Recent Developments on Fluid Structure Interaction in the Navier Stokes Multi Block (NSMB) CFD solver.

J.B. Vos∗, D. Charbonnier†
CFS Engineering, EPFL Innovation Park, Batiment A, CH-1015 Lausanne, Switzerland
T. Ludwig‡ S. Merazzi§
SMR Engineering and Development, Blumenstrasse 14-16, CH-2502 Bienne, Switzerland
A. Gehri¶ P. Stephani∥
RUAG Aviation, Aerodynamics Department, CH-6032, Emmen, Switzerland

Since 1997 the Swiss Airforce is operating the F/A-18 aircraft. To better understand the aerodynamic loads on the aircraft RUAG Aviation invested in developing a CFD capability that includes the possibility for Fluid Structure Interaction simulations. The paper will present recent developments on Fluid Structure Interaction and results of simulations for the F/A-18 fighter are discussed.

Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALE</td>
<td>Arbitrary Lagrangian Eulerian</td>
</tr>
<tr>
<td>AUSM</td>
<td>Advection Upstream Splitting Method</td>
</tr>
<tr>
<td>ASIP</td>
<td>Aircraft Structural Integrity Program</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CSM</td>
<td>Computational Structural Mechanics</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Model</td>
</tr>
<tr>
<td>FSI</td>
<td>Fluid Structure Interaction</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>NSMB</td>
<td>Navier Stokes Multi Block</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>TFI</td>
<td>Transfinite Interpolation</td>
</tr>
<tr>
<td>VSI</td>
<td>Volume Spline Interpolation</td>
</tr>
</tbody>
</table>

I. Introduction

The Swiss Airforce is operating the F/A-18 C/D aircraft since 1997. The maneuver spectrum for the Swiss usage of this aircraft is about three times more severe than the US Navy design, resulting in a different structural design. As a consequence an Aircraft Structural Integrity Program (ASIP) study was carried out by Boeing, and to validate this study a Full Scale Test Facility was built at RUAG Aviation and operated from 2003 to 2005. To better understand the aerodynamic loads on the aircraft RUAG Aviation looked
into methods to generate these loads for different flight conditions. A large investment was made in further developing the NSMB Navier Stokes Multi Block (NSMB) Computational Fluid Dynamics (CFD) code and in particular its extension for Fluid Structure Interaction (FSI) simulations.\textsuperscript{1,7} Procedures were developed for both static and dynamic Fluid Structure Interaction. Static Fluid Structure Interaction was put in place using a segregated two solver approach. A CFD calculation was made and run for a certain number of steps, the calculation was stopped, the aerodynamic loads were extracted and used as input for Nastran. Nastran computed the displacements of the aircraft structure that were transformed to surface grid point displacements. The CFD solver was then re-started and used these displacements to re-compute the grid, and then continued the calculation on this grid, typically for about 300 to 500 steps. Then the CFD solver was stopped and the process was repeated. In general 4 to 5 of these iterations were made to obtain a converged deformed geometry. Static FSI was only used to deform the wing of the F/A-18 fighter.

Dynamic FSI simulations were made to study vertical tail buffeting, and employed a linear structural model based on a modal formulation. In this case the CFD solver and the Computational Structural Mechanics (CSM) solver are tightly coupled.

Several developments were made over the past years that are presented in this paper. The grid used for the F/A-18 CFD simulations has evolved, the 2nd generation grid used for the studies reported in\textsuperscript{1} has been replaced by the 3rd generation grid that is employing the chimera method for easy change of control surface deflections and for adding or removing loads (fuel tanks and weapons). Using the chimera method for FSI calculations required several developments to make it work in an automatic fashion.

The need to provide quickly aerodynamic loads for statically deformed wings led to the decision to couple the NSMB CFD solver with the open-source Finite Element Analysis environment B2000++.\textsuperscript{2} This has led to the possibility to perform both static and dynamic FSI simulations using either a modal or a FEM approach without the need to interrupt the simulation.

This paper is organized as follows: first the CFD solver NSMB and the FEM Environment B2000++ are shortly presented. This is followed by a discussion on the geometrical coupling tool, the re-meshing procedure implemented in NSMB, and the Aero-elastic simulation environment. Finally results of simulations for the MDO Aircraft,\textsuperscript{3} the AGARD445.6 wing\textsuperscript{4} and the F/A-18 fighter will be presented.

II. The NSMB CFD solver

The Navier Stokes Multi Block solver NSMB was initially developed in 1992 at the Swiss Federal Institute of Technology (EPFL) in Lausanne, and from 1993 onwards in the NSMB consortium composed of different universities, research establishments and industries. Today NSMB is developed by IMF-Toulouse (IMF Toulouse, France), ICUBE (Strasbourg, France), University of Munchen (TUM, Germany), University of the Army in Munchen (Germany), Airbus Safran (France), RUAG Aviation and CFS Engineering. A variety of papers have been published on NSMB, examples are in\textsuperscript{5–15}

NSMB is a parallelized CFD solver employing the cell-centered finite volume method using multi block structured grids to discretize the Navier Stokes equations. To simplify the mesh generation for complex geometries NSMB uses the patch grid (also known as the sliding mesh) approach and the chimera method. The chimera method is also used for simulations involving moving bodies.

