A STAGGERED METHOD USING A MODAL APPROACH FOR FLUID-STRUCTURE INTERACTION COMPUTATION

François Debrabandere, Benoît Tartinville, Charles Hirsch

NUMECA International
Ch. de la Hulpe 189 Terhulpse Steenweg – 1170 Brussels – Belgium
francois.debrabandere@numeca.be

Keywords: Fluid-structure interaction, Reduced-Order-Model, modal.

Abstract: In the present paper, a new method for Fluid-Structure Interactions (FSI) predictions is introduced. A Reduced-Order Model (ROM) is used for the structure, represented by its mode shapes and natural frequencies. A linear structure is assumed as well as a Rayleigh damping. The structural deformations are calculated by the resolution of the modal equations wherein the loads exerted by the fluid are considered as external forces. The fluid solver uses these deformations to compute the flow solution according to the new shape of the structure. This loosely-coupled staggered approach ensures a two-way coupling between the fluid and the structure.

The method is firstly compared to a full-order model on a clamped beam configuration oscillating under the effect of von Karman vortices. As a second application, a turbomachinery case is considered. The influence of passing rotor blade wakes on the downstream stator blades is investigated. Finally, the modal approach is used for a flutter application with the AGARD 445.6 wing.

NOMENCLATURE

\[ \tilde{f} \] = fluid load vector
\[ f \] = modal projection of load vector
\[ \tilde{F}_c \] = convective flux vector
\[ \tilde{F}_v \] = viscous flux vector
\[ FSI \] = flutter speed index
\[ I \] = identity matrix
\[ L_{ref} \] = reference length
\[ M \] = mass matrix
\[ m_s \] = mass of the structure
\[ q \] = generalized displacement
\[ Q \] = source term
\[ \tilde{S} \] = surface vector
\[ t \] = time
\[ \tilde{u} \] = deformation vector
\[ U \] = conservative variables
\[ V_f \] = free stream velocity at flutter conditions
\( \dot{V}_g \) = mesh velocity vector  
\( \beta, \gamma \) = coefficients of Newmark’s algorithm  
\( \xi \) = damping ratio  
\( \rho_f \) = fluid density  
\( \vec{\phi} \) = mode shape vector  
\( \Phi \) = mode shape matrix  
\( \omega \) = natural frequency  
\( \omega_1 \) = frequency of 1st torsion mode  
\( \Omega \) = volume  

**Subscripts**  
\( k \) = mode number  
\( n \) = iteration number

### 1 INTRODUCTION

Various approaches to Fluid-Structure interactions can be considered, ranging from one-way coupling to two-way coupling, up to fully coupled methods based on monolithic codes. Although many problems can be handled by a one-way coupling, the prediction of phenomena such as flutter requires the consideration of a two-way coupling, whereby the aeroelastic problem is treated as a whole. It allows the consideration of nonlinear interactions that occur between the fluid flow and the structure deformation. Considering the importance of FSI in the design of light and robust structures, the industry needs an access to efficient and accurate tools for FSI and aero-elastic predictions. The present approach offers an efficient two-way coupling solution, whereby the structure is represented by its mode shapes and natural frequencies. The modal equations are solved inside the fluid flow solver in order to retrieve the deformation of the structure and to take it into account in the flow calculation. Compared to other methods using externally coupled solvers [1], it presents the advantage to involve only one solver reducing thereby the complexity of the computational set up.

### 2 METHOD

#### 2.1 Fluid solver

The commercial package Fine\textsuperscript{TM}/Turbo [2] is used for this study. The flow solver is a three-dimensional, density-based, structured, multi-block Navier-Stokes code using a finite volume method. Central-difference space discretization is employed for the spatial discretization with Jameson type artificial dissipation. A four-stage explicit Runge-Kutta scheme is applied for the temporal discretization. Multi-grid method, local time-stepping and implicit residual smoothing are used in order to speed-up the convergence. Unsteady computations are performed using a dual time stepping approach [3]. For FSI applications, the CFD computational domain deforms according to the solid displacement. Hence a mesh deformation module based on Radial Basis Function (RBF) interpolation algorithm [4] is included in the fluid solver. Additionally, it requires the resolution of Arbitrary Lagrangian-Eulerian (ALE) formulation of the Navier-Stokes equations stated by Eq. (1) in which \( U \) is the array of conservative variables, \( \vec{F}_c, \vec{F}_v \) are respectively the convective and viscous fluxes, \( Q \) is the array of source terms and \( \dot{V}_g \) is the mesh velocity.
\[
\frac{d}{dt} \int_{\Omega} U d\Omega + \int_{\Gamma} \left( \vec{F}_{\text{c}} - \vec{F}_{\text{t}} - \vec{U} \vec{V} \right) \cdot dS = \int_{\Omega} Q d\Omega
\]  

(1)

2.2 Structural deformation

The structure is represented by its natural frequencies and mode shapes. These are determined outside the flow solver and prior to any CFD computation, either by computation with a FEM structure solver or by experiments. Using these structural properties, the elastic body deformation under the action of the fluid loads is computed by a structural solver integrated inside the flow solver.

