A Time-accurate, Fast-running CFD Method for the Prediction of A Full Aircraft Flutter Boundary

H. Q. Yang[1], Ben Armstrong[2]
CFD Research Corporation, Huntsville, AL 35806, USA
hq.yang@cfdrc.com

In this paper, an efficient and robust fluid modal method through the use of a CFD solver is proposed to reduce the fully-coupled aeroelasticity problem to a second-order multi-degree-of-freedom (DOF) system while maintaining dominant nonlinearity and effects of all desired DOFs. A fully-coupled aeroelasticity problem is first formulated into a decoupled system, and a technique for rapid extraction of nonlinear fluid modal mass, damping, and stiffness from a CFD solver is developed. These fluid mass, damping, and stiffness are then used to construct a system of ordinary differential equations, thereby replacing the need for coupled CFD/CSD simulations. The proposed methodology is demonstrated on the AGARD 445.6 wing and a full aircraft. The fluid modal method simulations are shown to agree very well with CFD/CSD verification data. Validation again experimental data for flutter boundary of AGARD445.6 wing also showed good agreement. The proposed method provides a time-accurate, fast-running solution for describing the aeroelastic response of structures of air vehicles, including fighter aircraft, transport aircraft, and rotor blades.

I. Introduction

The accurate analysis of complex flows and the associated aeroelastic response is necessary for the design of next-generation flight vehicles. The coupling of computational fluid dynamics (CFD) solvers and computational structure dynamics (CSD) solvers can give accurate aeroelastic simulations. However, the increase in accuracy comes with a significant additional increase in computational cost. To mitigate this increased cost, the solution runs time, and the total number of solutions generated need to be minimized. Reduced-order modeling (ROM) is an accurate and cheap alternative to CFD/CSD simulations to study the dynamic aeroelastic response [1][2]

In this paper, an innovative "nonlinear fluid modal method" was developed to rapidly predict and offer unique physical insight into the nonlinear aeroelasticity of aircraft. The distinguishing factors of this effort are: (1) It is physics-based so that changes in aerodynamics, mass, inertia, and center of gravity are accounted for. (2) It is time-accurate and fast running. The coupled CFD/CSD problem is reduced to a set of ordinary differential equations, which can be solved in a matter of seconds compared to several hundred CPU hours. (3) It is CFD/CSD code independent. Any existing CFD solver can be used to build the nonlinear fluid modal model. (4) It is applicable to any geometry and flight condition.

---

[1] Chief Scientist, Aerospace and Defense Solutions, AIAA Associate Fellow
[2] Research Engineer, Aerospace and Defense Solutions

In this paper, the fully-coupled nonlinear Fluid-Structure Interaction aeroelasticity problem was first formulated into a decoupled time-accurate and fast running system of ordinary differential equations. Extraction techniques for the fluid modal properties from the high-fidelity CFD were then developed. The time-accurate and fast-running capability will then be demonstrated by numerically solving the constructed system of ODEs. The nonlinear fluid modal method was used to simulate the dynamic aeroelastic response of the AGARD 445.6 wing at several flight conditions. As shown, the simulated aerodynamic responses agree very well with CFD verification data and experimental validation data.

1 Simulation Model

The CFD mesh consists of 4.5 million cells of mixed elements and can be seen in Figure 1. An adiabatic no-slip wall boundary condition is applied on the Wing, Fuselage, and Tip Launcher Rail. A far-field boundary condition is applied on the outer boundary of the fluid domain. The freestream initial conditions are listed as follows: Mach = 0.96, Q (dynamic pressure) = 27 kPa, Density = 0.6 kg/m^3, Velocity = 300 m/s, and Altitude = 6864 m.