Space discretization schemes implemented in NSMB are the $2^{nd}$ and $4^{th}$ order central schemes with artificial dissipation and Roe and AUSM upwind schemes from $1^{st}$ to $5^{th}$ order. Time integration can be made using explicit Runge-Kutta schemes, or the semi-implicit LU-SGS scheme. Different methods have been implemented to accelerate the convergence to steady state, as for example local time stepping, multigrid and full multigrid, and low Mach number preconditioning. Unsteady simulations are made using the dual time stepping approach or using the 3rd order Runge Kutta scheme.

Turbulence is modelled using standard approaches as for example the algebraic Baldwin-Lomax model, the 1-equation Spalart model (and several of it’s variants) and the $k−\omega$ family of models (including the Wilcox and Menter Shear Stress models). Explicit Algebraic Reynolds Stress models and Reynolds Stress models have also been implemented, but are not used on a routine base. Transition to turbulence can be modelled by specifying transition lines or planes, or by solving the $\gamma−R_{0}$ transport equations.\textsuperscript{19} For unsteady CFD simulations different Hybrid RANS-LES models are available.

NSMB was from the start of its development designed for use of hypersonic applications, and includes different levels of chemistry modeling ranging from equilibrium chemistry to electronic non-equilibrium. A large number of non-equilibrium chemistry models for Air, CO2 and N2 have been implemented in NSMB.
For hypersonic flows NSMB includes the possibility to adapt the far field boundary to the bow shock. This is in particular important for blunt body shapes (like capsules) since this procedure removes the so-called carbuncle phenomenon introduced by the numerical scheme due to the non-alignment of the grid with the bow shock and it improves in particular the solution of the heat flux on the heat shield. The Arbitrary Langerian Eulerian (ALE) approach is employed for simulations using moving or deforming grids. When using the ALE approach it is not necessary that all blocks are moving or deforming, it is possible to define different groups of blocks each having their own movement. This is in particular useful for multi-body simulations.

To permit CFD simulations on deforming grids it is necessary to re-generate the grid. NSMB includes a remeshing algorithm\(^7\) and recent developments are discussed in more detail later on.

### III. The B2000++ solver

B2000++ is a Finite Element Method (FEM) solver which is being developed by SMR Engineering & Development. A wide range of problems in aerospace engineering can be studied, such as global aircraft as well as components or sub-components such as stiffened panels. The element library comprises shell elements, beam elements, point-mass elements, rigid-body elements, as well as 2D and 3D elements. Linear static analysis, linear dynamic analysis, free-vibration analysis and buckling analysis can be selected, as well as nonlinear static and dynamic analysis. For strength analysis, several failure criteria for isotropic materials and laminated composites are available.

A high computational effectiveness is one of B2000++’s strengths. Symmetric multi-processing (SMP) accelerates the element procedures, taking advantage of today’s multi-core CPU’s. The open-source matrix solver MUMPS provides distributed parallelism via MPI. Eigen-analysis is carried out with the implicitly restarted Lanczos solver that is implemented in the open-source package ARPACK.

The modular architecture facilitates the implementation of numerical methods. New material and element formulations, essential and natural boundary conditions, and solution methods can be added, requiring no or only a few modifications to the existing code. This flexibility enables the adaptation of B2000++ to specific problems like coupled fully nonlinear FSI.

### IV. Geometric Multi-Region Coupling

To transfer the aerodynamic forces from the CFD wetted surface to the structural FEM model, and to transfer the displacements from the FEM model to the CFD wetted surface, it is necessary to employ a spatial coupling procedure as the CFD and FEM meshes are non-matching.

The spatial coupling is implemented within the B2000++ FEM code and is capable of multi-region coupling. As example the F/A-18 wing consists of different control surfaces – inner leading edge flap, outer leading edge flap, trailing edge flap, and aileron – that can move independently. Thus, a multi-region spatial interpolation procedure is utilized where the wing surface and the control surfaces constitute different coupling regions, and C\(^0\) continuity constraints at the coupling region intersections are enforced.

#### IV.A. Single-Region Coupling Procedure

The following sets of points are distinguished: The set of structural grid points \(\mathcal{S}\), the set of fluid surface points \(\mathcal{F}\) where the displacements shall be interpolated, and the set of fluid surface points \(\mathcal{P}\) where the displacements are prescribed. \(\mathcal{P}\) may be empty. The set of support nodes is \(\mathcal{H} = \mathcal{S} \cup \mathcal{P}\), and without loss of generality it is assumed that the coordinates \(X^H\), displacements \(u^H\), and concentrated forces \(f^H\) are arranged as follows:

\[
X^H = \begin{bmatrix} X^S \\ X^P \end{bmatrix} ; \quad u^H = \begin{bmatrix} u^S \\ u^P \end{bmatrix} ; \quad f^H = \begin{bmatrix} f^S \\ f^P \end{bmatrix}
\]

Let \(n = |\mathcal{H}|\) and \(m = |\mathcal{F}|\). The volume-spline coupling operator\(^5\) \(G\) is a \(m \times n\) matrix

\[
G = AB
\]
Figure 1: Undeformed CFD wetted surface (yellow) and deformed CFD surface (silver) at the trailing edge flap and aileron (left) and at the wing tip (right).

