F/A-18 vertical tail buffeting calculation using unsteady fluid structure interaction

M. Guillaume, A. Gehri and P. Stephani
michel.guillaume@ruag.com
RUAG Aviation/Aerodynamics
Emmen, Switzerland

J. B. Vos
CFS Engineering
Lausanne, Switzerland

G. Mandanis
M@M GmbH
Kriens, Switzerland

ABSTRACT

The Swiss Airforce is operating F/A-18C/D Aircraft since 1997. Since the aircraft's structural design is different from the version operated by the US Navy it was necessary to carry out a structural integrity study (ASIP) which was done by The Boeing Company in St. Louis. To validate this study a full scale fatigue test facility was build at RUAG and operated from 2003 to 2005. When operating this facility difficulties were encountered with the aerodynamic loads data provided by Boeing (insufficient, not well documented, questionable data). As a result RUAG looked for alternative methods to provide the aerodynamic loads, and a large investment was made in the development of a Computational Fluid Dynamics (CFD) tool. The Navier Stokes Multi Block (NSMB) solver, which was developed in an international collaboration, was adopted. In a first phase the code was validated by comparing results of CFD calculations with wind-tunnel results, results from literature and flight test data results. In the second phase, discussed in this paper, a Fluid Structure Interaction (FSI) tool was developed to permit unsteady aero-elastic simulations. Particular attention is focused on the vertical tail since this component of the F/A-18 fighter is very sensitive to fatigue due to unsteady loads generated by buffeting phenomena.

NOMENCLATURE

ALE	arbitrary lagrange Eulerian
AoA	angle-of-attack
ASIP	aircraft-structural-integrity-program
BM	bending moment
CAD	computer aided design
CFD	computational fluid dynamics
FEA	finite element analysis
FFT	fast fourier transform
FSI	fluid structure interaction
FZ	force in global *z* direction
HT	horizontal tail
LE	leading edge
LEX	leading-edge extension
LBL	line by line
MI	modal integration
NSMB	Navier Stokes multi block
MES	master event spectrum
MPI	message passing interface
OEM	original equipment manufacturer
PC	personal computer
PSD	power spectral density
Q	dynamic pressure
RMS	root mean square
SFH	spectrum flight hours
SH	shear force
SL	sea level
SPMD	single program multiple data
TE	trailing edge
TEF	trailing-edge flap
TFI	trans-finite interpolation
TQ	torque or torsion moment
VSI	volume-spline interpolation
W/O	without

Paper No. 3582. Manuscript received 13 May 2010, revised version received 16 November 2010, 2nd revised version received 3 March 2011, accepted 13 March 2011.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

1.0 INTRODUCTION

For the Swiss F/A-18 aircraft, The Boeing Company performed an aircraft structural integrity study (ASIP) to analyse the structural integrity of the entire airframe based on the Swiss design spectrum. The Swiss manoeuvre spectrum was three times more severe than the US Navy design spectrum, but the dynamic spectrum was the same as for the US Navy. For the validation of the Swiss structural reinforcement a full scale fatigue test program was carried out at RUAG Aviation.

Only few relevant fatigue load cases for the entire aeroplane could be obtained from Boeing in St Louis, the F/A-18 Original Equipment Manufacturer (OEM). This forced RUAG Aviation to look for methods to generate independently aerodynamic loads for the F/A-18. As a result a considerable investment was made in the 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 flight envelope and to check load cases delivered by the OEM. The FSI tool follows the concepts developed by Farhat *et al*[1] 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.

The unsteady aero-elastic coupling tool was validated using the data from the well known AGARD 445.6 wing[2]. Three unsteady load cases were calculated and post processed in the frequency domain to compare with existing experimental test data. The results were in excellent agreement with published AGARD wing data.

For the F/A-18 vertical tail unsteady aero-elastic loads were calculated in order to obtain the buffeting load spectrum. This data will be used to determine the Swiss dynamic assessment of the F/A-18 vertical tail. The fatigue life reduction could be demonstrated.

The use of novel unsteady aero-elastic simulation methods will improve the design of modern aircraft since buffeting and flutter problems can be detected in the early design phase.

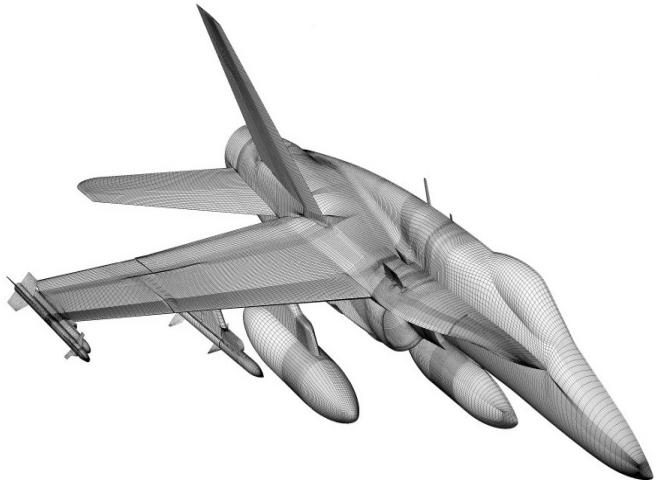


Figure 1. Surface mesh of the F/A-18 fighter.

2.0 COMPUTATIONAL FLUID DYNAMICS USING THE NSMB CODE

2.1 The NSMB flow solver

The calculations of the F/A-18 flow field were carried out 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[3,4]. Since 2004 NSMB is further developed in a new consortium lead by CFS Engineering and composed of RUAG Aviation (Emmen), EPFL (Lausanne), EHTZ (Zürich), IMFT (Toulouse), 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 discretise the Navier Stokes equations. Various space discretisation schemes are available to approximate the inviscid fluxes.

The space discretisation 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)
- pre-conditioning for low Mach number
- artificial compressibility for incompressible flows

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-ω
- Scale adaptive simulation (Menter SAS)

The ALE approach is available to simulate the flow on moving grids. Recently, a re-meshing 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 parallelised.

2.2 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 has to be set-up, and finally the mesh is generated. The latest mesh used in the F/A-18 fighter CFD simulations was generated by Mindware in collaboration with the RUAG Aviation Center of Aerodynamics. This mesh has 2,802 blocks and 13·9 million cells for one half of the aircraft. Figure 1 shows the surface mesh of the F/A-18 fighter, and as can be seen many geometrical details as for example antennas and the wing tip missile fins are modeled, explaining the large size of the mesh, see Fig. 1.

In all calculations discussed here it is assumed that the aircraft is perfectly symmetrical and only symmetrical load conditions were considered until now. Consequently only one half of the aircraft is used in the calculations. The current mesh is made using the replay feature of ANSYS ICEM CFD which permits to generate in a few seconds new mesh with, for example, different flap deflection angles.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

Comparison of flight-test load data with NSMB CFD calculations were presented at the ICAS conference in 2006 in Hamburg, see Ref. 5. The results match very good and support the use of CFD load calculations.

