GPPS-TC-2022-0024

Investigation of the Flutter Stability of a Transonic Fan with Inlet Distortion

Yanfeng Zhang
1Key Laboratory of Light-duty Gas-turbine, Institute of Engineering Thermophysics, Chinese Academy of Sciences
2University of Chinese Academy of Sciences
zhangyf@iet.cn
Beijing, China

Jieyun Pan
School of Aeronautics and Astronautics, Zhejiang University
Hangzhou, Zhejiang, China

Yalu Zhu
School of Aeronautics and Astronautics, Zhejiang University
Hangzhou, Zhejiang, China

Xu Dong
1Key Laboratory of Light-duty Gas-turbine, Institute of Engineering Thermophysics, Chinese Academy of Sciences
2University of Chinese Academy of Sciences
dongxu@iet.cn
Beijing, China

Jiaqi Luo
School of Aeronautics and Astronautics, Zhejiang University
jiaqil@zju.edu.cn
Hangzhou, Zhejiang, China

ABSTRACT

Flutter is a self-excited aeroelastic instability phenomenon which can cause material fatigue and, even worse leads to blade failure. The risk of flutter has to be avoided over the whole operating range especially near operating limits. This paper looks at the prediction of flutter instability of the transonic fan under inlet distortion. The flutter stability analysis with an in-house unsteady Reynolds-average Navier-Stokes (URANS) solver is performed by applying the so-called energy method. The phase-shifted boundary condition based on direct-store method is used, which enables reducing the computational domain to a one/two passages for all inter-blade phase angles (IBPAs), thus providing considerable saving in computing time. The more efficient double passage method is also adopted in our solver. The solver based on phase-shift boundary condition is first validated by the unsteadyaerodynamic force prediction of standard configuration 4 (STCF4) turbine. The results are compared with that by multi-passage solver. Then the flutter boundary of NASA Rotor 67 vibrating in its first natural mode is calculated and the influence of inlet distortion on flutter instability is investigated.

INTRODUCTION

The flutter of blades within compressors and turbines has been a serious cause of machine failure which has been difficult to predict and expensive to correct. There is clearly a need to avoid the possibility of related failures but until
recently this has been beyond the design capability. The advance in computational fluid dynamics (CFD) and computational structural dynamics (CSD) technology together with the greatly increased computational power have provided a powerful design tool for flutter analysis in both academic and industrial units.

Numerical flutter analysis can be performed either by a coupled method or an uncoupled method. In the coupled method, the governing equations of structure and fluid are solved simultaneously, then the aeroelastic stability of the system is determined by the time variant of the generalized displacement vector of the structural system. Sisto et al. (Sisto et al., 1991) applied a coupled method based on incompressible flow solver to study stall flutter in a linear cascade. Sadeghi (Sadeghi and Liu, 2005, 2007) developed a coupled CFD/CSD solver based on URANS equations and investigated the mistunning effects on cascade flutter.

However, the coupled method is too time-consuming to be applied in daily industrial design. The uncoupled method known as energy method (Carta, 1967) is another more efficient method for flutter analysis. In the uncoupled method, the blade row is usually assumed to be tuned so that all blades are forced to oscillate with the same frequency. Furthermore, the inter-blade phase angle (IBPA) is specified as constant throughout the blade row owing to Lane’s travelling wave model (Lane, 1956). Applying a particular oscillation frequency, IBPA, oscillation mode and amplitude, the unsteady flow around the blades is solved. The stability of the system is determined by the work done on the blade row by the aerodynamic forces.

A straightforward method for the uncoupled computation with non-zero IBPA is the multi-passage method (Ji and Liu, 1999; Sadeghi and Liu, 2001, 2005, 2007). But the number of blade passages need to be include becomes large for small value of IBPA. For some three-dimensional cases, all the blades in the blade row have to be included in the analysis, which makes the computational cost prohibitive. The most widely used uncoupled flutter analysis method is based on the phase-shifted periodic boundary condition. This method permit flutter analysis with only the simulation of one or two blades per row no matter what the IBPA is investigated. A conventional way of implementing this boundary condition is the direct store method proposed by Erdos (Erdos et al., 1977), in which the flow variables on the periodic boundaries are stored over a whole period of time. These stored time histories of the flow are then used to update solutions at corresponding periodic boundaries at a later time. When the computation reaches a periodic solution, the phase-shifted periodic boundary condition is then satisfied with the specified IBPA.

A main disadvantage of the phase-shifted boundary condition based on direct store method is the requirement of a large amount of computer storage, especially for an explicit time-marching scheme. He (He, 1990, 1992) proposed a Fourier-transform-based method called the shape correction method, in which the Fourier components of the flow variables are stored instead of the time history of the flow variables. Although the need for computer storage is greatly reduced in this method, extra computational load is added. Furthermore, the number of Fourier series terms need to be retained is case-dependent and should be determined by trail-and-error.

It was found that the computation converges slowly in reaching a final periodic solution by using a one passage phase-shifted boundary condition method comparing to that by using a multi-passage method (Ji and Liu, 1999). Biesinger (Biesinger et al., 2010) proposed a novel double-passage version phase-shifted periodic boundary condition based on the shape correction approach, in which a higher quality time signal is collected from the interface between two adjacent passages. They showed that the double-passage version converges in 5-10 times fewer cycles than the single passage version.