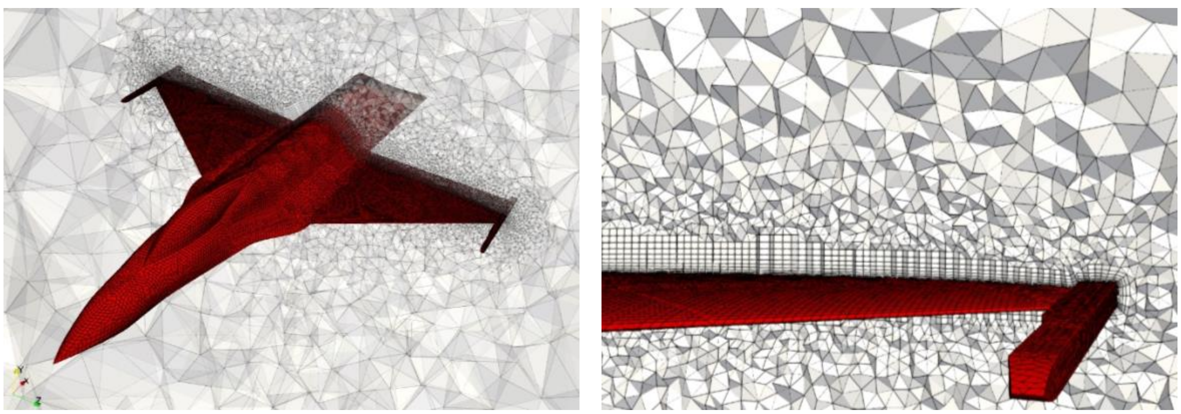

Figure 1. CFD mesh of the Open Source Fighter

The structural model shown in Figure 2 contains 10 modal shapes and has 3 structural components: Fuselage, Wing, and Stores. Stores are a pair of under-wing fuel tanks whose aerodynamic effects are neglected in the current CFD model.

Efficient Procedure for the Flutter Prediction Using Our Fluid Modal Method

The process employed for the demonstration of the nonlinear fluid modal method developed in this study can be broken down into the following five main steps:

1. Extract the fluid stiffness matrix, k;
2. Identify possible couplings between modes;
3. Calculate the critical dynamic pressure Q values and frequencies with the nonlinear fluid modal method
4. Run the fully-coupled FSI simulation;
5. Analyze and compare the predictions from the fully-coupled solution and the fluid modal method solution.