As the mode shapes are defined on a Finite Element mesh, some interpolation issues between structure and fluid data will occur [5]. The mode shapes are interpolated onto the fluid mesh prior to the coupled computation as suggested by Sayma et al. [6]. A RBF interpolation method is used.

Assuming a linear behavior, a Rayleigh damping, a stiffness not influenced by the frequency and using the normalization \( \Phi^T M \Phi = I \) for the mode shapes, the structure is characterized by a set of uncoupled modal equations:

\[
\frac{\partial^2 q_k}{\partial t^2} + 2\xi_k \omega_k \frac{\partial q_k}{\partial t} + \omega_k^2 q_k = \tilde{f} \phi_k^r
\]  

(2)

The fluid load \( \tilde{f} \) includes pressure and viscous forces acting on the structure.

With the assumptions made, a Complementary Function and Particular Integral (CF&PI) method can be used for the resolution of the modal equation for each mode [7]. This method appears to be more accurate than the usual Newmark’s algorithm [8] and the iterative solution is directly obtained from:

\[
\begin{bmatrix}
q_{n+1} \\
\dot{q}_{n+1}
\end{bmatrix} =
\begin{bmatrix}
a_{11} & a_{12} \\
a_{21} & a_{22}
\end{bmatrix}
\begin{bmatrix}
q_n \\
\dot{q}_n
\end{bmatrix} +
\begin{bmatrix}
b_{11} & b_{12} \\
b_{21} & b_{22}
\end{bmatrix}
\begin{bmatrix}
f_n \\
f_{n+1}
\end{bmatrix}
\]  

(3)

The coefficients \( a_{ij} \) and \( b_{ij} \) depends on the integration time step, the natural frequency and the damping ratio of the mode.

For validation, the integration method is applied to the equation:

\[
\frac{\partial^2 q}{\partial t^2} + 2\frac{\partial q}{\partial t} + 3q = 10\cos\left(\pi t + \frac{\pi}{4}\right)
\]  

(4)

This equation is quite similar to Eq. (2) and allows an analytical solution. Therefore, numerical and analytical results are compared on Figure 1. A numerical time step of 0.1 s has been selected here.

The differences between the analytical and the numerical solutions are illustrated on Figure 2. The error is defined by:

\[
\text{error} = q_{\text{analytical}} - q_{\text{CF&PI}}
\]  

(5)

In order to investigate the accuracy of the method, two different time step sizes are used. After the transient part, the amplitude of the error is about 1% of the amplitude of the solution for an integration time step size of 0.1 s. When dividing the time step size by a factor two, the amplitude of the error is decreased by a factor four. Hence, the method is second order accurate in time.
The resolution of Eq. (2) gives the generalized displacements corresponding to each mode. The physical structure deformation can be retrieved from these displacements by:

\[ \tilde{u} = \sum_{k=1}^{N_{mode}} q_k \tilde{\phi}_k \]  

(6)

2.3 Coupling algorithm

The coupling between the fluid and the structure is ensured by a staggered approach, illustrated on Figure 3. Starting with an initial flow solution, ideally the flow solution of a computation without coupling, the aerodynamic loads are computed. These loads are applied in the resolution of the modal equations for each mode providing their generalized displacement from which the structural deformation is retrieved. Knowing the new structural shape, the CFD mesh is deformed and the Navier-Stokes equations are solved. This leads to a
new flow solution that is used for the next time step calculation in an iterative process. As the equilibrium between the structure and the fluid is not fully ensured at the end of each iteration, the approach is typical for a weak coupling method.

![Flowchart of the coupling algorithm](image)

**3 APPLICATION**

**3.1 Vortex induced vibration beam**

The first application is related to a clamped beam oscillating under the action of von Karman vortices. The computation domain is illustrated in Figure 4. The flexible beam is characterized by a Young modulus of $2 \times 10^5$ Pa, a Poisson’s ratio of 0.35 and a density of $2 \times 10^3$ kg/m$^3$. The first five deformation modes are computed with the structural solver Abaqus [9]. Their natural frequencies are equal to 0.7, 4.2, 11.7, 22.9 and 37.7 Hz. The fluid is incompressible air ($\rho = 1.18$ kg/m$^3$, $\mu = 1.82 \times 10^{-5}$ Pa s). The laminar flow conditions correspond to a Reynolds number of 204. The initial condition is a pseudo-steady solution with a rigid beam.

