AEROELASTIC FORCED RESPONSE OF A BLADED DRUM FROM A LOW PRESSURE COMPRESSOR

JULIEN LAMOUROUX

Master of Science Thesis
Stockholm, Sweden Year
AEROELATIC FORCED RESPONSE OF A BLADED DRUM FROM A LOW PRESSURE COMPRESSOR

Julien Lamouroux

Master of Science Thesis
KTH School of Industrial Engineering and Management
Energy Technology EGI-2016-013MSC EKV1127
Division of Heat and Power Technology
SE-100 44 STOCKHOLM
Abstract

The purpose of this master thesis is to provide a reliable methodology to predict the forced response of a monoblock bladed drum from a low pressure compressor. Pre-test forced response calculations have already been made at Techspace Aero in 2013. Now that experimental data are available, the methodology has to be adapted to ensure the best numerical-experimental correlation possible. The final goal is that, at the end of the thesis, engineers at Techspace Aero will be able to launch reliable forced response simulations within a short amount of time. For the sake of confidentiality, some data are not revealed, such as the engine name, some numerical values (forced response, aerodynamic damping, frequency of the mode etc...) and axis scales.

In this paper, the study focuses on the forced response of a rotor blade from the first stage under the excitation from the upstream stator. The mode under investigation is the 2S2, the one that responded during the experiment. The TWIN approach is used to compute the forced response of the rotor blade. With this approach, a steady stage computation has first to be carried on as an initialization. Then two unsteady computations are necessary. The first, without blade motion, will provide the excitation aerodynamic forces. The aerodynamic damping will be extracted from the second one, where the motion of the blade is imposed on a given eigenmode. The forced response can then be computed with these two results and some additional structural data. The results will be compared to the experimental value.
Acknowledgements

I would first like to thank Cédric Cracco, aerodynamic engineer and my supervisor at Techspace Aero, who steered my work during the past 6 months. He took a lot of his time to answer my numerous questions and was always available to provide me with good advice and help. He put a great effort into following day after day the progress of my work, and was constantly up to date with the issues I was facing.

I would also like to thank all the engineers in the aerodynamic department at Techspace Aero who made this thesis a very rewarding experience to me, both from a professional and personal perspective. I felt very well integrated in this team of young and skilled engineers. Each one of them helped me in his own field of expertise.

I am also grateful to the engineers of the mechanical department I worked with. Even though they had plenty of work on other projects, they provided me the data that I needed in a very comprehensive and effective way.

I would also like to show my greatest gratitude to my supervisor at KTH, Nenad Glodic, professor at the Division of Heat and Power Technology and THRUST Program Director. He gave me very useful advice on the writing of this paper.

Finally, I would like to thank my examiner, Paul Petrie Repar, professor and researcher at the Division of Heat and Power Technology at KTH, and expert in turbomachinery aeroelasticity, for taking time to evaluate my work. I hope you will find this paper interesting.
# Table of contents

1. Introduction ...................................................................................10
2. Objectives ......................................................................................12
3. Theoretical background ....................................................................13
   3.1. Dynamical behavior of a bladed drum .......................................13
       3.1.1. Mode shapes and nodal diameters ................................13
       3.1.2. Resonance of a bladed drum .........................................14
           3.1.2.1. Campbell Diagram ................................................14
           3.1.2.2. ZZENF Diagram ...............................................15
           3.1.2.3. Zig-Zag Diagram ..............................................16
       3.2. Aeroelastic forced response methods .....................................17
           3.2.1. State of the art .....................................................17
           3.2.2. The TWIN method ...............................................18
               3.2.2.1. Assumptions ................................................18
               3.2.2.2. Analytical formulation ..................................19
                   3.2.2.2.1 Equation of motion ......................................19
                   3.2.2.2.2 Practical calculation of the GAF ..............20
                   3.2.2.2.3 Calculation of the forced response ..........20
4. TWIN Computation full procedure .................................................22
   4.1. Approach ..............................................................................22
   4.2. Selection of the aeroelastic response of interest .........................23
   4.3. Structural Analysis ..................................................................23
   4.4. Throughflow analysis ............................................................24
       4.4.1. The throughflow equations .........................................25
       4.4.2. S1SI Throughflow ....................................................25
       4.4.3. S1SI NS3D steady computation ..................................26
       4.4.4. S1SI Throughflow Reconstitution ...............................28
4.4.5. S1R2 Throughflow Extraction .................................................. 28
4.4.6. S1R2 Throughflow Optimization .............................................. 29
4.5. S1R2 NS3D Steady Computation .................................................. 30
  4.5.1. CFD Meshing ........................................................................ 30
    4.5.1.1. Node Distribution Along the radial direction .................. 30
    4.5.1.2. Blade to Blade (B2B) mesh configuration ...................... 30
    4.5.1.3. Mesh quality .................................................................. 31
    4.5.1.4. Modification of the blocks topology ............................. 31
  4.5.2. Boundary conditions .............................................................. 33
  4.5.3. Initialization ......................................................................... 34
  4.5.4. Configuration of the numerical model ................................... 35
  4.5.5. Convergence ......................................................................... 35
4.6. S1R2 NS3D Unsteady Computations ............................................. 37
  4.6.1. Allowing mesh motion ............................................................ 37
  4.6.2. Boundary conditions .............................................................. 37
  4.6.3. Chorochronicity .................................................................... 37
  4.6.4. Time step setup and number of iterations ............................ 38
  4.6.5. Initialization ......................................................................... 39
  4.6.6. Mode file generation .............................................................. 39
    4.6.6.1. First projection ............................................................... 40
    4.6.6.2. Second projection ........................................................... 41
    4.6.6.3. Checking the projection ............................................... 42
  4.6.7. Configuration of the numerical model ................................... 42
  4.6.8. Convergence ......................................................................... 42
  4.6.9. Calculation of the forced response ......................................... 43
    4.6.9.1. Extraction from the computation with fixed blade .......... 43
    4.6.9.2. Extraction from the computation with prescribed harmonic motion .... 44
    4.6.9.3. Forced response ............................................................. 44
4.6.10. Conversion in percentage of endurance limit ........................................... 45

5. Results ............................................................................................................. 47
  5.1. Throughflow analysis .................................................................................. 47
  5.2. Influence of time discretization .................................................................. 48
  5.3. Influence of the number of transmitted harmonics ..................................... 48
  5.4. Linearity assumption verification .................................................................. 49

6. Discussion ....................................................................................................... 49

7. Conclusion ....................................................................................................... 52

8. Future work ..................................................................................................... 53

References ........................................................................................................... 54
Table of Figures

Figure 1 – Conventional booster [20]..............................................................................10
Figure 2 – BluM® booster with composite carter [20].........................................................10
Figure 3 – First modes for a plate fixed on one end ...............................................................13
Figure 4 – First flexion mode with 8 nodal diameters............................................................14
Figure 5 – Campbell diagram. Taken from [9]......................................................................15
Figure 6 – ZZENF diagram at a given rpm [20].................................................................16
Figure 7 – Zig-Zag diagram for a bladed disk with 36 blades. Taken from [8]..............17
Figure 8 – TWIN procedure.................................................................................................22
Figure 9 – Throughflow analysis..........................................................................................24
Figure 10 – S1SI meridian plane .........................................................................................26
Figure 11 – S1SI 3D computational domain........................................................................26
Figure 12 – Booster performance map................................................................................28
Figure 13 – S1R2 meridian plane .........................................................................................28
Figure 14 – Static pressure radial distribution. Red: S1SI Throughflow, Doted Line:
tolerance, Black : S1R2 Throughflow unoptimized, Blue: S1R2 Throughflow optimized
...........................................................................................................................................29
Figure 15 – Flow paths in the meridian plane ..............................................................30
Figure 16 – B2B mesh............................................................................................................31
Figure 17 – Blocks topology modifications.........................................................................32
Figure 18 – Final 3D block topology....................................................................................32
Figure 19 – Color code for boundary conditions..............................................................33
Figure 20 – Splitter [3].........................................................................................................33
Figure 21 – Momentum azimuthal average at the mixing plane...........................................34
Figure 22 – Conservative variable and RHS residuals convergence..................................36
Figure 23 – Mass flow defect and mass flows convergence.................................................36
Figure 24 – Faces allowing mesh motion.............................................................................37
Figure 25 – Mode projection steps .....................................................................................40
Figure 26 – Reduced mesh on the pressure side.................................................................40
Figure 27 – FEM mesh..........................................................................................................41
Figure 28 – Modal deformation on the pressure side (above) and the suction side
(below) on the FEM mesh, the reduced FEM mesh and the CFD mesh...........................42
Figure 29 – Residuals and mas flow convergence for unsteady computation...............43
Figure 30 – Local excitation forces on the suction side (left) and pressure side (right) ..........................................................................................................................................43
Figure 31 – Local aerodynamic damping on the suction side (left) and pressure side (right) ..........................................................................................................................................44
Figure 32 – Normalized evolution of the forced response with respect to the normalized rotational speed ...........................................................................................................................................45
Figure 33: Goodman diagram .................................................................................................................................................................................................46
Figure 34: Normalized performance map .................................................................................................................................................................47
Figure 35: Normalized forced response convergence with time discretization .....................................................................................................................48
Figure 36: Turbulent kinetic energy at rotor tip ..........................................................................................................................................................50
Figure 37: Turbulent kinetic energy on the carter ....................................................................................................................................................50
Figure 38: Sensitivity of the forced response with respect to the Rayleigh structural damping coefficient ..................................................................................................................................................51
Nomenclature