3.0 DEVELOPMENT OF THE FLUID-STRUCTURE INTERACTION (FSI) TOOL

3.1 Fluid structure coupling interface

RUAG started the development of an unsteady aero-elastic simulation tool in early 2005. Essential elements of this tool are:

- a CFD solver using the ALE formulation which includes a remeshing algorithm to regenerate the CFD volume mesh after the movement of a surface;
- a geometrical coupling tool which permits to transfer the aerodynamic loads from the CFD mesh to the FE mesh, and to transfer the structural displacement into a CFD surface geometry displacement;
- a FEA 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 FEA solvers are coupled through the geometric coupling tool (employing 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 FSI 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[6].

A new version of the FSI/MI (Fluid Structure Interaction/Modal Integration) became available in 2006 which permits to couple CFD and structures 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.

3.2 Re-meshing algorithm

The NSMB flow solver includes a re-meshing procedure since 2004, which was improved in 2005 and 2006. The objective of the remeshing procedure is to generate a new grid inside the NSMB solver using the old coordinates as input together with the displacements of parts of the mesh (for example the deformation of the wing surface due to the aerodynamic loads). The initial re-meshing procedure is based on the Volume Spline Interpolation (VSI) method[7] and is carried out in two steps. In the first step information is collected on coordinates and their displacements, and a list of so called prescribed points is generated. A matrix of linear equations needs to be solved to compute the volume spline coefficients, which is done using the LAPACK-library[13]. For the second step three methods are available:

1. generate the new volume mesh using the volume spline interpolation method
2. generate the mesh on the block faces using the volume spline interpolation method and use 3D Trans Finite Interpolation (TFI) to generate the new volume mesh
3. generate the mesh on the block edges using the volume spline interpolation method, use 2D Trans Finite Interpolation to generate the mesh on block faces and use 3D (TFI) to generate the volume mesh

For all methods one needs to ensure that the sum of the original coordinates and displacements equal the new coordinates. The main shortcoming of method one is the large computational requirements (CPU time and memory), in particular for complex geometries such as the F/A-18. As mentioned before, at the start of the re-meshing procedure a list of so called prescribed points is generated. If n is the number of prescribed points, than the memory requirements for the VSI coefficient calculation is proportional to $½n^2$, and CPU time requirements follow the same trend, see Table 1.

Table 1
Memory and CPU time requirements VSI coefficient calculation F/A-18 fighter on a single processor PC

n	Memory (GB)	CPU (sec)
26,714	2·80	18,589
8,658	0·30	1,806
1,894	0·06	163

The simplest method to reduce memory and CPU time requirements is to reduce the number of prescribed points in the VSI coefficient calculation, and this can be done by using methods two and three. When using the VSI method to generate the volume mesh one needs to ensure that the displacements of the surface are correctly transferred into the new volume mesh, and as a result many prescribed points are needed on the moving surface. When using VSI to generate the block faces (method two), one only needs to select the block edges of the moving surface. The conservation of geometry is ensured automatically by setting the new coordinates of the moving block face as the sum of the un-deformed coordinates and their displacement. When using VSI to generate the block edges (method three), one only takes the vertices on the moving surface as prescribed points. Although methods two and three look simple in practice several difficulties were encountered. The first problem is typical of multi block meshes, and is due to the fact that several blocks share the same edge. In this case one needs to ensure that all blocks which share an edge receive a moving edge before generating the mesh on the block face. The second problem is related to the Trans Finite Interpolation (TFI) method. Many variants of TFI exist, and when the TFI method for the re-meshing algorithm is not exactly the same as the TFI method used to generate the original mesh large movements of the mesh may be observed in regions where nothing should happen. Since the TFI method of the original mesh is a priory unknown, this resulted in large problems.

The breakthrough is to use the TFI method (in 2D and 3D) on the displacements instead of the coordinates. The advantage of this method is that if the displacement of the edges is close to zero, the displacement of the coordinates 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 used for the F/A-18 studies can now be summarised as:

- compute the displacements of block edges using volume spline interpolation
- use 2D trans finite interpolation to generate the displacements of the coordinates on the block faces
- use 3D trans finite interpolation to generate the displacements of the coordinates in the volume

Today, it takes about ten CPU seconds (running on ten processors) to generate a moved mesh for the F/A-18 fighter having about 14 million grid points, and this mesh has almost the same quality as the original mesh.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

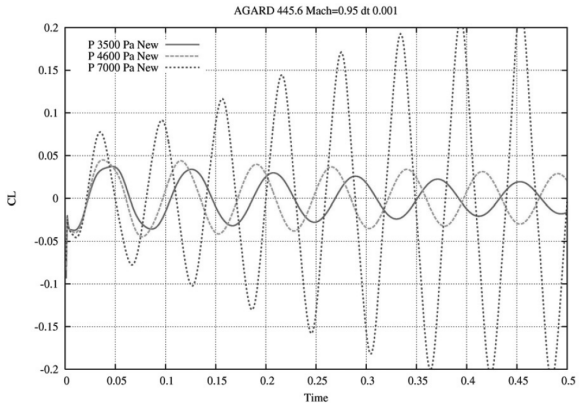


Figure 2. Lift coefficient as function of time for three cases (3,500Pa, 4,600Pa, 7,000Pa).

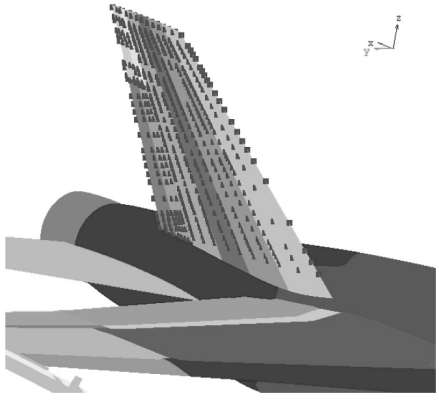


Figure 3. CSM grid on the vertical tail of the F/A-18 fighter.

3.3 Validation

To validate the unsteady aero-elastic simulation tool calculations are carried out for the AGARD 445·6 wing[2]. The AGARD 445·6 wing, made of mahogany, has a 45 deg quarter chord sweep, a half span of 2·5ft, a root chord of 1·833ft 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 mode three 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 FE model is built by SMR in Bienne (Switzerland), with the material properties taken from Ref. 8, see Table 2:

Table 2
Material properties AGARD 445·6 Wing

E1	$3{\cdot}15106 \bullet 10^6$	Pa
E2	$4{\cdot}16218 \bullet 10^8$	Pa
G	$4{\cdot}39218 \bullet 10^8$	Pa
ρ	381·98	kg/m^3
ν	0·31	

Only the first four mode shapes are considered, consisting of two bending and two torsion modes. CFD calculations are made using the free stream conditions given in Table 3 and for free stream pressures of respectively 3,500, 4,600 and 7,000 Pa.