where the $m \times (n+1)$ matrix $A$ is

$$A = \begin{bmatrix}
1 & |X^H_1 - X^F_1| & |X^H_2 - X^F_2| & \cdots & |X^H_n - X^F_n| \\
1 & |X^H_1 - X^F_2| & |X^H_2 - X^F_2| & \cdots & |X^H_n - X^F_2| \\
\vdots & \vdots & \vdots & \ddots & \vdots \\
1 & |X^H_1 - X^F_n| & |X^H_2 - X^F_n| & \cdots & |X^H_n - X^F_n|
\end{bmatrix}$$

(3)

and the $(n+1) \times n$ matrix $B$

$$B = (C^{-1})_{(1)(1)}$$

(4)

is a submatrix where the first column is removed from the inverse of the $(n+1) \times (n+1)$ matrix $C$.

$$C = \begin{bmatrix}
0 & 1 & 1 & \cdots & 1 \\
1 & 0 & |X^H_2 - X^H_1| & |X^H_3 - X^H_1| & \cdots & |X^H_n - X^H_1| \\
1 & |X^H_1 - X^H_2| & 0 & |X^H_3 - X^H_2| & \cdots & |X^H_n - X^H_2| \\
1 & |X^H_1 - X^H_3| & |X^H_2 - X^H_3| & 0 & \cdots & |X^H_n - X^H_3| \\
\vdots & \vdots & \vdots & \ddots & \ddots & \ddots \\
1 & |X^H_1 - X^H_n| & |X^H_2 - X^H_n| & |X^H_3 - X^H_n| & \cdots & 0
\end{bmatrix}$$

(5)

The displacements $u^F$ are interpolated as follows:

$$u^F = Gu^H$$

(6)

The forces acting on the structural grid points and on the constrained fluid surface nodes are obtained by applying the transpose of $G$ to the forces on the unconstrained fluid surface mesh nodes:

$$f^H = G^T f^F$$

(7)
IV.B. Multi-Region Coupling Procedure

Two coupling regions A and B are considered where some of the fluid surface mesh nodes belong to both regions (Fig. 2).

![Diagram of coupling regions A and B with structural points and fluid surface mesh nodes.]

Figure 2: Two coupling regions with structural points and fluid surface mesh nodes.

To maintain inter-region continuity of the displacements, it is necessary to proceed in three steps. Note that steps 2 and step 3 are interchangeable.

1. \( F \) contains the fluid interface nodes, \( S \) contains the structural nodes of region A and B, and \( P = \emptyset \).
2. \( F \) contains the fluid nodes of region A without the interface nodes, \( S \) contains the structural nodes of region A, and \( P \) contains the fluid interface nodes.
3. \( F \) contains the fluid nodes of region B without the interface nodes, \( S \) contains the structural nodes of region B, and \( P \) contains the fluid interface nodes.

The transfer of the concentrated nodal forces from the fluid wetted surface to the structural grid points is carried out in the opposite order (steps 1 and 2 are interchangeable).

1. \( F \) contains the fluid nodes of region A without the interface nodes, \( S \) contains the structural nodes of region A, \( P \) contains the fluid interface nodes, for which the calculated forces are added to the fluid forces.
2. \( F \) contains the fluid nodes of region B without the interface nodes, \( S \) contains the structural nodes of region B, and \( P \) contains the fluid interface nodes, for which the calculated forces are added to the fluid forces.
3. \( F \) contains the fluid interface nodes, \( S \) contains the structural nodes of region A and B, and \( P = \emptyset \).

The forces that are mapped from \( F \) consist of the fluid forces at these points plus the forces that were mapped in steps 1 and 2 to these points.

This procedure preserves the total force and is energy-conservative, which is important in flutter analysis. The deformed CFD surface is smooth except at the coupling region intersections. It is also possible to enforce zero displacements at coupling region boundaries. For example, the structural FEM model may consist only of a wingbox which is clamped at the root, while the CFD model is a wing-body configuration. In this case, the set \( P \) contains all fluid surface points of the body, and \( \mathbf{u}^P = \mathbf{0} \).
IV.C. Definition of Coupling Regions

To facilitate the definition of the different coupling regions, the interactive and graphical tool FSCON (Fluid-Structure CONnector) is being used. It allows to select and visualize individual parts of the CFD wetted surface and of the FEM model. For the CFD model, boundary codes are used to select the coupling regions, whereas nodesets, elementsets, group codes, etc. are used for the FEM model. Thus, the coupling region definitions are independent of the mesh size and aircraft configuration.

![Figure 3: The FSCON graphical user-interface, showing the MDO wing-body test case. The structural model consists only of the wing box. A part of the body fluid surface nodes is included in the coupling (colored in red, the cyan part is constrained to zero), since the FEM model is clamped at the symmetry plane and has nonzero displacements at the wing root. This way, problems with the remeshing algorithm are avoided.](image)

V. Remeshing algorithm

The remeshing algorithm implemented in NSMB is discussed in detail in,\textsuperscript{1} and is a combination of Volume Spline Interpolation (VSI)\textsuperscript{16} and Transfinite Interpolation (TFI). It consists of the following steps:

1. collect information on coordinates and their displacements and create a list of so called prescribed points. These prescribed points are then used to compute the Volume Spline Coefficients by solving a linear matrix systems of equations

2. compute the displacements of block edges using the VSI method. The advantage of working with the displacements instead of the coordinates is that if the displacement of the edges is close to zero, the displacement in the volume will be close to zero too, and the original coordinates are not changed.

