# Rapid Aeroelastic Analysis of Blade Flutter in Turbomachines

John H. Glenn Research Center, Cleveland, Ohio

The LINFLUX-AE computer code predicts flutter and forced responses of blades and vanes in turbomachines under subsonic, transonic, and supersonic flow conditions. The code solves the Euler equations of unsteady flow in a blade passage under the assumption that the blades vibrate harmonically at small amplitudes. The steady-state nonlinear Euler equations are solved by a separate program, then equations for unsteady flow components are obtained through linearization around the steady-state solution. A structural-dynamics analysis (see figure) is performed to determine the frequencies and mode shapes of blade vibrations, a preprocessor interpolates mode shapes from the structural-dynamics mesh onto the LIN-FLUX computational-fluid-dynamics mesh, and an interface code is used to convert the steady-state flow solution to a form required by LINFLUX. Then LINFLUX solves the linearized equations in the frequency domain to calculate the unsteady aerodynamic pressure distribution for a given vibration mode, frequency, and interblade phase angle.

FIRST MODE, 822 Hz   **SECOND MODE, 1822 Hz**
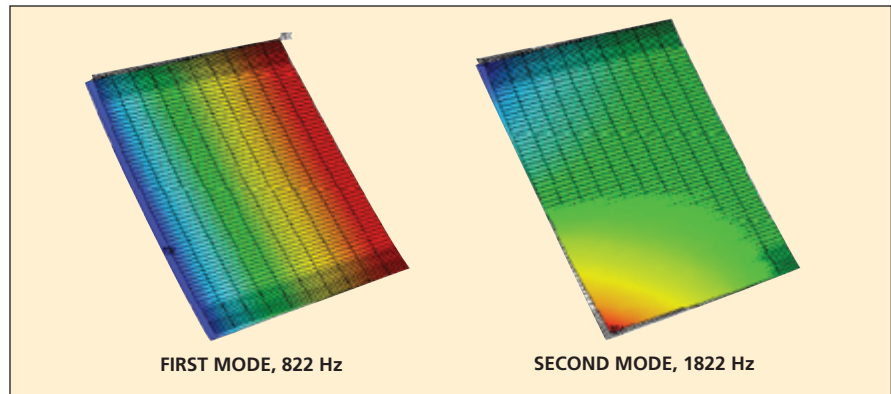
Blade Mode Shapes are shown for first and second modes. The first mode is a bending mode at 822 Hz, and the second mode is a torsion mode at 1,882 Hz.

A postprocessor uses the unsteady pressures to calculate generalized aerodynamic forces, response amplitudes, and eigenvalues (which determine the flutter frequency and damping). In comparison with the TURBO-AE aeroelastic-analysis code, which solves the equations in the time domain, LINFLUX-AE is 6 to 7 times faster.

This program was written by J. J. Trudell, O. Mehmed, G. L. Stefko, and M. A. Bakhle of **Glenn Research Center**; *T. S. R. Reddy of the University of Toledo; M. Montgomery of United Technologies; and J. Verdon of Ohio Aerospace Institute. Further information is contained in a TSP (see page 1).*

Inquiries concerning rights for the commercial use of this invention should be addressed to NASA Glenn Research Center, Innovative Partnerships Office, Attn: Steve Fedor, Mail Stop 4–8, 21000 Brookpark Road, Cleveland, Ohio 44135. Refer to LEW-17880-1.

General Flow-Solver Code for Turbomachinery Applications

Marshall Space Flight Center, Alabama

Phantom is a computer code intended primarily for real-fluid turbomachinery problems. It is based on Corsair, an ideal-gas turbomachinery code, developed by the same authors, which evolved from the ROTOR codes from NASA Ames. Phantom is applicable to real and ideal fluids, both compressible and incompressible, flowing at subsonic, transonic, and supersonic speeds. It utilizes structured, overset, O- and H-type zonal grids to discretize flow fields and represent relative motions of components.

Values on grid boundaries are updated at each time step by bilinear interpolation from adjacent grids. Inviscid fluxes are calculated to third-order spatial accuracy using Roe's scheme. Viscous fluxes are calculated using second-order-accurate central differences. The code is second-order accurate in time. Turbulence is represented by a modified Baldwin-Lomax algebraic model. The code offers two options for determining properties of fluids: One is based on equations of state, thermodynamic departure

functions, and corresponding state principles. The other, which is more efficient, is based on splines generated from tables of properties of real fluids. Phantom currently contains fluid-property routines for water, hydrogen, oxygen, nitrogen, kerosene, methane, and carbon monoxide as well as ideal gases.

This work was done by Daniel Dorney of **Marshall Space Flight Center** *and Douglas Sondak of Boston University. Further information is contained in a TSP (see page 1). MFS-32321-1*

Code for Multiblock CFD and Heat-Transfer Computations

John H. Glenn Research Center, Cleveland, Ohio

The NASA Glenn Research Center General Multi-Block Navier-Stokes Convective Heat Transfer Code, Glenn-HT, has been used extensively to predict heat transfer and fluid flow for a variety of

steady gas turbine engine problems. Recently, the Glenn-HT code has been completely rewritten in Fortran 90/95, a more object-oriented language that allows programmers to create code that is

more modular and makes more efficient use of data structures. The new implementation takes full advantage of the capabilities of the Fortran 90/95 programming language. As a result, the

Glenn-HT code now provides dynamic memory allocation, modular design, and unsteady flow capability. This allows for the heat-transfer analysis of a full turbine stage. The code has been demonstrated for an unsteady inflow condition, and gridding efforts have been initiated for a full turbine stage unsteady calculation. This analysis will be the first to simultaneously include the effects of rotation, blade interaction, film cooling, and tip clearance with recessed tip on turbine heat transfer and cooling performance. Future plans call for the application of the new Glenn-HT code to a range of gas turbine engine problems of current interest to the heat-transfer community. The new unsteady flow capability will allow researchers to predict the effect of unsteady flow phenomena upon the convective heat transfer of turbine blades and vanes. Work will also continue on the development of conjugate heat-transfer capability in the code, where simultaneous solution of convective and conductive heat-transfer domains is accomplished. Finally, advanced turbulence and fluid flow models and automatic gridding techniques are being developed that will be applied to the Glenn-HT code and solution process.