PREDICTION AND VALIDATION OF HIGH-PERFORMANCE CENTRIFUGAL COMPRESSOR IMPELLER FORCED RESPONSE

L. Toni\textsuperscript{1}, F. Moyroud\textsuperscript{2}, K. Ramakrishnan\textsuperscript{3}, V. Michelassi\textsuperscript{4}, E. Schurr\textsuperscript{5}

GE Oil & Gas - Via Felice Matteucci 2, 50127, Firenze, Italy, lorenzo.toni@ge.com
GE Oil & Gas - 480 Allée Gustave Eiffel, 71203 Le Creusot, France, francois.moyroud@ge.com
GE Global Research - 1 Research Circle, 12309-1027, Niskayuna, USA, ramakris@ge.com
GE Oil & Gas - Via Felice Matteucci 2, 50127, Firenze, Italy, vittorio.michelassi@ge.com
GE Oil & Gas - 300 Garlington Road, 29615, Greenville, USA, edward.schurr@ge.com

ABSTRACT

High-performance centrifugal compressors call for a reduction in weight to relax rotor-dynamic constraints, and a reduction in blade thickness as a way to improve efficiency. In this tight design space, synchronous excitations on impellers are often unavoidable and may represent a potential aeromechanic risk, especially for high speed stages operated at high pressures. Therefore, forced response analysis has become a fundamental design step. The aero/structure unsteady interaction in impellers, mainly driven by both stationary row wakes and potential fields, and the possible overlap with structural frequencies, must be carefully analyzed and quantified since they may lead to high cycle fatigue failures under critical operating conditions.

This paper focuses on the forced response analysis of an unshrouded high-speed centrifugal impeller incorporating aircraft engine technology designed for hydrocarbon processing. The methodology focuses on the effects of generic disturbances (inlet distortions, potential fields, aero-acoustic excitations) superimposed in amplitude and frequency on the time averaged flow field, and follows a rigorous numerical approach supported by validation against dedicated aeromechanic test data.

Aerodynamic forcing frequencies and amplitudes, and the corresponding aero-mechanic damping jointly contribute to the vibratory response. An unsteady CFD analysis determines the impeller aerodynamic load which, in conjunction with FE (Finite Elements) modal analysis, allows for computing the modal force. A time-linearized Navier-Stokes computation predicts the aerodynamic damping, and allows mapping the mode shapes onto the CFD grid in order to supply the displacement boundary conditions. A purely numerical Goodman diagram has been drawn and the response level for a specific structural mode shape analyzed. The predictions compare favorably with test data, providing confidence in this method for practical high-speed radial compressor design.

KEYWORDS

AEROMECHANICS, CENTRIFUGAL COMPRESSOR, FORCED RESPONSE, AERO-DAMPING, MODAL FORCE
NOMENCLATURE

\( d \) = Modal Displacement  
depends on normalization

\( f \) = Frequency  
[Hz]

\( F_{\text{aero}} \) = Aerodynamic Motion Independent Force  
[N]

\( F_{\text{modal}} \) = Modal Force  
depends on normalization

\( k \) = Blade to Blade Variability Factor  
[-]

\( P \) = Pressure  
[Pa]

\( Q \) = Damping Amplification Factor  
\( Q = \frac{1}{2\zeta} \)  
[-]

Greeks

\( \phi \) = Mode Shape  
depends on normalization

\( \rho \) = Density  
[kg/m\(^3\)]

\( \sigma \) = Stress  
[MPa]

\( \omega \) = Angular Frequency  
[rad/s]

\( \zeta \) = Overall Critical Damping Ratio  
[-]

\( \zeta_{\text{aero}} \) = Aerodynamic Critical Damping Ratio  
[-]

\( \zeta_{\text{mat}} \) = Material Critical Damping Ratio  
[-]