3. use 2D Trans Finite Interpolation to generate the displacement of the coordinates on the block faces

4. use 3D Trans Finite Interpolation to generate the displacement of the coordinates in the volume

5. sum the coordinates and displacements to obtain the new mesh

Several problems were encountered (and solved) for complex grids. For example it might occur that one block edge is shared between 3 blocks or more and not all blocks know about the mesh movement. At the start of a calculation all the information on these shared edges is collected and the displacements between these edges are exchanged during the remeshing process.
The most time consuming part in the process summarized above is the solution of the linear matrix system of equations to compute the Volume Spline Coefficients, and the costs are proportional to $\frac{1}{2}n^2$ with $n$ the number of prescribed points. A large effort was made to limit the number of prescribed points through a suitable combination of use of VSI and TFI. Today the remeshing of a F/A-18 third generation mesh having around 50 Million points takes less than a minute elapse time on present day HPC clusters.

A recent development in the remeshing procedure concerns the use of the chimera method. The chimera method implemented in NSMB employs blocks inside the structure (that are not computed) to determine intersections between grids. If the structure is moving it means that not only the fluid mesh is moved but one needs also to move the mesh inside the structure. This has been implemented through an exchange of surface displacements between fluid and solid meshes. The second problem concerned the remeshing itself, and is illustrated in Fig. 4.

![Figure 4: Illustration of the remeshing problem near the rudder of the F/A-18. From left to right: start grid, rudder position after a movement, grid for the vertical tail near the gap, corrected grid.](image)

When using the chimera method the rudder and vertical tail have different grids that are overlapping. Due to the movement of the rudder the block edges of the block attached to the vertical tail are moved too, leading to the deformed block shown in the figure. This problem has been solved by performing the remeshing procedure on each chimera grid independently of the other chimera grids.

### VI. Aero Elastic Simulation environment

The development of the Aero Elastic Simulation environment started in 2005, and consists in the current state of development of the following components:

- a CFD solver using the ALE formulation that includes a remeshing algorithm;
- an interactive graphical tool to set-up the spatial coupling;
- a FEM solver to compute the structural state and to carry out the spatial coupling;
- a spatial coupling tool that can be utilized independently of the FEM solver.

Until recently the Aero Elastic Simulation tool was used for two different applications:

1. Calculation of the static wing deformation at high angles of attack. This is a steady CFD calculation and the deformation of the structure is computed using a complete FEM model. In practice the steady CFD calculation was run until a stage of convergence, the loads on the structure were extracted from the CFD results and sent to an external FEM program which returned the displacements of the structure. These displacements were read by the CFD solver, a new mesh was generated and the calculation was continued. In general this process was repeated 5 times to obtain a converged wing position.

2. Calculation of the unsteady loads on the vertical tail. This is an unsteady CFD calculation and to reduce computational costs a linear structural model is used employing a modal formulation. The
time integration of the modal equations is made using the Newmark method. This can be completely integrated in the CFD solver. Once the mode shapes and the information on the geometrical coupling are available the CFD solver can perform the aero-elastic simulation. Both a weak and strong coupling approach have been implemented.

The Aero Elastic Simulation environment has been extended through a close coupling between the NSMB CFD solver and the B2000++ FEM solver so that for both applications mentioned above it is possible to use either the complete FEM solver or the modal analysis method. This also means that for the static wing deformation calculations it is not needed anymore to stop the CFD solver to transmit the loads to the FEM solver; in the new set-up the information is transferred at selected steps during the simulation.

![Figure 5: Schematic of the fully nonlinear aero-elastic toolchain. Orange: Data transferred to set up the spatial coupling. Red: Data exchanged during the inner time steps (strong coupling) or the outer time steps (weak coupling).](image)

Fig. 5 shows the different processes inside both the fluid and structural solvers when performing a fully nonlinear unsteady aero-elastic simulation. The solvers communicate via the Message Passing Interface (MPI), on top of which a high-level communication protocol has been implemented for structured data encapsulation. This application protocol makes use of the MemCom data manager functions that are both integrated in the NSMB and B2000++ solvers.

The NSMB CFD solver controls the time step and the time increment size. The spatial coupling is implemented in the B2000++ FEM solver. Due to the modular architecture, integration of other CFD codes is simplified as only the MPI communication interface needs to be adapted.

For the communication with the B2000++ FEM solver, a library has been implemented whose purpose is to send the coordinates of the wetted surface nodes, to communicate the current time increment size, and to send the integrated nodal forces to the B2000++ FEM solver at each time step. It also receives the interpolated displacements from the FEM solver.

Within the B2000++ FEM solver, the spatial coupling and the MPI communication with the NSMB CFD solver is encapsulated in a natural boundary condition. In the FEM model description, it is sufficient to add the directive “nbc_fsi_nsmb” to the analysis case description. This activates the natural boundary condition which in turn sets up the MPI communication with the NSMB CFD solver.