The double-passage version based on direct store method is used in the present work. Because we used an implicit method, fewer physical time steps are needed within each period of the blade oscillation. Hence, the memory needed to store flow variables on periodic boundary is much less than that by an explicit method, which is acceptable for a modern computer.

Most of large subsonic transports designed in the last few decades have employed podded engines, mounted by pylon, on the wings. With this placement, the engines receive a cleaner inlet flow. However, with the increased interest in the hybrid wing/body aircraft of which engine is embedded partially or fully submerged within the body, the jet engine running under inlet distortion need more investigation.

Originally, many researches focus on the impact of inlet distortion on engine aerothermodynamic performance. Hah (Hah et al., 1998), Fidalgo (Fidalgo et al., 2012), Sharma (Sharma et al., 2013) investigated the simple circumferential or sinusoidal total-pressure distortions to find the performance trends and flow features such as blade loading or shock location by URANS solvers. Du (Du et al., 2012), Cameron (Cameron et al., 2013), Nasri (Nasri et al., 2016) studied the performance of transonic axial compressor under a clean, hub-defect-distorted, and tip-defect-distorted total pressure inlet profiles. Their results showed that purely radial inlet total pressure distortion will affect the interaction between the main flow and tip-leakage-flow, thus considerably changing the stall margin.

Within the last few decades, researchers also developed interest in the impact of inlet distortion on blade aeroelasticity, because that more and more high aspect ratio and flexible blades susceptible to flutter are designed. Herrick (Herrick, 2010) and Bakhle (Bakhle et al., 2012) investigated the flutter stability of a boundary layer ingesting (BLI) fan with inlet distortion by an uncoupled energy method. Herrick (Herrick, 2010) investigated the impact of a planar-symmetric total pressure distortion pattern. Their results showed that the aerothermodynamic performance with inlet distortion plotted on a corrected flow basis was similar to that with clean inlet, and the aeroelastic stability was minimally impact by the inlet distortion.
distortion. In Bakhle’s (Bakhle et al., 2012) investigation, the distributions of total pressure, total temperature and total inflow angles at the fan inlet are extracted from an intermediate inlet design iteration. The inlet distortion only minimally reduced the aerodynamic damping of the blade vibrating in the first natural mode at near peak efficiency condition.

The effects of radial and circumferential distortions are coupled in above simulations. It may be more helpful to understand the impact of inlet distortion on blade flutter by investigating the radial and circumferential individually. Herrick (Herrick, 2014) investigated the flutter stability of a similar BLI fan with sinusoidal total pressure distortion by both coupled and uncoupled methods. For the uncoupled computations with inlet distortion, their results showed that part of blade row might exhibit unstable though the average blade indicated stability. This phenomenon becomes more prominent at less stable near stall condition. They further pointed out that due this phenomenon, the coupled method might show instability for the condition at which the uncoupled method shows a stable system.

Iseni (Iseni et al., 2016) calculated the flutter boundary of NASA Rotor 67 at different operating conditions (peak efficiency, near stall, near chock), their analysis was subject to a sinusoidal total pressure distortion. Their results indicated no risk of flutter for all first three modes of Rotor 67. They also showed that the time-dependent damping ratios were mostly dominated by the linear combination of disturbance frequencies of both inlet distortion and blade vibration.

The purely radial inlet distortion will considerably change the stall margin of a transonic compressor (Du et al., 2012; Cameron et al., 2013; Naseri et al., 2016). But in the authors’ knowledge, there is no public result about the impact of radial inlet distortion on the flutter stability of transonic compressor. Thus, we will focus on the radial inlet distortion in the present paper.

In the following sections, we first present the basic formulation of our URANS solver based on the phase-shifted periodic boundary condition. Then the phase-shifted periodic boundary is validated by test cases of STCF4 turbine. Results are compared to those obtained by a multi-passage method. Finally, the flutter analysis of NASA Rotor 67 at near peak efficient condition is performed, and the impact of radial inlet distortion on the flutter boundary is also investigated.

**METHODOLOGY**

**Governing Equations**

The unsteady Favre-average Navier-Stokes equations for compressible flow in integral form can be written as:

\[
\frac{\partial}{\partial t} \iiint_V \mathbf{W} dV + \iint_S (\mathbf{F}_c - \mathbf{F}_v) \cdot n dS = \iiint_V \mathbf{Q} dV
\]

where \( V \) is an arbitrary control volume with closed boundary surface \( S \), and \( n \) is the unit normal vector pointing to the outside of control volume. The vector of conservative variables \( \mathbf{W} \) is given by:

\[
\mathbf{W} = \begin{bmatrix}
\rho \\
\rho \mathbf{u} \\
\rho E
\end{bmatrix}
\]

where \( \rho \) is the density, \( \mathbf{u} = u, v, w^T \) is the velocity vector, and \( E \) is the total energy.