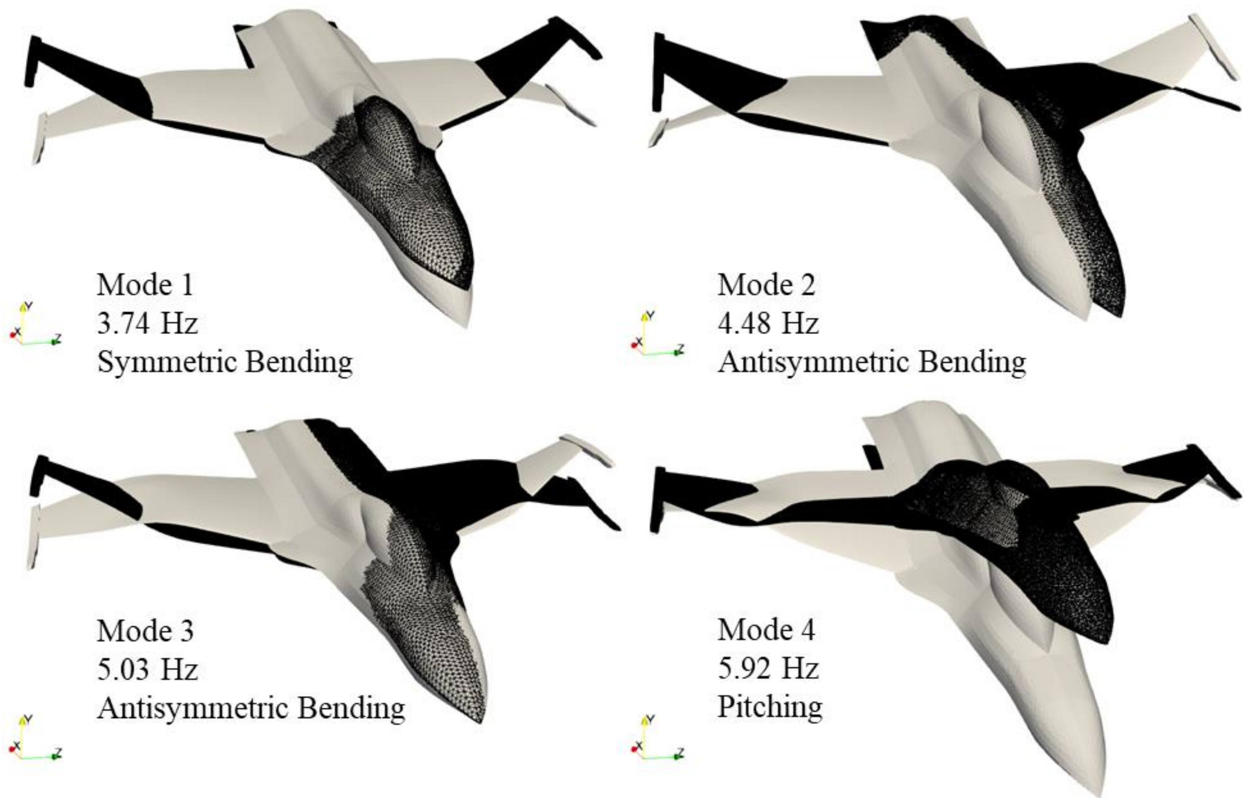

Figure 2. Structural Mode Shapes of the First Four Modes of the Open Source Fighter

2 Efficient Flutter Prediction Using Fluid Modal Method

In the following, we will demonstrate our procedure in obtaining the flutter boundary for the F-16 model.

Extraction of Stiffness Matrix

First, the fluid stiffness is extracted by displacing each mode by a specified value x_0 and holding that modal displacement until a steady-state solution is reached. An appropriate value for x_0 can be approximated using the following equation:

$$\frac{mode\ 1\ maximum\ displacement\ at\ wing\ tip}{wing\ span} * x_0 = 1\%$$

This ramp and hold simulation was conducted for each mode individually at the specified free-stream conditions. A "zero perturbation" simulation was also executed to obtain the modal load biases. This was achieved by forcing all the modes in the system to stay at an amplitude of 0 until a steady-state solution was reached. All simulations were conducted using the SA (Spalart-Allmaras) turbulence model.

Once the modal force is obtained, the stiffness is calculated using the following equation:

$$k_f = \frac{-(F_{f-s} - F_{bias})}{x_0}$$

F_{f-s} is the resulting modal load for each mode. F_{bias} is the resulting modal load of each mode from the "zero perturbation" simulation. Upon reaching a steady-state these modal loads values are extracted, and the nonlinear fluid modal stiffness is calculated. Table 1 shows the format of the modal stiffness matrix.

Table 1. Format of the Fluid Modal Stiffness Matrix

k_f	**Mode 1**	**Mode 2**	…
Mode 1	Mode 1 response to Mode 1 displacement	Mode 1 response to Mode 2 displacement	…
Mode 2	Mode 2 response to Mode 1 displacement	Mode 2 response to Mode 2 displacement	…
…	…	…	…

Table 2 presents the resulting fluid modal stiffness matrix of the Open Source Fighter.

