COMPUTATIONAL MECHANICS 2023
38th conference with international participation
Srní
October 23 - 25, 2023

Complex simulation procedure to predict flutter of steam turbine rotor blades

V. Sláma[a], D. Hron[b], A. Macálka[c]

[a]Department of Research and Development, Doosan Skoda Power, Tylova 1/57, 301 28 Plzeň, Czech Republic
[b]Department of Mechanics, Faculty of Applied Sciences, University of West Bohemia, Univerzitní 8, 301 00 Plzeň, Czech Republic
[c]NUM solution s.r.o., U Pergamenky 1145/12, 170 00 Praha 7 - Holešovice, Czech Republic

1. Introduction

The need for robust numerical tools to predict the flutter of rotor blades is a crucial aspect of the aeromechanical design of new modern last-stage blades for steam turbines. Therefore, a reliable flutter calculation procedure has been developed and validated in Doosan Skoda Power. As a result, a unique and úowerful tool FLTR was developed. It also automates the generation of transient input files and the evaluation described in this paper.

In general, the aerodynamic instability of blades, known as flutter, has been a crucial subject of many studies. Measurements were performed to identify the critical flutter parameters, and numerical solvers were used to predict turbine blade flutter and effective ways to keep steam turbine last stage blades' vibration amplitudes low were also proposed [1]. In this paper, the presented flutter calculation procedure is shown for the case of an 1100 mm long last-stage rotor blade, see Fig. 1 on the left, which was created for high backpressures [3].

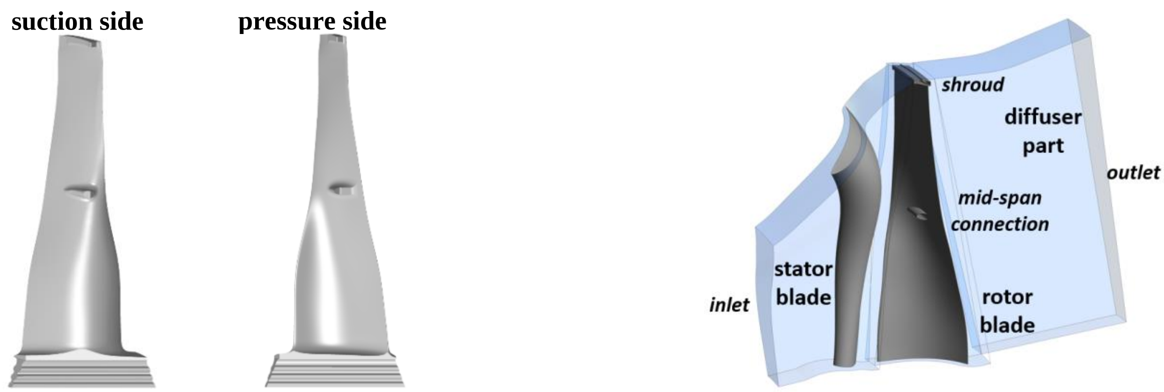

Fig. 1. 1100 mm long last stage rotor blade (left), CFD (right)

2. Flutter calculation procedure

The numerical investigation of flutter is performed using a one-way decoupled method with the workflow shown in Fig. 2 on the left. Higher fidelity CFD models [5] would be more precise. They could help to validate the currently used approach further. However, such models still need to be fully prepared, tested, and validated for such a complex geometry, which is a last-stage blade that includes all detailed features like a mid-span connection and a shroud and operates under different boundary conditions from transonic to supersonic cases.

The reliability of the used commercial code ANSYS CFX was proved by detailed studies described e.g. in [4]. Besides, the calculation procedure has been refined in recent years, as shown in Fig. 1 on the right. The analysis is defined as a transient blade row with the time integration method. This flutter calculation procedure provides following advantages: (i) pressure and other variables distributions are more precise at the rotor blade domain inlet which is crucial, especially for transonic flow cases which are very often for ultra-long LSBs (last stage blades), (ii) effects such as shock waves going from the stator domain to the rotor domain and vice versa are included, (iii) a calculation is stable.