Acronyms

CFD = Computational Fluid Dynamics

FEA = Finite Element Analysis

RANS = Reynolds Averaged Navier-Stokes

SG = Strain Gauge

INTRODUCTION

Centrifugal compressors are routinely used for both gas processing and aero-propulsion. Although the specifications and operating conditions are quite different (gas properties, tip speeds, durability, weight constraints and the like), there is a common growing demand for performance, power density, and weight reduction. Reducing weight by reducing blade, hub and shroud thickness does mitigate rotor-dynamic risks and improve performance. Nevertheless, the concerted action of the consequent structural eigen-frequencies reduction together with higher excitation frequencies and higher static stresses due to higher tip speeds required to meet higher power density, does increase the risk of incurring into aeromechanics and/or aeroacoustics risks. The complex nature of the flow field in a centrifugal compressor, where the fluid is subject to both strong meridional and blade-to-blade curvatures, and the mutual interaction between components calls for a detailed validation and screening of the aero design, as shown by Guidotti et al. (2014), and aeromechanical design methods, covered in this paper.

The importance of this topic is witnessed by the vast open literature. Dickmann et al. (2009) studied the aero mechanics performance of centrifugal compressor components by investigating the unsteady interaction of a turbocharger impeller with a vaned diffuser and exit scroll. Their analyses aimed at determining aeromechanic loads at both design point and part load. While showing how CFD can predict the measured vibrations, they also introduced a new method to account for real manufacturing mistuning effects.

Open-faced impellers are routinely used in aero-propulsion and turbocharging, and are becoming more popular in gas processing as well. Unshrouded impeller aeromechanics is very sensitive to inlet distortions, and Kammerer and Abhari (2009 Part I and II) completed a detailed experimental investigation in a closed loop rig. They intentionally created inlet flow distortions to excite three different resonant crossings, and measured fundamental and harmonic frequencies for a number of pressure settings, and compared with design predictions. While aerodynamic flow field
was characterized by FRAP (Fast Response Aerodynamic Probes), they determined how maximum response amplitudes were a function of inlet pressure, although the amplification with growing inlet pressure was not the same for all modes. The same authors determined damping, both mechanical and aerodynamic, as this is very important to understand if a specific mode at a given excitation frequency is truly dangerous at varying inlet pressure.

On top of forced-response driven by wakes and component-to-component interaction, described by Dickmann et al. (2009) and Kammerer and Abhari (2009), aeroacoustics proved to be a matter of concern, as described by Koenig et al. (2009) and by Koenig (2009). While failures of radial impellers are fortunately very rare, Koenig documented a case in which acoustic eigenmodes could be responsible for mechanical failures. The so-called triple-crossing, when structural resonance frequencies correspond to both aerodynamic and acoustic frequencies, may become risky especially in units that are asked to operate in a wide operating range and under variable inlet conditions. In particular, Koenig et al. analysis reveals that the design of centrifugal compressor cavities must be considered when screening aeromechanics. In view of the results of Koenig et al., Clancy et al. (2014) performed a linearized Navier-Stokes CFD investigation of a centrifugal compressor equipped with a shrouded impeller. Their unsteady simulations included both the hub and shroud cavities to reveal how the unsteady pressure field in the flow path may impact excitation of the impeller by eigenmodes of the impeller, cavity acoustics, and Tyler-Sofrin modes.

Centrifugal compressors are often equipped with vaned diffusers of both the low and high solidity type. In these cases, the complexity of the unsteady pressure field increases due to the presence of an additional blade row (the vaned diffuser). Furthermore, the small streamwise gap between impeller trailing edge and vaned diffuser leading edge, necessary not to jeopardize efficiency, results in strong wakes and bow waves with potential impeller-diffuser interaction that must be carefully scrutinized for aeromechanics risks, as performed by Ramakrishnan et al. (2011). The CFD analysis showed how acoustic modes can grow potentially dangerous as opposed to a vaneless diffuser configuration.

Mechanical and unsteady aerodynamics analyses can identify frequencies and forcing amplitude. Nevertheless, to translate this into stresses requires an estimate of damping. Damping is of material, mechanical, and aerodynamic nature (see Kammerer and Abhari, (2009)). The determination of the first two contributions requires specific mechanical test, while the third is much more problematic as it depends on the operating conditions, and may change with both operating pressure and flow regime (close to choke or to stall). Aerodynamic damping can be determined by using accurate measurements, as indicated by Kammerer and Abhari (2009, Part II), essentially by the difference between the measured overall damping during stage operation and the pre-measured mechanical and material damping. Still, it is desirable to determine aerodynamic damping in the early design phase to predict and overcome design problems. In this view, Toni et al. (2016) proved how CFD could led to a realistic estimate of damping, ultimately allowing to predict the vibratory response levels and the associated stresses.