![Vortex induced vibration beam](image)
Due to the presence of the rigid square, Von Karman vortices are shed along the beam. It induces pressure variations which lead to the deformation of the flexible structure. Figure 5 depicts the gauge pressure field and some streamlines corresponding to the maximal deflection of the beam. Due to the high flexibility of the beam, a small pressure difference is sufficient to produce relatively large deformations.

![Gauge Pressure](image1)

**Figure 5 – Instantaneous deformation of the beam at t = 9.1 s**

The beam’s tip displacement in the y direction calculated with the modal approach is depicted on Figure 6. Only, the first four structural modes are used for the computation. The effect of the fifth mode will be discussed afterward. The integration time step is 0.01 s. For comparison, the results obtained with a full order method using the coupling software MpCCI [1] and the structural solver Abaqus are also plotted. Both results are in excellent agreement. The frequency of the periodic oscillations is 0.84 Hz and its amplitude is equal to 0.022 m. These results are in accordance with those obtained by Hübner et al. with a monolithic method [10].

![Tip motion](image2)

**Figure 6 – Tip motion of the beam**

The frequency spectrum of the tip displacement over the three last oscillations is shown on Figure 7. Two peaks can be observed corresponding to the frequencies of the first, largely predominant, and the second modes. The effect of this last one can be seen on the tip motion.
curve which is not purely sinusoidal after the transient step. The contributions of the third and fourth modes are negligible.

Figure 7 – Contribution of each mode to the tip’s displacement of the beam

It has to be noted that the integration time step size must be chosen in accordance with the natural frequency of the highest structural mode. If there are not enough integration points by period, the resolution of Eq. (3) may diverge. It is the case if the same computation is performed with five modes instead of four. The vibration period of this mode is equal to 0.0265 s. Hence an integration time step of 0.01 s is too large for its proper resolution. By reducing the time step, the fifth mode can be accurately solved (Figure 8). A guideline is that the integration time step size should be less than one third of the period of the highest mode.

Figure 8 – Divergence of fifth mode with inappropriate integration time step size
3.2 Compressor stage

The second application is related to a rotor-stator compressor stage. The rotor has 16 blades. In order to reduce the computation domain, the number of stator blades is set to 32. Hence only one blade channel is meshed for the rotor and two for the stator. The computational domain depicted on Figure 9 is meshed with 146,000 nodes.

![Figure 9 – Geometry and mesh used for the compressor stage](image)

The deformation of the stator blades due to the passing rotor wakes is investigated. Each stator blade is free to deform independently of the other one. The blades are represented by their first ten vibration modes. Their natural frequencies are listed in Table 1.

<table>
<thead>
<tr>
<th>Mode index</th>
<th>Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1,168</td>
</tr>
<tr>
<td>2</td>
<td>2,215</td>
</tr>
<tr>
<td>3</td>
<td>4,918</td>
</tr>
<tr>
<td>4</td>
<td>5,786</td>
</tr>
<tr>
<td>5</td>
<td>7,202</td>
</tr>
<tr>
<td>6</td>
<td>8,980</td>
</tr>
<tr>
<td>7</td>
<td>9,870</td>
</tr>
<tr>
<td>8</td>
<td>10,986</td>
</tr>
<tr>
<td>9</td>
<td>12,448</td>
</tr>
<tr>
<td>10</td>
<td>17,174</td>
</tr>
</tbody>
</table>

The fluid is air considered as perfect gas. The averaged inlet Mach number is equal to 0.45. The inlet Reynolds number equals 660,000. The Spalart and Allmaras model [11] is used for the turbulence.

Two rotor rotation speeds, 20,000 RPM and 18,442 RPM, are considered in order to investigate their influence on the blade deformation. The first one corresponds to a Blade Passing Frequency (BPF) of 5,333 Hz. Hence, the stator blades are impacted by rotor wakes at a frequency between the third and the fourth structural natural frequency. The BPF related to the second rotation speed is 4,918 Hz. Such frequency induces an unsteady perturbation of
the stators blades at the third mode’s resonance frequency. For each rotation speed, the unsteady computations are performed with 80 time steps per rotor blade passing period. It allows having more than 20 time steps over the highest mode’s period ensuring its correct computation.