Latin symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BPF</td>
<td>Blade passing frequency</td>
</tr>
<tr>
<td>C</td>
<td>Damping matrix</td>
</tr>
<tr>
<td>$D_h$</td>
<td>Hydraulic diameter of the inlet</td>
</tr>
<tr>
<td>$dt$</td>
<td>Time step</td>
</tr>
<tr>
<td>$F$</td>
<td>Unsteady force</td>
</tr>
<tr>
<td>$f_k$</td>
<td>Frequency of mode k</td>
</tr>
<tr>
<td>$h_0$</td>
<td>Total enthalpy</td>
</tr>
<tr>
<td>$K$</td>
<td>Stiffness matrix</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$l$</td>
<td>Turbulent characteristic length</td>
</tr>
<tr>
<td>$M$</td>
<td>Mass matrix</td>
</tr>
<tr>
<td>$N_r$</td>
<td>Number of rotor blades</td>
</tr>
<tr>
<td>$N_s$</td>
<td>Number of stator blades</td>
</tr>
<tr>
<td>$n$</td>
<td>Number of nodal diameters</td>
</tr>
<tr>
<td>$P$</td>
<td>Wetted aera</td>
</tr>
<tr>
<td>$p_t$</td>
<td>Static pressure</td>
</tr>
<tr>
<td>$p_t$</td>
<td>Total pressure</td>
</tr>
<tr>
<td>$Q$</td>
<td>Mass flow</td>
</tr>
<tr>
<td>$r$</td>
<td>Radial coordinate</td>
</tr>
<tr>
<td>$S$</td>
<td>Cross section area</td>
</tr>
<tr>
<td>$s$</td>
<td>Entropy</td>
</tr>
<tr>
<td>$T_t$</td>
<td>Total temperature</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$th$</td>
<td>Throttling coefficient</td>
</tr>
<tr>
<td>$u$</td>
<td>Nodal displacement vector</td>
</tr>
<tr>
<td>$u_f$</td>
<td>Friction velocity</td>
</tr>
<tr>
<td>$V$</td>
<td>Velocity of the fluid</td>
</tr>
<tr>
<td>$y$</td>
<td>Wall distance</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Non-dimensional wall distance</td>
</tr>
<tr>
<td>$z$</td>
<td>Axial coordinate</td>
</tr>
</tbody>
</table>

Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>Amplitude factor</td>
</tr>
<tr>
<td>$\alpha_{aero}$</td>
<td>Amplitude factor of the prescribed harmonic motion</td>
</tr>
<tr>
<td>$\alpha_{fr}$</td>
<td>Amplitude factor of the forced response</td>
</tr>
</tbody>
</table>
$\delta$ Non-dimensional aerodynamic damping
$\varepsilon$ Dissipation rate
$\theta$ Angle in the cylindrical system
$\nu$ Cinematic viscosity
$\Pi$ Pressure ratio
$\sigma$ Stress
$\phi_k$ Modal deformation of mode k
$\Omega$ Engine rotational speed (round per minute)
$\omega$ Engine rotational speed (rad/s) or the Specific dissipation rate
$\omega_k$ Pulsation of mode k

Abbreviations

B2B Blade to blade
Booster Low pressure compressor
BPF Blade Passing Frequency
CFD Computational Fluid Dynamic
CFL Courant-Friedrichs-Lewy number
EO Engine Order
FE Finite Element
FEM Finite Element Model
FFT Fast Fourier Transform
GAF Generalized Aerodynamic Force
HCF High Cycle Fatigue
LE Leading Edge
NS3D 3D Navier Stokes
ODE Ordinary Differential Equations
RHS Right Hand Side
rpm rounds per minute
S1SI Refers to the full booster
S1R2 Refers to the first stage (stator+rotor)
TA Techspace Aero
TE Trailing Edge
1. Introduction

For both economic and ecological purposes, aircraft engine manufacturers put a great effort into reducing the fuel consumption of their new engines. The LEAP, the last engine developed by CFM International that will be mounted on the Boeing 737 MAX and the Airbus A320neo, is already the most sold aircraft engine in development of all time (already more than 10,000 orders and commitments). Its fuel consumption is lesser than its predecessor’s one, the CFM56, and the productions of noise and greenhouse gas have been decreased. What will be the specificities of the next generation engines ? While some projects such as ENOVAL [18] are developing high loaded boosters, the BluM® project [19] aims at reducing its weight by creating a monoblock rotor. This project is driven in parallel with the idea to implement carter in fibers composite materials (Figure 2). In a conventional booster (see Figure 1), the rotor blades are placed and fixed one by one around the drum. In a BluM® configuration (see Figure 2), the blades are welded by friction on the drum, forming a single solid piece. The elimination of the fixation systems (screw, bolts, etc..) should give a significant mass improvement of about 20%.

Figure 1 – Conventional booster [20]

Figure 2 – BluM® booster with composite carter [20]
Another consequence is the decrease of the mechanical damping due to the loss of the mechanical friction between the rotor blade and its attachment. A weak structural damping promotes flow induced vibrational problems such as forced response and flutter [13]. Forced response is a dynamical aeroelastic phenomenon where the unsteady periodic excitation produced by the upstream stator row makes the rotor blades enter a resonance vibration mode that can lead to HCF or even worse, engine failure. It is then a priority in the BluM® project to investigate and control the aeroelastic behavior of the monoblock rotor to ensure that the vibration amplitude will not trigger stresses that are above certification criteria.

This paper aims at presenting a reliable methodology to launch forced response computations of the first rotor under the excitation of the upstream stator, using the recent TWIN approach. The reliability of the results is measured through numerical-experimental matching. For the sake of confidentiality, this paper is more focused on the methodology since the actual numerical values of the results cannot be disclosed.
2. Objectives

- The main objective of this master thesis is to achieve the best matching possible between experimental and numerical results. Aeroelastic forced response experiments have been carried out on a bladed drum in 2014 at Liers, Belgium. The goal is to check if the tools available at TA are able to provide a reliable numerical solution for forced response.

- The difference between experimental and numerical results must be quantified. Moreover, potential improvements should be identified keeping in mind this kind of computations is meant to be used within an industrial process (limit in time and CPU resources).

- The procedure to launch such computations has to be described in details through several technical notes that will remain at TA. The goal is to make them as clear and detailed as possible so that future engineers at TA will be able to do the same within an acceptable amount of time.

- Some step of the procedure such as the meshing can be fastened by the use of scripts requiring few inputs. Such scripts will be developed during the thesis.
3. Theoretical background

3.1. Dynamical behavior of a bladed drum

3.1.1. Mode shapes and nodal diameters

The blades mode shapes are similar to the ones of a 2D plate fixed on one end. Figure 3 shows the first modes of such a structure.

