Chinese Journal of Aeronautics 24 (2011) 258-264

Contents lists available at ScienceDirect

Chinese Journal of Aeronautics

journal homepage: www.elsevier.com/locate/cja

# Coupled Fluid-structure Flutter Analysis of a Transonic Fan

ZHENG Yun, YANG Hui*

*School of Jet Propulsion, Beihang University, Beijing 100191, China*

Received 14 December 2010; revised 15 February 2011; accepted 7 April 2011

**Abstract**

A coupled fluid-structure method is developed for flutter analysis of blade vibrations in turbomachinery. The approach is based on the time domain solution of the fluid-structure interaction in which the aerodynamic and structural equations are marched simultaneously in time. The three-dimensional (3D) unsteady Reynolds average Navier-Stokes (RANS) equations are solved with a multiblock finite volume scheme on dynamic deforming grids to evaluate the aerodynamic force. Dual time-stepping technique and an efficient implicit scheme with multigrid are employed to march the solution in time. The blade vibration is modeled with an aeroelasticity model in which blade motion is computed by linear combination of responses of each mode under unsteady loads. The code is validated in prediction of the unsteady flow flutter behavior of an oscillating cascade and is applied to flutter analysis of a transonic fan at the design speed.

*Keywords:* transonic flow; unsteady flow; flutter; transonic fan; fluid-structure interaction; turbomachinery

## 1. Introduction

The design trends for modern commercial aircraft engine are towards high thrust/weight ratio, fuel efficiency and safety[1]. These requirements lead to design fans and compressors with high pressure ratio per stage, thin blades, making the blades more susceptible to fluid induced vibration. Flutter is known as a self-excited vibration of a body in fluid. It poses significant challenge to aero-engine safety. Furthermore, aeroelastic stability causes designers to be conservative in pushing the design of engines to new limits of performance[2]. Prediction flutter with numerical methods is critical for both the safety and the improvement of the aero-engine performance.

Flutter is a multidisciplinary subject combining unsteady aerodynamics and structural dynamics. The simultaneous integration of the equations of motion in time for the fluid and structure allows the correct assessment of the energy exchange between fluid and the blade(s). Flutter occurs when the blade absorbs energy from fluid and the blade vibration becomes unstable. When the aero-damping of fluid dissipates the vibrating blade energy, the amplitude of the vibration reduces, and thus the vibration is stable. When a balance between input energy and dissipation occurs, the blade enters auto-oscillation or limit-cycle oscillation mode (LCO)[3], in which the oscillation amplitude is critical to assess its impact on structure integrity. Flutter is usually associated with fan blades, though other compressor and low-pressure turbine blades may also suffer from such instabilities[4].

Prediction of blade flutter is proven to be a difficult task due to the perplexing phenomena of fluid-structure interaction in turbomachinery. In addition, people do not fully understand the consequences of many additional features, such as flow distortions due to blade-row interaction, coupling of assembly modes, and loss of spatial periodicity (aerodynamic or structural mistuning), acoustic resonance, etc. Time-linearized (frequency domain) approach has

*Corresponding author. Tel.: +86-10-82319391-105.
*E-mail address:* huiyang@buaa.edu.cn

Foundation items: Fan-Zhou Foundation of Beihang University (20090404); "111" Project (B08009)

 doi: 10.1016/S1000-9361(11)60031-9

been extended to model more realistic turbomachinery flows by several researchers, such as He[5], Hall[6] and Bakhle[7]. However, under transonic flow conditions, even small oscillations of the blade may lead to nonlinear flow behavior, such as intermittently choking by an oscillating shock[8], thus rendering the time-linearization methods inadequate in understanding the mechanism of fluid-structure interaction. While the linearized methods are particularly useful for parametric studies in the preliminary design stage[9], aeroelastic equations have to be solved in a time marching fashion in order to include all of the nonlinearities in both the structure and/or the aerodynamics. A review of current state of turbomachinery aeroelasticity was given by Marshall and Imregun[10] where flutter prediction methods were described in detail. The coupled fluid-structure method in solving the aeroelasticity problem has the advantages of modeling time history of the blade vibration, vibration amplitude and energy exchange between two or more discipline thus enhancing the understanding of characteristic of blade flutter. The principle of coupling between structure and fluid for wings was reviewed by Schuster, et al.[11], whereas cascade and fan applications were presented by Gnesin[8], Vahdati[12], Carstens[13], Srivastava[14], et al. with various degrees of accuracy. Cinnella, et al.[15] proposed a new deforming grid technique for aeroelasticity analysis. Aeroelasticity stability computations with energy method are reported by Srivastava[14], Gnesin[16], et al.

