Contents lists available at ScienceDirect
Aerospace Science and Technology
www.elsevier.com/locate/aescte

Numerical simulation of unsteady tip clearance flow in a transonic compressor rotor

Zhiqiang Wang, Bo Lu, Jiaxin Liu, Jun Hu

College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics, Jiangsu Province Key Laboratory of Aerospace Power System, Nanjing, Jiangsu, China

A R T I C L E I N F O

Article history:
Received 12 March 2017
Received in revised form 13 July 2017
Accepted 6 November 2017
Available online xxxx

Keywords:
Axial flow compressors
Unsteady flow
Tip leakage vortex
Rotating instability
Numerical simulation

A B S T R A C T

A three-dimensional, multi-passage unsteady numerical study was conducted to enhance the understanding of unsteady flow phenomena in the tip region of a transonic axial compressor rotor. Two different inlet conditions were applied to the transonic rotor to demonstrate the effect of the inlet condition on the unsteady flow phenomenon in the rotor tip region. The inlet conditions selected were axial inflow and 16-deg of co-swirl. The results show that different inlet conditions lead to different shock wave intensities and positions, which critically affects the unsteady flow structure in the tip region of the transonic rotor. Under the co-swirl inlet condition, the tip leakage vortex of each blade oscillates synchronously at the near stall point because of the weak interaction between the tip leakage vortex and the shock. Under the axial inlet condition, the rotating instability phenomenon appears as the "multi-passage structure," which propagates along the circumference occurring at the tip of the rotor in the stable operating range due to the strong interaction between the tip leakage vortex and the shock wave. With the decrease of the mass flow, the mode order of the "multi-passage structure" does not change, but the fluctuation frequency decreases gradually.

1. Introduction

The flow in the blade tip region is an important cause of fan/compressor stall [1,2]. Therefore, the flow field characteristic of the rotor blade tip region is the focus of active research. In studies of the tip region flow, a special type of unsteady flow phenomenon, the rotating instability (RI) phenomenon, has been described. The RI phenomenon differs from the modal wave and spike-type stall inception disturbances; it is an unsteady phenomenon that occurs in the tip region of axial compressor rotor during stable operation and has been observed in highly staggered rotors with relatively large tip clearance [3]. Various studies have shown that RI causes vibration of rotor blades and produces relatively large vibration stress, which reduces the fatigue life of blades and can even lead to blade destruction [4–8]. RI is also one of the main sources of noise inside the compressor [9,10].

Mailach et al. [11,12] conducted experimental studies using a low-speed axial compressor. They found that the number and morphology of unsteady flow structures are constantly changing in the stable working range of the compressor, and they hypothesized that the emergence of the RI is caused by unsteadiness of the tip leakage vortex (TLV). Marz et al. [13] observed another vortex that was different from the TLV, which is formed by the interaction of the mainstream, leakage flow and back flow at the end wall, by numerical simulation and dynamic surface pressure measurements in the tip region. They concluded that the unsteady fluctuation of the vortex caused the RI, and they termed the vortex a "rotating instability vortex." Kielb et al. [14] suggested that RI is caused by the common effect of the vortex shedding from the suction side and tip vortex flow instability. Bergner et al. [15] conducted experiments on a transonic rotor (Darmstadt rotor). By analyzing the casing wall static pressure, they found that the unsteady phenomenon at the tip flow field existed at near stall conditions, and in the shock wave and leakage vortex interference region, unsteady fluctuations became more intense. Therefore, they concluded that interference between the shock wave and the leakage vortex is the primary cause of the unsteady fluctuation of the tip region. Biela et al. [16] also measured the flow field of the Darmstadt rotor. They determined that the unsteady flow phenomenon in the tip region is caused by the oscillation of the leakage vortex. Holzinger et al. [17] experimentally investigated the flow field of the Darmstadt rotor. They found that the RI is an aerodynamic phenomenon with the potential to develop into self-excited vibrations, and they proved the dominant role of the tip clearance vortex for the RI-induced blade vibration. Hah et al. [3] conducted an unsteady

E-mail address: wangzq1981@126.com (Z. Wang).

https://doi.org/10.1016/j.ast.2017.11.012
1270-9638/© 2017 Published by Elsevier Masson SAS.

Nomenclature

Notation

Cp	Static surface pressure coefficient, $Cp = (P_s - P_{ref})/(0.5\rho U_t^2)$	
C_{P_rms}	RMS coefficient of the static pressure, $C_{P\_rms} = \sqrt{\frac{1}{n}\sum_{t=0}^{n-1}(P_s(t) - \overline{P_s})^2}/\frac{1}{2}\rho U_t^2$	
f	Oscillation frequency	Hz
H_n	Normalized helicity, $H_n = (\vec{\xi} \cdot \vec{W})/(\lvert\vec{\xi}\rvert \cdot \lvert\vec{W}\rvert)$	
P_s	Static pressure	Pa
T	Oscillation period	s
U_t	Tip speed of the rotor	m/s
$\vec{W}$	Relative velocity vector	m/s
W	Relative velocity magnitude	m/s
x/c	Relative chord length	
ρ	Air density	kg/m^3
$\vec{\xi}$	Vorticity vector	s^{-1}