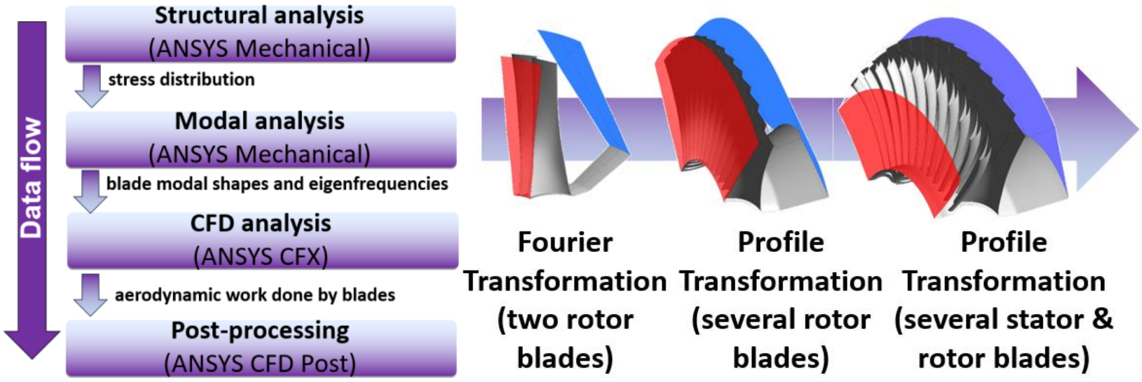

Fig. 2. Flutter analysis workflow, one way decoupled method (left), flutter calculation procedure development (right)

The structural analysis with defined periodic conditions is carried out concerning large deformations using element SOLID 186 with a defined contact at coupling elements consisting of a straight fir tree at the hub, a mid-span connection (also called a tie-boss) and an untwisted integrated shroud. It is shown in Fig. 3 on the left. This modal analysis respects all crucial phenomena, such as deformations of the blade caused by the operation speed, an imperfect blade root-disc groove connection, spin softening, and tensile stress softening. Further details are published in [2]. Besides the required modal analysis, other essential analyses are also carried out: (i) natural frequencies sensitivity to the size of contact areas of the connecting elements, which can differ due to the imperfect manufacturing, (ii) low cycle fatigue for an estimation of an allowed number of start-ups concerning a nonlinear material behavior, (iii) a rotor coupled analysis to investigate the interaction of the blades with the rotor.

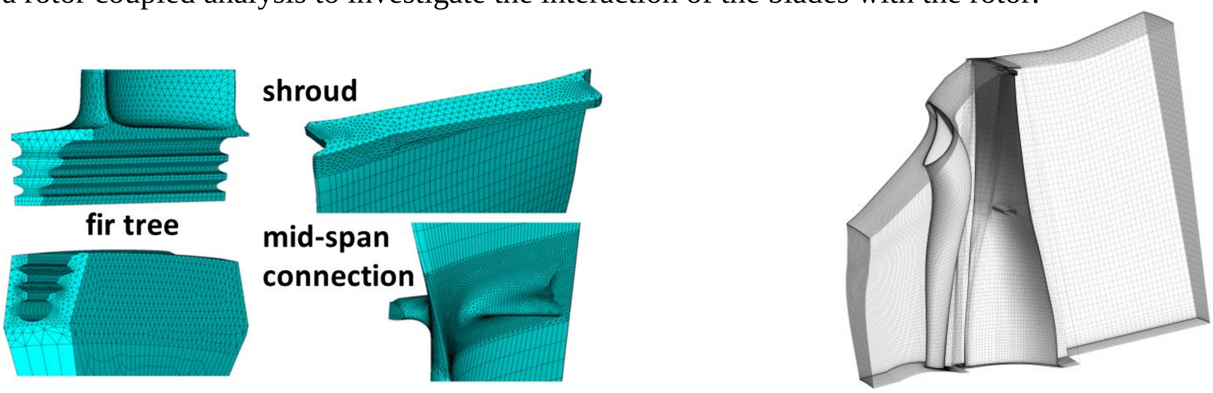

Fig. 3. Details of the structural model parts (left), CFD mesh (right)

The model for CFD analysis, Fig. 1 on the right, is created for ANSYS CFX using a mesh of finite volumes. The detail of the mesh is in Fig. 3 on the right. In order to reduce computation time, only fully structured meshes are used. It means that a hexahedral mesh with an appropriate boundary layer on the blades and the adjacent walls is created, ensuring that a maximum value of y+ is lower than 5. It is a recommended value for the used turbulent model.