With this in mind, this paper describes a method to compute aero-damping and modal force for the aeromechanic verification of centrifugal compressors. The justification for the proposed approach and its validation are discussed to prove the practical applicability to real designs and realistic operating conditions.

NUMERICAL METHODOLOGY

The CFD analysis was performed using a proprietary Reynolds Averaged Navier-Stokes flow solver. The code is a 3D finite-volume, multi-block, multi-grid, structured, explicit Runge-Kutta solver developed and validated for turbomachinery flows; further details on the CFD method can be found in Holmes et al. (1997). The grids adopted are the result of a grid sensitivity study that was conducted in a number of turbomachinery applications. In the present study, k-ω turbulence model with wall functions has been used and the y+ values generally range from 15-30 on the walls at the analyzed condition.
Damping Computation

The numerical method for time harmonic calculations for aeromechanics is described in detail in Clancy et al. (2014) and in Toni et al. (2016) and it will be briefly summarized here. A time-linearized Navier–Stokes method is used to predict the aerodynamic damping, where the perturbations to the steady base flow arising from blade motion are assumed to be small.

First, the steady flow around a periodic sector of the impeller is computed. Then, the modal displacements from a finite element model of the structure are mapped onto the CFD grid as a boundary condition to the linearized Navier-Stokes solver. Then, the unsteady pressures resulting from the blade motion are used to compute an aerodynamic work per cycle on the vibrating airfoil. Knowing both frequency and modal mass associated with the vibration mode, the aerodynamic work may be converted into an equivalent aerodynamic damping. For this analysis, phase lagged boundary conditions are enforced on the periodic surfaces of the model and non-reflecting boundary conditions are applied at the inlet and exit boundaries to minimize pressure wave reflections.

The linearized approach predicts aerodynamic damping levels in good agreement with the measured values in a range of cases; therefore the non-linear method was not applied.

Modal Force Computation

To compute the modal force, an unsteady CFD analysis of the impeller aerodynamic response to the inlet distortion is performed as described in the next section. The resulting unsteady pressure field at the frequency of interest is projected onto a basis of orthogonal vectors consisting of a set of natural mode shapes of the impeller. Like for the aerodynamic damping computation, the mode shapes are interpolated onto the CFD grid (generally finer than the FEA grid on the impeller surface). The reader can refer to Moyroud et al. (2000) for a discussion on the pros and cons of using the CFD grid to compute modal force.

For a given combination of an impeller mode shape and the unsteady pressure field at a given excitation frequency, the modal force is a complex valued scalar quantity defined as follows:

\[ F_{\text{modal}} = \langle \Phi_{\text{modal}}^c \rangle \{ F_{\text{aero}} \} \]  \hspace{1cm} [1]

Where \( \langle \Phi_{\text{modal}} \rangle \) is the transpose of the vector of complex nodal displacements for all modal displacement components at all CFD nodes, and \( \{ F_{\text{aero}} \} \) is a vector of nodal unsteady aerodynamic loads in complex form resulting from the surface integration of frequency domain unsteady pressure perturbations on the impeller walls at the excitation frequency of interest (or source of excitation). The C superscript denotes the complex conjugate operator. Generally, the mode shapes are mass normalized and therefore the corresponding modal response displacements or participation factors can be calculated as follows:

\[ d_{\text{modal}} (\omega_e) = \frac{F_{\text{modal}}}{\sqrt{(\omega_{\text{modal}}^2 - \omega_e^2)^2 + (\omega_{\text{modal}} \omega_e/Q_{\text{modal}})^2}} \]  \hspace{1cm} [2]