The convective fluxes tensor \( \mathbf{F}_c \) and viscous fluxes tensor \( \mathbf{F}_v \), are given by:

\[
\mathbf{F}_c = \begin{bmatrix}
\rho u_i u_j \\
\rho (u_i u_j + p) \\
(\rho E u_i + \rho u_j)^T
\end{bmatrix}, \quad \mathbf{F}_v = \begin{bmatrix}
0 \\
\tau_{ij} \\
(\tau_{ij} \cdot \mathbf{u} - \mathbf{q})^T
\end{bmatrix}
\]

Due to the moving mesh in the fixed coordinate system, the fluxes through the surface are expressed in terms of the contravariant velocity \( \mathbf{u}_r \):

\[
\mathbf{u}_r = \mathbf{u} - \mathbf{u}_g
\]

where \( \mathbf{u}_g \) is the grid velocity vector.

The shear stress tensor \( \tau \) and the heat flux vector \( \mathbf{q} \) are defined as:

\[
\tau_{ij} = (\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}(\mathbf{V} \cdot \mathbf{u}) \delta_{ij}, \quad \mathbf{q} = -(k + k_i) \nabla T
\]

where \( \mu \) and \( \mu_t \) are the molecular viscosity and eddy viscosity, respectively. \( k \) and \( k_i \) are the molecular thermal conductivity and eddy thermal conductivity, respectively. The one equation Spalart-Allmaras turbulence model (Spalart and Allmaras, 1992) is employed for calculating the eddy viscosity \( \mu_t \).

Disregarding the effect of body force, source terms only arise due to the representation of the velocity vector in a rotating reference frame in case of a rotor:

\[
\mathbf{Q} = \begin{bmatrix}
0 \\
-\Omega \times \rho \mathbf{u} \\
0
\end{bmatrix}
\]

3
where \( \Omega \) is the angular velocity of the rotating reference frame.

It is worth noting at this point that for an uniform flow in a stationary reference frame, the governing equations reduce to the “Geometric Conservation Law” (GCL) equation:

\[
\frac{\partial}{\partial t} \iiint_{V} dV = \iint_{S} \mathbf{u} \cdot \mathbf{n} dS
\]  

(7)

It was first pointed out by Thomas (Thomas and Lombard, 1979) that besides the conservation of mass, momentum and energy, the GCL must be satisfied in order to avoid error induced by a deformation of the mesh.

**Phase-shifted Periodic Boundary Condition**

![Figure 1 Double passage method](image)

The phase-shifted periodic boundary condition is widely used for unsteady aerodynamic force calculation in blade row with non-zero IBPA. The original version of this method based on the direct store method proposed by Erdos (Erdos et al., 1977) can reduce the computation domain to only one passage. As shown in Fig. 1, the upper and lower periodic boundaries for blade 1 have to be applied as:

\[
W_1(x,t) = W_2(x,t - \sigma / \omega), \quad W_2(x,t) = W_1(x,t - 2\pi / \omega + \sigma / \omega)
\]  

(8)

where \( \sigma \) is the IBPA, \( \omega \) is the blade vibration angular frequency. The flow variables at both boundaries 1 and 2 should be stored for a whole oscillation period.

However, Ji (Ji and Liu, 1999) showed that the phase-shifted periodic boundary condition method based on single passage reached the final periodic solution significantly slower than the multi-passage method. Biesinger (Biesinger et al., 2010) proposed a double-passage version of phase-shifted periodic boundary condition to accelerate the convergence. Biesinger’s phase-shifted periodic boundary is based on the shape correction method. We extend this technology to the direct store method in present work. In the present implementation, only the six conservative variables at the interface between two passages are stored for a period of oscillation. At every physical time step, the flow variables at both the upper and lower periodic boundaries (1, 3) are updated based on the stored values at interface boundary (2):

\[
W_1(x,t) = W_2(x,t - \sigma / \omega), \quad W_3(x,t) = W_2(x,t - 2\pi / \omega + \sigma / \omega)
\]  

(9)

Biesinger (Biesinger et al., 2010) shown that the double-passage version converged in 5-10 times fewer cycle than the single passage version, owing to the superior quality time history data collected in the interface.

**Numerical Method**

The basic numerical algorithm for solving the unsteady Navier-Stokes equations is a cell-centered finite-volume method. The semi-discrete form of the governing equations can be written for each cell as:

\[
\frac{d}{dt}(W \Delta V) = R(W)
\]  

(10)
where the residual $\mathbf{R}(\mathbf{W})$ is given by the discretized convective and viscous fluxes. The convective fluxes are discretized by the Roe’s (Roe, 1981) flux difference splitting method. Second-order accuracy is achieved by MUSCL extrapolation to the primitive variables. The viscous fluxes are discretized by a central scheme and the first derivatives are evaluated by using the Green’s theorem.

The time derivative is discretized by an implicit second-order backward-difference scheme:

$$\frac{1}{2\Delta t}[3(\mathbf{W}\Delta \mathbf{V})^{n+1} - 4(\mathbf{W}\Delta \mathbf{V})^n + 3(\mathbf{W}\Delta \mathbf{V})^{n-1}] = \mathbf{R}(\mathbf{W}^{n+1})$$

(11)

where $\Delta t$ is the physical time step, $n + 1$ denotes the current time level, and the two previous time levels are denoted by superscripts $n$ and $n - 1$.