The mid-span relative Mach number fields are shown on Figure 10 for the two rotation speeds. As the compressor is transonic, a weak shock is observable at the rotor inlet and a stronger shock between the stator blades.

The tip displacements at trailing edges of one stator blade are depicted on Figure 11. The plot window corresponds to five rotor blade passing periods. As can be seen the rotation speed of the rotor influences the evolution of the stator’s blade deformation. The curves show that the blade almost oscillates once over the time window for a 20,000 RPM rotation speed while at 18,442 RPM, the blade tends to vibrate with a higher frequency as five local maxima are observable.
Figure 12 shows two snapshots of instantaneous deformation fields of the stator’s blades. At 20,000 RPM, the deformation corresponds to a simple bending mode whereas we have a more complex shape when the passing blade rotates at 18,442 RPM.

![Figure 12 – Deformation of stator blades at 20,000 RPM (left) and 18,442 RPM (right)](image)

The Fourier transform of the tip motions of the stator blades are depicted on Figure 13 for the two rotation speeds considered. Several peaks can be observed corresponding to different relevant frequencies. For both rotation speeds, the main peak is observed around 1,180 Hz, corresponding to the natural frequency of the first vibration mode. A second major peak is observable around 2,170 Hz, close to the frequency of the second vibration mode. For the rotation speed of 20,000 RPM, three peaks of smaller amplitudes appear at 4,900, 5,330, and 5,760 Hz. They correspond respectively to the third vibration mode, the rotor BPF and the fourth vibration mode. At 18,442 RPM, a single peak of much more intensity appears at the frequency of the third mode also equals to the BPF.

![Figure 13 – Fourier transform of the tip motion of the stator blade](image)
These curves show that the deformations of the stator blades are mainly governed by the vibration modes of the structure. If the rotor BPF doesn’t coincide with a natural frequency, it induces a small peak in the vibration spectrum. However, when the BPF equals a natural frequency, a resonance occurs and induces a peak of higher amplitude.

3.3 AGARD 445.6 wing

This last application is related to the flutter of a wing experimentally studied by Yates [12]. The wing is formed by a NACA65A004 airfoil extruded with a sweep angle of 45° at the quarter chord line and a taper ratio of 0.6. The structural modal data are directly taken from Yates’ publication. The fluid is air considered as perfect gas. The Mach numbers are ranging from 0.5 to 1.14. The root chord based Reynolds numbers vary from $0.46 \times 10^6$ to $2.35 \times 10^6$. The turbulence is modeled with the Spalart-Allmaras model [11] and extended wall functions [13]. Two fluid domain meshes are used for this application. The first one (named Grid 2) is very coarse with 58,400 nodes. The second one (named Grid 1) is more refined and has 403,200 nodes (Figure 14).

![Figure 14 – Mesh used for the AGARD 445.6 wing](image)

The flutter conditions are identified by using the same approach as Pahlavanloo [14]. Unsteady FSI computations are performed with different free stream flow conditions. An integration time step of $5 \times 10^{-4}$ s is used. A lift perturbation is imposed during the first 0.05 s. Then the tip motion of the wing is monitored. A decreasing amplitude of oscillations indicates that we are below the flutter limit. An increasing amplitude of oscillations indicates that we are above the flutter limit. When the oscillations have constant amplitude, the flutter limit is reached (Figure 15 left). The static pressure is used to modify the flow conditions keeping the Mach number constant in the search of the flutter limit.

The Fourier transforms of the tip displacements (Figure 15 right) show that the deformation are characterized by a major peak at a frequency ranging from 12 Hz to 20 Hz for the different free stream Mach numbers considered. It has to be noticed that it doesn’t corresponds to the first structural vibration frequency, 9.6 Hz and 38.2 Hz for the two first modes [12]. With increasing Mach numbers, the influence of structural modes becomes of importance with clearly observable variations in amplitude at the structural frequencies.
Figure 15 – Tip motion in lift direction at flutter limit for the different Mach number

Figure 16 depicts the comparison between the undeformed and the deformed wing at the maximum tip displacement for the configuration at Mach 0.5. A scale factor is applied in the y-direction in order to enhance the visualization of the differences between both shapes. As can be seen, the deformation is a combination of bending and torsion. This last one induces a modification of the flow incidence along the span direction. This is observable with the Mach number field depicted on Figure 17. As a consequence, it induces a lift variation with the deformation, playing a major role in the aeroelastic coupling between the fluid and the structure.