![Mode Shapes Diagram]

Figure 3 – First modes for a plate fixed on one end

For a bladed drum mode shape, all the blades have the same modal deformed shape but with a phase difference. It is then necessary to introduce the notion of nodal diameter. A mode shape with \( n \) nodal diameters will have \( 2n \) phase changes around the drum. From that, it can be deduced that a rotor with \( N_r \) blades will have a maximum of:

- \( n = \frac{N_r}{2} \) nodal diameters if \( N_r \) is even
- \( n = \frac{(N_r-1)}{2} \) nodal diameters if \( N_r \) is odd

Moreover, some modes are qualified as “pure blade modes” if a certain amount (this limit is arbitrary) of strain energy is in the blades, as “pure drum modes” if the energy is mainly in the drum, and as “coupled modes” if the energy is in both. In this study, only pure blade modes will be investigated. Figure 4 displays the first flexion pure blade mode with 8 nodal diameters.
3.1.2. Resonance of a bladed drum

In case of a rotating structure presenting a condition of cyclic symmetry, two conditions must be satisfied for resonance to occur [15,16]:

- The excitation frequency must be equal to one of the natural eigenfrequencies of the structure. This condition gives the operating points at which resonance is likely to occur.
- The excitation pattern of the structure has to fit to the modal deformation. This condition is called appropriation. It is used to identify the operating points at which resonance is possible within the set of operating points at which it is likely to occur.

3.1.2.1. Campbell Diagram

The Campbell’s diagram is a tool that enables to see the operating points which satisfies the first condition. It displays the evolution of the different eigenmodes natural frequencies and the excitation frequency with respect to the engine rotational speed. Indeed, due to the stiffening effect of the centrifugal force, there is an evolution of the eigenfrequencies. The gyroscopic effects should also be taken into account, however, they are assumed negligible in this study.

A rotor blades experiences an unsteady and periodic load due to the disturbances generates by the stator. The excitation frequency is then a multiple of the engine rotational speed.
\[ \omega = \frac{N_s \Omega}{60} \] (1)

where \( \omega \) is the excitation frequency in Hz, \( N_s \) the number of upstream stators and \( \Omega \) the rotational speed in rpm. \( N \) is also called the engine order (EO). Figure 5 gives an example of a Campbell diagram.

![Campbell diagram](image)

*Figure 5 – Campbell diagram. Taken from [9]*

The crossing between a natural frequency and an excitation line indicates an operating point at which resonance is likely to occur. In order to check if resonance is actually possible or not, another tool is required.

### 3.1.2.2. ZZENF Diagram

ZZENF (Zig-Zag shaped Excitation line in the Nodal diameter versus Frequency diagram) diagrams have been first used at Stal-Laval Turbin (Finspong, Sweden) by J. Wildheim. It is meant to check whether the exciting pattern corresponds to the modal deformation. This directly refers to the notions of nodal diameters and cyclic symmetry.

A ZZENF diagram is used to illustrate the evolution of the eigenfrequencies of the structure as a function of the number of nodal diameters. The excitations lines are also represented. This diagram is done for a given rotational speed since the slope of the excitation line depends on it as well as the evolution of the eigenfrequencies. Spatial aliasing occurs at \( n = 0 \) and \( n = N_r/2 \) if \( N_r \) is even, and \( n = (N_r-1)/2 \) if \( N_r \) is odd. It means that, for a structure with 50 blades for example, an excitation of 70 EO will excite the modes with 20 nodal diameters.
Crossings between the mode and the excitation lines that correspond to an integer value of $n$ mean a possible resonance (red circle in Figure 6). If the crossing occurs on an excitation line with a positive slope, the mode will “turn” in the same direction as the excitation force (in the frame of reference of the rotor, the excitation of the stator is turning), this is called a forward wave. On the contrary, if the slope is negative, it will be a backward wave.

![ZZENF diagram at a given rpm](image)

*Figure 6 – ZZENF diagram at a given rpm [20]*

On the left and upper parts of the graph, the eigenfrequencies are very sensitive to the number of nodal diameters. It indicates drum modes. Indeed, the appearance of nodal diameters in the drum will affect the stiffness and then the eigenfrequency. On the other hand, pure blade modes have almost constant eigenfrequencies.

In some regions, the excitation line is passing very close to several modes (black circles in Figure 6) which can be hazardous from a structural point a view. These regions are called veering regions and must be avoided.

**3.1.2.3. Zig-Zag Diagram**

This diagram is used to determine the number of excited nodal diameters when the excitation is known in terms of shape, i.e. the number of upstream stators in this study.
\[ n = -N_s + mN_r \]

Figure 7 – Zig-Zag diagram for a bladed disk with 36 blades. Taken from [8]

From this graph, the number of excited nodal diameters is given by

\[ n = -N_s + mN_r \]  \hspace{1cm} (2)

where

- \( N_s \) is the number of excitations per turn, i.e. the number of upstream stators
- \( N_r \) is the number of responding blades, i.e. the number of rotors
- \( m \) is an integer such as \(-\frac{N_r}{2} < n \leq \frac{N_r}{2}\)

For a given rotational speed, since the number of obstacles is known, the frequency of the excitation can directly be calculated.

3.2. Aeroelastic forced response methods

3.2.1. State of the art

Numerous methods have been developed in the past decades to study the phenomenon of forced response for turbomachinery applications. Depending on the period and the author, they are sorted in different manners. J.G Marshall and M. Imregun give an exhaustive list of methods that are sorted within two categories: the classical methods, also called Decoupled methods, where the fluid flow does not affect the structural response of the blade, and the integrated aeroelasticity methods, also called Coupled methods, which take into account the coupling between the flow and the structure [5].

Decoupled approaches are based on the aerodynamic-independency of the modes, the superposition principle and the aerodynamic damping linearity. The aerodynamic-independency means that the flow does not influence the eigenmodes of the blade in terms of shape and frequency. Therefore, the structural analysis is carried out without
taking into account the aerodynamic forces. The superposition principle states that the excitation is independent from the blade motion and vice versa. Thus, the excitation forces and the aerodynamic damping, due to the fluid unsteadiness generated by the blade motion, are calculated separately. The aerodynamic damping is usually calculated from a single row configuration where the blade has a prescribed harmonic motion, while the excitation forces are computed from a stage configuration with fixed rotor blades. Moreover, the aerodynamic damping is assumed linear with respect to the amplitude of the blade motion, which means that the amplitude of motion can be chosen arbitrary (see further).

Coupled approaches do not rely on the three assumptions described above. A coupling between the fluid and the structure needs to be done at each time step. The forced response can then be computed without considering the restrictive superposition principle.

Some studies have shown the validity of the superposition principle [7, 11, 13] while others revealed that the blade-motion-independency assumption is not always valid and that the unsteadiness due to the blade motions can be significant [4, 12]. The Coupled methods should then be used in order to get the most reliable results possible. Unfortunately, the computing time for this type of calculation is substantial, and not acceptable from an industrial point of view.

3.2.2. The TWIN method

3.2.2.1. Assumptions

The TWIN method is a new approach which does not rely on the superposition principle. Indeed, the forced response is calculated from a stage configuration where the excitations forces from the stator and the prescribed harmonic motion of the rotor blade coexist. Since the blade motion is imposed, i.e. not affected by the upstream excitation, the convergence is much faster than for a Coupled approach, which makes it very interesting for industrial applications. This method takes into account the effect of the stator wake on the aerodynamic damping, and is then able to handle nonlinearities. Indeed, in cases where the aerodynamic damping is not linear, Newtonian iterations can be performed by updating the amplitude of the blade motion (see 5.4). In this case the method is called TWIN iterative.

The TWIN method has been used in recent studies and compared to some Coupled and Uncoupled approaches [9, 6]. They seem to give similar results. Moreover it shows a very good correlation with experimental forced response tests.
3.2.2.2. Analytical formulation

3.2.2.2.1 Equation of motion

The equilibrium of the forces of a structure submitted to the external unsteady force $F$ can be written as

$$ M\ddot{u} + C\dot{u} + Ku = F(t) \tag{3} $$

where $M$, $C$ and $K$ are respectively the mass, damping and stiffness matrices and $u$ is the nodal displacements vector.

If an harmonic motion is assumed, $u$ can be expressed as follows

$$ u(t) = \alpha \phi_k e^{i\omega_k t} \tag{4} $$

with $\phi_k$ the mode of interest, $\omega_k$ the related pulsation of mode $k$ and $\alpha$ a complex weighting coefficient characterizing the amplitude factor of the deformation and the dephasing with respect to the $k$ mode.