Where \( Q \) is the amplification factor calculated from the aero-mechanical modal damping (that is the sum of material, mechanical and aerodynamic damping, see also “Aero-Damping” section), and \( \omega_{\text{modal}} \) is the natural angular frequency of the impeller mode. Under resonance conditions, we have:

\[ d_{\text{modal}} (\omega_e = \omega_{\text{modal}}) = \frac{F_{\text{modal}} \cdot Q_{\text{modal}}}{\omega_{\text{modal}}^2} \]  \hspace{1cm} [3]
This single degree of freedom modal participation factor or displacement is used to scale the modal stresses and to compute physical alternating stresses at any node on the surface of the impeller FEA model, for representation in the numerical Goodman diagram.

**Unsteady CFD Analysis for Forced Response**

The aeromechanic response of centrifugal impellers requires the prediction of the unsteady flow field either by a non-linear or a linear approach. The experience so far, while validating forced response computations against test data, demonstrated that the more complex and time-consuming approach based on nonlinear unsteady CFD, leads to results much closer to experimental data, while a linearized approach was found appropriate to determine aerodynamic damping.

Figure 1 shows the computed unsteady pressure fields resulting from the two computational approaches, linear and non-linear, for a given frequency and inlet distortion. At first glance the time-linearized analysis predicts a sensibly lower level of unsteadiness as compared to the non-linear result. Both flow fields have then been used to compute the modal work for a selected mode shape and then compared to experimental values. Table 1 summarizes the obtained results, which have been normalized by the measured ones; the time-linearized approach under-predicts modal force level by approximately 70%, whereas the modal force computed by using the non-linear unsteady CFD is much closer to measurements.

Based on these results, confirmed by other test cases not reported here for the sake of brevity, we decided to abandon the time-linearized approach in favor of the non-linear unsteady computations to determine the modal force experienced by the impeller.

![Figure 1: Unsteady Pressure Fields; Linear Unsteady (Left), Non-Linear Unsteady (Right)](image)

<table>
<thead>
<tr>
<th>Approach</th>
<th>( \frac{F_{\text{modal}}}{F_{\text{modal \ Test}}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time-Linearized Unsteady</td>
<td>0.299</td>
</tr>
<tr>
<td>Non-Linear Unsteady</td>
<td>0.967</td>
</tr>
<tr>
<td>Test Result</td>
<td>1.000</td>
</tr>
</tbody>
</table>

**Table 1: Modal Force Results Comparison (Normalized Values)**

Hence, a full-annulus nonlinear time-unsteady CFD computation has been performed for the specified compressor stage. The analysis domain includes the design-intent flowpath upstream of the leading edge to a surface where an inlet distortion boundary condition is applied. The inlet boundary condition comes from a separate analysis of the inlet plenum geometry detailed in the “Excitation Sources” section. Downstream of the impeller blade trailing edges, a vaneless diffuser extension is modeled, terminating before the 180 degree bend.
The structured impeller grid was generated using Autogrid by NUMECA and contains ~56M cells. Within the domain, the clearance region is accurately modeled, along with the root fillet; Figure 2 shows CFD impeller grid details.

The steady-state flow field was converged using the proprietary solver, then the unsteady computation was started by imposing the conditions from the inlet plenum/impeller interface at the inlet of the impeller. Periodic convergence was achieved through running multiple revolutions of the rotor. The number of time-steps per revolution was set to the current best practice at GE. After convergence was achieved, two more revolutions were completed and averaged to output unsteady static pressures for post-processing. Based on prior GE-internal analyses, averaging the solution over multiple revolutions has been shown to improve the frequency discretization, especially for lower order frequency content. For this analysis, the primary focus was on predicted responses in the choke region. In the choke region, strong shocks are seen on both the main and splitter blades. These shocks are greatly influenced by the inlet pressure distortion. The modal forces and aerodynamic damping are also directly connected to the inlet distortion and its effect on the flow field.

![Figure 2: CFD Mesh Details](image)

In order to decompose the unsteady pressures from the full-annulus unsteady calculation into a suitable form for aeromechanical calculations, a Fast Fourier Transform was performed on the blades’ time-unsteady static pressures, converting to the frequency domain. This allowed for the specific harmonics of interest to be extracted for modal force calculations. In particular the 7th harmonic unsteady pressure field has been considered since, as detailed below, this was the main harmonic of interest at the choked-flow condition.