Figure 16 – Original (grey) and deformed (red) wing at maximum displacement (dimensions are scaled by a factor five in the lift direction)

Figure 17 – Mach number field at 10%, 50% and 95% span
The flutter condition is represented by the Flutter Speed Index (FSI):

\[
FSI = \frac{V_f}{L_{\text{ref}}} \frac{\Omega_{\alpha}}{\sqrt{\mu}}
\]

(7)

\[
\mu = \frac{m_r}{\rho_f \Omega_{\text{ref}}}
\]

It includes the ratio between the flow velocity at flutter \( V_f \) and the frequency of the first torsion mode \( \omega_{\alpha} \). A ratio \( \mu \) between the wing mass and the fluid density at flutter condition appears too.

The computed FSI are depicted on Figure 18. For all Mach numbers considered, the results obtained with the coarsest mesh are higher than those computed with the finest mesh. The gap showing the influence of the mesh is approximately constant and corresponds to about 0.0175 FSI. For subsonic flow the results with both meshes are very close to the experimental points. With increasing Mach numbers, the reduction of the FSI is well captured by the numerical computations. However the results obtained with the finest mesh underpredict the experimental values. Such underprediction of FSI value at Mach 0.9 and Mach 0.95 is consistent with other studies on the same case [14], [15]. Same behavior is observed for the results on Grid 2 although they appear to be closer to the experimental data due to the constant gap with the results on Grid 1. When reaching supersonic free stream flow, we can see an increase of the experimental FSI. However the numerical results at Mach 1.07 have a larger deviation from the data. The current method doesn’t succeed to reproduce the experimental behavior for the early supersonic Mach numbers. For the point at Mach 1.14, the increase of FSI is well captured even if the calculated FSI are again slightly lower than the experimental value. Such behavior is different from those observed by Pahlavanloo [14] or Beaubien and Nitzche [16] who have numerical values higher than the experimental ones. However, the mode shapes from Yates are not used in the two studies referenced above. Furthermore, the CFD calculations are only performed in Euler and laminar mode, whereas, the present study is based on RANS equations including Spalart and Allmaras turbulence model. The use of such a turbulent closure model should be of importance for transonic application.
The ratios between the frequency of the tip motion and the frequency of the first torsion mode are plotted on Figure 19. The same observations as for the FSI can be formulated.

![Figure 19 – Frequency ratio of the AGARD 445.6 wing](image)

### 4 CONCLUSION

A method for FSI prediction has been developed. It uses a reduced order model based on a modal synthesis for the structure. The computation of the structure deformation is directly performed by the flow solver in order to avoid data interpolation issues between structural and fluid meshes. The modal equations are solved by a second order time accurate complementary function and particular integral method.

The method has been successfully applied with success to three different applications. The computed vibration of a simple clamped beam under the action of von Karman vortices are the same as those computed by other numerical methods. In a second application, the method is applied on a more complex compressor stage configuration. The deformations of stator blades due to the wakes of passing rotor blade are calculated and the resonance between the rotation speed and a structural mode is highlighted. Finally, the method is used for the computation of flow conditions leading to flutter of a wing. It provides results in accordance with the experimental data.

Stability issues have been found when the integration time step size is too large regarding the frequency of the highest mode used for the structural computation. As a rule it seems that the integration time step size must be smaller than one third of this frequency.

The method introduced in this paper appears to be a simple and efficient approach for the FSI prediction even for complex configuration such as compressor stage. It will be used in a future work for the aeroelastic analysis of structures subject to load cases such as dynamic gust or manoeuvres for an aircraft.

A further extension would be the development of a strong coupling approach ensuring the equilibrium between the fluid and the structure at each time step. A second extension will be the coupling of this FSI approach with the Nonlinear Harmonic method [17] reducing considerably the CPU cost of aeroelastic computations.
5 ACKNOWLEDGMENTS

The research leading to these results has received funding from the European Community’s Seventh Framework Programme (FP7 / 2007-2013) under a grant agreement number 233665. FFAST (Future Fast Aeroelastic Simulation Technologies) is a collaborative research project aimed at developing, implementing and assessing a range of numerical simulation technologies to accelerate future aircraft design. Advances in critical load identification and reduced order modelling methods will potentially provide a step change in the efficiency and accuracy of the dynamic aeroelastic loads process. The partners in FFAST are: University of Bristol, INRIA, CSIR, TU Delft, DLR, IRIAS, University of Liverpool, Politecnico di Milano, NUMECA, Optimad Engineering, Airbus, EADS-MS and IITP.

6 REFERENCES


[16] BEAUBIEN R. J., NITZSCHE F. *Time and Frequency Domain Flutter Solutions for the AGARD 445.6 Wing*, 2005, Paper IF-102, IFASD.