Transient flow in a compressor blade row for a periodic vibration motion

Moumen Idres 1, Mohamed Labanie 1 and Mohamed Okasha 1

1Department of Mechanical Engineering, International Islamic University Malaysia (IIUM), Jalan Gombak, 53100 Kuala Lumpur, Malaysia

Email: midres@iium.edu.my

Abstract. The goal of this work is to conduct a transient compressor blade row flow simulation as part of blade flutter modeling. An integral step of blade flutter modeling is the calculation of the aerodynamic damping factor as a function of the possible vibration mode shapes. Using Fourier method, the number of blade passages required for transient flow analysis is kept to a minimum of two for all vibration modes. In this work, a compressor rotor blade row is considered. The vibration modes are obtained using ANSYS mechanical, then, unsteady flow is obtained for vibrating blades with a harmonic motion. Work of the flow on the blade is calculated and hence the aerodynamic damping is obtained.

1. Introduction

Aircraft gas turbine engine designers were faced with compressor blade flutter in the middle of 50s while developing the second generation of jet engines. At the present day, a huge theoretical and practical experience has been accumulated. Typically, regions of the various flutter types are plotted on the compressor operating map [1-6] as shown in Figure 1. This map clearly shows that possible flutter regions are mainly located near surge line or significantly lower than the compressor operating line. When operating in these regions, the air flow in blade passages is unsteady with vortices and shocks. The only exceptions are regions bounded by lines 3, 4, and 6. They lead to the blade flutter near operating line for unstalled flow.

Figure 1. Compressor operating map [2]: 1- surge line; 2-subsonic stalled flutter; 3- bending-torsional flutter line (expected); 4- supersonic unstalled flutter induced by shocks; 5-supersonic stalled flutter; 6-supersonic unstalled flutter; 7– choking flutter; 8-operating line.
The problem of numerical flutter prediction for compressor or fan blade wheels is associated with coupled aeroelastic problem, which needs a solution for unsteady flow in blade passages. Modern computational codes, such as ANSYS CFX, Star-CCM, Fluent, FlowVision, etc. provide reliable airflow parameters in blade passages near the operating line, where the flow is un stalled. For surge line and for stalled flow, it is very difficult (and usually impossible) to determine reliable flow parameters numerically. However, for the design phase of modern gas turbine engine compressors and fans the most important is to suppress flutter around the operating line with required safety margins.

In this paper, a rotor blade row of an axial compressor is considered. The blades are vibrating with mode shape and frequency corresponding to one of the vibration modes. Using Fourier Transformation model, transient flow requires the usage of a double passage. Simulation is performed using ANSYS CFX 16.2 employing Transient Blade Row Model. The Fourier Transformation method is less expensive and is reliable when it comes to the point of flutter and aerodynamic damping.

2. Methodology
For a metal blade vibrating in air, the influence of the flow on vibration modes is negligible. This assumption is valid for the case of sufficiently stiff blades, when flow disturbances excited by small blade vibration have no significant effect on blade eigenmodes. Therefore, the airflow can result only in small additional damping (for stability case), or additional energy inflow (for flutter case) without change of natural modes and frequencies.

\[ \Delta E = W = \int_{t_o}^{t_o + T} \int_S p(x,y,z,t) \cdot \vec{n}(x,y,z,t) \cdot \vec{v}(x,y,z,t) \, ds \, dt \]  

(2.1)

Where \( T = 1/f \) is the blade oscillation period (\( f \) is the natural frequency). \( S \) is the blade surface, \( p \) is the flow pressure, \( n \) is the blade surface normal, \( v \) is velocity of the blade. The key goal is then estimating the work per cycle performed by the fluid on the blades. Positive work per cycle of the fluid on the blades results in negative aerodynamic damping. Unless sufficient mechanical damping is available in the system to cancel out the negative aerodynamic damping, flutter would occur.