**EXPERIMENTAL SET-UP**

An unshrouded high Mach number and high flow coefficient impeller for Oil & Gas application has been selected as a test case for aeromechanics validation. The tested impeller has been designed with 11 main flow channels. Contained in each main channel is a splitter blade.

**Rig Layout and Instrumentation**

Model testing was performed at the GE Oil & Gas Technology Laboratory (OGTL) in Florence. The test bench is a closed loop rotating single stage rig, normally employed for performance measurements of different centrifugal compressor designs; further details can be found Toni et al. (2010), Bianchini et al. (2012) and in Guidotti et al. (2011).

The impeller has been tested in a first stage arrangement, thus an inlet plenum is included; see Figure 3. This plenum is designed to achieve minimum losses and low flow distortions and comprises three non-uniformly spaced straightener vanes. A vaneless diffuser and return channel are included in the stage as well, approximating conditions in a product.
During performance testing, total pressure, total temperature, and flow angle measurements were taken at different locations along the flow-path. During aeromechanic tests, only few sections have been instrumented so as to avoid as much as possible any source of spurious interactions. In order to characterize distortions coming from the inlet plenum and to derive its harmonic content at the impeller inlet (Section 10), total pressure rakes were mounted on a rotating conveyor which enabled the acquisition of high resolution measurements in the tangential direction. Rakes at Section 10 were then removed during aeromechanic test. Tests have been performed using Nitrogen as working gas, with almost constant inlet temperature and varying inlet pressure from atmospheric conditions up to 3.5 bar; test case considered for the comparison was run at 1.5 bar.

The strain gauges (SG) used for the aeromechanic validation were installed on main blades, splitter blades and disk. Each gauge position has been replicated over 3 sectors on the impeller to guarantee redundancy and to estimate the sector-to-sector variability. Regarding gauge positioning, the best practice requires a minimum value for sensitivity and a low local stress gradient in order to minimize the measurement error due to sensor position installation tolerances. A telemetry system was used for strain gauge signals acquisition.

![Test Rig Layout](image)

**Figure 3: Test Rig Layout**

**Test Data Processing**

Strain gauge signals are continuously analyzed and recorded with a sampling rate of 51 kHz, against a blade passing frequency of ≈7 kHz. Both on-line and off-line analyses have been performed using proprietary software. Signals are converted from the time domain to the frequency domain, then the vibration amplitudes corresponding to different engine orders are extracted and an experimental Campbell diagram is constructed.

Figure 3 shows an example of experimental Campbell diagram: frequency is represented as a function of shaft rotational speed, whereas the fan lines are the so-called engine orders, that is to say running frequency harmonics. Whenever a structural mode is excited, a frequency modulation of the synchronous forced response in the Campbell diagram is found: in this case a resonant crossing along 7/Rev excitation line is highlighted. Through a proper fitting of response peaks, using a single degree of freedom numerical model, modal damping is extracted from test data; amplitude and frequency are obtained too. Further details about data processing may be found in Toni et al. (2016).
RESULTS

Excitation Sources
Inlet distortions coming from the upstream plenum have been considered as sources of excitation for the impeller: the analysis of circumferential harmonic content of the inlet plenum flow field is therefore a fundamental step to identify both the resonant crossings that are likely to occur in the design phase as well as correctly analyze experimental data. Inlet plenum steady state CFD computations were then performed using ANSYS CFX and the predictions compared to the results obtained from the high resolution flow field scanning; a 65M elements tetrahedral mesh with prism layers on the walls was made using ANSYS ICEM. The CFD results have been used both to obtain the boundary conditions for impeller unsteady runs and to compute the circumferential harmonic content at impeller inlet which was compared to measured one.