Table 2. Fluid Modal Stiffness Matrix of Open Source Fighter at M = 0.96 and Q = 27kPA

k_f	**Mode 1**	**Mode 2**	**Mode 3**	**Mode 4**	**Mode 5**	**Mode 6**	**Mode 7**	**Mode 8**	**Mode 9**	**Mode10**
Mode1	-21.25	-3.75	-3.75	26.25	-2.5	-3.75	162.5	1.25	162.5	-1.25
Mode2	0	76.25	-65.75	0	0	-3	0	-215	0	173.75
Mode3	0	57.5	-31.625	0	0	-1.875	0	-129.5	0	72.25
Mode4	-27.5	0	0	-262.5	0	0	-87.5	0	-3.75	0
Mode5	-0.625	-0.2	-0.125	-0.4625	-0.125	-0.125	2.5	-0.0625	3	-0.125
Mode6	0	-1.0875	0	0	0	-0.3375	0	2.8375	0	-1.1625
Mode7	-56.25	-1.25	-0.5	-105	0	0	178.75	8.75	278.75	1.25
Mode8	0	-105	100	0	5.625	5.625	0	290	0	-257.5
Mode9	-48.75	0	0	-156.25	0	0	12.5	0	145	0
Mode10	0	-34.625	-7.7	0	0.875	0.875	0	53.375	0	-6.6

Identification of Mode Coupling

The possible coupling modes are identified by the color marks. These modes are coupled because they have opposite signs in the off-diagonal terms. These are highlighted in the table above and listed here:

- $k_{1,4}$ and $k_{4,1}$
- $k_{1,7}$ and $k_{7,1}$
- $k_{1,9}$ and $k_{9,1}$
- $k_{2,3}$ and $k_{3,2}$
- $k_{2,10}$ and $k_{10,2}$
- $k_{3,8}$ and $k_{8,3}$
- $k_{3,10}$ and $k_{10,3}$
- $k_{5,8}$ and $k_{8,5}$
- $k_{5,10}$ and $k_{10,5}$
- $k_{6,10}$ and $k_{10,6}$
- $k_{8,10}$ and $k_{10,8}$

Calculate the Critical Q Values and Frequencies

Our fluid modal method needs the following inputs for each modal coupling: omega1, omega2, kappa11, kappa12, kappa21, kappa22, and g_{mass}. Where 1 represents the first mode of the coupling and 2 the second. The equations for these inputs are as follows:

- $\omega_x = 2 * \pi * f_{x1}$, where f_{x1} is the natural frequency of mode x,
- $\kappa_{xx} = k_{xx}/Q$, where k_{xx} comes from the stiffness matrix and Q is the dynamic pressure,
- and g_{mass} is the generalized mass term and for this case it has a value of 1.

The algebraic equation for the Nonlinear Fluid Model Method is as follows:

$$r^2 = \frac{1}{2}[(\omega_2{}^2 + \kappa_{22}q + \omega_1{}^2 + \kappa_{11}q) \pm \sqrt{(\omega_2{}^2 + \kappa_{22}q - \omega_1{}^2 - \kappa_{11}q)^2 + 4k_{12}k_{21}q^2}]$$

Flutter Frequency — Flutter Q Value

Solving for r (root) will result in a pair of critical Q values (due to the ±) which can then be used to calculate corresponding critical frequencies. If a Q value is negative, it is expected not to exist, but this will need verification.

The output of the Nonlinear Fluid Modal Method for the Open Source Fighter is shown in Table 3. Note that the – and + represent the solution from using either the – or + before the square root in the equation. All cells highlighted in grey either have a negative Q value or a corresponding frequency that resulted in *NAN* (not a number). The modal couplings with these results are expected to not occur.

Table 3. Results of the Nonlinear Fluid Modal Method

Coupled Modes	$\boldsymbol{Q-}$	$\boldsymbol{Q+}$	$\boldsymbol{f-}$	$\boldsymbol{f+}$
$k_{1,4}$ and $k_{4,1}$	75,919	119,431	3.8	2.9
~~$k_{1,7}$ and $k_{7,1}$~~	-12,989,777	-291,754	*NAN*	6.8
$k_{1,9}$ and $k_{9,1}$	10,737,977	-366,807	26.4	7.2
$k_{2,3}$ and $k_{3,2}$	24,107	-368,578	4.8	3.9
$k_{2,10}$ and $k_{10,2}$	783,993	-2,581,353	11.5	4.8
~~$k_{3,8}$ and $k_{8,3}$~~	-1,186,052	-203,057	*NAN*	7.3
~~$k_{3,10}$ and $k_{10,3}$~~	8,172,596	-2,507,093	*NAN*	12.5
~~$k_{5,8}$ and $k_{8,5}$~~	-243,563	-241,580	8.0	8.0
$k_{5,10}$ and $k_{10,5}$	19,597,892	24,057,391	8.2	7.3
$k_{6,10}$ and $k_{10,6}$	16,621,896	32,417,051	8.7	5.0
$k_{8,10}$ and $k_{10,8}$	130,838	1,118,370	13.4	17.6