Jin and Yuan[17] performed flutter analysis of a turbine blade under a single vibration mode (torsion) with a coupled fluid-structure procedure. However for more general cases, modeling the motion of a tubomachine blade generally requires the inclusion of several modes. Zhang, et al.[18] conducted flutter analysis with an influence coefficient method. Hu and Zhou[19] simulated two-dimensional (2D) unsteady flows around oscillating cascades.

In this study, a fully coupled numerical method to simulate fluid-structural interaction is developed for the flutter analysis of turbomachinery applications. The code is based on solving three-dimensional (3D) unsteady Reynolds averaged Navier-Stokes (RANS) equations for compressible flows on multiblock structured grids. Structural dynamic equations are marched simultaneously with aerodynamic equations in a strong coupled fashion in order to accurately simulate the interaction of aerodynamic force and the structure motion. The accuracy of the procedure is validated with unsteady flow simulation of the 4th Standard Configuration of turbomachinery aeroelacticity and applied to predicting the flutter of a transonic fan at the peak and near stall conditions.

## 2. Aerodynamic Models

Aerodynamic model for fluid-structure interaction is the 3D unsteady RANS equations in relative frame of the reference. The governing equations in integral form are written for a dynamic deforming grid, as

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega(t)} \boldsymbol{q}\mathrm{d}\Omega + \oint_{S(t)} (\boldsymbol{F}_\mathrm{c} - \boldsymbol{F}_\mathrm{v})\cdot\hat{\boldsymbol{n}}\mathrm{d}S = \int_{\Omega(t)} \boldsymbol{H}\mathrm{d}\Omega \qquad (1)$$

where $\boldsymbol{q}$ is conservative variable vector, $\Omega(t)$ and $S(t)$ are control volume and its boundary, $\boldsymbol{n}$ is the out-pointing norm vector of $S(t)$, $\boldsymbol{F}_\mathrm{c}$ and $\boldsymbol{F}_\mathrm{v}$ represent the convective and viscous fluxes, and $\boldsymbol{H}$ is the source term vector consisting of non-inertia force due to constant rotation of the system and other external contribution to the conservation law. The turbulence contribution to viscous fluxes is modeled with several Boussinesq eddy-viscosity models.

For turbomachinery applications, the flow angle, total temperature and total pressure are prescribed at the inlet, whereas the inlet Mach number is given to initialize the flowfield. The exit static pressure is determined with a simple radial pressure balance procedure.

Both the Navier-Stokes equations and turbulence model equations are discretized with a cell-centered finite volume scheme for multiblock grids. The convective fluxes at the cell interface are computed with Roe’s approximate Riemann solver, whereas the diffusion fluxes are evaluated with a second order center difference. An upwind-biased MUSCL scheme with limiter is employed to achieve second order accuracy.

An efficient implicit scheme with local timestep with multigrid is used to advance the flow to steady state. The turbulence model equation is decoupled with the main equations and solved separately. For time accurate solution, such as flutter problems, Jamerson’s dual time-stepping technique is adopted with several sub-iterations to reduce time-linearization errors.

## 3. Structural Model

There are a number of more refined and realistic structural models[20] of the blade vibration under aerodynamic loads. However, in the context of coupled fluid-structure solution, such model generally requires expensive computing resource which is not currently available. Linear aeroelasticity model is proven to be sufficient for most flutter problems[21]. The structural dynamic equation for the blade vibration is written as

$$\boldsymbol{M}\ddot{\boldsymbol{x}} + \boldsymbol{C}\dot{\boldsymbol{x}} + \boldsymbol{K}\boldsymbol{x} = \boldsymbol{Q} \qquad (2)$$

where $\boldsymbol{M}$ is the mass matrix, $\boldsymbol{C}$ the damping matrix, $\boldsymbol{K}$ the stiffness matrix, $\boldsymbol{x}$ the displacement vector and $\boldsymbol{Q}$ the force vector.