The harmonic motion of the blade will lead to an harmonic force with the same pulsation $\omega_k$

$$ F(t) = f_k e^{i\omega_k t} \tag{5} $$

where $f_k$ is the complex first harmonic of the signal decomposition of the force by a Fast Fourier Transform (FFT).

Introducing (4) and (5) in (3) and multiplying by $\bar{\phi}_k^T$ yields to

$$ \alpha (\bar{K}_k - \omega_k^2 \bar{M}_k + i \omega_k \bar{C}_k) = \bar{\phi}_k^T f_k \tag{6} $$

where the reduced matrices $\bar{K}_k$, $\bar{M}_k$ and $\bar{C}_k$ have been introduced with $\bar{K}_k \equiv \bar{\phi}_k^T K \bar{\phi}_k$.

By definition, the right hand side of this equation represents the Generalized Aerodynamic Forces (GAF) and is equal to

$$ GAF \equiv \left( \phi_r f_r + \phi_i f_i \right) + i (\phi_r f_i + \phi_i f_r) \tag{7} $$

where the indexes $i$ and $r$ denote the imaginary and real parts.
3.2.2.2.2 Practical calculation of the GAF

The four terms of the right hand side of equation (7) can be computed from the evolution of the pressure on each cell \( p_{\text{cell}}(t) \). The pressure force can be evaluated by multiplying this scalar by the vector normal to the cell \( \vec{n}_{\text{cell}}(t) \)

\[
F_{\text{cell}}(t) = p_{\text{cell}}(t) \cdot \vec{n}_{\text{cell}}(t)
\]  
(8)

The forces \( F_{\text{cell}}(t) \) are then multiplied by \( \Phi_k^T \) to get a complex signal \( \Phi_r F(t) - i \Phi_i F(t) \). The application of a FFT on the real and imaginary parts of this signal gives the four terms \( \Phi_r f_r, \Phi_i f_i, \Phi_r f_i \) and \( \Phi_i f_r \).

3.2.2.2.3 Calculation of the forced response

In the TWIN approach, the excitation forces are superposed to the forces due to the prescribed harmonic blade motion. The GAF computed is then a combination of both forces

\[
GAF_{\text{TWIN}}(\alpha) = GAF_{\text{forcing}} + GAF_{\text{aero}}(\alpha)
\]  
(9)

Equation (6) becomes

\[
\alpha (K_k - \omega_k^2 \vec{M}_k + i \omega_k \vec{C}_k) = GAF_{\text{TWIN}}(\alpha)
\]  
(10)

The forced response is then the solution of the equation

\[
F(\alpha) = 0 \quad \text{where} \quad F(\alpha) = GAF_{\text{TWIN}}(\alpha) - \alpha (K_k - \omega_k^2 \vec{M}_k + i \omega_k \vec{C}_k)
\]  
(11)

By applying the Taylor expansion and taking into account only the first order terms, the following linear system is obtained [6]

\[
\frac{\partial F}{\partial \alpha} \bigg|_n (\alpha^{n+1} - \alpha^n) = -F(\alpha^n)
\]  
(12)

The Jacobian is very difficult to compute because of the nonlinear behavior of the Navier-Stokes equations. A finite difference formula can be used to approximate the Jacobian

\[
\frac{\partial F}{\partial \alpha} \bigg|_n = \frac{F(\alpha^n + \epsilon) - F(\alpha^n)}{\epsilon}
\]  
(13)

where \( \epsilon \) is a small value. Hence, (12) becomes

\[
\frac{F(\alpha^n + \epsilon) - F(\alpha^n)}{\epsilon} (\alpha^{n+1} - \alpha^n) = -F(\alpha^n)
\]  
(14)
The solution of this equation is given by
\[
\alpha^{n+1} - \alpha^n = \frac{GAF_{TWIN}(\alpha^n) - (\bar{K}_k - \omega_k^2 \bar{M}_k + i \omega_k \bar{C}_k) \alpha^n}{(\bar{K}_k - \omega_k^2 \bar{M}_k + i \omega_k \bar{C}_k) - \frac{GAF_{TWIN}(\alpha^n + \varepsilon) - GAF_{TWIN}(\alpha^n)}{\varepsilon}}
\]  
(15)

The non-dimensional aerodynamic damping is defined as
\[
\delta \equiv -\frac{1}{2\bar{K}_k} Im \left( \frac{GAF_{TWIN}(\alpha^n + \varepsilon) - GAF_{TWIN}(\alpha^n)}{\varepsilon} \right)
\]  
(16)

For the case where \(\alpha^0 = 0\) and \(\varepsilon = \alpha_{aero}\), the first Newton iteration of equation (15) gives
\[
\alpha_{fr} = \frac{GAF_{TWIN}(0)}{(\bar{K}_k - \omega_k^2 \bar{M}_k + i \omega_k \bar{C}_k) - \frac{GAF_{TWIN}(\alpha_{aero}) - GAF_{TWIN}(0)}{\alpha_{aero}}}
\]  
(17)

The forced response can then be calculated from a computation without blade motion and a computation with a prescribed harmonic motion of amplitude \(\alpha_{aero}\).

If the variation of the aerodynamic damping with respect to the amplitude of the blade movement \(\alpha_{aero}\) is highly nonlinear, a Newtonian iterative method is possible: the amplitude of the harmonic motion is replaced by the calculated forced response and computations are repeated until a converged solution is obtained.

The benefits of the TWIN approach compared to the Decoupled approaches are the following:

- The effect of the exciting forces on the aerodynamic damping is taken into account
- The same steady computation is used to initialized the two unsteady ones
- The same S1R2 mesh is used for the three computations
- The two unsteady computations only differ from the value of the imposed amplitude (zero in one case and non-zero in the other one)
4. TWIN Computation full procedure

4.1. Approach

Figure 8 gives an overview of the main steps of a TWIN computation. As it has been presented in the previous section, two unsteady computations are required to compute the forced response. These unsteady computations have to be initialized by a steady one in order to speed up the convergence. Before that, a throughflow analysis must be done to ensure that the boundary conditions correspond to the experimental conditions. In order to make the blade move along a given mechanical mode, this mode has to be projected from the FEM mesh to the CFD mesh. The projection is divided in two steps. Each of these steps will be detailed in the next paragraphs.

![Figure 8 - TWIN procedure](image)

The throughflows are solved with GOUX, an intern code used in SAFRAN Group (no reference available).

The CFD meshing is done with the softwares Autogrid and IGG, developed by NUMECA International. They are full automatic hexahedral structured grid generators for all types of rotating machinery.

The CFD computations are using the elsA code [17]. It is a software simulation tool developed by ONERA (the French National Aerospace Research Centre) since 1997, and in collaboration with CERFACS (European Centre of Research and Advanced Training in Scientific Computation) since 2000 and Cenaero since 2009. elsA solves the
compressible, Reynolds-averaged Navier-Stokes (RANS) equations, in integral form, in fixed or moving reference frames. Turbulence is modeled by either algebraic or transport equation models. Numerical procedure is based on a finite-volume conservative formulation on block-structured meshes. Both steady and unsteady computations can be performed.

For forced response analysis, pre-processing has to be done with CardGenerator which is a tool developed by CenAero whom main task is to generate the topological information and the description of the boundary conditions that are required during the elsA computations.

4.2. Selection of the aeroelastic response of interest

With the tools described in 3.1.2, the mechanical department is able to determine the modes and the corresponding operating points for which forced response should be investigated. However, considering the complexity of a forced response analysis, the number of candidates might have to be reduced to a couple of modes. Therefore, some new criteria are defined to make this pre-selection:

- The crossing between the mode and the excitation in the Campbell diagram should occur for high rotational speeds
- The mode should be excited by an upstream aerodynamic obstacle
- High strain energy in the blade

Since BhuM® are designed (for the moment) from an existing engine, some data from the same engine which is under study but with a conventional booster are available. Two more criteria can be defined:

- The mode should have shown a significant response (in terms of endurance limit)
- The predicted and observed resonance regimes should be close

This is the procedure that should be done in a development phase. In this study the mode of interest is the one that responded during the.