Since the FEM solver is nonlinear, it is possible for highly nonlinear problems that convergence of the Newton iterations cannot be obtained in a single increment that corresponds to the time increment size as is specified by the CFD solver. In this case, the FEM solver performs several increments with smaller time increment sizes, while ensuring that the sum of them matches the CFD time increment size. When this point is reached, the FEM solver initiates the transfer of the interpolated displacements to the CFD solver.
VII. Test cases

VII.A. MDO Aircraft

The MDO aircraft is a large transport aircraft with a flexible wing resembling the Airbus 380, and was the result of an EU funded project in the 1990's. The geometry consists of a fuselage and a wing, and the wing has a jig-shape needed for static aero-elastic computations. This case was also used in the EC funded project UNSI (Unsteady Viscous Flow in the Context of Fluid-Structure Interaction) that finished in the year 2000. The results of this project are published in a book.

Calculations were made for the so called Case A conditions that are summarized in Table 1. Both the modal approach (considering 10 elastic modes) and the FEM approach using B2000++ were used in the calculations.

| Altitude | 37000 | ft |
| Mach | 0.85 |
| $p_\infty$ | 21662.3 | Pa |
| $\rho_\infty$ | 0.34832 | $kg/m^3$ |
| $T_\infty$ | 216 | K |
| Re/meter | 6.1581*10^6 | 1/m |
| $C_L$ | 0.4581 |

Table 1: MDO fuselage-wing Case A conditions.

The grid employed in the calculations had 123 blocks and about 1.15 million cells, and the surface grid is shown in Fig. 6.

All calculations were made using central space discretization scheme with artificial dissipation, the LUSGS scheme for the time integration and the $k-\omega$ Menter Shear Stress turbulence model. The calculations ran for 7000 steps, and the static wing deformation was executed 5 times, after respectively 4000, 4500, 5000, 5500 and 6000 steps. Figure 7 shows the wing position (including a zoom of the wing tip) at different iterations. The zoom of the wing tip shows clearly the different iteration steps: the first deformation (orange color) shows a large movement upwards followed by a downward movement for the 2nd deformation. The last two deformations (light blue and blue) are close to each other.

Table 2 summarizes the computed angle of attack to reach $C_L = 0.4581$ for this case, and the results are compared with results reported by SAAB in . Note that the results obtained by SAAB are for the wing only. Figure 8 shows the deformation of the leading and trailing edge of the wing, together with the results obtained by SAAB. $\eta$ is the normalized y-coordinate ($y/y_{max}$). As can be seen the NSMB results...
obtained by using B2000++ are in good agreement with the results by SAAB. One can also observe that the maximum deflection of the leading edge of the wing near the tip is slightly less than 3 m.

<table>
<thead>
<tr>
<th>CFD Approach</th>
<th>CSM model</th>
<th>( \alpha )</th>
</tr>
</thead>
<tbody>
<tr>
<td>SAAB Euler</td>
<td>Modal</td>
<td>0.7450</td>
</tr>
<tr>
<td>NSMB Turbulence</td>
<td>Modal</td>
<td>0.7145</td>
</tr>
<tr>
<td>NSMB Turbulence</td>
<td>B2000++</td>
<td>0.5330</td>
</tr>
</tbody>
</table>

Table 2: Computed angle of attack MDO Test Case A.

VII.B. AGARD 445.6 Wing

An important problem in the development and validation of dynamic aero-elastic simulation tools is the lack of available experimental data for assessment and validation. Reasons for this are that the experiments by definition are destructive and that they require special models for the correct scaling of the frequencies. One the most cited experimental test case is the AGARD 445.6 wing.\(^4\) The AGARD445.6 wing, made of mahogany, has a 45° 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 for the AGARD 445.6 was build by SMR. The material properties for the structural model were taken from:

\[
\begin{align*}
E_1 &= 3.15106 \times 10^6 \text{ Pa} \\
E_2 &= 4.16218 \times 10^8 \text{ Pa} \\
G &= 4.39218 \times 10^6 \text{ Pa} \\
\rho &= 381.98 \text{ kg/m}^3 \\
\nu &= 0.31
\end{align*}
\]

Table 3: Material properties AGARD445.6 wing.

with \(E_1\) and \(E_2\) the moduli of elasticity in the longitudinal and lateral directions, \(G\) the shear modulus, \(\nu\) the Poisson ratio and \(\rho\) the wing density. Only the first four mode shapes are considered.

Inviscid calculations were made using a 31 block grid having about 170'000 cells. Grid refinement studies were made indicating that this mesh was sufficient. Table 4 lists the free stream values used in the calculations. Three different values of the free stream pressure were used, respectively 7000 Pa, 4600 Pa and 3500 Pa. Due to the change in speed of sound, this results in three different values of the free stream velocities, and thus to three different values of the flutter speed index, see Table 5. The experimental data\(^4\)
Table 4: AGARD445.6 wing free stream conditions.

<table>
<thead>
<tr>
<th>$\text{Mach}$</th>
<th>0.95</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho_\infty$</td>
<td>0.061 kg/m$^3$</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Table 5: AGARD445.6 wing free stream pressure and velocity.