The structural equation of motion is a set of linear differential equations that can be solved with a modal approach in eigen-space. Once the eigen-solution is obtained, the vibration of structure can be expressed as a series of linearly independent modal vectors, so the deflection of blade can be written as a linear combination of the modes included in the analysis, as

$$\boldsymbol{x} = \boldsymbol{\Phi\xi} \tag{3}$$

where $\boldsymbol{\Phi}$ is the mass averaged mode-shape matrix, and $\boldsymbol{\xi}$ the generalized coordinate vector. Putting Eq.(3) back to Eq.(2) and pre-multiplying it by $\boldsymbol{\Phi}^{\mathrm{T}}$ yields

$$\boldsymbol{\Phi}^{\mathrm{T}}\boldsymbol{M\Phi}\ddot{\boldsymbol{x}} + \boldsymbol{\Phi}^{\mathrm{T}}\boldsymbol{C\Phi}\dot{\boldsymbol{x}} + \boldsymbol{\Phi}^{\mathrm{T}}\boldsymbol{K\Phi}\dot{\boldsymbol{x}} = \boldsymbol{\Phi}\dot{\boldsymbol{x}}^{\mathrm{T}}\boldsymbol{Q} \tag{4}$$

With the assumption of proportional damping (Rayleigh damping), the damping matrix can be expressed as a combination of the mass and stiffness, and Eq.(4) can be written as

$$\ddot{\boldsymbol{\xi}} + \mathrm{diag}(2\xi_i\omega_i)\dot{\boldsymbol{\xi}} + \mathrm{diag}(\omega_i^2)\boldsymbol{\xi} = \boldsymbol{\Pi}(t) \tag{5}$$

where $\omega_i$ is the $i$th vibration circular frequency.

When the blade motion is dominated by the lower modes in turbomachinery applications, the equations of motion can be reduced to smaller number of degree of freedom (DOF) by removing high-frequency modes or modes which are not interested in. The generalized force vector can be expressed as

$$\boldsymbol{\Pi}(t) = \boldsymbol{\Phi}^{\mathrm{T}}\boldsymbol{Q} = \left\{ \frac{\rho_\infty V_\infty^2}{2} \int_{S(t)} \varphi_i C_p \mathrm{d}S \right\}_{1 \le i \le N} \tag{6}$$

where $\rho_\infty$ is the free-stream density, $V_\infty$ the free-stream velocity, $\varphi_i$ the $i$th mode shape, $C_p$ the pressure coefficient.

Eq.(5) is solved by a state-space representation method[22]. With a first order state-space model, the decoupled $N$ modes second order differential equations are reduced to $2N$ sets of first order differential equations and solved by a convolution integral method.

## 4. Fluid-structure Coupling

In this study, an integrated method is used to solve coupled aeroelasticity problems. It involves the solution of the fluid and structural equations simultaneously. The boundary condition information exchanges between two domains at each time step, so that the solution from one domain is used as a boundary condition for the other. At each time step, the aerodynamic force vector is computed and sent to the modal equation solver. The structural solver evaluates the generalized forces for each vibration mode and solves the time-dependant modal equation. Then the displacement vector of the blade is computed by a linear combination. The resultant displacement and velocity of the boundary are updated in the aerodynamic grid with a grid deformation technique.

## 5. Grid Motion and Deformation

In a time-accurate coupled fluid-structure solution, the motion of the blade defines new boundaries for flow domain at each time step. The aerodynamic grid has to be updated in order to be conformed to the aeroelastic or prescribed blade shape. It is desirable to move the aerodynamic grid according to the instantaneous position of blade. A deforming grid technique based on transfinite interpolation (TFI) method is developed with the consideration of turbomachinery applications. The deforming boundary faces are interpolated to interior points based on a decay function[23]. A special movement condition is imposed on the surface grid of the casing when the tip-gap is small, so that the grid point on the casing would slip on the revolving surface. The other boundary surfaces remain stationary. This is proven to be beneficial to the quality of grid in that region.

## 6. Numerical Results

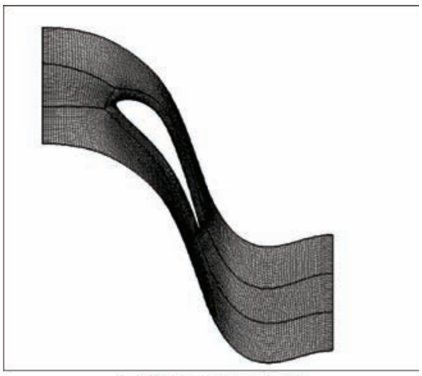
The proposed method has been applied to predicting flutter instability of the transonic flows in a turbine cascade with bending motion of the blade to validate the unsteady flow simulation algorithm. Steady flow and flutter analysis are performed on the Rotor 67 to demonstrate the capability of the current method.

