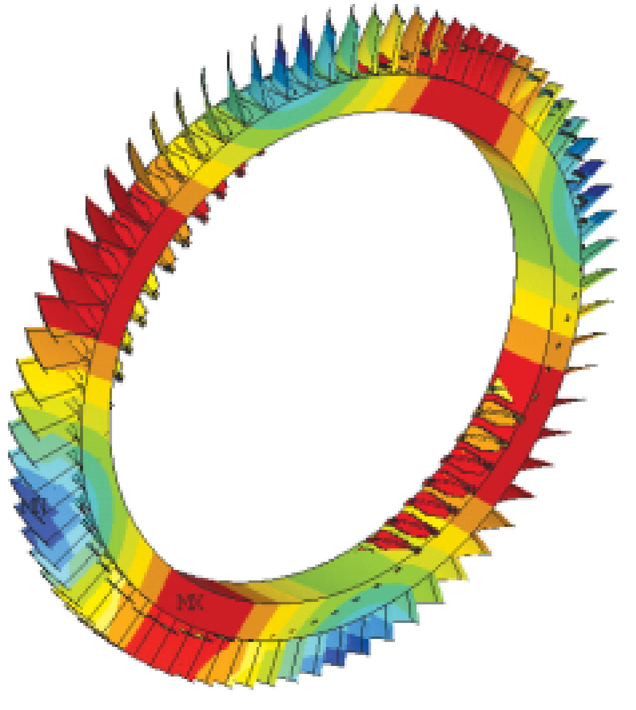
Streamlined Flutter Analysis

Integrated fluid structure interaction enables high-fidelity turbomachinery blade flutter analysis.

By Robin Elder and Ian Woods, PCA Engineers Limited, Lincoln, U.K.
Simon Mathias, ANSYS, Inc.



ANSYS Mechanical analysis tools can predict vibration modes that occur over an entire wheel from a single blade component model. Shown here are exaggerated deformations for a four-nodal diameter mode shape, meaning that the mode repeats itself four times over the entire wheel circumference. Engineers are interested in determining whether vibration modes such as these will be amplified by interaction with the fluid or safely damped out.

The "flutter" of blades within compressors and turbines is a serious cause of machine failure that is difficult to predict and expensive to correct. This aeromechanical phenomenon usually occurs at a blade natural frequency and involves sustained blade vibration resulting from the changing pressure field around the blade as it oscillates. For the process to occur, it is necessary that, over one cycle, there is an input of energy from the gas stream to the blade of a sufficient magnitude to overcome the mechanical damping.

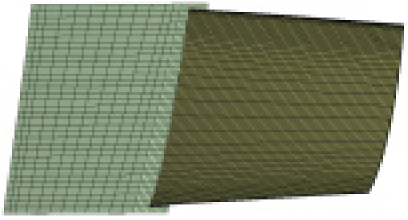
Clearly, flutter is dependent on both the aerodynamic and structural characteristics of the blade, and, until recently, it has been beyond the design capability to satisfactorily investigate and avoid this phenomenon. Historically, empirical design criteria have been used based on parameters involving blade natural frequencies and flow transit times, but these methods fail to take into account generally found vibrational modes or the influence of adjacent blades.

Improvements in unsteady computational fluid dynamics (CFD) capability combined with the ability to easily and accurately transfer information between CFD and finite element analysis (FEA) has enabled the development of an advanced yet efficient and cost-effective methodology for analyzing forced vibration processes.

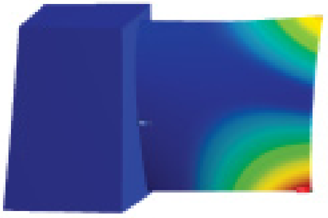
A key enabling development now provided by ANSYS, Inc. is the ability to deform the CFD computational grid in response to deformations at the fluid structure interface and integrate this with unsteady flow computations. The process is straightforward to set up and is facilitated by the intuitive and intrinsic functionality of the user interface and layout in the ANSYS Workbench platform. PCA Engineers Limited, based in the U.K., has utilized this capability by mapping time-dependent deformations computed from a finite element analysis to the CFD computational grid.

As a rule, blade flutter occurs at a blade natural frequency that is determined together with its corresponding mode of vibration using traditional finite element techniques.