Following a dual-time approach of Jameson (Jameson, 1991), the unsteady problem is reformulated as a steady-state problem in a pseudo-time $t^*$ to solve for the solution at the current real-time step ($n + 1$):

$$\frac{d}{dt^*}(\mathbf{W}^{n+1} \Delta V^{n+1}) = \mathbf{R}(\mathbf{W}^{n+1}) - \frac{1}{2\Delta t}[3(\mathbf{W}\Delta \mathbf{V})^{n+1} - 4(\mathbf{W}\Delta \mathbf{V})^n + 3(\mathbf{W}\Delta \mathbf{V})^{n-1}]$$

(12)

There are different kinds of method to take into account the GCL for the unsteady calculation with deforming mesh, we follow the implementation of Biedron (Biedron and Thomas, 2009). The temporal discretization scheme of the GCL Equation 7 is identical to the Naiver-Stokes equations:

$$\frac{1}{2\Delta t}[3\Delta V^{n+1} - 4V^n + 3\Delta V^{n-1}] = R_{GCL}(\mathbf{u}_g^{n+1})$$

(13)

The $\Delta V^n$ in the residual term of Equation 12 is replaced by the discretized GCL equation (13). On the other hand, the grid velocity due to the mesh deformation should be calculated in discrete form thought it can be calculated by an analytic method in some harmonic oscillation cases:

$$\mathbf{u}_g = \frac{1}{2\Delta t}[3\mathbf{x}^{n+1} - 4\mathbf{x}^n + 3\mathbf{x}^{n-1}] + \Omega \times \mathbf{r}^{n+1}$$

(14)

The last term comes from solving the absolute flow variables of Navier-Stokes equations described in a rotating reference frame. $\mathbf{r}$ is the radial distance between the field point and rotational axis, and $\Omega$ is the constant angular velocity of the rotating reference frame.

An implicit LU-SGS method (Jameson and Yoon, 1987) is used to integrate the semi-discrete Equation 12. The convergence acceleration techniques, such as local time-stepping, and multigrid method are also adopted.

The turbulence model equation is solved by a similar discretization method, the only difference is that the convective flux is discretized by a first order upwind scheme.

**Interpolation Method for Mode Shapes**

In the aeroelastic calculation, the grid used for modal analysis does not usually coincide with that used for aerodynamic calculation. Moreover, the modal analysis tends to use some simplified geometry models such as plate and beam, where the aerodynamic grid locates outside the structural model. Deformation obtained on the structural grid has to be transferred to the aerodynamic grid by an interpolation or extrapolation method.

Popular interpolation techniques such as the infinite plate spline (IPS) method and thin plate spline (TPS) method work well as long as the structural grid points and the aerodynamic grid points locate within or near within the same surface. However, for aerelastic analysis in turbomachinery, most of the blades are thick in the profile or highly twisted in space, thus a three-dimensional structural model is preferred and a three-dimensional interpolation or extrapolation method is needed.

Goura (Goura, 2001) presented a Constant-Volume Tetrahedron (CVT) method for three-dimensional interpolation. Each aerodynamic node is attached to a triangular structural element, forming a tetrahedron. The orientation between the aerodynamic node and the structural element remains constant, while the distance is adjusted to obtain a constant volume tetrahedron. This method is efficient and easy to implement, so it’s adopted in the present work.

**Mesh Motion**

While the mesh displacements of blade surfaces are given based on the specified mode shape, amplitude and IBPA, and mesh displacements on other boundary surfaces (hub wall, shroud wall, inlet, outlet, periodic boundary) are set to zero, the mesh of inner flow domain has to be deformed by a mesh deformation algorithm.

A multi-block mesh deformation approach proposed by Hartwich (Hartwich et al., 1997) is implemented in our solver. Several control points on the block surface, such as block corner are selected and categorized into master point and slave point. The control point which locates on the boundary surface with specified mesh deformation is a master point, otherwise
it’s a slave point. The deformation of a slave point $\mathbf{x}_s$ is calculated based on the mesh deformation of closest master point $\mathbf{x}_m$.

$$\mathbf{x}_s = D_{ms} \mathbf{x}_m, \quad D_{ms} = \min(1, e^{-A}), \quad A = \beta_2 (\Delta r_{ms}/\Delta r_{\text{max}} - \alpha_2)$$

where $\Delta r_{ms}$ is the distance between the corresponding master point and slave point, $\beta_2$ and $\alpha_2$ are the parameters for decay control. A smaller value of $\beta_2$ and a larger value $\alpha_2$ give a more extensive rigid motion region around the blade. Typical values for decay parameters are:

$$\beta_2 = 1 - 10, \quad \alpha_2 = 0.005 - 0.05$$

Then, the deformation of mesh lines between control points, boundary surfaces and inner flow domain can be calculated by the transfinite interpolation (TFI) method step by step.