### *6.1. 4th Standard Configuration*

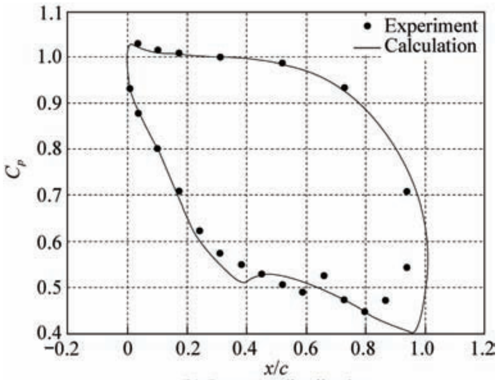
The 4th Standard Configuration has been investigated experimentally in the non-rotating annular cascade tunnel[24] and computationally to validate numerical methods for unsteady flows[15] or flutter analysis. The same flow conditions and blade oscillation amplitude and frequency for the updated Case 624 are imposed for the following computation.

The flow is transonic with isentropic exit Mach number 1.04, the inlet flow is inclined 21.4° with respect to the axial direction. The computational grid (see Fig.1(a)) consists of one "O" block and five "H" blocks for the rest of the domain.

The steady computation is performed by solving the RANS equation. Fig.1(b) provides the compare-



(a) Computational grid




(b) Pressure distribution

Fig.1 Computational grid and pressure distribution.

son of pressure coefficient distribution, where $x/c$ represents local coordinate along the chord $c$. The agreement with the experimental data is reasonably well except in the trailing edge and on the suction surface where the weak shock exists.

The steady solution is used as the initial condition for unsteady computations. Theinter-blade phase angle (IBPA) considered in this work is 180°, thus the computation domain consists of two passages. The oscillation amplitude is 0.321 mm and the oscillation frequency is 149 Hz. The bending direction is 60° with respect to the chord.

Fig.2 shows the normalized amplitude and phase of the first harmonic of the pressure for IBPA=180°. The results agree well with experimental data and show improvement in the prediction of the phase, compared with similar cases in recent publications[25]. This indicates that the present method is able to accurately predict unsteady pressure around oscillating blades.

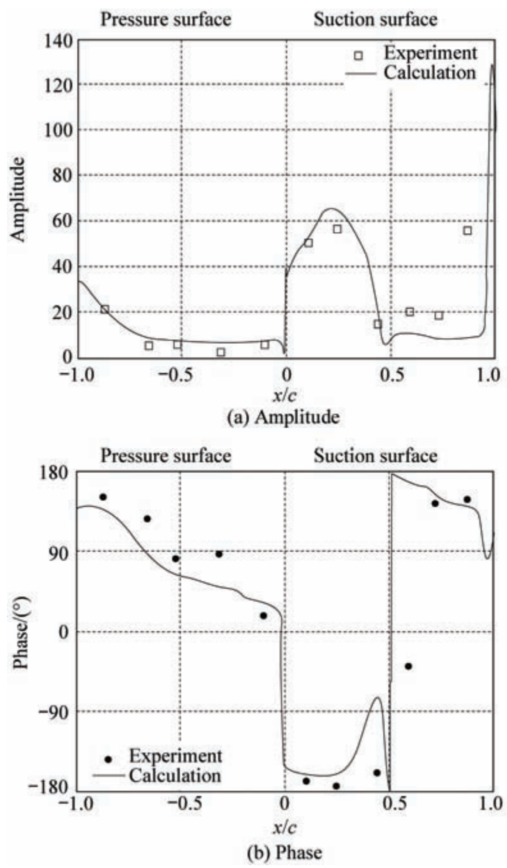



Fig.2 Amplitude and phase.

## 6.2. Fan flutter analysis

Rotor 67 is a popular test case for 3D viscous flow calculations. It is designed with 22 blades with rotational speed of 16 043 r/min at design point. The total pressure ratio is 1.63 at the peak efficiency. Coupled fluid-structure analysis has been carried out by Sadeghi and Liu[26], Doi and Alonso[27] with a closed coupled finite element method, and Vahdati, et al.[12] with an unstructured grid method.

The computational fluid dynamics (CFD) grid is a multiblock grid of 7 blocks. There are 17 grid lines within the tip-gap regions at a tip clearance of 0.7% span. Fig.3 shows the grid on the $S_1$, $S_2$ surface at mid-span and in the tip gap region. The total number of grid points for one passage is 683 442.