Based on our theory, the possible mode couplings sorted in ascending Q value (dynamic pressure) are shown in Table 4. From the table, it can be seen that at a dynamic pressure of 24K, one can expect modes 2 and 3 to be coupled together and have a coupling frequency of 4.8 Hz. Even though several other mode couplings are listed, this coupling is the most important since it will occur first.

Table 4. Possible Mode Couplings of the Open Source Fighter

Coupled Modes	*Q*	*f*
$k_{2,3}$ and $k_{2,3}$	24 kPa	4.8 Hz
$k_{1,4}$ and $k_{4,1}$	76 kPa	3.8 Hz
$k_{1,4}$ and $k_{4,1}$	119 kPa	3.0 Hz
$k_{8,10}$ and $k_{10,8}$	131 kPa	13.4 Hz
$k_{2,10}$ and $k_{10,2}$	784 kPa	11.5 Hz
$k_{8,10}$ and $k_{10,8}$	1.1 MPa	17.6 Hz
$k_{1,9}$ and $k_{9,1}$	10.7 MPa	26.4 Hz
$k_{6,10}$ and $k_{10,6}$	16.6 MPa	8.7 Hz
$k_{5,10}$ and $k_{10,5}$	19.6 MPa	8.2 Hz
$k_{5,10}$ and $k_{10,5}$	24 MPa	7.3 Hz
$k_{6,10}$ and $k_{10,6}$	32.4 MPa	5.0 Hz

3 Verification of Predicted Flutter Values Using Fully Coupled Solution

The fully coupled FSI simulation will be used to verify the accuracy of the nonlinear fluid modal method. The fully coupled aeroelastic simulation can be conducted in two steps: 1) obtaining a steady-state solution of the flow field and 2) ping the structure (start of FSI) and observing the unsteady response.

Steady-State Solution

Obtaining a steady-state solution ensures a good initial condition before any structural motion occurs. This can be done by running a fully coupled CFD and structural code with global time-stepping or by using a larger time-step with startup iterations with local time stepping. An example steady-state solution result can be seen in Figure 3. The free-stream conditions of this case are: Mach = 0.96, Q = 60 kPa, Density = 0.6 kg/m^3, Velocity = 447 m/s, and Altitude = 717 m. Note that the dynamic pressure for this example is a little more than double that which was used for extracting the stiffness matrix. Lift, drag, and pitch coefficients are all converged. As expected, several shocks are present in the flow field along the fuselage and on the wing.