4.3. Structural Analysis

The structural analysis is meant to provide the structural eigenmode associated to its corresponding eigenfrequency. This is done by the mechanical department through two consecutive computations:
A static computation aiming at determining the pre-stresses induced by the centrifugal force, and the pressure and temperature fields
A dynamic computation accounting for these pre-stresses. The modal displacements and the superficial constraints are then computed

4.4. Throughflow analysis

In order to get the best match possible between the computations and the experimental results the boundary conditions at the booster inlet and outlet have to be chosen such that it is representative of the testing conditions. In theory, the forced response computation should be done on a full booster configuration, but the amount of time and resources required by this kind of computation is not acceptable from an industrial point of view. This is why the computational domain is restricted to the first stator S1 and the first rotor R2 (R1 is the fan). However a full booster computation is still required for two reasons:

- To compute the booster performance and check the consistency with the experiment
- To extract the boundary conditions at S1R2 inlet and outlet

The full booster computations itself needs inlet and outlet boundary conditions. They are extrapolated from a quasi-3D flow field that is computed by solving the “throughflow” problem. Figure 9 displays the different steps required before the S1R2 NS3D steady computation.

![Diagram](image-url)

*Figure 9 – Throughflow analysis*
4.4.1. The throughflow equations

As a first approximation, the flow field in a turbomachine can be assumed to be periodic and axisymmetric. All the planes that contain the engine axis are then equivalent and called the meridian planes. Under the additional hypothesis that the flow is adiabatic, inviscid and steady, the momentum equations in the cylindrical system \((r, z, \theta)\) are [3]

\[
\frac{V_r}{r} \frac{\partial V_r}{\partial r} + \frac{V_z}{r} \frac{\partial V_r}{\partial z} = 0
\]

(18)

\[
V_r \left( \frac{\partial V_z}{\partial r} - \frac{\partial V_r}{\partial z} \right) - \frac{V_\theta}{r} \frac{\partial V_r}{\partial z} = T \frac{\partial s}{\partial z} - \frac{\partial h^0}{\partial z}
\]

(19)

\[
V_z \left( \frac{\partial V_r}{\partial z} - \frac{\partial V_z}{\partial r} \right) - \frac{V_\theta}{r} \frac{\partial V_z}{\partial r} = T \frac{\partial s}{\partial r} - \frac{\partial h^0}{\partial r}
\]

(20)

where \(s\) is the entropy and \(h_0\) the total enthalpy. There are several ways to solve this system depending on which data are available. In this study, the geometry of the booster is already known and fixed, a so called “direct method” is then used to solve the throughflow problem.

The variation of the total enthalpy across a blade row is given by the Euler equation

\[
\Delta h_0 = \Delta (V_\theta r \omega)
\]

(21)

where \(\omega\) is the rotational speed of the engine.

The variation of the entropy across a blade row can be evaluated from some upstream conditions and a loss factor. This loss factor represents the loss in total pressure due to the fluid deviation and the development of boundary layers on the blade surface. Theoretically, it should be derived from an interblade azimuthal plane computation. Both throughflow and interblade problems would then be solved in a coupled way. In order to save some time, the loss factor can be approximated with loss correlations. These correlations are specific to each company and built on years of booster design experience.

A very in depth explanation on how these equations are solved is available in [3].

4.4.2. S1SI Throughflow

In order to solve the system of equations presented above, the user has to impose the radial distribution of some variables between each blade rows. Some variables are directly imposed based on experimental measurements while the others have to be assessed based on similar engines, empirical correlations and previous NS3D computations.
Once the throughflow computation is done, the solution is extrapolated in 3D to be used as an initialization for the S1SI NS3D steady computation. In particular, the values at the inlet and outlet borders, which are constant in the azimuthal direction, will serve as boundary conditions.

4.4.3. S1SI NS3D steady computation

Among all the computations that are done during the whole process, this one is the most realistic. It takes into account the whole booster, turbulence, tip gap clearance, recirculation under stators (derived from empirical laws) and others 3D effects. The setup of such a computation (meshing, numerical parameters etc...) is presented further. The goal is to make sure that this computation is as close as possible from the experiment. To do so, the performance of the booster is post processed through two parameters:
The reduced mass flow

\[ Q_{\text{reduced}} = Q \sqrt{\frac{T_{\text{upstream}}}{288.15}} \frac{p_{\text{upstream}}}{101325} \]  (22)

The pressure ratio

\[ \Pi = \frac{p_{\text{downstream}}}{p_{\text{upstream}}} \]  (23)

On a performance map, both the numerical and experimental results are plotted. In case the matching is judged unsatisfying, the user has to change the radial distributions imposed in the throughput analysis. Post processing tool FFTRO, developed by Snecma, enables to plot the radial distribution of all the variables anywhere in the booster. According to these results, the user has to adapt the radial distributions that are imposed in the throughput analysis.

In practice, several S1SI NS3D computations are launched from the same throughput initialization with different outlet boundary conditions.

The outlet boundary condition is a so called “\( p_s/Q \) objective” valve condition where \( p_s \) is the outlet static pressure at a given height (the whole radial distribution is derived from the radial equilibrium) and \( Q \) is the mass flow. At each iteration \( p_s \) is adapted so that the ratio \( p_s/Q \) converges until \( \frac{p_{\text{target}}}{Q_{\text{target}}} \). Usually the target ratio is expressed as follow:

\[ \frac{p_{\text{target}}}{Q_{\text{target}}} = \frac{p_{\text{ref}}}{Q_{\text{ref}}} (1 + \theta) \]  (24)

where \( \theta \) is the throttling coefficient, and \( p_{\text{ref}} \) and \( Q_{\text{ref}} \) are the reference outlet static pressure and mass flow obtained by the throughput analysis. The variation of \( \theta \) enables to study several operating points for the same rotational speed of the booster. On the booster “map” (see Figure 12), from the reference point, a positive \( \theta \) will go up on the iso rotational speed curve, while a negative value will go down.
After a few iterations, and once the computation is judged close enough from the experiment, a S1SI throughflow can be reconstituted.

4.4.4. S1SI Throughflow Reconstitution

The 3D converged solution is averaged in the azimuthal direction to give a new S1SI throughflow.

4.4.5. S1R2 Throughflow Extraction

In any CFD computation, the computational domain frontiers have to be far enough from the obstacle, here the blades, in order to avoid disturbances at the boundary conditions. As one can see on Figure 10, the second stator is right behind the first rotor,
which would give an outlet too close to the rotor. The domain is then virtually stretched out. Upstream of the stator, the domain is also slightly different.

4.4.6. S1R2 Throughflow Optimization

Since the geometry of the domain has been modified, the throughflow must be recomputed. As for the S1SI throughflow computation, the radial distribution of some variables must be imposed by the user at given axial positions. All these values are directly taken from the S1SI reconstituted throughflow. However, since the geometry is different, the static variables at these positions will be different. An optimization of the S1R2 has then to be done in order to match the static flow conditions from the S1SI throughflow. Figure 14 shows the result of the optimization (black before, blue after).

![Graph](image)

**Figure 14** – Static pressure radial distribution. Red: S1SI Throughflow, Doted Line: tolerance, Black : S1R2 Throughflow unoptimized, Blue: S1R2 Throughflow optimized

Once this step is done, it is assured that the inlet and outlet boundary conditions of the S1R2 domain are consistent with the experiment. As for the S1SI NS3D steady computation the S1R2 throughflow is extrapolated in 3D and used as an initialization for the S1R2 NS3D steady computation.
4.5. S1R2 NS3D Steady Computation

4.5.1. CFD Meshing

As it has been explained previously, the stage mesh consist of one rotor and one stator blades. This mesh will be used for the steady and the two unsteady computations.

4.5.1.1. Node Distribution Along the radial direction

This distribution is meant to generate the flow paths and is done following several steps

- Specification of the number of cells along the span
- Specification of the first cell width at the wall

This value is obtained on basis of a Reynolds-like non-dimensional number at the wall $y^+ \ [1]$ such that

$$y^+ = \frac{u_f y}{v} \approx 1 \quad (25)$$

where $u_f$ is the friction velocity at the nearest wall, $y$ is the distance to the nearest wall and $v$ is the local cinematic viscosity of the fluid.