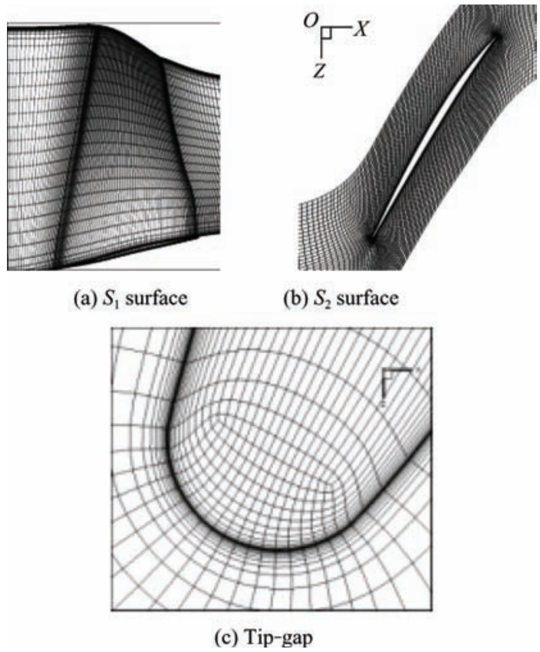



Fig.3 Computational grid.

The computed chocking mass flow rate is 34.94 kg/s at 100% speed, which is in a very good agreement with experimental data at 34.96 kg/s[28]. Fig.4 shows the total pressure ratio and adiabatic efficiency against normalized mass flow. The computed pressure ratio and efficiency show good agreement with experimental data by Strazisar[28], except for slightly under predicting pressure ratio and missing the peak efficiency spike. The total pressure $p_t$ and temperature $T_t$ distribution against inlet pressure $p_{inf}$ and temperature $T_{inf}$ at the rotor outlet are plotted at Fig.5. The calculations show an overall better agreement than previous publications [16-17,29].

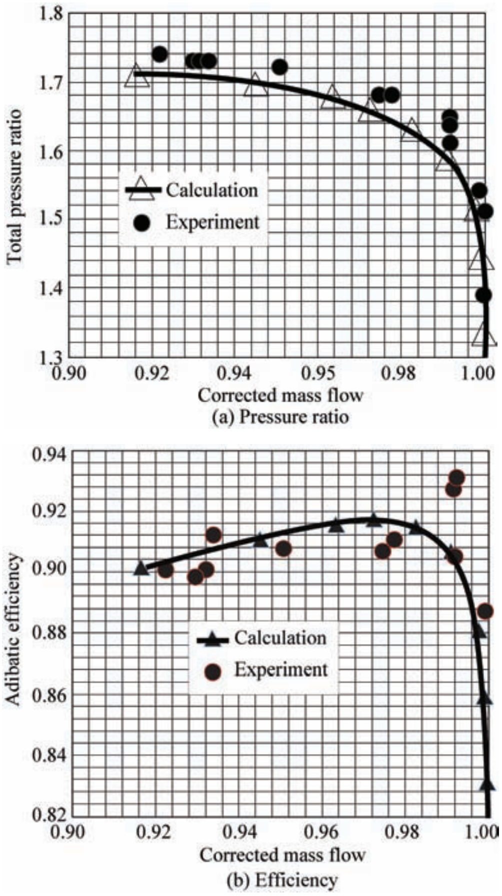



Fig.4 Performance map of Rotor 67.

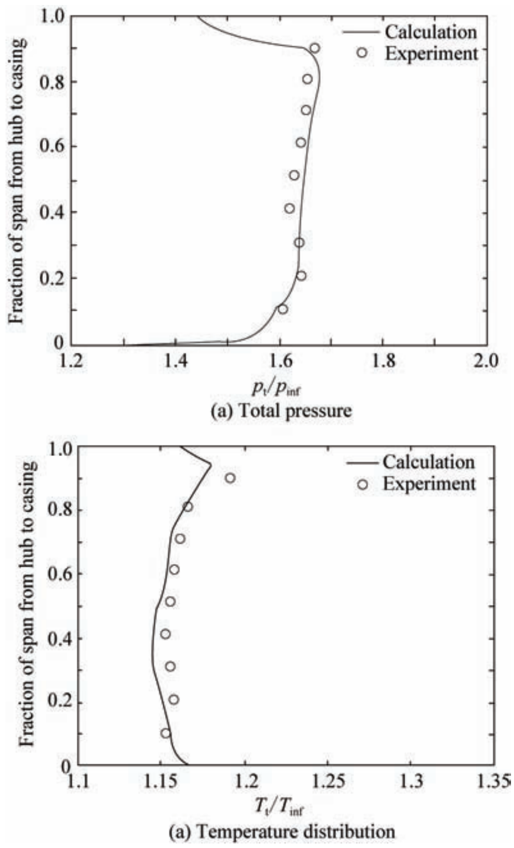



Fig.5 Total pressure and temperature distribution at outlet.