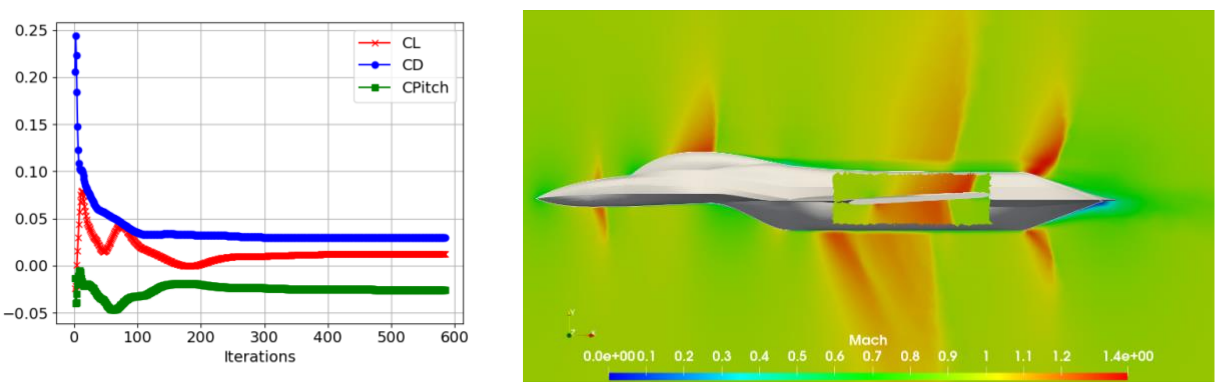

Figure 3. Steady-State Flow Field and Convergence Properties of the Q = 60 kPa Run

Ping the Structure and Observe the Unsteady Response

The structure can be "pinged" by supplying an initial velocity to all modes. With the ping, each mode will begin to oscillate. After some time, the modes will either decay or grow. The modes that grow are unstable. For example, the modal displacements of Modes 2 and 3 for the same case from Figure 3 are shown in Figure 4. The predicted critical Q value for flutter onset was calculated to be 24 kPa. Since the simulation was conducted well above that region at a dynamic pressure of 60 kPa, one should expect the modes that lead to instability to grow without bound, as shown in Figure 4.

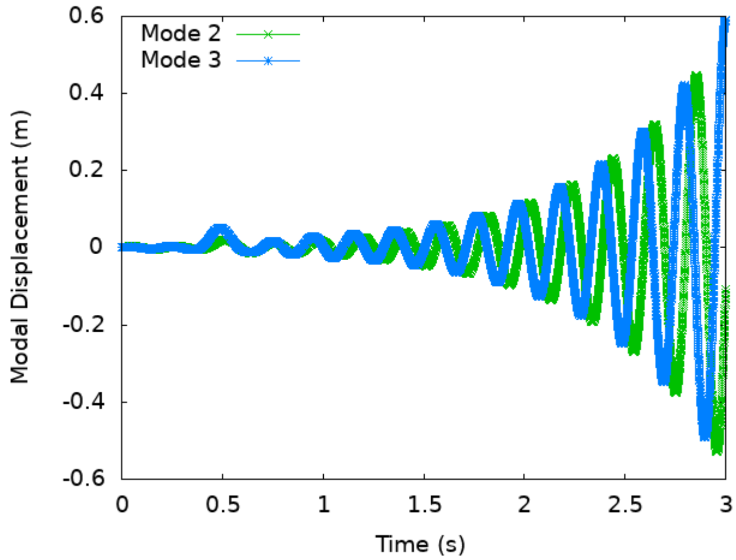

Figure 4. Modal Displacements of Modes 2 and 3 for the Q = 60 kPa Run

Capturing the Critical Q value and Frequency of Flutter Onset

For this study, flutter was defined as when the coefficient of Roll of the simulation resulted in a damping factor of zero. A total of 8 dynamic pressure values ranging from 10 kPa to 60 kPa were simulated to narrow in on the critical Q value. Figure 5 shows the damping factors of the 27.5 kPa, 30 kPa, and 35 kPa runs. A quadratic fit was employed to extrapolate the critical Q value. The critical Q value extrapolated from the simulation runs is very close to that predicted by the nonlinear fluid modal method, resulting in a percent error of less than 2%. To determine the flutter frequency, the FFT (fast Fourier transform) of the coefficient of Roll of the Q = 40 kPa simulation run is shown in Figure 6. The theoretical critical frequency determined by the nonlinear fluid modal method was 4.8 Hz, as given in Table 5. The plot of Figure 5, which determines the frequency content of the cRoll data, shows a peak at 4.8 Hz as well.