An accurate prediction of the unsteady pressure distribution consisting of its amplitude and phase with respect to the blade displacement provides a mean of validation of the CFD method when compared to experimental results. Under the assumption of having the blades of the disk vibrating at the same frequency, mode shape and amplitude with a phase lag on time when they reach their
maximum amplitude, it is said that the blades are vibrating in a travelling wave mode (TWM). The phase lag between the vibratory motions of two adjacent blades is known as Interblade Phase Angle (IBPA) and it depends on the number of blades (N) and the nodal diameter (ND). The nodal diameters are lines of zero displacement during the vibration of the disk. They go across the disk passing through its center. For each nodal diameter, a forward and a backward travelling wave mode exists. Figure 4 demonstrates an example of the nodal diameter pattern and the resulting instantaneous wave mode.

![Figure 4. Nodal diameter modes and wave modes for a blade row.](image)

For modeling forward (or backward) traveling wave, which is typical for cascade flutter phase lag (-\(\alpha\)) and lead (+\(\alpha\)) with respect to the middle blade are specified for neighboring blades, where the phase shift (\(\alpha\)) = \(2\pi m/N\) (\(m\) is the number of nodal diameters). The work done by steady component of pressure during one harmonic oscillation is zero. The work associated with the steady pressure can be non-zero if there is a phase shift between different blade points (i.e., eigenmodes are complex), as explained in [7]. However, we assume that each blade oscillates in form of standing wave, such that there is no phase shift between points of each blade (though the phase shift between different blades is non-zero), hence the eigenmodes are real. In this case, the work done by the steady pressure is zero. Therefore, we will assume that the pressure (\(p\)) is the unsteady pressure due to blade oscillation.

Flutter analysis consists of four stages:
1. Modal analysis of elastic blades. Interpolation of mode shapes by Lagrange polynomials.
2. Steady-state flow analysis.
3. Unsteady flow analysis with blades oscillating (i.e. fluid mesh moving) in a specified vibration.
4. Calculation of work done by pressure for the middle blade and check of flutter criterion.

Steps 3 and 4 are executed for each vibration mode potentially sensitive to flutter. Work is calculated for the last of several simulated cycles of oscillations, such that the flow response to the blade oscillations is pure harmonic. Three periods is typically enough to have harmonic response. Structural modal analysis is performed using ANSYS Mechanical FE software. For fluid flow analysis, we use ANSYS CFX 16.2.

### 3. Results and Discussion
In this work, we will model a nodal diameter (ND) of four using the Fourier transformation approach with two passages. The machine is rotating at 1800 [rad/s]. The inlet boundary condition is modeled as total pressure and total temperature in the stationary frame, with a specified flow direction in the cylindrical components. The outlet boundary condition is set to an average static pressure of 138 [kPa], varying in the radial direction only. The blade vibration is modeled as forced periodic motion at a fixed frequency with a specified interblade phase angle. The frequency and displacement profile (mode shape) are obtained from cyclic symmetry calculations in ANSYS Mechanical using a single blade model, and exported to a file that is been used later as input for ANSYS CFX 16.2. For this case the vibration frequency is 1152.13 [Hz], and the maximum displacement for the mode shape is 0.00129 [m]. The surface of revolution mesh motion boundary condition is used at the shroud to
model the sliding of the mesh along the surface. Figure 5 shows the two passages needed for Fourier method. The interface between the two passages is used to store Fourier coefficients. The mesh and boundaries are shown in figure 7. Phase-shifted periodic boundary conditions are used at the interface between passages.

![Figure 5. Double passages for Fourier Transformation Method.](image)

Figure 5. Double passages for Fourier Transformation Method.

![Figure 6. Mesh and boundary condition conditions.](image)

Figure 6. Mesh and boundary condition conditions.