- Specification of the percentage of mid-flow cells
- Specification of the nodes distribution along the different connecting edge

Figure 15 displays the flow paths in the meridian plane.

![Flow paths in the meridian plane](image)

*Figure 15 – Flow paths in the meridian plane*

4.5.1.2. Blade to Blade (B2B) mesh configuration

It is then necessary to define the nodes distributions around the blade at a specific spanwise position. The default topology is O4H: an O block around the skin, and four H
blocks (see Figure 17). The node distribution at the interface must be chosen carefully. Indeed, this mesh will also be used for the unsteady computation where the stator wake is transmitted to the rotor (it is not the case in the steady computation, see 0). In order to minimize the wake dissipation across the interface, the mesh at the interface must be such that:

- The cell width along the azimuthal direction is constant on both sides
- The cell width used on both sides is similar
- The cell width in the axial direction is constant (as much as possible) up to the blade

Figure 16 displays the final B2B mesh.

![B2B mesh](image)

*Figure 16 – B2B mesh*

### 4.5.1.3. Mesh quality

The quality of the generated 3D mesh is checked through three criteria:

- No negative cells
- Minimum skewness angle greater than TA criteria
- Maximum expansion ratio smaller than TA criteria

### 4.5.1.4. Modification of the blocks topology

The block topology automatically generated by Autogrid doesn’t fit the requirements of CardGenerator. Some modifications have then to be done by the user on IGG. The first step is to split and merge some blocks such that there is only one face at the inlet, the outlet, and the mixing plane. This is depicted in Figure 17.
Then the global axis have to be rotated such that:

- the X axis corresponds to the engine axis and is directed toward the rear
- the Z axis is radial, positive outwards
- the axes system is right hand positive

Each block has a local axes system (I,J,K). A great attention should be paid on their orientation. For some practical reason concerning the post-processing, each block local axes system should be reoriented to match the global (X,Y,Z) axes system.

Once the periodicity of each block and block naming have been done by the user, all the faces that have the same boundary condition should be named identically (see 4.5.2).

It is important to note that the radial tip gap between the rotor and the shroud is composed of an H and O blocks. Therefore, the faces that belong to the blade and to the shroud should be named accordingly. The final 3D topology is presented in Figure 18.
4.5.2. Boundary conditions

The color code for boundary conditions is presented on Figure 19.

![Figure 19 – Color code for boundary conditions](image)

- Inlet (Blue) and Outlet (Green)

These boundary conditions are extracted from the throughflow analysis (see 4.4).

- Walls (Orange)

The walls are assumed adiabatic with adherence condition, i.e. velocity equal to zero at the wall. It is important to note that, for the rotor, the velocity is set to zero at the wall in its own rotating frame of reference.

- Upstream stator (Red)

Upstream of the stator, a wall slip condition is imposed on the upper face. Indeed, as one can see on Figure 20, the beginning of the shroud, called the splitter, is right before the stator. As first approximation, it can be assumed that the splitter starts at the same position than the stator LE.

![Figure 20 – Splitter [3]](image)
• Interface (Purple)

For a steady computation, the wake of the stator is not transmitted to the rotor. Instead, the so called Mixing Plane method is used. At each spanwise location, the flow is averaged on the azimuthal direction, as it is shown in Figure 21.

![Figure 21 – Momentum azimuthal average at the mixing plane](image)

• Periodic boundaries (not colored)

For a steady computation the periodic boundaries are treated as symmetry planes.

4.5.3. Initialization

The 3D flow field is initialized from the extrapolation of the throughflow solution (see 4.4.6). Since the NS3D computation takes into account turbulence, the turbulence variables need to be initialized as well. The turbulence models available are k-ω, k-ε and k-l. The following equations give the relation between the dissipation rate ε, the specific dissipation rate ω, the turbulent kinetic energy k, and the turbulent characteristic length l [1]:

\[
\varepsilon = \frac{C_\mu \beta_1^{1/3} k^{3/2}}{l \sqrt{2}} \quad (26)
\]

\[
\omega = \frac{\beta_1^{1/3} \sqrt{k}}{l \sqrt{2}} \quad (27)
\]

with \( C_\mu = 0.09 \) and \( \beta_1 = 18 \).
Practically, the non-dimensional turbulent kinetic energy $k^*$ is used, and approximate with

$$k^* \approx \frac{3l^2}{2}$$

(28)

If $l$, is not provided, a good rule consist in taking $l = 0.07D_h$, where $D_h$ is the hydraulic diameter of the inlet, i.e. $D_h = 4S/P$, where $S$ is the cross section area and $P$ is the wetted area of the section.

4.5.4. Configuration of the numerical model

Table 1 presents the configuration of the numerical model. All the options available are presented in [2]. The number of iterations has been chosen so that the flow is well converged to avoid any crash of the unsteady computations.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>k-ε</th>
</tr>
</thead>
<tbody>
<tr>
<td>Space discretization</td>
<td>Jameson</td>
</tr>
<tr>
<td>Time discretization</td>
<td>Backward Euler</td>
</tr>
<tr>
<td>CFL</td>
<td>Ramp [0-50]</td>
</tr>
<tr>
<td>Number of iterations</td>
<td>50,000</td>
</tr>
</tbody>
</table>

*Table 1 – Configuration of the numerical model*

4.5.5. Convergence

The quality of the convergence can be checked through 4 graphs:

- The L2 norm residuals for each conservative variable
- The RHS residuals
- The mass flow defect which is defined by $|Q_{inlet} - Q_{outlet}|$
- The mass flows across the inlet, the outlet, and the mixing plane
Since the role of this steady calculation consists in initializing the flow fields for the unsteady computations, it is required to have a well converged steady computation. The mass flow defect should then be small and a decrease in the residuals of minimum two order of magnitude should be achieved.
4.6. S1R2 NS3D Unsteady Computations

As it has been explained previously, the two unsteady computations only differ from the amplitude of vibration of the blade. The pre-processing is then exactly the same for both computations. These paragraphs detail the few modifications that have to be done compared to the steady computation pre-processing.

In addition to this, since the blade is vibrating on a specific eigenmode, a great effort is put into explaining how to interpolate the eigenmode from the FEM mesh to the CFD mesh, the so-called “mode projection”.

4.6.1. Allowing mesh motion

Since the blade is moving, the mesh has to be modified at each time step. To do so, the “Aeroelastic” option must be activated and the faces where mesh motion is allowed must include the tag “free” in their name (see Figure 24).

![Figure 24 – Faces allowing mesh motion](image)

The small block at the inlet of the rotor does not allow mesh motion to prevent from interactions between mesh motion and the interface which could trigger bad wake transmission.

4.6.2. Boundary conditions

The only difference with the steady computation is that the chorochronic boundary condition is imposed at the periodic boundaries and at the interface between the stator and the rotor.

4.6.3. Chorochronicity

The unsteadiness in a turbomachinery flow comes from several phenomena: relative motion of the rows, mistuning, upstream or downstream distortions, interaction between
shock wave and boundary layers and so on. Ideally the whole rows should then be taken into account. This strategy may lead to prohibitive calculation costs and memory occupation.

The chorochronicity method aims at reducing the cost of such an application by preserving as computational domain one inter-blade channel for each row. The main assumption is that the unsteadiness is only due to the relative motion of the two rows. In this case, since the periodicity of the flow is greater that the extension of the channel, the conditions have to be space lagged and phase lagged on the external meridian boundaries of the interblades channels and at the interface between the two rows. It should be noted that only a prescribed number of harmonics are transmitted to the rotor row. More information on the chorochronic periodicity and on the mathematical treatment at the interface can be found in [8].

4.6.4. Time step setup and number of iterations

The unsteady algorithm is used to solve unsteady chorochronic computations. This paragraph aims at explaining how to choose the time step and the number of iteration in such a case.

The unsteady flow in the frame of reference rotating with a given row is periodic with the fundamental frequency of the Blade Passing Frequency (BPF) of the blades of the adjacent blade row

\[
BPF_1 = \frac{1}{T_1} = \frac{N_2|\Omega_2 - \Omega_1|}{2\pi}
\]

\[
BPF_2 = \frac{1}{T_2} = \frac{N_1|\Omega_2 - \Omega_1|}{2\pi}
\]

where \(N_1, N_2\) are the periodicity of the first and second blades rows and \(\Omega_1\) and \(\Omega_2\) their rotational speed in rad/s.