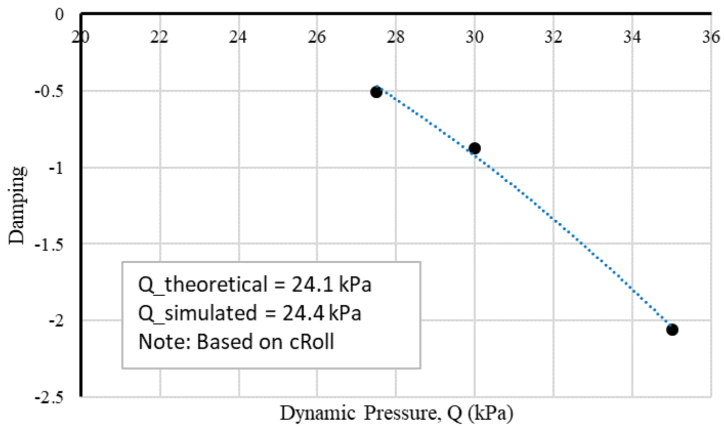

Figure 5. Extrapolation of the Simulated Critical Q Value of the Open Source Fighter at Mach = 0.96

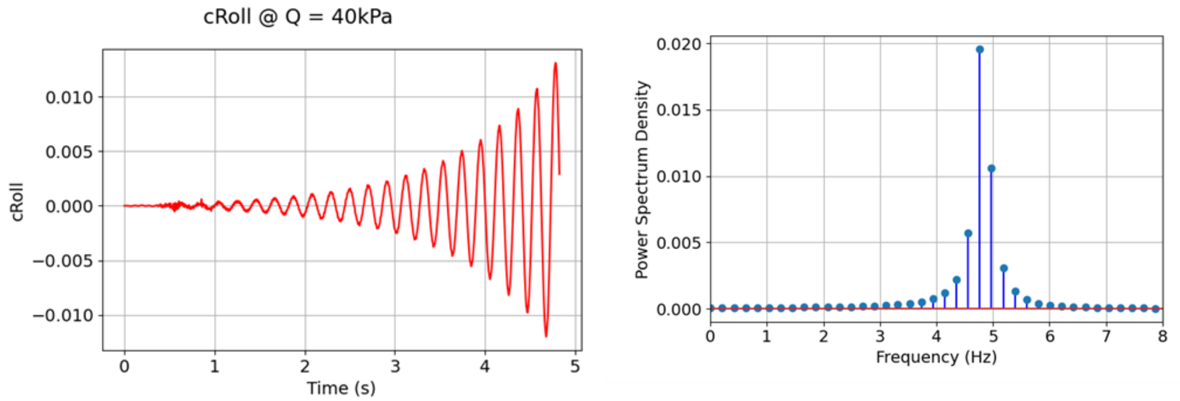

Figure 6. FFT of the Coefficient of Roll of the Q = 40 kPa Simulation Run

Table 5 shows the comparison of dynamic flutter pressure and flutter frequency from the fully coupled solution and the current fluid modal method. One can see very good agreements.

Table 5. Comparison of Flutter Dynamic Pressure and Flutter Frequency

	Current Fluid Modal Method	Fully Coupled Solution
Flutter Dynamic Pressure (kPa)	24.1	24.4
Flutter Frequency (Hz)	4.80	4.80

4 Prediction of Flutter Boundary

All results up to this point have only been dealing with the onset of flutter at a Mach number of 0.96. However, from past aeroelastic analysis, the critical Q value has been shown to be dependent on Mach number. Several methods for predicting flutter have been developed over the years. One such method known as the Schur method was implemented on the Open-Source Fighter Geometry by Marques *et al.* [5]. These results are presented in Figure 7, along with the calculated critical Q value determined by the nonlinear fluid modal method. Since the Schur method included Euler solutions, the stiffness matrices for each Mach number were extracted using an Euler solver. As seen in the figure, the nonlinear fluid modal method compares well with the Shur method. The critical Q values calculated for the nonlinear fluid modal method were determined using only the coupling between Mode 2 and Mode 3.