**RESULTS AND DISCUSSION**

**Flutter Analysis of the STCF4 Turbine Stator**

STCF4 of the turbomachinery test case compiled by Bölcs and Fransson (Böelcs and Fransson, 1986) is an annular turbine blade row operating under subsonic to transonic conditions. The blades are mounted on the elastic beams which allow for blade oscillation in a quasi-bending motion. This case is originally proposed as a two-dimensional test case. The midsection of the blade passage is approximated as a planar stream-surface and the cascade is subject to a plunging displacement. The geometry of cascade and experimental data, i.e. unsteadypressure distributions over the midsection are report in Ref. (Böelcs and Fransson, 1986)

![Figure 2 Mesh for STCF4 computation](image)

In this paper, test case 627 is studied by two-dimensional calculation. The average inlet Mach number is 0.19, the average inlet flow angle is $-15.2^\circ$, and the average isentropic exit Mach number is 0.85. The flow is subsonic through the entire blade passage. The blades oscillate at a reduced frequency of 0.119 (based on half-chord and exit velocity). The oscillation amplitude is about $4 \times 10^{-3}$ chord-length. Fig. 2 shows the grid with 10864 cells for a single blade passage. $y^+ < 5$ is ensured as required by Spalart-Allmaras turbulence model for viscous sublayer resolution.

In the numerical simulation, the nominal inlet flow angle was changed to $-25^\circ$ to match the inlet Mach number in the experiment. The steady isentropic Mach number distribution over the blade surface is shown in Fig. 3. The result calculated by Sadeghi (Sadeghi et al., 2004) based on the corrected inlet flow angle as well as the experimental data are shown in Fig. 3. These two computational results agree well with each other, and both of them agree well with the experimental data.

Figs. 4 through 6 show the distributions of the magnitude and phase of the first harmonic unsteady pressure coefficient for IBPAs of $180^\circ$, $90^\circ$ and $-90^\circ$. The unsteady pressure coefficient is defined as $C_p = \hat{p}/[h(p_{0,in} - p_{in})]$, where $\hat{p}$ is the first harmonic unsteady pressure, $p_{0,in}$ and $p_{in}$ are the total pressure and static pressure at the inlet and $h$ is the oscillation amplitude. The results calculated by the phase-shifted (PS) boundary condition method are denoted as 'PS'.
In order to validate the implementation of PS method, calculations based on multi-passage (MP) method were also performed by our solver. In MP computation, the number of passages \( N_p \) needed depends on the IBPA: \( N_p = \frac{2\pi}{|\sigma|} \), and periodic boundary condition is still applied at the upper and lower boundaries in the circumferential direction.

The unsteady pressure coefficients calculated by the PS method agree well with that by MP method for all three IBPA conditions investigated. Minor differences are observed in the trailing edge for IBPAs of \( 90^\circ \) and \( 90^\circ \). Because that the upper or lower boundaries which are fixed in PS method is movable as a block interface in MP computation, the mesh deformations are not exactly the same in these two methods. And the unsteady force at the separation region near the blunt trailing edge may be susceptible to the mesh deformation.

Fig. 7 shows the damping ratio as a function of IBPA calculated by a PS method. The damping ratio is defined as:

\[
\Lambda = -\frac{C_w}{\pi h^2}, \quad C_w = -\int_{t_0}^{t_0+T} \int_S \rho \mathbf{u}_b \cdot \mathbf{n} dS \, dt
\]  

where \( C_w \) is the coefficient of work performed on a blade within one oscillation period, \( \mathbf{n} \) points into the flow domain, \( \mathbf{u}_b \) is the grid velocity due to blade vibration, and \( h \) is the oscillation amplitude. The aeroelastic system is stable if \( \Lambda \) is positive, otherwise it’s unstable. The present computation correctly predicts the unstable range between IBPA = \( -120^\circ \) and \( -10^\circ \). The damping ratio calculated by Sadeghi (Sadeghi et al., 2004) with a MP method is also shown in Fig. 7. These two computational results agree well with each other in qualitative, which further validates our implementation of PS method. The quantitative difference between these two methods may due to the difference in mesh resolution or turbulence model.

**Modal Analysis of the NASA Rotor 67**

NASA Rotor 67 is chosen as a challenging and practical representative example for which detailed geometry and flow information are available. The geometry for Rotor 67 is provided by Strazisar (Strazisar et al., 1989). All the flutter analyses were performed at design speed near peak efficiency operating conditions. The rotational speed is 16043 rpm, which results in an inlet tip relative Mach number of 1.38.

A pre-stressed modal analysis was carried out using ANSYS Mechanical on a single rotor blade with fixed support at the hub. The three-dimensional finite element mesh generated by the ANSYS automatic algorithm consisted of 552 mixing elements. Titanium alloy with material properties shown in Tab. 1 was used for modal analysis.

The natural frequencies of the first three modes are shown in Tab. 2, and the corresponding mode shapes are shown in Fig. 8. Contours represent the y-direction deformation. As it can be seen in Fig. 8, the deformation of tip region indicates...
that the first two modes are bending modes and the third mode is a torsion mode. Both the natural frequencies and mode shapes are consistent with that by Iseni (Iseni et al., 2016).

Only the first mode was used in the flutter analysis, and the blades were forced to oscillate at the natural frequency of the first mode ($f = 579.9$ Hz). The mode shape was scaled such that the maximum deformation of the blade was about 0.5% of the hub chord-length.

**Numerical Setup for NASA Rotor 67**

Non-reflecting inlet and outlet boundary conditions based on the Riemann invariant were used for both the steady and unsteady simulations. For clean inlet condition, uniform total pressure ($p_{0,in}$), total temperature ($T_{0,in}$) and inlet flow angles ($\alpha = \arctan(u_{0}/u_r)$, $\beta = \arccos(u_z/|u|)$) were specified at the inlet boundary. The outlet static pressure is determined by the radial equilibrium equation with static pressure specified at hub ($p_{out,hub}$). The inlet and outlet boundary conditions for near
peak efficiency operating condition are shown in Tab. 3, which were identical for both steady and unsteady simulations.

