

UNSTEADY AERO-ELASTIC COUPLING FOR THE F/A-18 VERTICAL TAIL BUFFETING

M. Guillaume^{*}, J. Vos^{**}, A. Gehri^{*}, B. Bucher^{*}, S. Merazzi^{***}, Th. Ludwig^{***}, G. Mandanis^{****} *RUAG Aerospace, **CFS Engineering, ^{***}SMR SA, ^{****}M@M GmbH

Keywords: Computational Fluid Dynamics, Loads, Fluid Structure Coupling

Abbreviations

ALE	Arbitrary Lagrange Eulerian		
AoA	Angle of Attack		
Alt	Altitude		
ASIP	Aircraft-Structural-Integrity-		
	Program		
CAD	Computer Aided Design		
CFD	Computational Fluid Dynamic		
CSM	Computational Structural Mechanic		
CSS	Conventional Serial Staggered		
Е	Elastic Modulus		
FEM	Finite Element Model		
FSI	Fluid Structure Interaction		
G	Elastic Shear Modulus		
HSTAB	Horizontal Tail		
IGES	3D CAD format		
LEF	Leading Edge Flap		
LEX	Leading Edge Extension		
NSMB	Navier Stokes Multi Block		
Nz	Load factor		
OEM	Original Equipment Manufacturer		
ρ	density		
Re	Reynolds Number		
SPC	Single Points Constraints		
TEF	Trailing Edge Flap		
TFI	Trans Finite Interpolation		
ν	Elastic Contraction Ratio		

Abstract

For the Swiss F/A-18 Aircraft, the Boeing Company performed an Aircraft Structural Integrity Study (ASIP) to analyze the structural integrity of the entire airframe based on the Swiss design spectrum. The Swiss maneuver spectrum was three times more severe than the US Navy design spectrum, but the dynamic spectrum was not more severe than the US Navy dynamic design spectrum, see figure 1. For the validation of the Swiss Redesign a full scale fatigue test was carried out at RUAG Aerospace. Only few relevant fatigue load cases for the entire airplane were obtained from The Boeing Company in St. Louis, the F/A-18 Original Equipment Manufacturer (OEM).

This situation pushed RUAG Aerospace to search for methods to generate independently aerodynamic loads for the F/A-18 and as a result a considerable investment was made in the further development of the Computational Fluid Dynamics (CFD) solver NSMB, and the application of this solver to steady and unsteady simulations.

To take the structural response of the structure into account a Fluid Structure Interaction (FSI) tool was developed. In a first step only static aero-elastic deformation was considered, in a second step the tool was extended to permit dynamic fluid structure interaction simulations for the analysis of the F/A-18 Vertical Tail buffeting. This novel tool permitted to better understand the complicated flow field over the entire F/A-18 full flight envelope and to check some load cases delivered by the OEM. The FSI tool follows the concepts developed by Farhat et al. in which the structural part is solved using a modal integration with the Newmark method. A semi-implicit coupling approach is used between the fluid and structural mechanics solvers in which the structural mechanics solver is called in the inner-loop of the dual time stepping method used by the CFD solver.

The unsteady aero-elastic coupling tool was validated using the data from the well known

AGARD445.6 wing. The results were in full agreement with the experimental data.

For the F/A-18 Vertical Tail unsteady aeroelastic loads were calculated for the development of a buffeting spectrum. This data will be used to determine the dynamic Swiss severity of the F/A-18 Vertical Tail.

The use of novel unsteady aero-elastic simulation should improve the design of modern aero structures due to buffeting and flutter problems in an early design phase.

1. Computational Fluid Dynamics using the NSMB Code

1.1 The NSMB Flow Solver

The calculations of the F/A-18 flow field were made using the NSMB Structured Multi Block Navier Stokes Solver. NSMB was developed from 1992 until 2003 in a consortium which included Airbus France and SAAB Military Aircraft. Since 2004 NSMB is further developed in a new consortium lead by CFS Engineering and composed of RUAG Aerospace (Emmen), EPFL (Lausanne), EHTZ (Zürich). (Toulouse), IMFT **IMFS** (Strassbourg), the Technical University of München and the University of the Army in München.

NSMB employs the cell-centered Finite Volume method using multi block structured grids to discretize the Navier Stokes equations. Various space discretization schemes are available to approximate the inviscid fluxes.