The main settings used in the simulations are provided in Table 1. It should be noted that double precision in the calculations is required when using the FT method to avoid truncation errors when storing the temporal Fourier series data on the sampling interface. According to recommendations from the ANSYS CFX user’s manual, the number of time steps per period when using the FT method should be dependent on the nodal diameter. Convergence criteria for steady and transient simulations are different. For steady simulations, the solution is considered converged when the residuals of mass, momentum, energy and turbulence equations reach $10^{-6}$. The solution of transient simulations is considered converged when fluid properties as pressure, velocity, temperature and integrated flow quantities as force and work on the blade reach a periodic behavior. The transient simulations included in this study converged to less than 0.01% variation in work per cycle between periods.

| Property                  | Value/Option                                      |
|---------------------------|---------------------------------------------------|
| Analysis Type             | 3D RANS - Transient Blade Row (TBR)               |
| TBR Model                 | Fourier Transformation                            |
| Flow Type                 | Subsonic (axial inlet velocity = 1 m/s)          |
| Turbulence                | K-ε (5% intensity)                                |
| Inlet Pressure            | 101 kPa                                           |
| Inlet Temperature         | 288 K                                             |
| Outlet Pressure           | 138 kPa                                           |
| Vibration Mode            | 1152.13 Hz, max. amplitude of the mode shape 0.00129 m, ND=4 |
| Periods                   | 10                                                |
| Cycles Per Period         | 72                                                |
| Mesh Stiffness            | Finite Volume Dependent                           |
| Number of Nodes           | 116190                                           |
| Number of Elements        | 105568                                            |

The results showed in figure 7 represent the cycle and number of runs per cycle that was needed till the aerodynamics damping coefficients reached to the convergence results. The run was set to have 10 cycles, with a 72 steps per cycle. Eventually, the simulation needed less than the specified number of cycles to reach convergence values. Curves in figure 7 represent the aerodynamic damping for the two blades considered using one cycle time integration or full time integration. All curves converge to final value of the aerodynamic damping (0.0053). In figure 8, a contour plot of instantaneous work on the is shown. The whole geometry of the cascade is assumed to be perfectly tuned in terms of mass, geometry and material stiffness. Higher work near to the tip of the blade is observed leading to
instability effect. Work distribution along the blade surface might be used to suggest geometry modifications to reduce flutter instability.

![Image of Aerodynamic Damping Convergence](image1.png)

**Figure 7.** Aerodynamic damping convergence.

![Image of Work Distribution](image2.png)

**Figure 8.** Work distribution along the rotor blades.

### 4. Conclusion

Unsteady flow simulation for a vibrating compressor rotor blade is conducted using Fourier transformation method in ANSYS CFX. The method allows the use of two passages for all vibration modes. Blades are forced to vibrate at one of the natural modes, then, the unsteady flow forces are calculated. Solution was converged after seven oscillation cycles. The aerodynamic damping is calculated which allows the evaluation of flutter in later aeroelastic analysis.
Acknowledgement
The authors gratefully acknowledge the financial support from kulliyah of engineering, International Islamic University Malaysia (IIUM).

References
[1] Shrinivasan A V 1997 *Journal of Engineering for Gas Turbines and Power* **119** 742-775
[2] Bendiksen O 1988 *Journal of Propulsion and Power* **4** 164-172
[3] Petrie-Repar P, McGhee A, Jacobs P and Gollan R 2006 *ASME Turbo Expo* (Barcelona: Spain)
[4] Marshall J G and Imregun M 1996 *Journal of Fluids and Structures* **10** 237-267
[5] Böles A and Suter P 1986 *Transsonische Turbomaschinen* Wissenschaft und Technik Taschenausgabe, Karlsruhe, G. Braun.
[6] Khorikov A A, Danilkin S Y 2011 *Herald of Samara State Aerospace University* **2** 57-63
[7] Kiell R., Barter J, Chernysheva O and Fransson T 2006 *Unsteady aerodynamics, aeroacoustics and aeroelasticity of tubomachines* Springer.