For flutter analysis, the possible IBPAs for a three-dimensional blade row are defined by:

$$\sigma = \frac{2\pi (ND)}{N_{blade}}$$  \hspace{1cm} (18)

where $N_{blade}$ is the number of blades in the blade row and $ND$ is the nodal diameter.
The damping ratio in this case is defined as:

\[ \Lambda = -\frac{C_w}{M_i q_i^2 \omega_i^2}, \quad C_w = -\int_{t_0}^{t_0+T} \left( \oint_S \rho u_b \cdot n S \right) dt \]  

where \( M_i, q_i \) and \( \omega_i \) are the modal mass, modal amplitude and natural frequency of the natural mode investigated, respectively.

Three different meshes with approximately 400,000, 720,000 and 1,440,000 cells per blade passage were used to study the effect of mesh refinement on the damping ratio. \( y^+ < 5 \) is ensured for all three meshes as required by Spalart-Allmaras turbulence model for viscous sublayer resolution.

The effect of the mesh refinement was studied for a representative nodal diameter of 11. For investigation of mesh refinement, 44 physical time steps per oscillation cycle was chosen and all calculations were run for 16 cycles. After 16 cycles, the oscillation amplitude of damping ratio was below 0.1% of the mean value. It was observed that the percentage changes in the damping ratio are about 11.3% going from coarse to medium mesh and 0.7% going from medium to fine mesh. The result is not fully mesh-independent, but we will show that the error introduced by other numerical parameters is the same order as that introduced by mesh. Fig. 9 shows the medium mesh used for flutter analysis. The 1.01mm tip clearance is modeled by a butterfly mesh.
Flutter Analysis of the NASA Rotor 67 with Radial Inlet Distortion

Table 2 The first three natural frequencies for NASA Rotor 67

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency (Hz)</td>
<td>579.9</td>
<td>1239.0</td>
<td>1834.6</td>
</tr>
</tbody>
</table>

Table 3 Inlet and outlet boundary conditions for NASA Rotor 67

<table>
<thead>
<tr>
<th>$P_{ref}$</th>
<th>$P_{0,in}/P_{ref}$</th>
<th>$T_{0,in}$</th>
<th>$\alpha$</th>
<th>$\beta$</th>
<th>$P_{out, hub}/P_{ref}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>101325 Pa</td>
<td>1</td>
<td>288.15 K</td>
<td>0°</td>
<td>0°</td>
<td>1.035</td>
</tr>
</tbody>
</table>

The effect of physical time steps per oscillation cycle was also studied for a representative nodal diameter of 11. Three different sets of physical time steps (22, 44, 66 steps / cycle) with 16 oscillation cycles run were investigated for the medium mesh. Tab. 4 indicates that the percentage change of damping ratio is about 0.13% going from 44 steps / cycle to 66 steps / cycle, which is on the same order as that introduced by mesh refinement. Thus, 44 physical time steps per oscillation cycle was chosen and all the following calculations were run for 16 cycles.

Flutter Analysis of the NASA Rotor 67 with Clean Inlet

Figs. 10a, 10b, 10c show the steady relative-Mach-number contours at the near peak efficiency condition with clean inlet. The bow shock at the mid-span region (see Fig. 10b) and the passage shock at the near tip region (see Fig. 10c) are well produced by the simulation.

Fig. 11a shows the steady pressure coefficient contour and limit streamlines on the suction side of the blade. The steady pressure coefficient is defined as

$$C_p = 2(p_{ref} / P_{0, clean} - 1) / (\gamma M_{ref}^2),$$

where $M_{ref}$ is the reference Mach number. The trailing edge separation region at the corner of the suction and hub surfaces and the tip separation bubble induced by the strong passage shock are also well predicted by the present simulation.

In order to further validate the PS method, the unsteady pressure coefficient distributions over blade surface for $ND = 11$ case are shown in Fig. 12. The first harmonic unsteady pressure coefficient is defined as

$$C_{p1} = \tilde{p}/(A_{max} P_{ref}),$$

where $\tilde{p}$ is the first harmonic unsteady pressure and $A_{max}$ is the maximum deformation amplitude. The results calculated by our MP solver are also shown for comparison. For all three blade-to-blade sections investigated, the results calculated by PS method agree well with that calculated by MP method. Minor difference of the magnitude of the first harmonic of unsteady pressure is shown at the section near hub. The good agreement between the unsteady pressure calculated by PS method and that calculated by MP method further validates the implementation of PS method in our solver.

For flutter analysis, unsteady calculations were performed for the cases with even nodal diameter and the case at $ND = 11$. Fig. 14 shows the damping ratio as a function of IBPA calculated by a PS method. It shows that the damping ratio remains positive over the whole range of IBPA investigated, indicating a stable system. The minimum damping occurs at $ND = 0$. This flutter stability behavior is consistent with that by Iseni (Iseni et al., 2016).