The space discretization leads to a system of ordinary differential equations, which can be integrated in time using either the explicit Runge Kutta scheme or the semi-implicit LU-SGS scheme. To accelerate the convergence to steady state the following methods are available:

- local time stepping
- implicit residual smoothing (only with the Runge Kutta scheme)
- full multi grid (grid sequencing)
- multi grid

- pre-conditioning for low Mach number
- artificial compressibility for incompressible flows
- scale adaptive simulation (Mentor SAS)

Different turbulence models have been thoroughly tested and validated for NSMB:

- Baldwin-Lomax algebraic model
- Spalart-Allmaras 1 equation model
- Chien k-ε 2 equations model
- Wilcox k-ω 2 equations model
- Menter Baseline and Shear stress $k-\omega 2$ equations model
- scale adaptive Simulation (Menter SAS)

The ALE approach is available to simulate the flow on moving grids. Recently a remeshing algorithm was implemented in NSMB to permit the simulation of the flows on deforming grids, as found for example in Fluid Structure Interaction problems.

The NSMB code was originally written in Fortran 77 and the code is at present a mix of Fortran 77 and Fortran 90. The code is fully parallelized.

1.2 The F/A-18 Mesh

The most time consuming process in a CFD simulation is the generation of the grid. This involves different steps. First (if required) the CAD surface needs to be cleaned up, then a multi block topology needs to be set-up, and finally the mesh is generated. The latest mesh for the F/A-18 fighter was generated by Mindware in collaboration with the Center of Aerodynamics of RUAG Aerospace. This mesh has 2802 blocks and 13.9 million cells (see figures 2 and 3) for one half of the aircraft. Many geometrical details as for example antennas and the SIWA fins are modeled, explaining the large size of the mesh.

In all calculations discussed here it is aircraft assumed that the is perfectly symmetrical symmetrical and only load considered conditions were until now. Consequently only one half of the aircraft was used in the calculations.

2 Unsteady State and Transient Calculations

2.1 Calculation for C2S825 Load Case

calculations fluid Unsteady without structure interaction were made for the Boeing load case C2S825, which concerns a 8.5 G manoeuvre at Mach=0.7, Altitude 15'000 feet and angle of attack 26.6 deg. The dual time stepping approach was used with a constant outer time step of $2.5 \ 10^{-4}$ seconds. Two thousand time steps were made, and 0.5 seconds of real time was simulated, see figure 4. The pressure and skin friction vector were saved each outer time step to permit the analysis of the unsteady aerodynamic loads on the aircraft. The calculation ran for several days on a PC cluster and generated 350 GBytes of data. Comparison of the mean unsteady aerodynamic loads with the loads obtained using a steady calculation showed significant differences in loads on the Aft Fuselage, Vertical Tail, Rudder, Trailing Edge Flap, Aileron and Horizontal Stabilizer, indicating that unsteady flow effects are important on these components of the aircraft.

2.2 Transient Analysis for the F/A-18 Vertical Tail

The unsteady flow field generated by the Leading Edge Extension (LEX) vortex produces a time and location dependant pressure field on the Vertical Tail surface. In order to capture these unsteady forces the fin surface has been divided into 54 (6x9) trapezoidal panels as it is shown in the figure 5. By each time step the resulting force of the pressure acting on both sides of each panel has been calculated using unsteady CFD calculations.

A detailed finite element model of the Vertical Tail with the six stubs and the flexible attachment to the fuselage originally developed for stress analysis was available by RUAG. 54 lumped masses were included to the model, attached at the central point of each panel and corresponding to the mass of each trapezoidal domain. This dynamic model has been used to calculate the first five "Eigenmodes" of the Vertical Tail structure and its rudder. The resulting deformations and frequencies were perfectly satisfactory compared to the measured modes founded in the documentation of the aircraft OEM manufacturer. The frequencies lie between 15 and 85 Hz.

The time dependant aerodynamic panel forces mentioned above have been applied on each node of the lumped masses and a transient calculation of the Vertical Tail has been achieved for a real duration of 0.5 second and 2000 time steps. This time range is quite short but acceptable with regard to the frequencies, which are interesting for our dynamic investigation.

For the transient analysis of the Vertical Tail structure the simplified Nastran CQUAD4 model was used. The shape and size was adjusted to the geometry of the aircraft Vertical Tail FEM. The mechanical properties were selected and the thickness was uniform and adjusted, such that the first bending eigen frequency was close to that of the aircraft. The rudder was not modeled and the fixation was done with SPC's.