The structured eigen-solutions of a single blade clamped on the hub are obtained with the commercial FEA package NASTRAN®. Because the material of the blade is omitted in the original publications[29], the blade is assumed to be solid and a titanium-alloy is used in this work based on the modern design trend for fan blades. The structured mesh along with the first three vibration modes interpolated to the fluid mesh is displayed in Fig.6. The computed frequencies are consistent with recent publication with similar material[26-27].

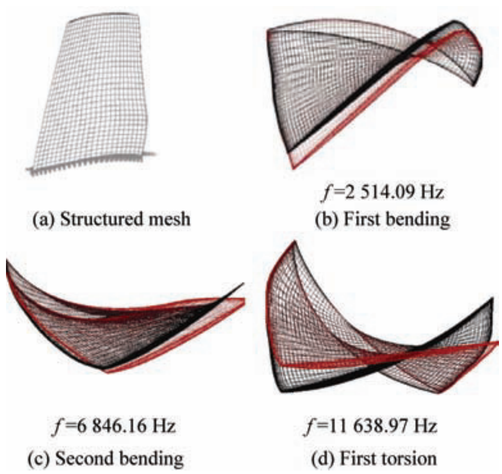



Fig.6 Structured mesh and modes.

The coupled fluid-structured flutter analysis is performed at the operation point of the peak efficiency and near stall. Although the flutter characteristic of a fan requires analysis of all IBPA conditions, the IBPA=0° is considered in this work due to constrains of the computation resource.

The converged steady solutions are used as initial flow field. The global time step is 1/90 of a period of the highest vibration mode in order to accurately simulate the blade response.

Fig.7 shows the time history of the modal displacements of three modes at the peak efficiency. It is evident that the first bending mode dominates the blade motion and the amplitude reduces with time. Fig.8 gives the vibration history of the trailing edge

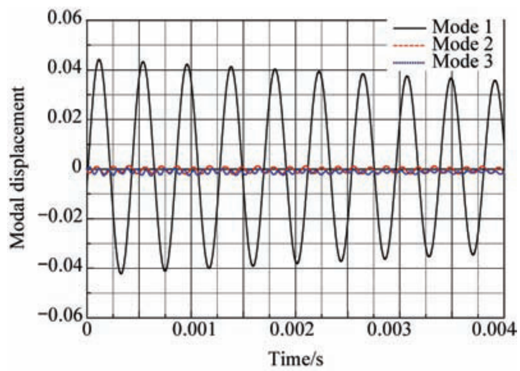



Fig.7 Time histories of modal displacement (peak efficiency).

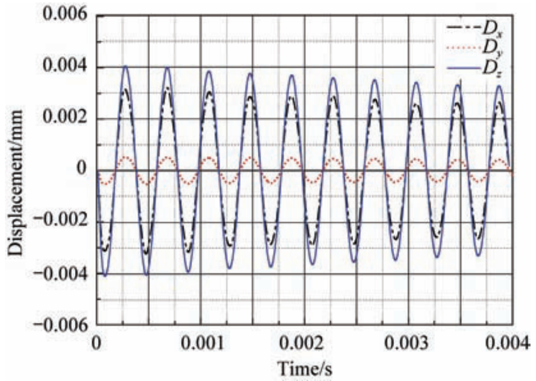



Fig. 8 Movement of trailing edge point at blade tip (peak efficiency).

point at the blade tip. It shows similar trends with the modal displacements and the blade is stable. The maximum displacement (square root of the maximum motion at *X*, *Y* and *Z* directions) of the motion is about 0.005 mm.

Figs.9-10 are the time histories of modal displacement and the blade tip motion, respectively, at the operation point near stall. It is obvious that the vibration amplitude reduces with time, and thus the blade is stable. It is also evident that the amplitude of the modes is increased, especially the second and the third modes, which suggests that the blade stability is reduced as the operation point moves towards the stall region. The maximum displacement of the blade tailing edge is about 0.006 mm.

