COMPUTATIONAL MECHANICS 2021
36th conference with international participation
Srní
November 8 - 10, 2021

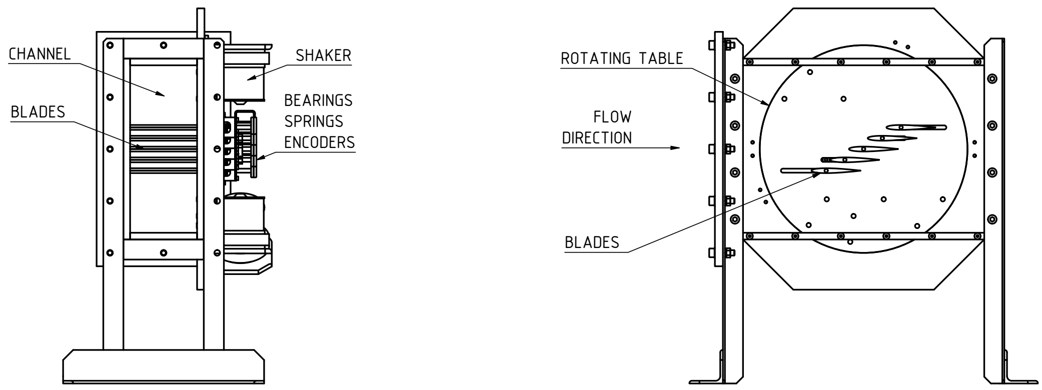
Numerical and experimental investigation of the flow field in five blade linear cascade in subsonic flow

P. Šnábl[a], S. Chindada[a], O. Bublík[b], P. Procházka[a], C. S. Prasad[a]

[a] *Institute of Thermomechanics of the CAS, v. v. i., Dolejškova 1402/5, 182 00 Praha 8, Czech Republic*
[b] *Faculty of Applied Sciences, University of West Bohemia, Univerzitní 8, 301 00 Plzeň, Czech Republic*

In large steam turbines, last stage blades are very long and must be designed very thin and with no shroud to minimize the centrifugal force which leads to low eigenfrequencies and low structural damping. In this case, aero-elastic damping plays important role on last stage bladed disc's dynamics. Three major aero-elastic issues found in turbomachinery as described in [2] are forced response, non-synchronous vibrations, and flutter. Flutter is an unstable, self-excited vibration resulting from coupling between the structural vibrations and unsteady aerodynamic forces. It is clear that this unstable behaviour must be avoided and predictions of flutter behaviour need to be performed during design stage of the turbine.

An experimental blade cascade with five NACA 0010 profiles with one degree of freedom (rotation) has been built in the Institute of Thermomechanics to study flutter phenomenon. The schematic picture of the cascade is shown in Fig. 1 and it is described in [1].



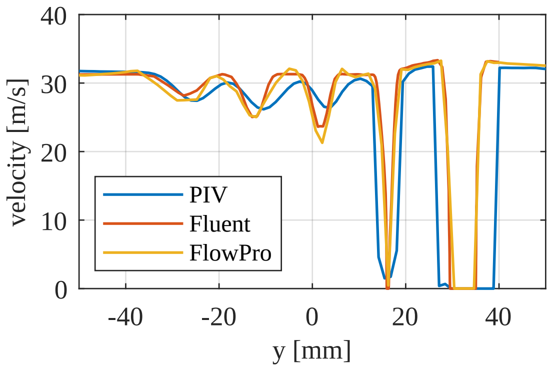
(a) front view (b) side view

Fig. 1. Assembly of the blade cascade

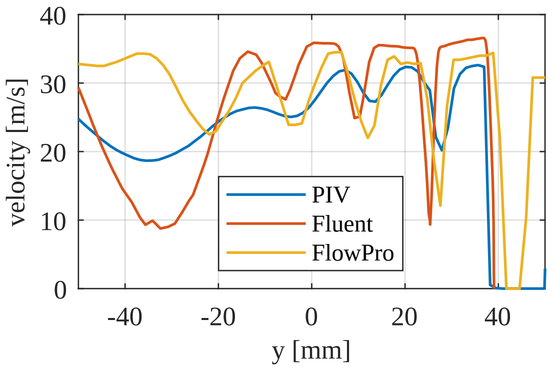
Laser *New Wave Research Pegasus PIV* and high speed camera *Vision Research Phantom* were used for PIV measurement of the flow field in the experiment. The results were compared with calculations in Ansys Fluent and in FlowPro.

Ansys Fluent is well-known commercial software that is used for simulation of flow, turbulence, heat transfer and reactions. 2D model of the cascade was simulated using compressible flow with K-ω SST turbulence model. The fluid parameters correspond to air, turbulent intensity was set to 0.05% and viscosity ratio was set to 1. The velocity inlet was prescribed at the inlet and the value of static pressure was prescribed at the outlet.

FlowPro is a multipurpose open-source CFD software designed for complex fluid flow simulations. The current version of the numerical software is capable of solving for example the system of Euler equations, Navier-Stokes equations, shallow water equations or equations of ideal magnetohydrodynamics. 2D model of the cascade was simulated using the incompressible fluid flow model together with Spalart-Allmaras turbulence model. The fluid parameters correspond to air, boundary conditions were set the same as in Fluent. The second order method in space and time was used.



(a) angle of attack of the cascade 0°



(b) angle of attack of the cascade $-8°$

Fig. 2. Comparison of flow velocities from the experiment (PIV) and calculations (Ansys Fluent and FlowPro) in the vertical plane placed 100 mm from the leading edge of the middle blade

Fig. 2 shows streamwise velocity in vertical plane placed 100 mm downstream from the leading edge of the middle blade. Results from the experiment are compared with the results from both Ansys Fluent and FlowPro for two angles of attack of the cascade, $0°$ and $-8°$. While the calculated results for $0°$ in Fig. 2a correspond well with the measured ones in terms of wake vertical position y and velocities in the wake, the results for $-8°$ in Fig. 2b look very different. Especially the wake region behind the first blade (the region with low velocity on the left side of the graph). There, the results from Ansys Fluent show similar position of the wake but the velocity gradient between the wake and surrounding flow is much higher. Interestingly, the result from FlowPro is different both from experiment and Ansys Fluent result. The wake is positioned higher in the y coordinate and is more narrow. There is an ongoing research to discover the reasons of those discrepancies in the results and to tune the simulations that will simulate also the cases with dynamic movement of the blades and the flutter phenomenon.

Acknowledgement

This research is supported by the research project of Czech science foundation No. 20-26779S "Study of dynamic stall flutter instabilities and their consequences in turbomachinery application by mathematical, numerical and experimental methods".

References

[1] Šnábl, P., Pešek, L., Prasad, C. S., Bula, V., Cibulka, J., Experimental setup and measurement for evaluation of blade cascade stall flutter instability, Proceedings of the 27th International Congress on Sound and Vibration, 2021.

[2] Waite, J. J., Physical insights, steady aerodynamic effects, and a design tool for low-pressure turbine flutter, Department of Mechanical Engineering and Materials Science, Duke University, 2016.