The analysis was the performed using the modal solution sequence Nastran/Patran (SOL112), see figure 6. This analysis showed a reasonable approach to study the buffeting at the Vertical Tail. For further validation of the simplified transient approach a full coupled calculation will be done.

3 Unsteady Aero-Elastic Coupling Development

To take into account the full structural response due to dynamic aero loads an unsteady aero-elastic simulation tool is required. RUAG started the development of such a tool in early 2005. Essential elements of this tool are:

- a CFD solver using the ALE formulation and which includes a re-meshing algorithm to regenerate the CFD volume mesh after the movement of the surface;
- a geometrical coupling tool which permits transfer of the aerodynamic loads from the CFD mesh to the CSM mesh, and transfers the structural

displacement into the CFD surface geometry displacement;

• a CSM solver to compute the structural state. To reduce computational costs a linear structural model is used, which is further simplified by using a modal formulation. The time integration of the structural equations is made using the Newmark method.

The CFD and CSM solvers are coupled through the geometric coupling tool (the so called segregated or partitioned approach). Within this approach different coupling schemes can be formulated and in a first step the so called Conventional Staggered Scheme (CSS) [1] was implemented. When using the dual time stepping approach for the CFD solver, no coupling between CSM and CFD solver takes place inside the so called inner-loop. This may be sufficient for small deformations of the surface, but for larger deformations a stronger coupling approach may be needed [2].

A new version of the FSI/MI (Fluid Structure Interaction/Modal Integration) library became available in 2006 which permits to couple CFD and CSM in the inner loop of the dual time stepping procedure. Although the computational costs are higher compared to the old implementation, the new implementation permits to use larger outer time steps without the loss of accuracy.

To validate the unsteady aero-elastic simulation tool calculations were made for the AGARD 445.6 wing [3]. The AGARD445.6 wing, made of mahogany, has a 45 deg quarter chord sweep, a half span of 2.5 ft, a root chord of 1.833 ft and a constant NACA64A004 symmetric profile.

Flutter tests were carried out at the NASA Langley Transonic Dynamics Tunnel, were published in 1963 and re-published in 1987. Various wing models were tested (and broken) in air and Freon-12 for Mach numbers between 0.338 and 1.141. The case most often used in the literature is the so called weakened model 3 at zero angle of attack in air. The model was weakened by holes drilled through the surface of the original model to reduce its stiffness.

The linear structural model was build by SMR, with the material properties taken from [4]:

E1	3.15106 106	Pa
E2	4.16218 108	Ра
G	4.39218 108	Ра
ρ	381.98	kg/m3
ν	0.31	

Only the first four mode shapes are considered, consisting of two bending and two torsion modes. CFD calculations were made for the following free stream conditions:

Mach	0.95	
ρ∞	0.061	kg/m3
AoA	0	deg
Re	1.196 106	1/m
μ	234.93	

using free stream pressures of respectively 3500, 4600 and 7000 Pa. Experimental data showed that flutter occurs when the value of the flutter speed coefficient (or flutter index) is around Vf = 0.32 (Vf = U ∞ / (bs $\omega a \sqrt{\mu}$) with U ∞ the free stream velocity, bs the half span, ωa the frequency of the first torsional mode and μ the mass ratio. Flutter should occur for the highest free stream pressure, which has a flutter index of 0.383.

The coupled aero-elastic calculations were started from the steady CFD result for the same conditions. Then a 2.5% deflection of the first bending moment was given to the structure and the unsteady simulation was started. A structural damping of 2% was used in the CSM calculation. In figures 7 and 8 the contour plot of the Mach number is represented for two extreme wing deflections. In figure 9 one can observe that for the case of 7000 Pa the deflections are amplified while for the other 2 cases (3500 Pa & 4500 Pa) they are damped. This is in agreement with the experimental data and the flutter index.