Subscripts

1	Co-swirl inlet condition
2	Axial inlet condition

Abbreviations

BPF	Blade-passing frequency
CFD	Computational fluid dynamics
FFT	Fast Fourier transformation
IGV	Inlet guide vane
PE	Peak efficiency
RI	Rotating instability
RMS	Root mean square
TLV	Tip leakage vortex

numerical simulation on the same transonic rotor. They found an unsteady vortex similar to that found by Marz [13] and termed it an "induced vortex." The existence of the induced vortex leads to the occurrence of an unsteady phenomenon in the tip region. They supplemented their transonic rotor research with a full annulus large eddy simulation, and found that several blade passages were occupied by the induced vortex. As a result, they concluded that the RI phenomenon is probably caused by the induced vortex [18]. Wu et al. [19,20] characterized the RI phenomenon on a subsonic rotor experimentally and with numerical simulation and on a transonic rotor using numerical simulation. Their results show that the periodic oscillation of "secondary clearance flow" is the primary cause of the RI phenomenon. However, Du [18] suggested that the RI phenomenon is just a special case, due to a small disturbance on the self-excited unsteadiness of the TLV in a multi-passage environment.

Although many previous works have identified the RI phenomenon, the variety of complex flow structures in the rotor tip region and the complex interaction between various flow structures indicate that a unified understanding of the formation mechanism of the RI phenomenon has not been well determined. Furthermore, controversy remains regarding whether RI is an independent flow phenomenon [21,22]. Therefore, to enhance understanding of the RI phenomenon, we conducted a three-dimensional multi-passage unsteady numerical simulation on a transonic axial flow compressor rotor. We first induce the RI phenomenon by changing the inlet flow direction, then study the unsteady flow characteristics in the tip region and analyze the flow signal when the RI phenomenon occurs.

2. Numerical models and methods

Numerical calculations were conducted for a transonic rotor, which is the first-stage rotor of a multistage high-speed axial flow compressor with an inlet guide vane (IGV). The blade number, tip clearance and design speed of the rotor were 40, 1.5 mm and 12,000 r/min, respectively. The primary design parameters of the rotor are shown in Table 1.

Table 1
Design parameters of the transonic compressor rotor.

Parameters	Values
Blade number	40
Casing radius (mm)	306
Mass flow (kg/s)	25.5
Pressure ratio	1.45
Efficiency	0.89
Tip solidity	1.145
Hub-tip ratio (leading edge/trailing edge)	0.70/0.74
Tip clearance (mm)	1.5
Design speed (r/min)	12000

The three-dimensional Reynolds averaged Navier–Stokes equations were solved using the commercial computation fluid dynamics (CFD) flow solver, EURANUS, which was developed by NUMECA International Corporation. The equations were discretized in space using a cell-centered finite volume formulation, and viscous fluxes were determined in a central differencing manner with Gauss's theorem. To more accurately characterize the shock wave and TLV near the casing, a second-order upwind scheme based on a flux difference splitting formulation implementing the van Albada limiter was chosen to evaluate the inviscid fluxes. For steady simulations, the equations were solved using an explicit four-stage Runge–Kutta method to obtain convergent solutions. Local time stepping, implicit residual smoothing and multigrid techniques were used to reduce the computational cost. For unsteady simulations, equations were solved using the implicit dual time stepping method, in which pseudo-time-derivative terms are added to the time-dependent equations. A previous study by Copenhaver et al. [23] showed that a time step of 2.5×10^{-4} s. was necessary to capture shock motion in a transonic rotor. For the present study, the unsteady nature of the flow originates primarily from the interaction of the shock with the TLV. Therefore, a physical time step of 5×10^{-6} s. was used for all unsteady flow simulations, which was sufficiently small to capture the motion of the shock and the TLV. Therefore, 25 time steps existed in one blade-passing period. Within each physical time step, 50 pseudo-time iterations with a CFL number of 3 were performed. When the integrated residuals of all the equations are reduced by four orders of magnitude from their initial values during the subiteration, the solution is advanced to the next time step. Turbulent flow was modeled using a one-equation Spalart–Allmaras turbulence model. The calculation was conducted on 40 processors on I620-G10, a Sugon Linux Cluster at the Lab of Turbomachinery Aerodynamics of NUAA.

The computational grid was generated using Autogrid 5. To ensure grid orthogonality, the HOH structured grid (H-block inlet, O-block blade and H-block outlet) was employed for the main channel of the rotor, and the butterfly grid was applied to the gap region of the rotor. The entire region was separated into five blocks for a single blade passage; the detailed grid points are listed in Table 2. In this table, I, J and K denote azimuthal, spanwise and axial directions, respectively. The grid near the wall was densified. The minimum grid spacing on the solid walls was 2×10^{-6} m, to ensure a minimum grid spacing of $y^+ < 3$ at the walls. The to-

Please cite this article in press as: Z. Wang et al., Numerical simulation of unsteady tip clearance flow in a transonic compressor rotor, Aerosp. Sci. Technol. (2017), https://doi.org/10.1016/j.ast.2017.11.012