Figs. 15a and 15d show the damping ratio distortion for $ND = 0$ case, which were calculated by Equation 19 without the surface integral. Most of the damping ratio is contributed by the unsteady pressure near the tip region of the suction side and the leading edge region of the pressure side. For the suction side, the unsteady pressure ahead of the passage shock stabilizes the system, while that behind the shock destabilizes the system. For the pressure side, the damping ratio contribution only concentrate on the region near the reflected passage shock. But the contributions of unsteady pressure before and behind the shock are opposite to that in the suction side. The region with positive damping ratio is much larger that with negative damping ratio, which indicates a stable system in this situation.

Flutter Analysis of the NASA Rotor 67 with Radial Inlet Distortion

For flutter analyses with inlet distortion, the inlet pressure distributions are defined as:

$$p_0(r) = p_{0, clean} + D_I (r - \frac{r_{hub} + r_{shroud}}{2}) / (r_{shroud} - r_{hub}) p_{0, clean}$$

(20)

where $D_I$ is the distortion intensity, $p_{0, clean}$ is the inlet total pressure used for clean inlet calculation. While an uniform total temperature and uniform inlet flow angles with the same values as that in clean inlet calculation are applied at the inlet boundary.

The outlet boundary condition is still based on the radial pressure equilibrium equation with the same hub static pressure as that in clean inlet calculation.

For axial transonic compressor, Du (Du et al., 2012), Cameron (Cameron et al., 2013), Naseri (Naseri et al., 2016) showed that the radial inlet distortion can dramatically affect the interaction between tip leakage flow and incoming main flow in the tip gap, thus changing the stall margin of compressor. We want to further investigate the impact of radial inlet distortion on aeroelastic stability of transonic compressor. Two different radial distortion patterns, namely hub defect and
Table 4 Damping ratios of different physical time steps, NASA Rotor 67

<table>
<thead>
<tr>
<th>Steps/ cycle</th>
<th>22</th>
<th>44</th>
<th>66</th>
</tr>
</thead>
<tbody>
<tr>
<td>Damping ratio</td>
<td>0.10837</td>
<td>0.10770</td>
<td>0.10756</td>
</tr>
</tbody>
</table>

Figure 10 Steady relative-Mach-number contours of blade-to-blade section at different spans from hub, NASA Rotor 67

Tip defect distortion, are investigated, which correspond to $D_I = 10\%$ and $D_I = -10\%$, respectively. The spanwise total pressure distributions of these two kinds of distortion are shown in Fig. 13.

Figs. 10d through 10i show the steady relative-Mach-number contours at conditions with inlet distortion. For hub defect distortion, the higher total pressure in the tip region increase the axial velocity of main flow, forming a weaker passage shock on the suction side of blade. The limit streamlines in Fig. 11b indicate that the separation bubble at the tip region becomes much smaller comparing to the case with clean inlet. As shown in Fig. 11e, the reflected passage shock in pressure side moves toward to the leading edge, which almost disappears in the 90% B2B section (see 10e).

For tip defect distortion, the pressure side branch of the passage becomes stronger, which tend to become a normal shock in the 90% B2B section (see Fig. 10i). Due the total pressure defect in the tip region, much stronger adverse pressure
gradient is formed in the tip region of the blade, which induces a much larger boundary layer separation region. The limit streamlines in Fig. 11c indicate that the separation bubble area extends in both the streamwise and spanwise directions. Fig. 11f shows that the reflected passage shock in pressure side moves toward to the leading edge and the hub region. And the reflected passage shock almost disappears in the tip region. Above results show that the radial inlet distortion considerably changes the flow patterns of steady flow near the tip region.

The damping ratios as a function of IBPA for cases with inlet distortion are shown in Fig. 14. It shows that the blade row is still stable over the whole range of IBPA with the impact of inlet distortion. And the minimum damping ratio still occurs at $ND = 0$. For most of the IBPAs, the hub defect distortion improves the stability of blade row while the tip defect distortion degrades the stability. But the difference produced by inlet distortion is marginal, especially for hub defect distortion. Tab. 5 shows the minimum damping ratio for cases with different inlet distortions, the percentage difference introduced by inlet distortion is about 10%.

**Table 5 Minimum damping ratio with different inlet conditions, NASA Rotor 67**

<table>
<thead>
<tr>
<th>Distortion</th>
<th>Clean</th>
<th>Hub defect</th>
<th>Tip defect</th>
</tr>
</thead>
<tbody>
<tr>
<td>Damping ratio</td>
<td>0.0157</td>
<td>0.0170</td>
<td>0.0146</td>
</tr>
</tbody>
</table>

Fig. 15e, 15b, 15f and 15c show the damping ratio distributions for $ND = 0$ case with inlet distortion. For hub defect distortion, the positive damping region becomes larger while the negative damping region becomes smaller on the pressure side, due to the moving forward of the reflected passage shock. On the other hand, the negative damping region on the suction side also become larger comparing to the case with clean inlet.

For tip defect distortion, the most significant change is that the positive damping region near the tip leading edge of pressure side changes to negative, which may attribute to the disappearance of reflect passage shock near the tip region. And the positive damping region on the pressure side also become smaller comparing to that with clean inlet. The impact of tip defect distortion on damping ratio distribution on the suction side is less significant comparing to that on the pressure side, though a much larger tip separation bubble is shown in steady flow at this condition.

Above results indicate that the damping ratio distribution is mainly dominated by the position of the passage shock but not the separation region.