According to [20], the greatest time step \(\Delta t\) such that both \(T_1\) and \(T_2\) can be discretized is then

\[
\Delta t = \text{gcd}(N_1, N_2) \frac{2\pi}{|\Omega_2 - \Omega_1|N_1N_2}
\]
In order to get a stable computation, the computational time step should be a fraction of $\Delta t$

$$dt = \frac{\Delta t}{nquo} \tag{32}$$

where $nquo$ is an integer which is chosen such that a full wheel turn is done in a given number of time steps $n_{steps}$. The time for a full wheel turn is

$$t_{full} = \frac{2\pi}{|\Omega_2 - \Omega_1|} = n_{steps} dt \tag{33}$$

Inserting (35) in (33) yields to

$$nquo = \text{int}\left(n_{steps} \frac{\gcd(N_1,N_2)}{N_1 N_2}\right) \tag{34}$$

The number of iterations to complete the period $T_1$ is then

$$N_{iter_1} = \frac{N_1}{\gcd(N_1,N_2)} nquo \tag{35}$$

and the number of iterations for completing a full wheel turn is given by

$$N_{iter_{full}} = N_{iter_1} N_2 = \frac{N_1 N_2}{\gcd(N_1,N_2)} nquo \tag{36}$$

4.6.5. Initialization

The converged flow from the steady computation is injected at each node of the whole mesh.

4.6.6. Mode file generation

Since the mesh used in the FEM analysis is different from the one used in the CFD computation, it is necessary to project the FEM nodal displacements of the mode of interest on the CFD mesh. To do so, an intermediate mesh called the reduced mesh is used. Figure 25 shows the different steps of this projection.
4.6.6.1. First projection

The reduced mesh is composed of two sets of coordinates, one for the suction side and one for the pressure side. These coordinates are normalized in the radial and chord direction, so they are in $[0,1]$. This projection is done with Aeromeca, a software developed at Snecma.

The first step of the projection from the FEM mesh to its reduced mesh is to extract the coordinates of both the suction side and the pressure side. To do so the user needs to provide several parameters enabling the program to distinguish the two sides of the blade. These need to be chosen carefully or some points of the suction side might be considered as part of the pressure side and vice versa. On Figure 26 one can see that some points of the suction side have been interpolated on the pressure side.

![Reduced mesh on the pressure side](image)

**Figure 26 – Reduced mesh on the pressure side**

As aerodynamic department launches computations on the blades only while the mechanical department studies the blades and the drum together (see Figure 27), some
other parameters allow the split of the blade on the mechanical model. This enables to have consistent domains before the projection.

![Figure 27 – FEM mesh](image)

4.6.6.2. Second projection

The next step is to project the mode from the FEM reduced mesh to the CFD reduced mesh, and from the CFD reduced mesh to the CFD mesh. These two projections are done consecutively by a similar tool that has been developed at Sncema.

The projection can only be done on a single row configuration. A simple rotor CFD mesh has then to be created. In order to get a good interpolation, the nodal distribution should be the same as the one used for the stage mesh, i.e. same flow paths and same B2B mesh parameters.

The tool that does this projection has been developed to provide modes for the pre-processing chain Comet. The problem is that the rotor blade has to be oriented in SHAV (turning clockwise when looked from the front of the engine) in Comet while it has to be oriented in SHAR (turning clockwise when looked from the rear of the engine) in
CardGenerator. Therefore, the mode have to be reoriented before launching the computation on CardGenerator.

4.6.6.3. Checking the projection

At the two projections, the user should check that the mode has been well interpolated during the process as it is shown in Figure 28 (the reduced CFD mesh is not available).

![Modal deformation on the pressure side (above) and the suction side (below) on the FEM mesh, the reduced FEM mesh and the CFD mesh](image)

**Figure 28 – Modal deformation on the pressure side (above) and the suction side (below) on the FEM mesh, the reduced FEM mesh and the CFD mesh**

4.6.7. Configuration of the numerical model

The only difference with the steady computation is that the Unsteady algorithm is used to solve the problem. The choice of the time step and the number of iterations has been explained in 4.6.4.

4.6.8. Convergence

Since it is an unsteady computation, there is no reason for the residuals to decrease. However, they should have a periodic pattern (see Figure 29). The behavior of the mass flow is the main criteria to judge the quality of the convergence. It should have a periodic behavior at both inlet and outlet as well as the same constant mean value.
4.6.9. Calculation of the forced response

The formula used to calculate the forced response is reminded below.

\[ \alpha_{fr} = \frac{GAF_{TWIN}(0)}{(\bar{K}_k - \omega_k^2 \bar{M}_k + i \omega_k \bar{C}_k) - \frac{GAF_{TWIN}(\alpha_{aero}) - GAF_{TWIN}(0)}{\alpha_{aero}}} \]  

(37)

4.6.9.1. Extraction from the computation with fixed blade

\( GAF_{TWIN}(0) \), which is equal to \( GAF_{forcing} \), is extracted from the computation with fixed blade. Figure 30 displays the local excitation forces on both the suction side and pressure side.

Figure 30 – Local excitation forces on the suction side (left) and pressure side (right)
4.6.9.2. Extraction from the computation with prescribed harmonic motion

\( GAF_{\text{TWIN}}(\alpha_{\text{aero}}) \) is extracted from the computation with prescribed harmonic motion of the rotor blade. It is now possible to compute the aerodynamic damping

\[
Damping = -\text{Im} \left( \frac{GAF_{\text{TWIN}}(\alpha_{\text{aero}}) - GAF_{\text{TWIN}}(0)}{\alpha_{\text{aero}}} \right) = -\text{Im} \left( \frac{GAF_{\text{aero}}(\alpha_{\text{aero}})}{\alpha_{\text{aero}}} \right)
\]  

(38)

Positive values (in blue on Figure 31) indicates an effective damping, i.e. which tends to decrease the amplitude of motion of the rotor blade.

\begin{center}
\textbf{Figure 31 – Local aerodynamic damping on the suction side (left) and pressure side (right)}
\end{center}

From there, the non-dimensional global aerodynamic damping can be calculated with equation (16).

4.6.9.3. Forced response

The forced response can now be calculated with equation (37). The value of the structural damping is difficult to evaluate because it comes from both the material itself and the friction at the blade-platform interface. The structural damping can then be derived from experimental vibration testing or/and be expressed as a Rayleigh structural damping coefficient [20]

\[
\xi = \frac{\omega_k \tilde{c}_k}{2 \tilde{K}_k}
\]

(39)

The evolution of the forced response with respect to the engine rotational speed is investigate around the regime of interest considering that the structural parameters and
the GAF are constant in that interval. The green point in Figure 32 corresponds to the crossing in the Campbell diagram. One can note that this point is not exactly the maximum. This is the consequence of the assumption which neglects the influence of the fluid on the mode and its frequency. The frequency shift is given by [6]

\[
\omega_{\text{shift}} = \sqrt{\frac{1}{M_k} Re \left( \frac{GAF_{\text{TWIN}}(\alpha_{\text{aero}}) - GAF_{\text{TWIN}}(0)}{\alpha_{\text{aero}}} \right)}
\]  

(40)

![Figure 32 - Normalized evolution of the forced response with respect to the normalized rotational speed](image)

The value that is taken as a result is the maximum of this curve since it corresponds to the resonance of the blade.

4.6.10. Conversion in percentage of endurance limit

The forced response given by the post processing tools is a maximum amplitude of vibration in mm. Converting it into a stress, and then into a percentage of the endurance limit gives a better understanding of the mechanical behavior of the blade. The endurance limit is the amplitude of cyclic stress that gives an infinite life time of the blade, $10^7$
cycles in aeronautics. In order to make this conversion, a so-called “weaklink” analysis is done by the mechanical department.

The weaklink analysis requires the Goodman diagram of the material used to manufacture the blade. This diagram gives the endurance limit as a function of the static stress. In turbomachinery the static stress is caused by the pressure and temperature fields as well as the centrifugal force.

