Modal analysis of the nozzle guide vane in low pressure turbine system of aircraft engine

R Robak¹,² and M Szczepanik³

¹ Silesian University of Technology, Faculty of Energy and Environmental Engineering, Konarskiego 18, 44-100 Gliwice, Poland
² AvioPolska Sp. z o.o., Michała Grażyńskiego 141, 43-346 Bielsko-Biała, Poland
³ Silesian University of Technology, Faculty of Mechanical Engineering, Konarskiego 18A, 44-100 Gliwice, Polska

*Corresponding author’s e-mail: Rafal.Robak@avioaero.it

Abstract. The aim of the article is to present modal analysis of nozzle guide vane in low pressure turbine system of aircraft engine. Comparison of results obtained by means of finite element analysis with results of experimental one in terms of frequencies and modal forms are presented. Additionally, the article presents consistent results which will be used in further analyses and researches related to the optimization of dynamic characteristics with application of artificial intelligence.

1. Introduction
Modal analysis [1], [2] is fundamental assessment in the design phase of nozzle guide vane in low pressure turbine system. Evaluation is crucial for new concept design but also in case of design modification domain. Technical requirement is to ensure appropriate durability level (number of flight cycles) and reliability of the system. Understanding of dynamic behavior is one of a key element in the high cycle fatigue (HCF) evaluation. Finite element method (FEM) analysis [3] are widely used in new product introduction phase to verify modal characteristics with respect to operating range and engine orders. Verification at early stage of design is providing an opportunity to address dynamic issues before first prototype enters in the engine certification tests.

2. Modal analysis
The result of the analysis is a natural frequency and mode shape. Basing on the modal shape additionally from analysis can be obtained modal strain and stresses. Mode shapes are normalized to the unitary modal mass or unitary displacement therefore obtained strains and stresses are not real and need to be adjusted using dedicated experiment or frequency response analysis. Nevertheless, information about strain and stress patterns are important in order to optimize strain gage position and identify critical location from HCF perspective.

2.1. Analysis theory
Basic system to demonstrate relationship between modal and mechanical characteristics is mass-spring system presented in figure 1.
Natural frequency of the system is evaluated using equation 1.

\[ f = \frac{\omega}{2\pi} \text{ Hz}; \quad \omega = \sqrt{\frac{k}{m}} \text{ rad/s} \]  

(1)

where: \( f \) and \( \omega \) – natural frequency, \( k \) – stiffness, \( m \) – mass.

Understanding of basic system and relationship between stiffness, mass and natural frequency is crucial to develop proper strategy to resolve dynamic problems (increase stiffness and/or reduce mass in order to raise a natural frequency).

2.2. **Campbell diagram**

Information about dynamic characteristic needs to be verified together with engine order and assessed in the operating range against potential resonance points (aligning of natural frequency and excitation source frequency; harmonic function) as presented in figure 2. Identified crossing not necessarily means resonance because structure could still have enough damping to prevent amplitude growth however such condition is not desired from durability standpoint (high dynamic stress or extensive wear problems).

Identified resonance appears together with assumption of not adequate damping.

Typical engine input in the turbine are mechanical extortion caused by rotor imbalance and aeromechanical due to blade passing phenomena (down and upstream excitation).

2.3. **FEM analysis**

Finite element model in order to evaluate natural frequency and modal shape is expressed by equation 2.

\[ M\ddot{x} + Kx = 0 \]  

(2)
where: \( M \) – mass matrix, \( K \) – stiffness matrix, \( x \) – displacement

In order to successfully perform analysis by means of finite element method [4] the applied numerical model should be possibly consistent with real device and should include all effects which may affect results. Typically, in the analysis of the nozzle guide vane, the important factors are temperature and presence of adjacent components since are impacting model stiffness.

![Diagram of nozzle guide vane components](image)

**Figure 3.** First system mode – pendulum - axial

Figure 3 shows first system mode of nozzle guide vane with pendulum axial mode shape. Making more detailed verification on strain energy distribution above system can be simplified to the basic mass-spring system.

| Mass-spring system                     | Strain energy contribution |
|----------------------------------------|----------------------------|
| Nozzle guide vane (mass element)       | 5%                         |
| Nozzle support cone (spring element)   | 95%                        |

**Table 1.** Modal strain energy distribution for pendulum mode-shape

Crossing of natural frequency with engine input can be eliminated by increasing a stiffness associated with high strain energy contribution and reducing a mass on nozzle side.

3. **Modal analysis validation**

Main purpose of the validation is to benchmark airfoil modal results between FEM model and component test (real hardware) results. This step demonstrates finite element model consistency in terms
of frequencies and modal shapes. Experiment is conducted in ambient temperature by use of the special fixture. The device (special grip) is subjected to the harmonic loads with frequencies identified by ping test. FEM model has been constrained by use of the zero displacements in all three directions at outer band forward and rear hook as presented in figure 4.

![Component fixed at outer band forward and rear hook.](image)

**Figure 4.** FEM model constrains

Model discretization has been completed with tetrahedral elements considering mid side nodes (quadratic shape function) in order reflect correct model stiffness. Modal analysis uses Block Lanczos eigen solver method with results normalized to the unitary modal mass.

**Table 2.** Comparison of modal analysis results coming from FEM and experiment tests

|                  | Numerical prediction | Component test results |
|------------------|----------------------|------------------------|
| Fifth flexural airfoil mode (5F) |                      |                        |

**Table 3.** Comparison of modal analysis results coming from FEM and experiment tests

|                  | Numerical prediction | Component test results |
|------------------|----------------------|------------------------|
| The second flexural airfoil mode (2F) |                      |                        |
Mode shape for both FEM model and component test results present the same modal family; fifth flexural mode for table 2 results and the second flexural mode reported in table 3. Maximum amplitude occurs at trailing edge section of the aerodynamic profile. Numerical model prediction for presented in table 2 mode shape frequency is 7497 Hz compared to the component test results 7560 Hz. Difference is well below 1% and acceptable from design quality standpoint. Comparison proves good accordance between results obtained by means of FEM analysis and real system tests. All aerodynamic profiles of the nozzle guide vane show the same displacement pattern however in real hardware slight variation can be observed. The main reason is the fact that numerical model by the definition accounts nominal condition of the geometry (design intent) while in real component variation within tolerance can be observed and affects a modal pattern.

4. Summary and conclusions
The purpose of the test was to confirm the correctness of the FEM model in the context of the obtained modal forms and thus also the deformation distributions, which are crucial in the subsequent phases of validating the structure dynamics. Numerical models take into account not only the complex geometry of the component, but also high-tech materials (directly solidified or single crystal alloys) that exhibit the characteristics of orthotropic materials. Numerical simulations of real system with complex geometry made and materials, gives proper results which are in good accordance with real experiment. Positive outcome from validation is providing confidence in the sequential steps basing on numerical models (e.g. definition of strain gage position and orientation, prediction of HCF critical location).

References
[1] Geradin M and Rixen D J 2015 Mechanical Vibrations (Wiley)
[2] Case J., Chilver A. H. 1986 Strength of materials and structures (Baltimore: Edward Arnold)
[3] Zienkiewicz O.C. 2013 The Finite Element Method: Its Basis and Fundamentals, ELSEVIER
[4] https://www.ansys.com/