<table>
<thead>
<tr>
<th>$p_\infty$</th>
<th>7000</th>
<th>4600</th>
<th>3500</th>
<th>Pa</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_\infty$</td>
<td>381</td>
<td>309</td>
<td>269</td>
<td>m/s</td>
</tr>
<tr>
<td>$V_f$</td>
<td>0.383</td>
<td>0.311</td>
<td>0.271</td>
<td></td>
</tr>
</tbody>
</table>

Figure 10: MDO Aircraft test case A: Computed $C_p$ on the undeformed and statically deformed wing.

shows that at $\text{Mach} = 0.95$ the flutter boundary is around $V_f = 0.32$, hence the highest pressure condition should yield flutter while for the other 2 conditions no flutter should be found.

All calculations were made using the central scheme with artificial dissipation. The time integration was carried out using dual time stepping employing the 2$^{nd}$ order implicit backward difference scheme. The LU-SGS scheme was used to converge the equations in the inner loop. The unsteady calculations were started from a steady state calculation. A 2% deflection of the first bending moment was given to the structure, and a ramping procedure was used during the first 25 outer time steps to impose this deflection smoothly. Different outer time steps were studied for the case with a free stream pressure of 7000 Pa, indicating that an outer time step of $10^{-3}$ is a good compromise. Using smaller time steps does not change the results significantly, while the results for larger time steps show differences, in particular in the amplitude of the oscillations.

Figure 11 shows the computed $C_L$ as function of time for the different calculations carried out. The left figure shows that flutter is obtained for the highest pressure, as was to be expected. The right figure shows the influence of the CFD-CSM coupling approach for the case with a free stream pressure of 4600 Pa. The weak coupling approach (couple the fluid and structure only each outer time step) leads to undamped
oscillations while with the strong coupling approach (couple the fluid and structure each step in the inner time stepping loop) the oscillations are not amplified. With the strong coupling approach it is possible to switch off the CFD-CSM coupling after a certain number of timesteps. Performing the coupling only the first 25 steps of the inner time stepping loops yields results close to the results obtained by making the coupling each step.

Figure 11: Results AGARD445.6 case, left lift coefficient as function of time for different free stream pressures, right influence of coupling approach on the lift coefficient for a free stream pressure of 4600 Pa.

VII.C. F/A-18 fighter

VII.C.1. Introduction

The Aero Elastic Simulation environment is employed for two different types of simulations in the context of the Swiss F/A-18 fighter. Steady CFD simulations are carried out to determine the aerodynamic loads on the aircraft components, and at high angle of attack the wing deformation needs to be taken into account. These calculations are made using the full FEM model of the F/A-18 fighter. Unsteady simulations have been and will be made to obtain the unsteady loads on the vertical tail. For these calculations a modal analysis approach is used for the structure and the objective of the simulations is to generate unsteady loads that can be used for fatigue analysis.

VII.C.2. F/A-18 Grids

The CFD studies for the F/A-18 fighter started in 2001, and the first generation F/A-18 grid was generated by ICEMCFD in collaboration with the Aerodynamics Department of RUAG Aviation. Scripts were used for control surface deflections and the sub-topology strategy was used to add or remove external stores (fuel tanks, weapons). To permit the extraction of the aerodynamic loads on different aircraft components different solid wall boundary condition types were employed. This first generation grid had 939 blocks and 7.6 million cells for half an aircraft. Many aircraft details (antenna, gaps between components, missiles) were not modeled. The resolution of the mesh in the boundary layer was not sufficient with only 11 cells in the boundary layer. Based on results of CFD simulations it was decided to refine the mesh in 2005, leading to the 2nd generation grid. The ICEMCFD replay function was used for control surface deflections and for adding or removing external stores. This 2nd generation grid had 2802 blocks and 13.8 million cells for half an aircraft. Compared to the first generation mesh many aircraft details were now modelled, and the number of grid cells in the boundary layer was increased to 16. Still the grid resolution in the boundary layer was not sufficient, and each new aircraft configuration (change of control surface deflections, adding/removing external loads) took several days of work to implement. To remove these short comings and to introduce new store configurations as the AMRAAM dual launcher a 3rd generation F/A-18 grid was created in 2012. Since a first version of the chimera method had become available in NSMB it was decided to use this method because it permits to generate the grid of each aircraft component independent of the other components. This has two advantages:
• The overall grid becomes simpler and has a higher quality

• Aircraft components can be easily rotated or added/removed when assembling the grid required for a simulation, removing the need of manual intervention. While it took about 2-3 days to generate a 2nd generation F/A-18 grid for a particular load case, it takes a few seconds for the 3rd generation F/A-18 grid using python scripts.

The principal disadvantages of the chimera method are the need of a robust algorithm to detect the overlap between the different grids, and the increased CPU costs of the simulations.

The 3rd generation F/A-18 grid consists of 13 components that each have their own grid. The first component is the base aircraft without control surfaces and loads. There are 8 grids related to control surfaces, and 4 grids related to external loads. The grids for the different aircraft components have been made to permit 2 levels of multigrid, which also means that it is possible to generate a coarse grid by taking every 2nd grid point. This is in particular useful for testing and debugging purposes. Close to the walls the grids are made using the recommendations for grid generation used for the AIAA Drag Prediction Workshops. The first version of the 3rd generation grid has about 5000 blocks and around 50 Million cells (the exact numbers depend on the aircraft configuration). Since then the grids of some components were improved to increase the mesh resolution in specific areas and to reduce the gaps between grid components. This has been done using the partially or fully patched grid approach. Figure 12 shows the different grids used for the F/A-18 studies. For the 3rd generation grid only every 2nd grid line is shown, and the colors are used to indicate the different aircraft components.