Normalized tangential total pressure distributions measured from the model test are shown in Figure 5 for different span-wise positions along with the two-dimensional distributions. The presence of the three vanes may be easily seen, as well as the increase in wake strength when moving radially outwards. Figure 6 represents the measured and predicted Fourier decomposition obtained from the tangential total pressure distribution at 96.7% span; only the location close to the shroud is shown where strong wakes are present in that region.

A fairly good agreement is observed, even though CFD tends to overestimate harmonic content (i.e. wake strengths). Although many harmonics have low magnitudes, they may interact with the downstream impeller generating unsteady loading variations; indeed, as it will be evident when analyzing the vibratory response due to 7/Rev excitation, a relatively low amplitude circumferential harmonic may lead to non-negligible response level due to the aerodynamic-mechanical mode shapes coupling. Note also that the inlet plenum contains a wide range of circumferential harmonics due to the interactions between straighteners and the intrinsic asymmetry of this component.
In order to determine if the impeller is exposed to resonant vibratory response and to prevent high cycle fatigue failures during service, an FEA cyclic symmetric pre-stressed modal analysis was conducted during the design cycle and prior to testing to extract all natural modes in the band of relevant aerodynamic excitation frequencies. In this type of analysis, the impeller is assumed to be perfectly tuned and the effects of sector-to-sector variability are ignored.

Figure 5: High Resolution Flow Field Scanning, Normalized Total Pressure

Figure 6: Inlet Plenum Harmonics in Shroud Region – CFD vs. Test

These impeller modal results were combined with the 7/Rev inlet plenum aerodynamic excitation frequencies to produce the interference diagram presented in Figure 7. The Fourier orders of the tuned impeller modes are identified and the modes are grouped according to the impeller structural component with the dominant modal participation. The impeller has splitter blades, i.e. a set of long blades paired by a set of shorter blades. Since the impeller main blades are more flexible than the splitter blades and the disk, the lowest modes form a family of main blade first bending mode shapes with 0 to 5 nodal diameters. The 7/Rev excitation is represented by a red vertical line covering the range of shaft speed from 80% to 105% of the test rig nominal speed. The impeller strain gauge responses clearly show a fully resolved resonance peak, Figure 3.
It is well established that the aeromechanical damping is the sum of material, mechanical and aerodynamic damping generated during one cycle of vibration in any particular impeller mode shape. The material damping is related to internal energy dissipation, which may change due to metallurgical processes, see also Gibert et al. (2012). For one-piece impellers no energy dissipation due to friction is present so the second term is commonly neglected. The aerodynamic damping is an inviscid phenomenon resulting from the unsteady pressure response of the impeller primary flow path and secondary flow path as the impeller walls vibrate in resonance in a natural mode of vibration. Both the material damping and the aerodynamic damping are driven by the mode shapes and resonance frequency. Typically, the higher the resonance frequency the higher the material damping and aerodynamic damping. The effect of the mode shape is difficult to appreciate and this is why having a numerical prediction capability is important.

For any impeller natural mode, the critical damping ratio may then be expressed as the sum of pressure dependent (or better, density dependent) aerodynamic damping and the pressure independent material damping:

$$
\zeta_{\text{model}} (\rho) = \zeta_{\text{mat}} + \zeta_{\text{aero}, \rho}
$$

Material damping values for the different mode shapes have been obtained through a dedicated ping test campaign; the $\zeta_{\text{mat}}$ value for the main blade first flexural mode under investigation has been added to numerical predictions. Material damping values found in present test case are within the ones which have been obtained, in other cases, for first flexural mode shapes.

Figure 8 (left) shows the measured damping values for the mode shape under analysis in red symbols. A linear increase of aeromechanical damping with density is observed in accordance with the physical dependency on density. The spread in experimental data at constant density is mainly related to gauge-to-gauge and sector-to-sector variability.

Predicted aeromechanical damping, obtained as per Equation 4 adding material damping to aerodynamic damping, is then compared to experimental data. The value of predicted damping, shown by the green symbol, is in good agreement with experimental data. Since the predicted
damping is within range of data variation, this value will be used for the response level prediction of the impeller.