4 Unsteady Aero-Elastic Coupling Development for F/A-18 Vertical Tail

4.1 Re-Meshing Algorithm

The NSMB flow solver includes a remeshing procedure since 2004, which was improved in 2005 and 2006. The initial remeshing procedure was based on the Volume

Spline Interpolation method. In this method the new coordinates are computed using the surface displacements at a number of so called prescribed points. The main short coming of this algorithm is the large computational requirements (CPU time and memory), in particular for complex geometries such as the F/A-18. For this reason other methods were implemented and tested, as for example the Trans Finite Interpolation (TFI) method. In this method first the mesh on the block edges is generated followed by the generation of the mesh on the block faces. In the final step the volume mesh is generated from the mesh on the block faces. Bunching laws are all taken from the original grid un-deformed grid. Although this method is straightforward to implement and computationally fast it led to problems in regions where the geometry is not moving due to the fact that the implemented TFI method was not exactly the same as the TFI method from the mesh generator. This resulted in problems when using the coupled CFD-CSM method for the F/A-18. For example the calculations showed a sudden increase in pressure was observed in the gap between and Trailing Edge Flap, which Aileron propagated down stream until it arrived at the high Vertical Tail where the pressure completely deformed the structure leading to a crash of the calculation.

The breakthrough was to use the TFI method on the displacement of the edges. The advantage of this method is that if the displacement of the edges is close to zero, the displacement in the volume will be close to zero as well, and as a result the original coordinates are unchanged. The second advantage of this procedure is that it is independent of the method used to generate the original coordinates.

The re-meshing method can be summarized as:

- compute the displacement of block edges using Volume Spline Interpolation
- use 2D Trans Finite Interpolation to generate the displacement of the coordinates on the block faces
- use 3D Trans Finite Interpolation to generate the displacement of the coordinates in the volume

• sum the coordinates and displacements to obtain the new mesh

The implementation of this algorithm in NSMB solver was straightforward, and removed the problems encountered with the coupled CFD-CSM calculations for the F/A-18 fighter.

Today, running in parallel, it takes between 10 to 25 CPU seconds to generate a new mesh for the F/A-18 geometry, and this mesh has sufficient quality to continue the calculation.

4.2. Unsteady Coupling Calculation Strategies

The unsteady coupled CFD-CSM calculations all employed the same numerical approach, used before when performing unsteady calculations without the movement of the Vertical Tail.

The time integration was made using the dual time stepping method of Jameson in which the outer loop is used for the advancement in time, and an inner time-stepping loop to solve the equations at each time step. The inner timestepping loop employed the same numerical method as far the steady state calculations. The Detached Eddy Simulation (DES) approach was used to model the unsteady turbulence.

Three strategies for the CFD-CSM coupling procedure were investigated:

- **1 MI**: the coupling is made only each outer time step
- ALL MI: the coupling is made each inner time step
- **STD MI**: the coupling is made the inner time steps which yields an integer when taking the square root (inner time steps: 1,4,9,25, 36, etc.)

No differences could be observed for the AGARD445.6 wing when comparing the STD and ALL MI results indicating that for this calculation is not need to perform the CFD-CSM coupling each inner time step. However, performing the coupling only each outer time step appeared to be insufficient.

The structural model used for the F/A-18 Vertical Tail was the same as prepared for transient analysis. The initial version of the structural modal included 100 mode shapes, of which only the first 5 mode shapes ("Eigenmodes") were used for these calculations

4.3. Results for two Load Cases

4.3.1 Transonic Load Case (M=0.9)

The load case has the following parameters:

Mach	Alt	AoA	LEF	TEF	HSTAB
	feet	deg	deg	deg	deg
0.9	5'000	0.0	0.0	0.0	0.0

This load case is not in the buffeting environment and so only a small well damped displacement will be expected.

The coupled CFD-CSM calculation was about twice as expensive as the uncoupled calculation. Time spend in the re-meshing algorithm was between 4 and 30 seconds.

The run was interrupted after 0.3 seconds real time, which took 28 days to compute and generated close to 700 Gbytes of data.

To analyze the movement of the Vertical Tail, 5 points were selected which are shown on figure 10. Points 1 and 2 are on the top of the Vertical Tail, points 3 and 4 near the gap between Rudder and Vertical Tail, the other points are near the trailing edge.

Most dominant is the movement in z-direction (outboard/inboard), there the initial perturbation is clearly visible when starting the coupling calculation. This initial perturbation is followed by an oscillatory movement of the Vertical Tail, which is damped out in time. It can be observed that the asymptotic position to which the Vertical Tail seems to move is not the unperturbed position, but one which is slightly displaced.

The unsteady simulation without CSM coupling for this load case showed a steady flow on the Vertical Tail, which is confirmed by the fully coupled calculation. The initial perturbation at the start of the coupled simulation is damped out in time, indicating that there is no mechanism which maintains an unsteady flow behavior.