VII.C.3. Preparation of the FSI Simulations using B2000++

The preparation of the structural FEM model consisted of two steps. In the first step, the Nastran BDF file of the F/A-18 FEM model was converted to a B2000++ FEM model. In the second step, the FEM model was modified for the spatial coupling procedure with the CFD model.

The F/A-18 structural FEM model was converted from Nastran Bulk Data Deck (BDF) format to a B2000++ input file, using an automatic conversion tool. This conversion tool:

• Accounts for the differences in the element formulations between the Nastran and the B2000++ FEM solvers. As an example, Nastran shell and beam elements allow for infinite shear correction factors, whereas the B2000++ shell and beam elements accept arbitrary but finite values for the shear correction factors.

• Is capable of changing the units. The structural FEM model is specified in the lb-in (force) imperial system, and the CFD model is specified in the SI system.

The Nastran model of the wing relies on the AUTOSPC option. With this option enabled, the FEM solver “fixes” singularities that are present in the FEM model. In general this makes it easier to create and to analyze FEM models. On the other hand, the accuracy of the direct sparse matrix solver is reduced, and serious modelling errors might be overlooked. For the validation of the conversion process, AUTOSPC would make it hard to determine whether a difference in the numerical result of the two codes was due to

• an error in the conversion of the FEM model,

• a different element formulation,

• a different implementation of AUTOSPC,

• or different values for the numerical parameters related to AUTOSPC.

For this reason, the B2000++ FEM model was modified such that it would work without AUTOSPC. To this end the various sources of singularities in the FEM model were identified. The conversion tool was updated such that it would recognize certain modelling problems and report on them. The MUMPS matrix solver’s capability to calculate the singular modes in conjunction with Python post-processing scripts allowed the graphical visualization of regions with singularities. In the wing, these singularities included:

• Some beam cross-section properties did not have any stiffness associated to them. Hence, the beam elements have rotation degrees-of-freedom which may be unconstrained.
• The skin of the control surfaces was modelled with shear panel elements. These elements have no bending stiffness.

• The RBE3 elements at the load stations were not fully constrained.

These issues were resolved on one hand by adding special rules to the conversion tool which can be activated, i.e., there is an option to stabilize beam elements, and another option allows to replace shear panel element by shell elements. To fix the singularities at the RBE elements, single-point constraints were added to the FEM model.

These activities resulted in a well-conditioned FEM model that could be analyzed with very good ac-
accuracy. For the numerical verification, the FEM model was clamped at the wing attachments and a single concentrated force was applied at the outer wingbox (Fig. 13).

Figure 13: Verification of the structural FEM model. The wing is clamped at the attachments and subjected to a concentrated load. The deformations are amplified.

The relative difference between the displacements that were reported by NX Nastran and those that were calculated by B2000++ did nowhere exceed 5%.

The spatial coupling is based on the volume-spline method, for which a dense system of equations has to be solved. The size of this system corresponds to the number of structural grid points \( n \) that are included in the coupling. Thus, the numerical effort is \( O(n^3) \).

The mesh of structural FEM model is rather dense. The computational effort for the spatial coupling would be quite large if all structural grid points were included in the coupling. For this reason, a subset of the grid points was selected for the spatial coupling. This significantly reduces the computational effort for the spatial coupling. These grid points are still close enough to each other to allow for good quality of the spatial mapping, see Fig. 14.
Figure 14: The structural grid points involved in the spatial coupling. The colours indicate the different coupling regions.

Near the wingtip, the structural model is rather coarse, and it does not cover the full wetted surface. Preliminary analyses have shown that the quality of predicted CFD surface displacement was insufficient near the wingtip. Figures 15a and 15b show images of the grid of the SIWA and launcher for a static FSI calculation using the original FEM model (the coarse F/A-18 grid was used for these tests), and one clearly sees the oscillations in the geometry.

To amend this, several grid points were added at SIWA and launcher to the structural FEM model; these grid points were connected by means of rigid-body elements to the structure. This measure resulted in much better quality of the spatial coupling. Fig. 15c shows the additional coupling points and Fig. 15d shows the mesh obtained after static deformation using the corrected model.

Structural grid points were also added at the load stations (not displayed in Fig. 14). If loads are attached, the spatial coupling will be able to transfer the loads and displacements between the two meshes with good accuracy.

VII.C.4. Steady FSI Simulations

To demonstrate the Aeroelastic simulation environment for the F/A-18 aircraft a calculation was carried out for the ASIP load case C1S825 conditions, for an aircraft equipped with loads (the C1S825 is in reality a so-called ‘clean’ configuration). This load case concerns a 8.25 g maneuver, and the free stream conditions are summarized in Table 6.

A coarse 3rd generation F/A-grid was used for this simulation, and consisted of 6700 blocks having in total 7.5 Million grid cells. The calculation was made using the central space discretization scheme with artificial dissipation, time integration was carried out using the LU-SGS scheme and turbulence was modelled using the $k-\omega$ Menter Shear Stress turbulence model. The static wing deformation was carried out three times. Figure 16 shows views of the aircraft with the undeformed (gray) and deformed wing (red). For this case the wingtip moves about 0.4 m and one also observes a slight twist of the wing.
Figure 15: Corrections F/A-18 FEM model for Static FSI calculations.