Due to the fact that the total damping ratio is an integration value, it’s more convenient to investigate the effect of inlet
Figure 12 Unsteady pressure coefficient of blade-to-blade section at different spans from hub at \( ND = 11 \), NASA Rotor 67

distortion by the spanwise distributions of damping ratio. The spanwise distributions of the damping ratio for IBPA = 0° cases are shown in Fig. 16. The spanwise distributions were calculated by replacing the domain of area integral in Equation 19 with the band regions on blade between different blade-to-blade stream surfaces. It shows that the unsteady aerodynamic forces at different spanwise positions always contribute positive damping, and the contributions concentrate at about 80% of the spanwise position from hub. The major influence of the inlet distortions also occur near the damping ratio peaks. The damping ratio peak value with hub defect distortion is large than that with clean inlet, while the damping ratio peak value with tip defect distortion is small than that with clean inlet, which is consistent with the result shown in Fig. 14.

Though the radial inlet distortions significantly change the flow pattern, such as shock position and separation region near tip region, the distributions of damping ratio on blade surface are changed quantitatively only.

CONCLUSIONS

In the paper, we develop an URANS solver to perform flutter analysis for turbomachinery by the energy method. The phase-shifted boundary condition method based on direct store approach is used to calculate the aerodynamic damping of blade row with non-zero IBPA. The double-passage version of this method is implemented to accelerate the convergence.

The phase-shifted method is firstly validated by comparing the unsteady aerodynamic forces of the STCF4 turbine and NASA Rotor 67 to that calculated by a multi-passage method. Then we perform a flutter analysis for the first natural mode of NASA Rotor 67 at the near peak efficiency condition, and investigate the impact of two types of radial inlet distortion on the flutter boundary.

The results show that the Rotor 67 remains stable over the whole range of IBPA at conditions with different types of
inlet distortion. The hub defection distortion improve the stability of blade row while the tip defect distortion degrade the stability, but their impact is marginal.

The damping ratio distribution mainly depends on the passage shock position but not the separation region. Both types
of inlet distortion investigated significantly change the passage shock position and separation region near the tip region, and thus the detail distribution of damping ratio. But the gross impact is less prominent.
NOMENCLATURE

Abbreviations

CFD Computational Fluid Dynamics
CSD Computational Structural Dynamics
CVT Constant-Volume Tetrahedron
GCL Geometric Conservation Law
IBPA Inter-Blade Phase Angle
IPS Infinite Plate Spline
MP Multi-Passage
PS Phase-Shifted
STCF4 Standard Configuration 4
TFI Transfinite Interpolation
TPS Thin Plate Spline
URANS Unsteady Reynolds-Averaged Navier-Stokes

Variables (Latin)

\( n \) Unit Normal Vector
\( q \) Heat Flux Vector \( \text{W} \text{m}^{-2} \)
\( u \) Velocity Vector \( \text{m} \text{s}^{-1} \)
\( u_g \) Grid Velocity Vector \( \text{m} \text{s}^{-1} \)
\( u_r \) Contravariant Velocity Vector \( \text{m} \text{s}^{-1} \)
\( W \) Vector of Conservative Variables
\( \bar{x} \) Mesh Deformation Vector \( \text{m} \)
\( C_w \) Coefficient of Work \( \text{J} \)
\( E \) Total Energy \( \text{J} \text{kg}^{-1} \)
\( k \) Molecular Thermal Conductivity \( \text{W} \text{m}^{-1} \text{K}^{-1} \)
\( k_t \) Eddy Thermal Conductivity \( \text{W} \text{m}^{-1} \text{K}^{-1} \)
\( M_i \) Modal Mass \( \text{kg} \text{m}^{2} \)
\( p \) Pressure \( \text{Pa} \)
\( q_i \) Modal Amplitude
\( t \) Physical Time \( \text{s} \)
\( t^* \) Pseudo-Time \( \text{s} \)
\( V \) Control Volume \( \text{m}^3 \)
\( x \) Mesh Coordinates \( \text{m} \)

Variables (Greek)

\( \tau \) Shear stress tensor \( \text{N} \text{m}^{-2} \)
$\beta_2$, $\alpha_2$  Decay Coefficients for Mesh Motion

$\Lambda$  Damping Ratio

$\mu$  Molecular Viscosity  kg m$^{-1}$ s$^{-1}$

$\mu_t$  Eddy Viscosity  kg m$^{-1}$ s$^{-1}$

$\Omega$  Angular Velocity of the Rotating Reference Frame  rad s$^{-1}$

$\omega$  Angular Frequency of Blade Vibration  rad s$^{-1}$

$\omega_n$  Natural Frequency  rad s$^{-1}$

$\rho$  Density  kg m$^{-3}$

$\sigma$  Inter-Blade Phase Angle  rad

ACKNOWLEDGMENTS

Financial support for the work presented is provided by the National Natural Science Foundation of China (Grant No. 51876202), K. C. Wong Education Foundation, National Science and Technology Major Project (2017-II-0006-0020) and Natural Science Foundation of Zhejiang Province of China (LXR22E060001).

References


Jameson, A. (1991), Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings, in ‘10th Computational Fluid Dynamics Conference’, Honolulu, Hawaii, USA.