4.3.2 Buffeting Load Case C2S825

The load case C2S825 is indeed affected by severe buffeting based on the flight test results performed by The Boeing Company for the assessment of dynamic impact on the Vertical Tail. The load case is characterized by following parameters:

Mach	Alt	AoA	LEF	TEF	HSTAB
	feet	deg	deg	deg	deg
0.7	15'000	26.6	26.0	0.0	-6.1

Due to time constraints only initial results for this calculation are available. As for the transonic load case 5 points were selected to show the movement of the Vertical Tail. Figures 11 and 12 show the movements of these 5 points in time in y- and z-direction respectively. The calculation is clearly in the initial phase, since the amplitudes of the movement are still increasing. Compared to the transonic load case the movement of the Vertical Tail is much more important, see figures 13, 14, and 15 for small time steps. One also observes that points 3 and 4, which are near the gap between Rudder and Vertical Tail, move in opposite directions.

5 Conclusions

Component loads for structural and fatigue analysis were calculated using the RUAG inhouse CFD solver NSMB. The calculated loads were in good agreement with the flight loads data.

The interaction of the aerodynamic pressure over wing with the structural stiffness is important and must be considered for the loads calculation using fluid structure interaction.

With today's computer performance an unsteady state CFD calculation brings more information into buffeting and flutter behavior of modern airplanes. With the NSMB unsteady capabilities real flow field for 0.5 seconds over the F/A-18 were processed. A simple transient analysis using Nastran/Patran SOL 112 solution sequence can provide first information on the aero-elastic behavior. For full information an unsteady fluid structure coupled analysis is necessary.

The buffeting impact for the F/A-18 Vertical Tail was simulated using unsteady aero-elastic coupling algorithm. This method uses implicit conventional serial staggered coupling in the inner loop. An efficient re-meshing procedure within the flow solver is important for complex geometries such as a F/A-18 fighter. The CPU time and memory is still very high but can be managed with today's computer capacity but further improvements are needed.

The tool chain for CFD-CSM unsteady coupling was validated by the experimental data for AGARD445.6 wing from the literature.

First successful calculations for buffeting load cases were done for the F/A-18 Vertical Tail. These results will be used to develop a fatigue spectrum for the Swiss F/A-18 usage under buffeting environment condition to assess the structural integrity.

The buffeting and flutter should be addressed in an early design phase of a modern airplane. RUAG Aerospace CFD dynamic fluid structure coupling tool may provide an early answer to these problems.

References

- [1] C. Farhat, M. Lesoinne, N. Maman. Mixed explicit/implicit time integration of coupled aeroelastic problems: three-field formulation, geometric conservation and distributed solution. International Journal of Numerical Methods in Fluid, Vol. 21, 1995, no. 10, pp.807-835.
- [2] J. Smith, Aeroelastic Functionality in Edge Initial Implementation and Validation, FOI Report R-1485-SE, 2005.
- [3] E.C. Yates Jr. AGARD Standard Aero-elastic Configurations for Dynamic Response 1: Wing 445.6 AGARD R-765, 1988, NASA TM-100492, 1987.
- [4] G.S. L. Goura, Time marching analysis of flutter using Computational Fluid Dynamics. Ph.D thesis, University of Glasgow, 2001.

Copyright Statement

The authors confirm that they, and/or their company or institution, hold copyright on all of the original material included in their paper. They also confirm they have obtained permission, from the copyright holder of any third party material included in their paper, to publish it as part of their paper. The authors grant full permission for the publication and distribution of their paper as part of the ICAS2008 proceedings or as individual off-prints from the proceedings.



Figure 1: Buffeting at Vertical Tail due to LEX



Figure 2: Details of grid at Center Fuselage



Figure 3: Details of grid at Outer / Inner Wing



Figure 4: Unsteady calculation at two time steps



Figure 5: 54 Trapezoidal zones on Vertical Tail



Figure 6. Model for transient analysis (SOL112)



Figure 7: Down bending for AGARD445.6 wing (Mach data along elastic plane)



Figure 8: Up bending for AGARD 445.6 wing (Mach data along elastic plane)



4600 Pa, 7000 Pa)



Figure 11: Y (out/inside direction) displacement for the 5 points at Vertical Tail



5 points at Vertical Tail



Figure 13: Pressure at 0.004 seconds on Vertical Tail



Figure 15: Pressure at 0.041 seconds on Vertical Tail



Figure 14: Pressure at 0.03 seconds on Vertical Tail