Table 6: F/A18 simulation conditions.

<table>
<thead>
<tr>
<th>Altitude</th>
<th>Mach</th>
<th>$p_\infty$</th>
<th>$\rho_\infty$</th>
<th>$T_\infty$</th>
<th>Re/meter</th>
<th>Qbar</th>
</tr>
</thead>
<tbody>
<tr>
<td>5000</td>
<td>0.7</td>
<td>84300</td>
<td>1.05651</td>
<td>216</td>
<td>$1.42 \times 10^7$</td>
<td>28915</td>
</tr>
</tbody>
</table>

Figure 17 shows the computed $C_p$ on the aircraft for the undeformed and deformed wing. One clearly observes differences in $C_p$, in particular near the trailing edge and near the wing tip. The front view figure (Fig. 17b) clearly shows the deformation and the twist of the wing.

Analysis of the loads on the different aircraft components show differences upto 15% between the undeformed and deformed aircraft. This is in line with studies in the past where steady FSI studies were performed on the 2nd generation F/A-18 grid using Nastran as FEM solver.

VII.C.5. Unsteady FSI Simulations

Unsteady FSI simulations for the F/A-18 fighter concern mostly the vertical tail since it is well known that at high angles of attack the breakdown of the vortex coming from the LEX will generate buffeting in this
region. This in turn will affect the fatigue life of aircraft components linking the vertical tail to the fuselage.

Unsteady FSI simulations for the vertical tail were made using a modal approach, considering the first 5 mode shapes (torsion and bending moments). To facilitate the extraction of the aerodynamic loads the vertical tail is divided into 6 strips, see Fig. 18. Each strip is further cut by horizontal lines in 9 parts leading to the creation of 54 panels shown on the right of Fig. 18.

During the calculation the position and pressure of the 5 points indicated in Fig. 18 were extracted as function of time.

Calculations were made for the Boeing load case C2s825. This case corresponds to an 8.5 g pull up maneuver at Mach=0.7 and the calculation conditions are summarized in Table 7.

<table>
<thead>
<tr>
<th>Mach</th>
<th>Altitude</th>
<th>$\alpha$</th>
<th>LEF</th>
<th>TEF</th>
<th>HSTAB</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.7</td>
<td>15000</td>
<td>26.6°</td>
<td>26.0°</td>
<td>0.0°</td>
<td>−6.1°</td>
</tr>
</tbody>
</table>

Table 7: C2S825 conditions.
Figure 17: $C_p$ on the F/A-18 fighter for the deformed and undeformed wing, C1S825 conditions.

Figure 18: Division of the vertical tail of the F/A-18 in panels.
The unsteady calculation was made using the central space discretization scheme with artificial dissipation. The dual time stepping approach was used for the integration in time, while LU-SGS scheme was used for the convergence of the equations in the inner loop. Turbulence was modelled using the Spalart turbulence model. The calculations were made using a time step of $2.5 \times 10^{-4}$ seconds. A strong FSI coupling approach was employed during the first 100 time steps and at maximum 200 innerloop steps were made. The unsteady calculation was started from a steady state calculation.

Figure 19 shows the $y-$ and $z-$ movement of the 5 points on the vertical tail. One can observe large movements, in particular of point2 (trailing edge of the vertical tail) and point4 (top of the rudder). One can also observe that the rudder and vertical tail have sometimes opposite movements (compare point3 and point4).

![Figure 19: Movements of 5 points on the vertical tail, Boeing load case C2S825, strongly coupled FSI calculation.](image)

Figure 20 shows the position of the vertical tail and rudder and the pressure at two different times. One clearly observes the different position of the rudder. Pressures on the vertical tail vary considerably.

![Figure 20: Vertical tail and rudder position (left) and pressure on the vertical tail at two time instances (right), Boeing load case C2S825.](image)

This can also be seen in Fig. 21 which shows the pressure in the 5 selected points as function of time. Pressure variations can be large, in particular for point2.

VIII. Conclusions

Since 2002 the NSMB CFD solver has been used to provide loads for the F/A-18 fighter. Several developments were made during the past years to improve this simulation capacity, in particular for aeroelastic CFD simulations.
Figure 21: Pressure as function of time in 5 points on the vertical tail, Boeing load case C2S825, strongly coupled FSI calculation.

The NSMB CFD solver has been successfully coupled to the B2000++ open-source Finite Element Analysis environment. A detailed discussion of the geometrical coupling tool is provided.

The aero-elastic simulation environment was used to simulate well known cases as the MDO aircraft and the AGARD445.6 wing, and results are in good agreement with results obtained previously or with results reported in the literature.

The aero-elastic simulation environment was also used to perform simulations for the F/A-18 fighter. Difficulties encountered were discussed and results for steady and unsteady simulations presented. This new environment is since the start of the year deployed at RUAG and used on a routine base.

References

2http://www.smr.ch/products/b2000/
12Shinde V., Marcel T., Hoarau Y., Deloze T., Harran G., Baj F., Cardolaccia J., Magnaud J.P., Longatte E., Braza M.,


