Vol. 33, No. 5, September-October 1996, pp. 995-1002.

10. Cowan, T.J., "Efficient Aeroelastic CFD Predictions Using System Identification," *Masters Thesis*, Oklahoma State University, May 1998.