The CFD analysis consists of two steps: the steady-state analysis and the unsteady analysis. For the steady-state analysis, the Reynolds stresses terms of the RANS equations are computed using the two-equation eddy-viscosity SST k-ω turbulence model with an AWF (automatic wall function). The equilibrium steam model based on the IAPWS-IF97 steam properties database is used. A total pressure profile, a total enthalpy profile, vectors of flow velocity components, and turbulence intensity define the inlet boundary condition. An average static pressure and a radial equilibrium condition define the outlet boundary condition.

A convergence of the steady-state analysis is said to be achieved when averaged residuals of momentum and mass variables reach the target value of $1 \cdot 10^{-4}$, and the monitored thermodynamic efficiency oscillations are lower than 0.01%. Furthermore, the global imbalances have to be lower than 0.01%. The unsteady analysis uses a full-scale time-marching 3D viscous model to obtain the solution of the URANS equations in the time domain. A high-resolution scheme deals with the advection term, and a second-order backward Euler scheme is used for the transient term. The time period is specified as the reciprocal value of the blade eigenfrequency according to the relevant ND (nodal diameter). The total time duration of each simulation is defined as the total number of periods per run. The blade motion is defined according to physical deformations obtained from the modal analysis results.

3. Post-processing

The energy method is used to compute aerodynamic forces during the unsteady simulation. It assumes that flutter occurs with the blade's natural mode shape and makes a flutter prediction by calculating the energy exchanged between the blade and the flow field. The required aerodynamic work *W* per one vibration cycle is calculated as

$$W = -\int_{t_0}^{t_0+T}\int_{A} p\mathbf{Vn}\,\mathrm{d}A\mathrm{d}t \tag{1}$$

where *T* is the period of one vibration cycle, t_0 is the time at the start of the cycle, *p* is the static fluid pressure, *V* is the velocity of the blade due to the imposed vibrational displacement, *A* is the surface of the blade and *n* is the normal unit vector to the surface *A*.

In order to generalize the results, the aerodynamic work *W* has to be normalized. There are two basic types of normalization: the normalization using the inlet flow energy or the normalization using the blade average kinetic energy. In this case, the second option is used. It means that the damping ratio ξ is calculated as:

$$\xi = \frac{-W}{8 \cdot \pi \cdot E} = \frac{-W}{2 \cdot \pi \cdot m \cdot \omega^2 \cdot a^2} \tag{2}$$

where *E* is the blade´s average kinetic energy, *m* is the blade (equivalent) modal mass, *a* is the modal amplitude (maximal displacement), and ω is the angular frequency. When the damping ratio is negative, it indicates that the blade is unstable and there can be a danger of flutter.

In this numerical method, mechanical and material damping of the blade is not considered. It means that the blade can be stable even for some negative values of the damping ratio. Therefore, the flutter onset was defined from the results of on-site vibration measurement during the turbine shut-down operations where atypical operating conditions were allowed. The flutter threshold was determined when the vibration limit was close to the defined maximal values. However, in practice, it is not the threshold of flutter. It appears later. In other words, if the values of the damping ratios of the newly designed blade are greater than those of the measured referential blade, the new blade can be considered flutter-free and, hence, safe.

Fig. 4 on the left shows the damping ratio results according to Eq. (2) for various NDs for one turbine operating point. The positive NDs stand for the forward traveling waves, and the negative ones for the backward traveling waves. Results show that the damping ratios are negative for a few negative NDs. In this region, the new blade damping ratios are greater than the ratios of the referential blade. It means that no problems with flutter are expected for the presented operating point of the new blade.