Table 3
Free stream conditions AGARD 445.6 Wing

Mach	0·95	
ρ_∞	0·061	kg/m^3
P	3,500/4,600/7,000	Pa
AoA	0	deg
Re	$1{\cdot}19^* 10^6$	1/m
μ	$1{\cdot}952\ 10^{-8}$	

Experimental data showed that flutter occurs when the value of the flutter speed coefficient (or flutter index) is around $Vf = 0{\cdot}32$ ($Vf = U_\infty / (b_s\ \omega_a\ \sqrt{\mu})$ with U_∞ the free stream velocity, b_s the half span, ω_a 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 are started from the steady state CFD result for the same conditions. Then an initial 2·5% deflection of the first mode (bending) is given to the structure and the unsteady simulation is started. Figure 2 shows the lift coefficient as function of time for the three calculations. One can observe that for the case of 7,000Pa the initial deflection is amplified while for the other two cases (3,500Pa and 4,600Pa) it is damped. This is in agreement with the experimental data and the value of the flutter index.

4.0 ANALYSIS OF UNSTEADY CALCULATIONS F/A-18 FIGHTER

4.1 Simulation parameters

Unsteady simulations are made for the F/A-18 fighter using different aircraft configurations and free stream conditions. All calculations are carried out using the dual time stepping method, in which the outer loop is used for the advancement in time, while in the inner loop a pseudo time stepping method is used to solve the equations at each outer time step. The Detached Eddy Simulation (DES) variant of the *k*-ω Menter Shear Stress turbulence model is used to model the unsteady turbulence. All unsteady calculations are started from a converged steady state calculation for the same free stream conditions. An outer time step of $2{\cdot}5\ 10^{-4}$ or $5\ 10^{-4}$ seconds is used, and the pressure and skin friction vector were saved each outer time step. For the aero-elastic calculations the fluid structure coupling is executed each inner time step corresponding to a square number (inner step 1, 4, 9, 16,..). Initial tests show that this is sufficient and computationally less expensive compared to making the coupling each inner time step.

The structural model of the vertical tail uses for the aero-elastic calculations was prepared by RUAG Aviation based on the dynamic data provided by Boeing, for the details of the structural model see Ref. 9.

For the vertical tail only the first four or five mode shapes are important (bending and torsion moments), and it was decided to use only the first five modes in the coupled calculations.

Three different F/A-18 conditions/load cases are computed, a transonic case, the Swiss load case C2S825, and a subsonic case for which experimental data were reported in the literature. For all configurations calculations are carried out with and without FSI.

4.2 Transonic load case (Mach = 0·9)

The definition of this configuration and the free stream conditions are summarised in Table 4.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

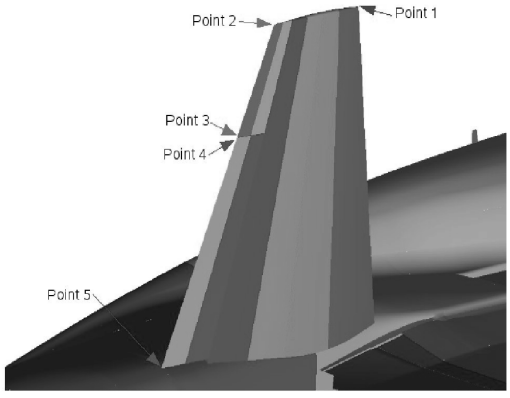


Figure 4. Selected points at the vertical tail.

Table 4
Specification of the transonic load case

Mach	Alt (feet)	AoA (deg)	LEF (deg)	TEF (deg)	HSTAB (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 excitation will be expected.

The coupled CFD-FSI calculation is computationally twice as expensive as the uncoupled calculation. Time spent for the remeshing algorithm is between 4 and 30 seconds for each re-meshing.

To analyse the movement of the vertical tail, five points are selected. Points one and two are on the top of the vertical tail, points three and four near the gap between Rudder and vertical tail and the last point is near the trailing edge, for details see Fig. 4.

Figure 5 shows the movement in *Y*-direction (outboard/inboard) of the five points. Most dominant is the movement on the top of the vertical tail (points one and two), where the initial perturbation is clearly visible when starting the coupled calculation. This initial perturbation is followed by an oscillatory movement of the vertical tail, which is damped out in time indicating that there is no mechanism which maintains an unsteady flow behaviour.

The unsteady simulation without FSI coupling for this load case shows a steady flow on the vertical tail, which is confirmed by the fully coupled calculation.

4.3 Results Boeing load case C2S825

The Boeing load case C2S825 (Swiss condition) is affected by severe buffeting at the vertical tail. This case is characterised by the following parameters, see Table 5:

Table 5
Specification of the Boeing load case C2S825

Mach	Alt (feet)	AoA (deg)	LEF (deg)	TEF (deg)	HSTAB (deg)
0·7	15,000	26·6	26·0	0·0	–6·1

This load case corresponds to an 8·5g steady state pull up manoeuvre at Mach = 0·7. The calculation without FSI coupling shows significant differences in the averaged unsteady aerodynamic loads compared to the loads obtained using a steady calculation, in particular on aft fuselage, vertical tail, rudder, trailing edge flap, aileron and horizontal stabiliser. This is a clear indication that unsteady flow effects are important on these components of the aircraft.

An initial coupled simulation is performed for this load case running up to 0·5 seconds real time. As for the transonic load case five points are selected to show the movement of the vertical tail, see Fig. 6. Compared to the transonic load case the movement of the

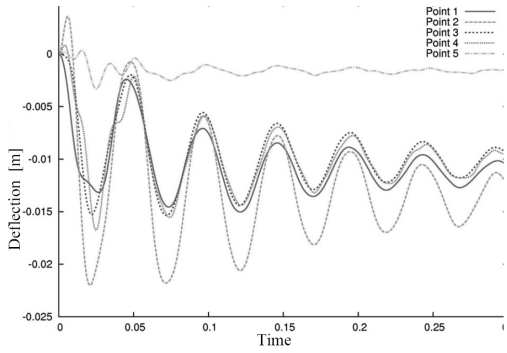


Figure 5. Movement of the vertical tail at five selected points, transonic case.