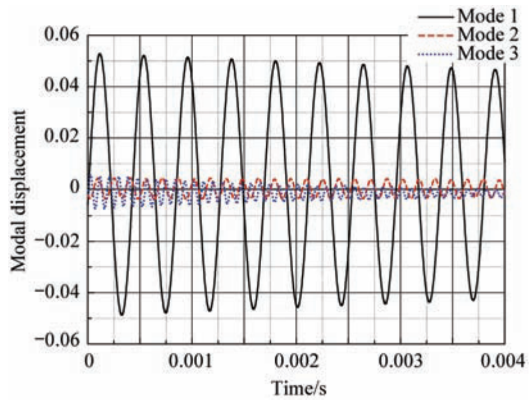



Fig. 9 Time histories of modal displacement (near stall).

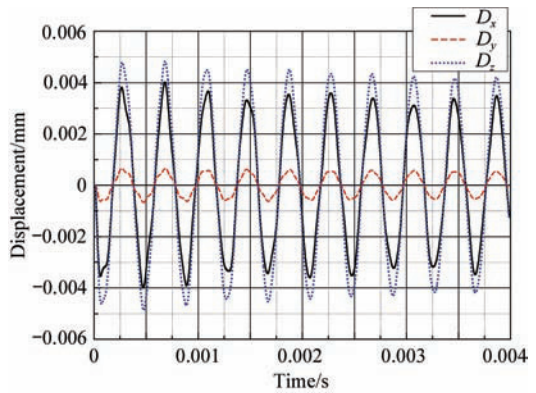



Fig. 10 Movement of trailing edge point at blade tip (near stall).

The flutter analysis indicates that rotor blade at 100% speed is stable for 0 IBPA at the peak efficiency and near the stall boundary, and the maximum blade vibration amplitude increases with the aerodynamic loads. The blade motion is dominated by the first bending mode.

## 7. Conclusions

A fully integrated numerical method for flutter analysis with a coupled fluid-structure interaction is presented. The structural dynamic equation and the fluid dynamic equation are integrated in time simultaneously.

The accuracy of the code is demonstrated by flutter analysis of transonic unsteady flows over the 4th Standard Configuration cascade. Steady analysis of the NASA Rotor 67 shows the accuracy of the numerical method used in simulation of transonic flows with strong shock-boundary layer interaction. Coupled fluid-structure flutter analysis shows that the rotor is stable both at peak efficiency and near stall at 100% rotation speed.

Coupled fluid-structure method is not only capable of predicting the stability of the blade by analysis of the vibration history, but also able to compute the amplitude and frequency of the vibration, which is important to evaluate the dynamic stress for reliability and life analysis.

## References

[1] Chen M Z. Development of fan/compressor techniques and suggestions on further researches. Journal of Aerospace Power 2002; 17(1): 1-15. [in Chinese]

[2] Gill J D, Capece V R. Experimental investigation of flutter in a single stage unshrouded axial-flow fan. AIAA-2004-686, 2004.

[3] He L. Harmonic solution of unsteady flow around blades with separation. AIAA Journal 2008; 46(6): 1299-1307.

[4] Rzadkowski R, Gnesin V, Kolodyazhnaya L. Unsteady aerodynamics, aeroacoustics and aeroelasticity of turbomachines, in unsteady aerodynamics, aeroacoustics and aeroelasticity of turbomachines. Netherlands: Springer, 2006; 569-580.

[5] He L, Chen T, Wells R G, et al. Analysis of rotor-rotor and stator-stator interferences in multi-stage turbomachines. Journal of Turbomachinery 2002; 124(4): 564-571.

[6] Hall K C, Kielb R E, Ekici K, et al. Recent advancements in turbomachinery aeroelastic design analysis. AIAA-2005-14, 2005.

[7] Bakhle M A. Fan flutter computations using the harmonic balance method. AIAA-2008-4743, 2008.

[8] Gnesin V, Rzadkowski R. A coupled fluid-structure analysis for 3-D inviscid flutter of IV standard configuration. Journal of Sound and Vibration 2002; 251(2): 315-327.