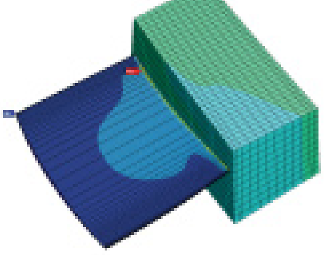
A bladed disc assembly can be classified as a rotationally periodic structure, and, therefore, the mode shape of adjacent blades within a row are fully defined by a phase



Finite element (FE) mesh at the fluid–structure interface



A typical torsional blade mode, where the relative amplitude of each node point on the gas swept surface of the blade is known as a function of time



Equivalent stresses

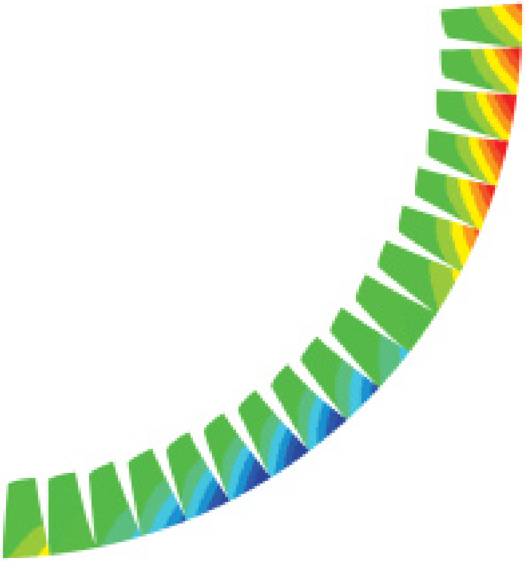
difference. This phase difference (the inter-blade phase angle or IBPA) depends on the number of blades in the row and the number of patterns repeating around the annulus. This latter parameter is often called the nodal diameter (ND) and can move either in the direction of rotation or against the direction of rotation.

The significant development is that this modal displacement information now can be applied to the computational grid and the resulting time varying flow through a blade row as well as the dynamic pressure field over each defined blade calculated using ANSYS CFX software. The computed dynamic pressure distribution and the corresponding modal displacements then are used to compute the work done on the blade over one complete cycle. If the net work done on the blade is positive, then work is being imparted to the blade, creating negative damping, a potentially unstable situation leading to a self-sustained vibration (flutter) likely to cause a material fatigue failure. On the other hand, if the aerodynamic work done on the blade is negative, the blade motion is doing work on the fluid and leads to a stable or damped vibration.

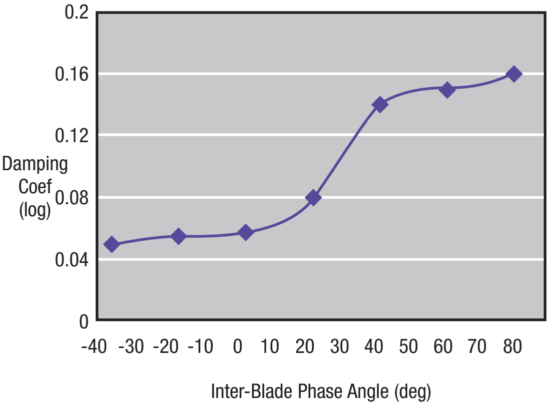
In the aerodynamic damping case illustrated, the blade is stable (no flutter) because the damping is always positive. This information is critical to the designer as blades are relatively easy to modify before manufacture but extremely costly to rectify in an operational plant. By utilizing blade flutter prediction early in the design cycle, costly damage and repairs can be avoided. This integrated design and analysis approach in multiphysics technology from ANSYS can lead to improved quality and dependability of the design process, realizing further cost benefits to clients.

ANSYS, Inc. and PCA Engineers now are applying such technology to a wide range of applications extending from large steam turbines to small turbochargers. These techniques are assisting engineers to design compressor and turbine blading in which both aerodynamic efficiency and structural integrity are paramount over the operational range of the machine. ■

www.pcaeng.co.uk



Deformations of a four-nodal diameter which repeat once over each quarter of the wheel, were exported from the modal analysis vibration mode to ANSYS CFX software as a boundary profile. The mode shape is used to create a periodic boundary motion in the CFD software and to evaluate the net work input due to the blade motion.



Damping coefficients can be calculated from the CFD results. Negative net work input due to blade motion results in a positive damping coefficient. Negative damping coefficients induce sustained blade vibration, or flutter, which could lead to blade failure. These results show positive damping for all inter-blade phase angles.