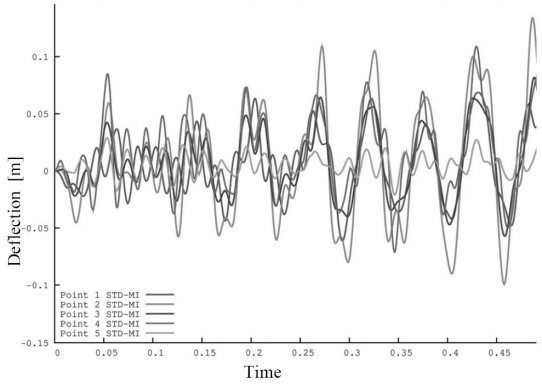


Figure 6. Movement of the vertical tail in five selected points, Boeing load case C2S825.

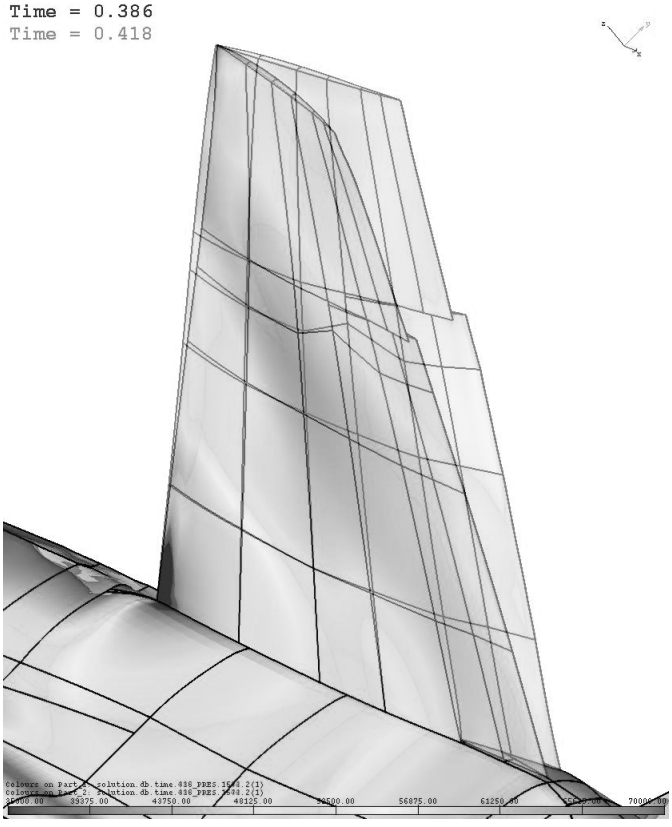


Figure 7. Vertical tail displacement at two time steps of 0·386 and 0·418 second.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

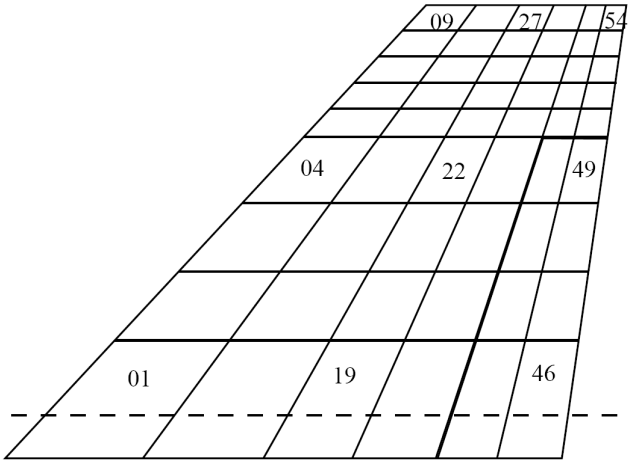


Figure 8. 54 panels on the vertical tail for CFD-FSI-calculation with labelling of panels.

vertical tail is much more important, up to 13cm in *Y*-direction compared to a little more than 2cm. One also observes that points three and four, which are near the gap between rudder and vertical tail, move in opposite directions. The results for this case also show many small oscillations.

To compare data transformed from the time domain to the frequency domain the simulation of 2·5 seconds is not enough. Therefore a second coupled calculation was made running up to 4 seconds real time. This calculation ran on ten PC processors and took about 30 days to complete.

In order to quantitatively take profit of the results of the CFD-FSIcalculation on the F/A-18 the vertical tail surface is divided into 54 panels and the pressure and the motion at each panel centroïd is extracted (see Fig. 8).

The PSD has been calculated for the pressure of nine panels using the FFT-algorithm after conditioning the signal with a Hanningwindow. The results show that no particular frequency can be distinguished (see Fig. 9). Calculating for the same panels the autocorrelation function the random character of the time signal has been definitively recognised (see Fig. 10), nevertheless some frequencies are emerging above the noise at about 40Hz in the panels 04 and 22 (Fig. 8), which are situated at 60% of the span.

The pressure distribution over the 54 panels as well as their displacement in time gives us the ability to determine the dynamic loading of the connection between the vertical tail and the fuselage(1).

$$\vec{M}(t) = (BM(t), PM(t), TQ(t)) = \sum_{1}^{54} \vec{r}_i x (A_i \vec{p}_i(t) - m_i \vec{a}_i(t)) \quad \ldots (1)$$

With *BM*, *PM* and *TQ* the bending, pitch and torsion moment at the root of the vertical tail, A_i the panel surface, p_i panel pressure, m_i lumped mass and a_i the acceleration. The most interesting aspect for RUAG Aviation is the fatigue damage caused by the vertical tail. In order to check the plausibility and the feasibility of a fatigue calculation based on the F/A-18 CFD-FSI, the peak-valley sequence in a critical location of the stub 557·5, which is the forward connection beam vertical tail to fuselage has been calculated.

The strain survey measurements accomplished during the Swiss F/A-18 full scale fatigue test delivered us the coefficient A and B of the next formula, and thus the relation between the strain at gauge position SF6001 (see Fig. 11) and the moments *BM* and *TQ*.

$$\varepsilon_c(t) = A \cdot BM(t) + B \cdot TQ(t)$$

$$\sigma_c(t) = E \cdot \varepsilon_c(t)$$

E: elastic modulus

A part of the $\sigma_c(t)$-function (2) for the buffeting peak valley sampling is shown in Fig. 12. To evaluate the buffeting effect on the global fatigue damage this peak-valley sequence which corresponds to a buffeting period of 2·5 seconds has been inserted in the Swiss design test spectrum called MES used for the full scale fatigue test, considering that buffeting occurs when the AoA exceeds a limit value (of 20°, 22°, 24°). Such a mixed spectrum of maneuver and buffeting loads with onset of 20° is presented in Fig. 13. Finally a crack initiation calculation based on Neuber Notch approach with strain life data[12] has been calculated with different mixed spectra of maneuver and buffeting cycles, and the results are shown in the Fig. 14. The buffeting, depending at which AoA it appears (20°, 22°, 24°), can reduce the fatigue life of the vertical tail with a factor in the order of 9 to 50 depending on the stress level, see for example the influence at 60Ksi. This study clearly demonstrates the sensitivity of buffeting cycles for fatigue evaluation.