[9] Wang D X, He L. Adjoint aerodynamic design optimization or blades in multistage turbomachines—Part I: methodology and verification. Journal of Turbomachinery 2010; 132(4): 210-221.

bomachinery 2010; 132(4): 210-221.
[10] Marshall J G, Imregun M. A review of aeroelasticity methods with emphasis on turbomachinery applications. Journal of Fluids and Structures 1996; 10(3): 237-267.
[11] Schuster D M, Liu D D, Huttsell L J. Computational aeroelasticity: success, progress, challenge. Journal of Aircraft 2007; 40(5): 843-856.
[12] Vahdati M, Sayma A I, Marshall J G, et al. Mechanisms and prediction methods for fan blade stall flutter. Journal of Propulsion and Power 2001; 17(5): 1100-1107.
[13] Carstens V, Belz J. Numerical investigation of nonlinear fluid-structure interaction in vibrating compressor blades. Journal of Turbomachinery 2001; 123(1): 402-408.
[14] Srivastava R, Bakhle M A, Keith T G, et al. Aeroelastic stability computations for turbomachinery. NASA TM-2001-210693, 2001.
[15] Cinnella P, Cappiello E, de Palma P. A numerical method for 3D turbomachinery aeroelasticity. Journal of Turbomachinery 2004; 126(4): 310-316.
[16] Gnesin V I, Kolodyazhnaya L V, Rzadkowski R. The numerical analysis of the aeroelastic behaviour for the isolated blade row in 3D viscous flow. Proceedings of the 8th International Symposium on Experimental and Computational Aerothermodynamics of Internal Flows Lyon. 2007; 1-11.
[17] Jin Y, Yuan X. Numerical analysis of 3D turbine blade's torsional flutter by fluid-structure coupling method. Journal of Engineering Thermophysics 2004; 25(1): 41-44. [in Chinese]
[18] Zhang Z Q, Zou Z P, Wang Y R, et al. Investigation of flutter prediction method for transonic fan. Journal of Aerospace Power 2010; 25(3): 132-138. [in Chinese]
[19] Hu Y C, Zhou X H. Numerical simulations of two dimensional unsteady viscous flow around oscillating cascades. Chinese Journal of Applied Mechanics 2003; 20(3): 79-81. [in Chinese]
[20] Bendiksen O O. Aeroelastic problems in turbomachines. AIAA-1990-1157, 1990.
[21] Marshall J G, Imregun M. A review of aeroelasticity methods with emphases on turbomachinery applications. Journal of Fluids and Structures 1996; 10(3): 237-267.
[22] Cunningham H J, Batina J T, Bennett R M. Modern wing flutter analysis by computational fluid dynamics methods. Journal of Aircraft 1982; 25(10): 376-382.
[23] Potsdam M A, Guruswamy G P. A parallel multiblock mesh movement scheme for complex aeroelastic applications. AIAA-2001-0716, 2001.
[24] Bolcs A, Fransson T H. Aeroelasticity in turbomachines: comparison of theoretical and experimenttal cascade results. ADA181763, 1986.
[25] Ji S, Liu F. Flutter computation of turbomachinery cascades using a parallel unsteady Navier-Stokes code. AIAA Journal 1999; 37(3): 320-327.
[26] Sadeghi M, Liu F. Coupled fluid-structure simulation for turbomachinery blade rows. AIAA-2005-0018, 2005.
[27] Doi H, Alonso J. Fluid/structure coupled aeroelastic computations for transonic flows in turbomachinery. GT-2002-30313, 2002.
[28] Strazisar A J, et al., Laser anemometer measurements in a transonic axial-flow fan rotor. NASA TP-2879, 1989.
[29] Chima R V. Viscous three-dimensional calculations of transonic fan performance. NASA TM-103800, 1991.

## Biographies:

**ZHENG Yun** Born in 1971, he received B.S (1994) and M.S. (1998) degrees from Beihang University, and Ph.D. degree (2002) from Durham University (UK). He worked as a research assistant at Imperial College London during 2003-2005. Then he worked in Beihang University as a lecturer in 2006. His main research interests are CFD, turbomachinery, aeroelasticity.
E-mail: zheng_yun@buaa.edu.cn

**YANG Hui** Born in 1970, she received B.S.(1992) and M.S.(1997) degrees from Beihang University, and Ph.D. degree (2004) from Durham University (UK). She worked as a postdoctor in Beihang University during 2006-2009. Then she worked in Beihang University as a lecturer in 2009. Her main research interests are experimental and numerical research in blade flutter in turbomachinery.
E-mail: huiyang@buaa.edu.cn