The weaklink analysis consist of taking the stress of the critical element from the dynamic analysis $\sigma_{\text{max\_dynamic}}$ and project it on the Goodman curve at the static stress given by the static analysis. The weaklink analysis also gives the maximal displacement of the dynamic analysis $d_{\text{max\_dynamic}}$ (the maximal displacement does not occur at the same location than the maximal stress). From this it is then possible to convert the forced response from a maximal displacement $\alpha_{fr}$ to a maximal stress $\sigma_{\text{max\_fr}}$. If a linear mechanical behavior is assumed, the forced response in percentage of endurance limit is given by the formula

$$
\% \text{EL} = \frac{\sigma_{\text{max\_fr}}}{\sigma_{\text{Goodman}}} = \frac{\alpha_{fr} \cdot \sigma_{\text{max\_dynamic}}}{d_{\text{max\_dynamic}} \cdot \sigma_{\text{Goodman}}}
$$

Figure 33: Goodman diagram
5. Results

5.1. Throughflow analysis

It is reminded that the goal of the throughflow analysis was to make sure that the boundary conditions of the restricted computational domain were representative of the experiment. For that, an iterative process including S1SI NS3D computations has been used. Figure 34 presents the results of these computations on the normalized performance map with respect to the experimental working point at which forced response occurred.

![Normalized performance map](image)

*Figure 34: Normalized performance map*

The best matching gave a difference of 0.16% for the reduced mass flow and 0.03% for the pressure ratio. A throttling coefficient of 0.08 has been used for this computation.

As it is explain in 4.4.6, the last step of the throughflow analysis is to optimize the S1R2 domain so that it matches the static conditions of the full booster configuration. Additional S1R2 NS3D computations were launched to check this matching in terms of performances. It gave a difference of 0.06% in reduced mass flow and 0.01% in pressure ratio.
5.2. Influence of time discretization

A time discretization convergence has been carried on. The unsteady computations have been launched with approximately 10.000, 20.000, 30.000, 40.000 and 50.000 thousands iterations per full turn. The value of the Rayleigh structural damping coefficient used to calculate the forced response is taken from experimental tests. The results are presented on Figure 35 where the numerical value of the forced responses is normalized by the experimental one.

![Normalized forced response convergence with time discretization](image)

*Figure 35: Normalized forced response convergence with time discretization*

As one can see on Figure 35 the convergence was obtained with 30.000 and 40.000 thousands iterations per turn. These results give a very good correlation with the experimental value. However, the computation with 50.000 iterations per turn seems to diverge. This divergence might be due to some numerical issues. In the scope of this study, it has been assumed that this last value is not relevant and that the result of the computation with 40.000 iterations per turn is the most realistic.

5.3. Influence of the number of transmitted harmonics

With the use of a chronochochronic boundary condition at the interface between the stator and the rotor, the pressure signal is decomposed using FFT and only a given number of harmonics is transmitted to the rotor. In this study, computations have been launched
with 20 and 40 harmonics. The difference between the results are smaller than 1%. It is then advised to only use 20 harmonics since it is more time saving.

5.4. Linearity assumption verification

The TWIN procedure relies on the linearity of the response of the surrounding flow. It means that the aerodynamic forces due to the vibration of the blade are linear with respect to the amplitude of vibration. One way to check this assumption is to take the value of the forced response obtained by a computation, use it as the amplitude of vibration, and recompute the forced response. This is what has been done with an unsteady computation with 40,000 iterations per turn and 20 harmonics. The new result is 2.5% smaller. As a first approximation, the linearity assumption is satisfied. One has to remember that the mode 2S2 has a pretty high eigenfrequency and a complex deformation shape. Thus, non-linear phenomena are more likely to occur which makes 2.5% a very acceptable value. Moreover, computations with 40,000 iterations take about 50 hours, it is then not advised to make relaunches.

6. Discussion

The boundary conditions reconstitution gave extremely good results. Considering the amount of time required to launch S1SI NS3D computations, it would have been worthless to make more iterations in order to get a better matching.

The forced response computations with 30,000 and 40,000 gave an excellent correlation with the experimental value (less than 3%). However, some approximations have been made:

- The value of the Rayleigh structural damping coefficient used in this study has been measured for the first flexion mode with 0 nodal diameter and at 0 rpm
- The unsteady computations do not take into account the flow recirculation under the stator
- The blade used by the mechanical department to compute the mode is not exactly the one that has been tested during the experiment

The divergence at 50,000 iterations per turn is also an issue since it is not possible to state that convergence is reached. Some solid arguments are required in order to reject this result with certitude. A visualization of the turbulent kinetic energy on the carter wall shows a non-physical phenomenon that appears only for this computation. As
depicted on Figure 36 some turbulent kinetic energy is created right in front of the stator. This corresponds exactly to the frontier of the upstream block were a wall slip condition is applied.

![Image](image_url)

Figure 36: Turbulent kinetic energy at rotor tip

If the same visualization is done on the carter, one can see the source of this creation.

![Image](image_url)

Figure 37: Turbulent kinetic energy on the carter

This is clearly a numerical issue that appears for small time steps. However this issue couldn’t be studied in depths during this thesis.

Another discussion that could be done is about the uncertainty on the Rayleigh structural damping coefficient. As it is show in equation (39), this coefficient depends on the reduced matrices which depend on the mode of interest. The value used for these computations
has been measured for the first flexion mode with 0 nodal diameter at 0 rpm. It was however the best approximation available at this time. The sensitivity of the forced response with respect to the Rayleigh structural damping coefficient is shown Figure 38. The axes are normalized by the value used for the computations and the corresponding forced response. As one can see, an uncertainty of 50% on the coefficient gives an uncertainty of about 5% on the forced response, which is acceptable. In the worst case, the correlation is then still acceptable.

![Figure 38: Sensitivity of the forced response with respect to the Rayleigh structural damping coefficient](image-url)
7. Conclusion

The goal of this thesis was to compute the forced response of the mode 2S2 of the first rotor of a low pressure compressor under the excitation of the upstream stator using TWIN methodology. The benefit of such a method is that it is more accurate than decoupled approaches and that the same mesh is used for all the computations. This study was made on a booster including a bladed drum, since this new design where the rotor blades are directly welded on the drum is very sensitive to aeroelastic phenomena. The results were compared to an experimental value in order to prove the reliability of the computations and the method.

In order to get the best numerical-experimental correlation possible, the boundary conditions have been reconstituted thanks to full booster NS3D computations. The matching is measured through the performance of the booster: 0.16% difference for the reduced mass flow and 0.03% for the pressure ratio. The boundary conditions are then adapted to the restricted computational domain used for the forced response computation which is composed of the first stator and rotor.

In the TWIN methodology, two unsteady computations are required: one with a fixed rotor, and another one where the rotor has a prescribed harmonic motion along the mode 2S2. Since the computational domain is the same for both computations, the same steady computation is used as an initialization. In order to make the blade vibrate in the CFD computations, the mode 2S2 had to be projected from the FE mesh to the CFD mesh. From the two unsteady computations, the aerodynamic forces generated by both the wake of the stator and the vibration of the blade can be extracted. From there, the TWIN methodology gives an analytical expression of the maximal amplitude of vibration that is converted into a percentage of endurance limit with a weaklink analysis.

The results showed a very good correlation with the experiment. However, a non-physical phenomenon appears for the smallest time-step and has to be studied in depths. Despite this issue, the engineers at TA were very satisfied with these results and are enthusiastic for the future of forced response prediction.

The TWIN methodology proved to be predictive and applicable in an industrial process. In a phase of development, the boundary conditions reconstitution step is non-existent which makes the procedure faster. This thesis also highlighted the fact that forced response computation requires a good cooperation between the aerodynamic and mechanical departments.
8. Future work

- Recompute the forced response with a mechanical analysis that is done on the exact same rotor blade as the one used in the experiment.
- Investigate the numerical issues that occur at small time-steps. One could for example launch computations with 60,000 and 70,000 iterations per turn to see if the result keeps diverging.
- Compute forced response of other modes and other engines to show the robustness of the method.
- Take into account the recirculation under the stator (not possible at the moment).
- Take into account mistuning effect in order to get a more statistical approach.
References


[20] TA internal documents
Dept of Energy Technology
Div of Heat and Power Technology
Royal Institute of Technology
SE-100 44 Stockholm, Sweden