4.4 Results of a low speed load case

L. A. Meyn published in 1996 results of buffeting measurements on a full scale F/A-18 in the wind tunnel at NASA Ames Research Center[10]. The test conditions were Mach = 0·15, *SL* at different angles-of-attack.

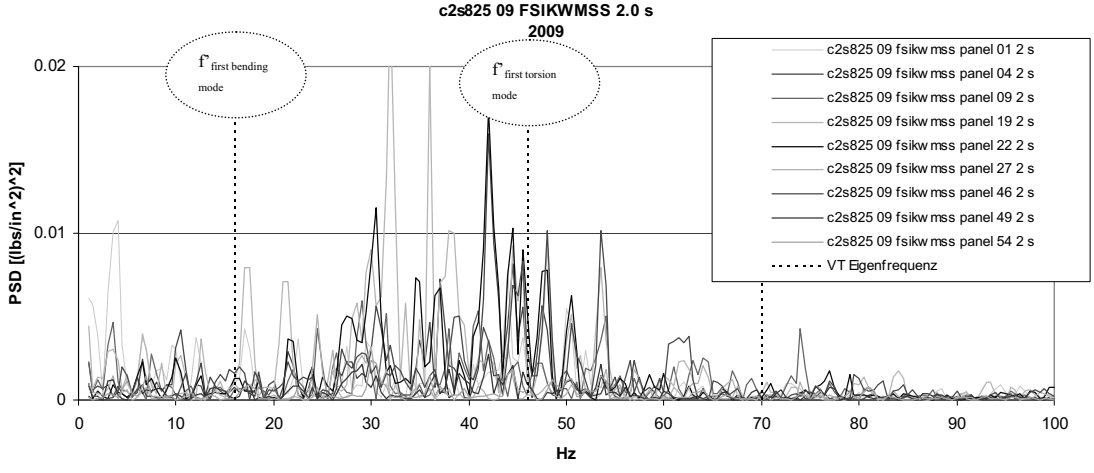


Figure 9. Power Spectral Distribution (PSD) function of nine different panels.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

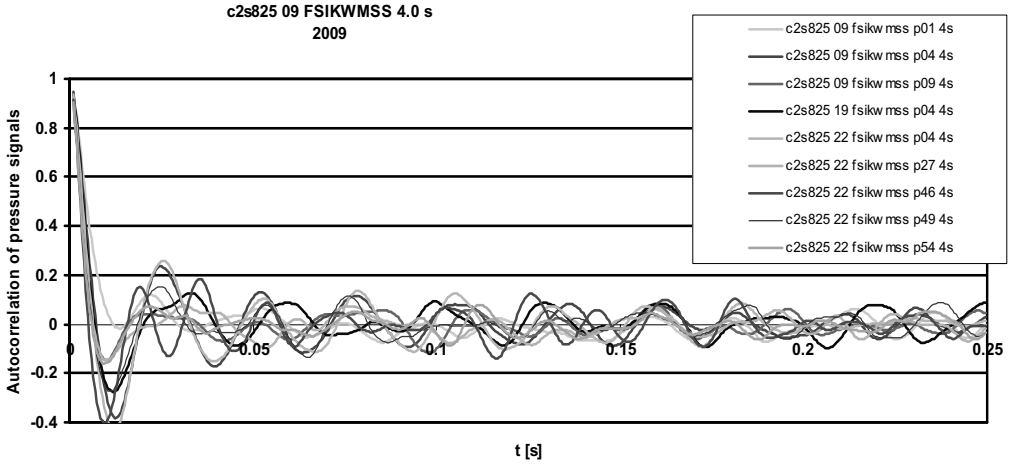


Figure 10. Autocorrelation function of nine different panels.

Calculations with and without FSI coupling were made for a simulation time of 0·5 seconds. The PSD has been calculated for panels of the LE, of the TE and of the vertical tail torsion box and represented, like in Ref. 5, over the reduced frequency defined as follows:

$$f' = fL/V$$

with f the signal frequency in Hz, L (11·52ft) the mean aerodynamic chord and V (168ft/s) the flight velocity. The so called pressure power coefficient is the dimensionless value of the PSD defined as fellows:

$$\text{Pressure Power Coefficient} = \frac{l}{VQ^2} PSD \qquad \ldots (4)$$

These results are displayed in the Figs 15 and 16. They show the same trend as in Ref. 10. In our case the calculation with FSI is similar to the one without FSI, nevertheless the pressure-power-peak at $f' = 0{\cdot}3$ seems to be less pronounced and wider with FSI coupling.

Other interesting results can be found in Ref. 11, where RMSpressure Coefficient of CFD-computations, wind-tunnel test and flight tests are published. This dimensionless number is defined as follows:

$$\text{RMS Pressure Coefficient} = \frac{1}{Q}\sqrt{\frac{1}{N}\sum_{1}^{N}(p_i - \bar{p})^2} \qquad \ldots (5)$$

With Q the dynamic pressure, p the pressure and N the number of considered values. This coefficient has been established for the 45% chord and 60% span area of the vertical tail conforming to Ref. 11. Figure 17 shows that the results of our CFD calculations with and without FSI are pretty well in the domain of the reported results.

5.0 CONCLUSION

The paper presented the development of an unsteady Fluid Structure Interaction (FSI) CFD tool to permit the computation of the deformation of an aircraft due to unsteady aerodynamic loads. An efficient re-meshing procedure was developed an implemented in the NSMB CFD solver which permits a fast regeneration of the mesh after a movement of the aircraft surface. For a calculation running on ten processors it takes about ten CPU seconds to generate a new mesh for the F/A-18 fighter having about 14 million grid points.

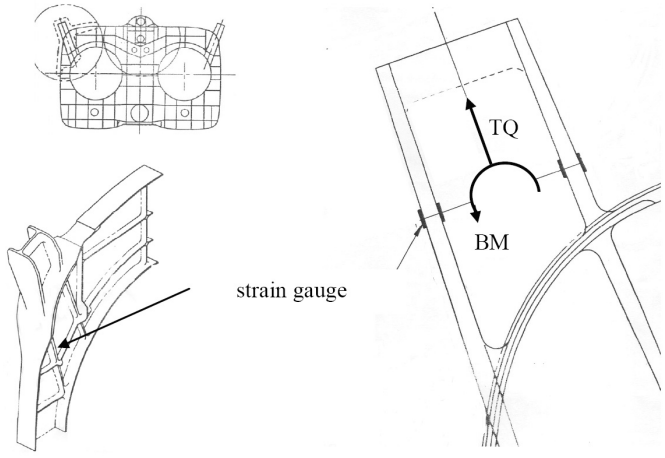


Figure 11. First stub FS 557.5 at F/A-18 vertical tail root.