Figure 8: Main Blade 1st Flex - Normalized Damping (left) and Normalized Response Level (right)

Numerical Goodman Diagram

Now that the details of both aerodynamic and mechanical modal shapes have been covered, the physical alternating stresses acting on impeller surfaces can be computed starting from the modal force computation, and using the predicted damping. With a single mode structural dynamics approach, the physical stresses $\sigma_{phys}$ at any node of the impeller finite element model can be expressed as follows:

$$
\sigma_{phys} = \frac{F_{modal} \cdot Q_{modal}}{(2\pi f_{modal})^2} \cdot k \cdot \sigma_{modal}
$$

[5]

Where $f$ is the impeller mode natural frequency (predicted with a finite element analysis) and $\sigma_{modal}$ is the modal stress component at a node of the impeller finite element model; $Q$ is the aeromechanical damping amplification or quality factor and $k$ is a factor introduced to take into account blade-to-blade or sector-to-sector variability due to the manufacturing and balancing processes.

To begin with, the main blade first flex mode response level will be analyzed. Response levels are expressed as a percentage of a maximum allowable strain at the SG position that makes the Goodman diagram critical; i.e., reaching maximum strain at the gauge position means that one reaches the Goodman line.

Figure 8 (right) represents all the data acquired from the main blade strain gauges as function of inlet flow coefficient; different colors refer to the different instrumented sectors. We have averaged strain gauges response over all sensor positions and operating conditions and used this value to normalize the presented strain gauge response.

An increasing response level trend with flow coefficient is evident. Alternating stresses increase by a factor of 3-5 from lower to higher flow coefficients. Moreover, as also highlighted while damping was analyzed, a sector-to-sector variability is present, making the response level at choke variable by ±40% between the three instrumented impeller sectors (1, 2 and 3).

At choke flow, the predicted response level for the tuned impeller is represented with the green marker in Figure 8 (right); a $k$ value of 1 has been considered. Prediction falls exactly in between the variability range, though stressing the importance of taking into account the blade-to-blade variability, as expressed by factor $k$, to predict the level of the maximum responding impeller sector. For the present case, with a $k = 1.4$ maximum measured response may be obtained.
The tuned response level shown in Figure 8 (right) has been used to compute alternating stress that is represented on the numerical Goodman diagram in Figure 9 for the Main Blade 1st flexural mode shape with 4 nodal diameters under analysis. The impeller surface is decomposed into the three impeller components; in blue the main blade passage walls, in red the splitter blade and in black the disk rear surface in the secondary flow region. Predicted alternating stresses, which are consistent with experimentally determined scope limits, moreover showed that no high cycle fatigue issues are expected for this design.

**Figure 9: Main Blade 1st Flexural Mode Goodman Diagram for Predicted Response Level**

**CONCLUSIONS**

This paper describes the numerical procedure to investigate centrifugal impeller forced response, and its validation as applied to a real case. The methodology predicts both aerodynamic damping and modal force, thereby allowing the determination of a purely numerical Goodman diagram.

The procedure has been validated against test data obtained through an aeromechanical test campaign performed on a single stage rig. In particular, a main blade first flexural mode shape excited by inlet plenum distortions was selected as the test case for the comparisons.

The predicted level of aerodynamic damping is in good agreement with measurement. This value together with the modal force obtained by combining the unsteady pressure field, coming from a non-linear unsteady calculation, with the modal displacements, determined by a finite element analysis, allowed estimation of the vibratory stress level. Notably, the predicted vibratory response is quite close to mean value observed in an experimental test, in which blade-to-blade variability was also investigated.

The analysis conducted in the framework of this work also demonstrated that the analysis of the circumferential harmonics of the predicted flow field alone is not enough to assess the risk level associated to a particular excitation. In fact, the unsteady analysis and the coupling between the resulting pressure field, i.e. the aerodynamic forcing mode shape, and the mechanical mode shape are indeed necessary.

The quality of the prediction is generally good, thereby allowing the use of this method to real design cases that require a special aeromechanic attention. This level of comparison has been also
achieved and published in previous works with reference to damping computation for different mode shapes and operating conditions.

This prediction process is now being used routinely as part of the design verifications to quantify aeromechanical risk for new products.

REFERENCES