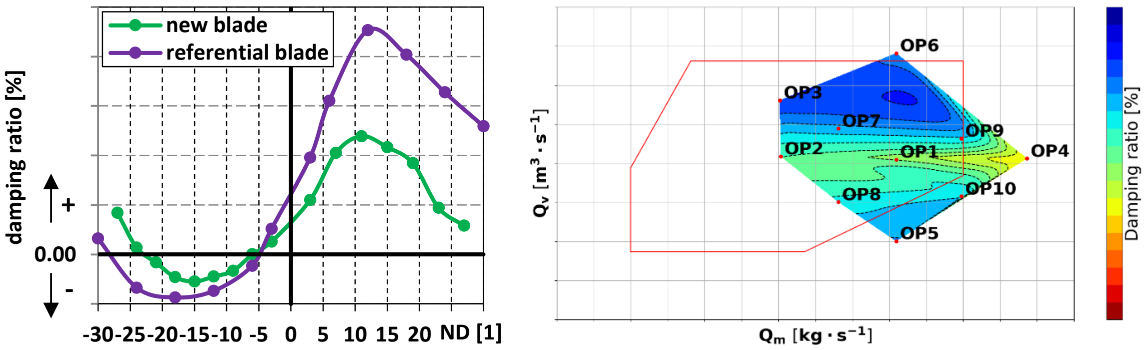

Fig. 4. Damping ratio dependence on nodal diameter (left), flutter map for the newly developed LSB (right)

The results presented in Fig. 4 on the left are just for one operating condition. However, it is necessary to perform flutter analyses for a wider range to avoid the danger of flutter for all turbine operating conditions. Therefore, in order to evaluate and clearly show the blade sensitivity to flutter, the flutter maps were developed. An example of one such flutter map can be seen in Fig. 4 on the right. It shows the dependence of the damping ratio on the mass flow Q_m going through the last stage and the volumetric flow rate Q_v. The red lines depict the region of the LSB operating range. The flutter map is created for the NDs with the most negative values. As a result, the areas of damping ratios with low negative values can be found. Hence, a potential danger of flutter can be detected in the preliminary design phase. If the values inside the operation range were lower than the referential value, using the blade for such operating points would be dangerous. In this case, the blade must be redesigned.

4. Conclusion

The in-house flutter calculation procedure (FLTR tool), which is based on a commercial numerical code ANSYS, has been developed and validated to predict a danger of flutter. Details of this procedure, along with all crucial settings, were shown. Using the S-shaped curves and the flutter maps, which are supposed to be an original and complex approach developed by Doosan Skoda Power Company in cooperation with NUM Solution Company, all required areas in the operating range of the blade were investigated. By comparing the damping ratio results with the referential blade results, it was shown that the new blade satisfies the safety requirements from the flutter occurrence point of view in the required operating range. Thanks to this procedure, any blade can be checked whether it is flutter-free or not.

Acknowledgement

The work of David Hron was supported by the student project SGS-2022-008.

References

[1] Drozdowski, R., Völker, L., Häfele, M., Vogt, D. M., Experimental and numerical investigation of the nonlinear vibrational behavior of steam turbine last stage blades with friction bolt damping elements, ASME Turbo Expo 2015, GT2015-42244, Montreal, Canada, 2015.

[2] Misek, T., Kubin, Z., Duchek, K., Static and dynamic analysis of 48” steel last-stage blade for steam turbine, ASME Turbo Expo 2009, GT2009-59085, Orlando, USA, 2009.

[3] Novak, O., Bobcik, M., Slama, V., at al., Development of highly efficient and robust ultra-long last stage blade for high backpressure, ASME Turbo Expo 2019, GT2019-90350, Phoenix, USA, 2019.

[4] Slama, V., Eret P., Pinelli, F., at al., Experimental and numerical study of controlled flutter testing in a linear turbine blade cascade, Acta Polytechnica 20 (2018) 98-107.

[5] Zorn, J., Davis, R., Clark, J., High fidelity simulation of fluid-structure interaction in a 1½ stage axial turbine, ASME Turbo Expo 2019, GT2019-90972, Phoenix, USA, 2019.