This tool was validated using the experimental data for the AGARD 445.6 wing[2]. Flutter was observed for the highest free stream pressure, which is in agreement with results reported in the literature. Specific post processing tools were developed for the F/A-18 fighter to analyse buffeting on the vertical tail. Calculations were made for the conditions of full scale wind-tunnel measurements[11], and a good agreement with measurements of the computed RMS pressure coefficient for 45% chord and 60% span of the vertical tail was observed.

The effect of buffeting on the global fatigue damage was assessed using tools used for the full scale test facility, and show that, depending on the angle of attack above which buffeting occurs (20°, 22° or 24°), the fatigue life of the vertical tail is reduced by a factor between 9 and 50.

Future usage of the CFD-FSI tool is the computation of buffeting loads on the F/A-18 vertical tail for stress and fatigue analysis for the Swiss usage of this fighter. Further development of the simulation technology will help during the design phase to improve detailed understanding of the flow field around the airframe and get much earlier information on the aero-elasticity and buffeting issues as seen on the F/A-18 vertical tail.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

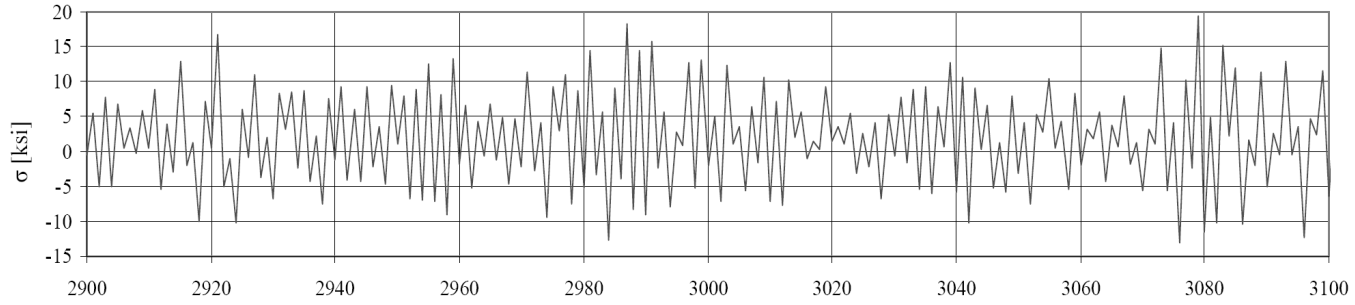


Figure 12. A sample of the $\sigma_c(t)$-function for the buffeting peak valley sampling.

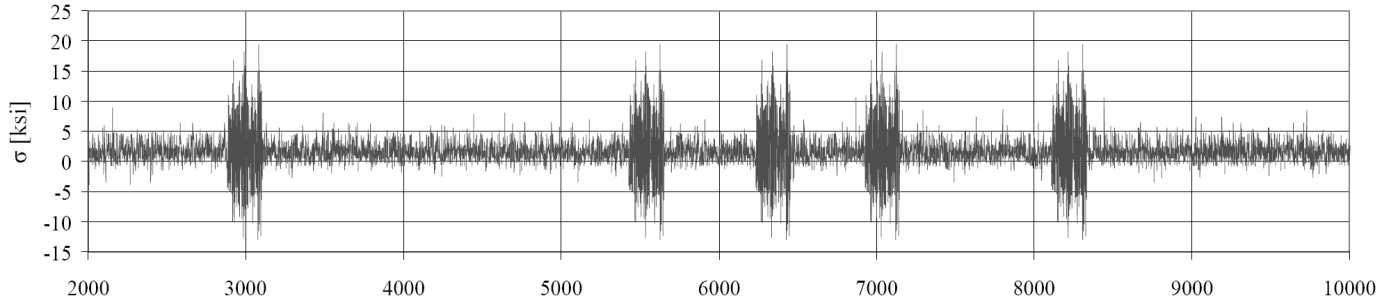


Figure 13. mixed spectrum with AoA limit for buffeting at 20°.

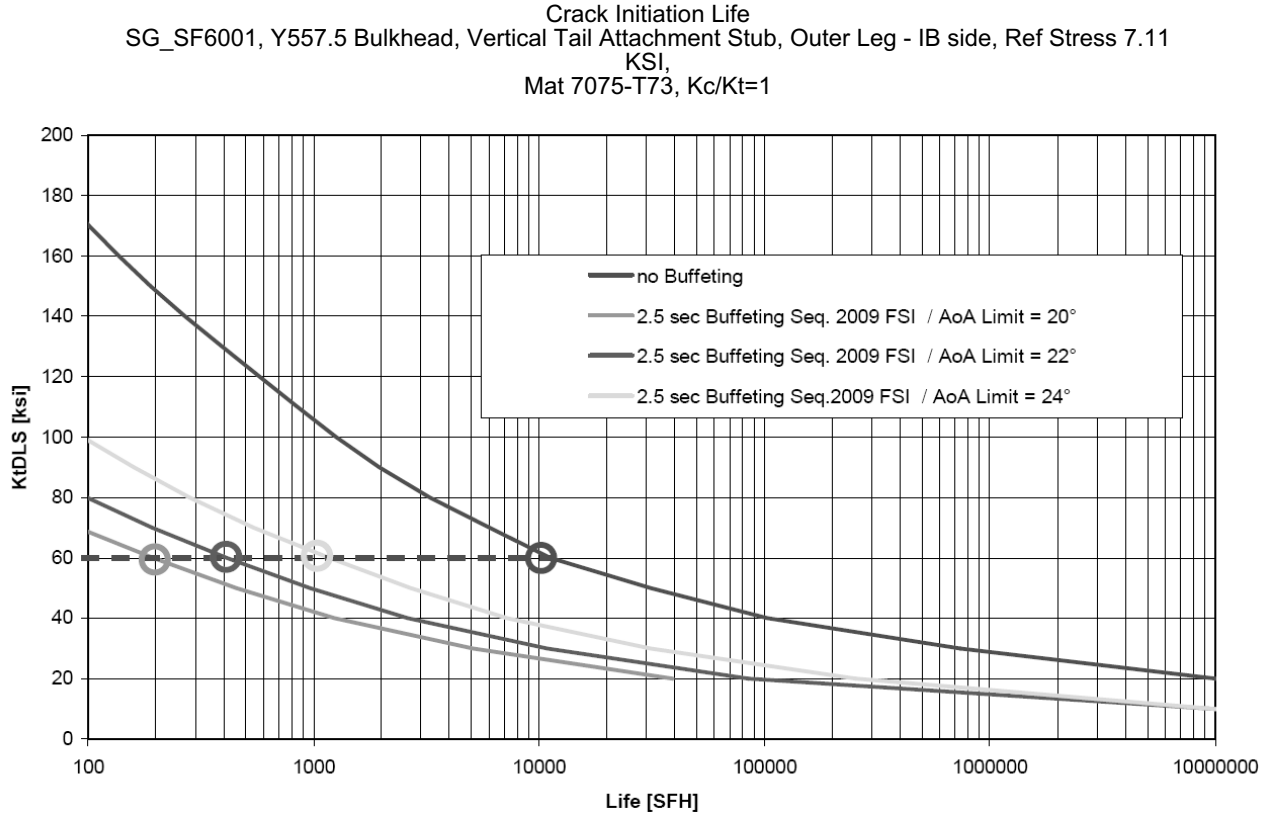


Figure 14. Kt DLS curves for different mixed spectra.