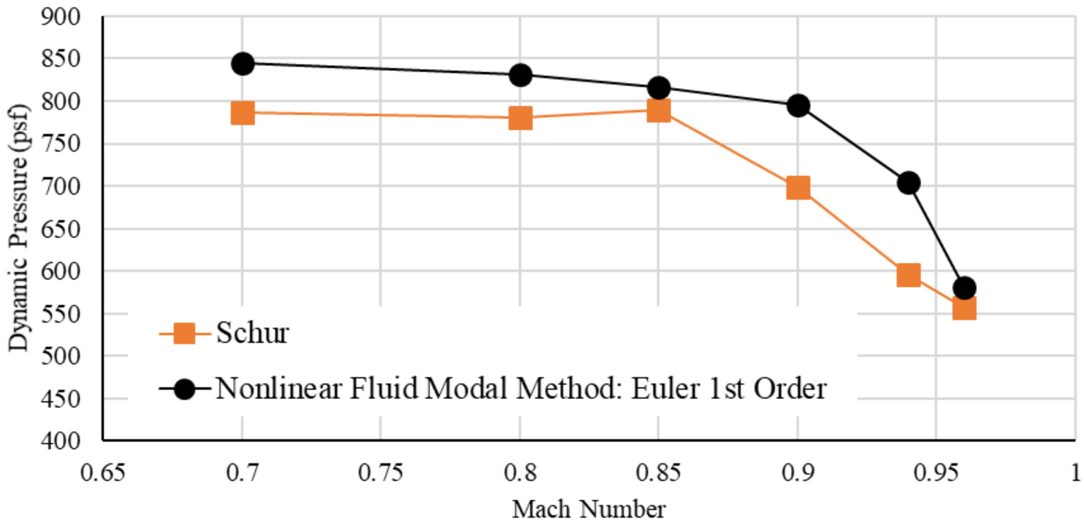

Figure 7. Flutter Boundary for the Open-Source Fighter with Comparisons between the Schur and Nonlinear Fluid Modal Methods

Acknowledgments

This work was supported by the Air Force Research Laboratory at Wright-Patterson Air Force Base under a Small Business Innovation Research Program Phase I. The authors would like to thank Dr. Brian Barlow of the AFRL for their support and encouragement during this effort.

References

[1] Silva, W.A. and Bartels, R.E. “Development of Reduced-Order Models for Aeroelastic Analysis and Flutter Prediction Using the CFL3Dv6.0 Code,” J. Fluids Struct. 2004, 19, 729-745.

[2] Silva, W.A., Vasta V.N., and Biedron, R.T. “Development of Unsteady Aerodynamic and Aeroelastic Reduced-Order Models Using the FUN3D Code,” 2009, IFASD Paper No. 2009-30.

[3] Biedron, Robert T., et al., “FUN3D Manual: 12.9,” NASA-TM-2016-219012, 2016.

[4] H. Q. Yang and R. E. Harris. "Development of Vertex-Centered, High-Order Schemes and Implementation in FUN3D," AIAA Journal, Vol. 54, No.12, pp. 3742-3760, 2016

[5] Yates, E. C. Jr., AGARD standard aeroelastic configuration for a dynamic response, candidate configuration I.-Wing 445.6, NASA TM-100492, 1987.

[6] Yates, E. C. Jr., Land, N. S., and Foughner, J. T. Jr., Measured and calculated subsonic and transonic flutter characteristics of a 45° sweptback wing planform in air and in Freon-12 in the Langley transonic dynamics tunnel, NASA TN D-1616, 1963.

[7] Silva, W., Chwalowski, P., and Perry, P. “Evaluation of Linear, Inviscid, Viscous and Reduced-Order Modeling Aeroelastic Solutions of the AGARS 445.6 Wing Using Root Locus Analysis.” AMS Seminar. April 2015.

[8] Ridders, C. F. J. “A New Algorithm for Computing a Single Root of a Real Continuous Function.” IEEE Trans. Circuits Systems 26, 979-980, 1979.