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

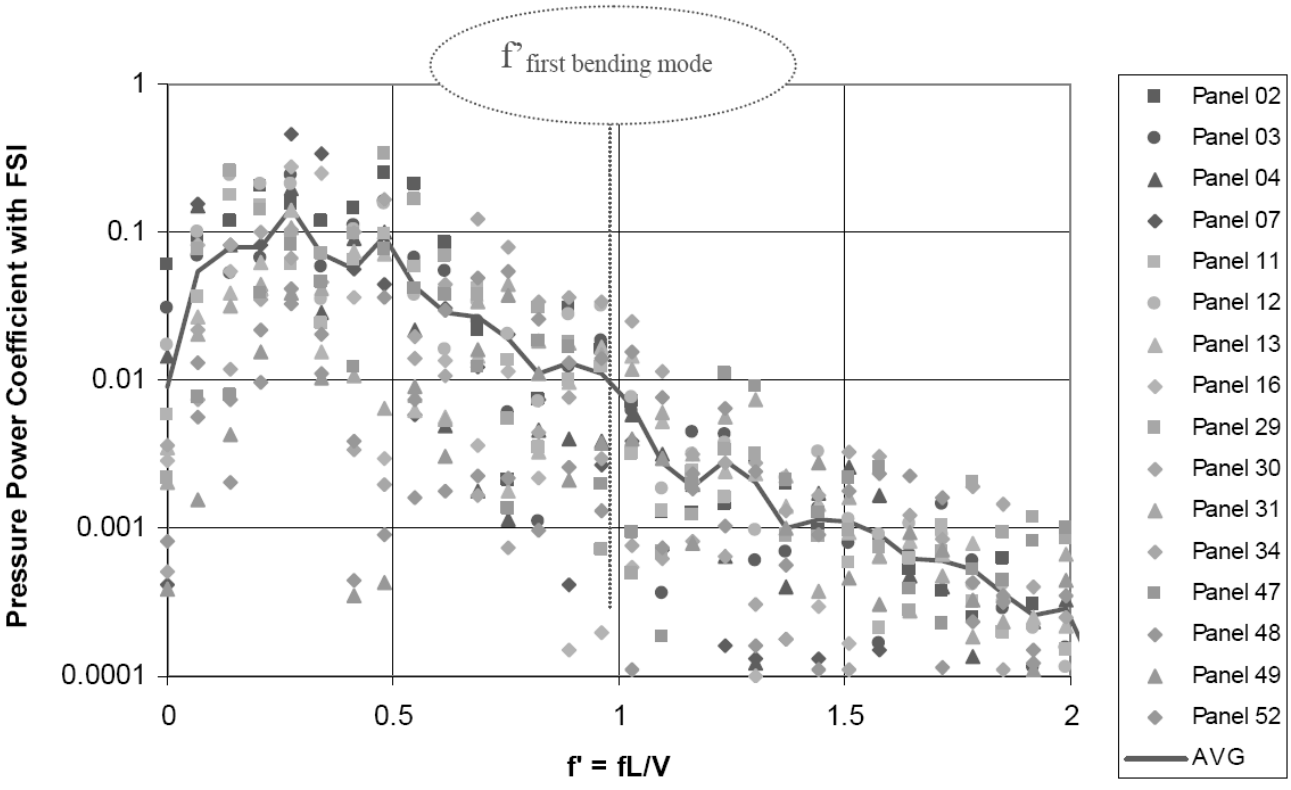


Figure 15. Power spectral distribution for 16 different panels with Fluid Structure Interaction (FSI).

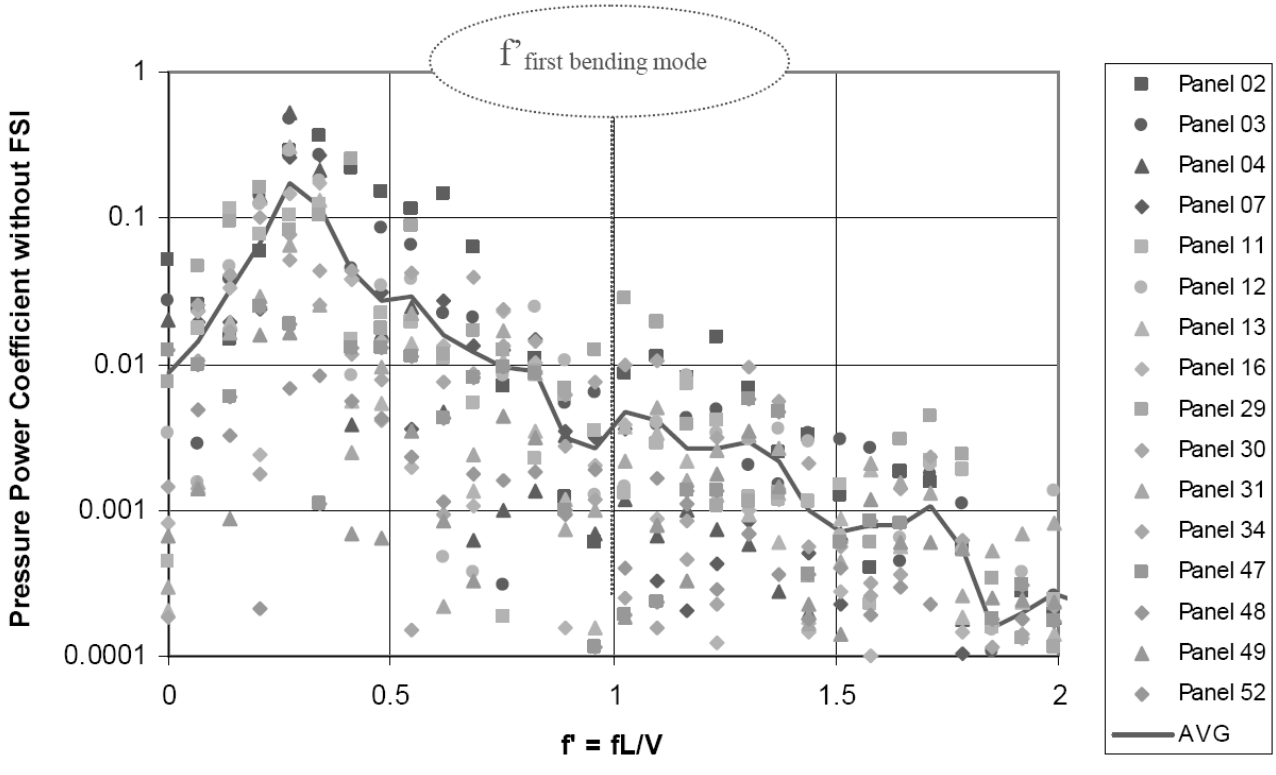


Figure 16. Power spectral distribution for 16 different panels without Fluid Structure Interaction (FSI).

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press

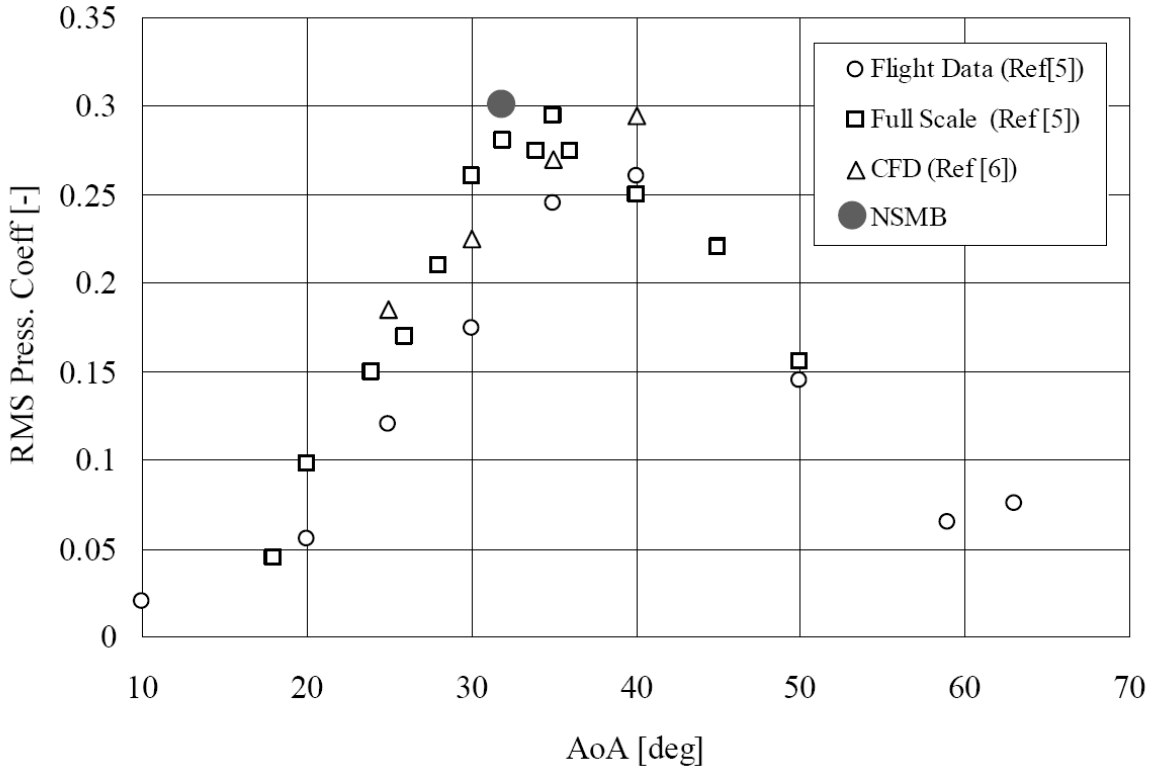


Figure 17. RMS pressure coefficient at 45% chord and 60% span.

REFERENCES

1. FARHAT, C., LESOINNE, M. and MAMAN, N. Mixed explicit/implicit time integration of coupled aeroelastic problems: three-field formulation, geometric conservation and distributed solution, *Int J Numerical Methods in Fluid*, 1995, **21**, (10), pp 807-835.
2. YATES, JR E.C. AGARD Standard aero-elastic configurations for dynamic response 1: Wing 445.6 AGARD R-765, 1988, NASA TM- 100492, 1987.
3. VOS, J.B., RIZZI, A.W., CORJON, A., CHAPUT, E. and SOINNE E. Recent advances in aerodynamics inside the NSMB (Navier Stokes Multi Block) Consortium. AIAA 36th Aerospace Sciences Meeting, January 1998.
4. VOS, J.B., RIZZI, A.W., DARRACQ, D. and HIRSCHEL, E.H. Computational aerodynamics for industrial airframe design using Navier Stokes solvers, *Progress in Aerospace Sciences*, 2002, **38**, pp 601-697.
5. GUILLAUME, M., GEHRI, A., MANDANIS, G. and VOS, J.B. Calculation of F/A-18 fatigue loads and wing deformations using computational fluid dynamics, Proccedings 2006-10.5.2, ICAS Hamburg, Germany, September 2006.
6. SMITH, J. Aeroelastic functionality in edge initial implementation and validation, FOI Report R-1485-SE, 2005.
7. SPEKREIJSE, S.P., PRANANTA, B.B. and KOK, J.C. A simple, robust and fast algorithm to compute deformations of multi block structured grids, NLR-TP-2002-105, 2002.
8. GOURA, G.S.L. Time Marching Analysis of Flutter Using Computational Fluid Dynamics. PhD thesis, 2001, University of Glasgow, Glasgow, UK.
9. BUCHER, B., GUILLAUME, M., GEHRI, A., VOS, J.B., LUDWIG, T., MERAZZI, S. and MANDANIS, G. Dynamic response and fatigue loading of the Swiss F/A-18 vertical tail due to buffeting, NAFEMS World Congress, Vancouver, Canada, May 2007.
10. LARRY, A.M. Full scale wind-tunnel studies of F/A-18 tail buffet, *J Aircr*, May-June 1996, **33**, (3).
11. ESSAM, F.S. Alleviation of vertical tail buffeting of F/A-18 Aircraft, *J Aircr*, March-April 2004, **41**, (2).
12. STEPHENS, R.I., FATEMI, A., STEPHENS, R.R and FUCHS, H.O. *Metal Fatigue in Engineering*, 2nd ed; John Wiley & Sons, 2001.
13. LAPACK-Library, http://www.netlib.org/lapack/

https://doi.org/10.1017/S0001924000005777 Published online by Cambridge University Press