Applied Thermal Engineering 103 (2016) 282–290

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

A numerical study on the tip clearance in an axial transonic compressor rotor

Xiaodong Ren [a], Chunwei Gu [b,*]

[a] Department of Mathematics, School of Science, Hong Kong University of Science and Technology, Hong Kong, China
[b] Key Laboratory for Thermal Science and Power Engineering of Ministry of Education, Department of Thermal Engineering, Tsinghua University, Beijing, China

HIGHLIGHTS

- An improved blockage indicator is proposed to quantify the blockage.
- Tip clearance flow has a significant influence on the blockage in the rotor.
- Tip clearance flow in the transonic rotor can be divided into two parts.

ARTICLE INFO

Article history:
Received 14 December 2015
Accepted 18 April 2016
Available online 19 April 2016

Keywords:
Tip clearance flow
Transonic compressor
Spike stall
Discontinuous Galerkin method

ABSTRACT

The tip clearance flow in a transonic compressor rotor has been studied numerically. The three-dimensional Navier–Stokes equations with Spalart–Ammaras (SA) turbulence model are discretized in the physical space by using a discontinuous Galerkin method. The flow field is assumed to be periodic and a single-passage steady state model is used for the numerical simulation. An improved blockage indicator has been proposed to estimate the blockage in the transonic rotor. The results show that the tip clearance has a significant influence on the blockage in the rotor. The tip clearance flow and the main flow is interacted with each other, and the interface between them is becoming parallel to the blade leading edge plane as the mass flow rate is decreasing. The tip clearance flow in the transonic rotor can be divided into two parts, and the interface between them is in contact with the adjacent blade trailing edge in the near stall condition. A part of the tip clearance flow is ejected into the next blade passage when the mass flow rate is small enough. Besides, the shock and the tip clearance flow interaction can be seen clearly. Due to the tip clearance flow, the shock wave bends and its intensity is weakened. Meanwhile, the tip leakage vortex diffuses after the interaction with the shock.

1. Introduction

Tip clearance flow has significant influence on the compressor overall efficiency and stability. Various techniques, such as the casing treatment [1], curved or swept blade and endwall [2], suction-side winglet [3,4] and normal synthetic jet [5,6], have been proposed to control the tip clearance flow to make the compressor achieve a higher overall efficiency and better stability. However, as Day [7] says that “few of our achievements have made the step from qualitative understanding to improve engine designs. What should we be doing to make our research more useful?” In his suggested topics we can find that improving CFD (computational fluid dynamics) predictions for flow conditions near the surge line is a main point.

Nowadays, the 2nd order finite volume method (FVM) is commonly used for the compressor design. The calculations have been proved reliable near the design point, but the inability to predict flow near the surge line is a hindrance [7]. Turbulence models ineffectiveness at near stall conditions is one of the main reasons which hinder CFD predications of off-design performance [7]. Another point to be considered is the high numerical dissipation of the numerical method. Flow at near stall conditions always accompanies large separations. It is best to use a numerical method with lower numerical dissipation to predict these phenomena. The authors have pointed out that the discontinuous Galerkin method (DGM) has lower numerical dissipation and can produce more reliable results [8] at the design condition. Here we will use the DGM to study the tip clearance flow at both design and off-design conditions, though its capacity to predict the flow near the surge line is still uncertain.

* Corresponding author.
E-mail addresses: maxdren@ust.hk (X. Ren), gcw@mail.tsinghua.edu.cn (C. Gu).

http://dx.doi.org/10.1016/j.applthermaleng.2016.04.082
1359-4311/

Dring et al. [9], Smith and Cumpsty [10] and Denton [11,12] have pointed out that the tip clearance flow is one of main sources of the loss in the compressor. The tip clearance flow would lead to an additional blockage in the endwall region of the compressor, especially for the near stall conditions [13,14]. In this paper, the influence of the tip clearance flow on the endwall blockage in a transonic rotor for both design and off-design conditions will be studied. An improved methodology to quantify the endwall blockage, which is based on the method introduced by Khalid et al. [15], will be proposed.

Moreover, the tip clearance flow plays an important role on the compressor stability [14,16–19]. "When stall occurs, the flow breakdown process nearly always begins in the vicinity of the rotor tips [7]". Some special features of the tip clearance flow can be used as the stall criterion. Based on our survey [17–21], we know that there are three features when the spike disturbance, which is one of the well-established routes to rotating stall, in the rotor passage forms. The first is the slope of the total-to-static pressure rise is still negative. The second is the interface between the tip clearance flow and the main flow becomes parallel to the leading edge plane. The last one is the initiation of backflow at the trailing edge plane near the rotor tip. In this paper, these features will be checked to make sure if the DGM will give us the same features or not, and whether other additional features will be presented for the transonic flow.

The paper is organized as follows. Section 2 briefly describes the numerical method. Section 3 shows the boundary condition and mesh convergence. An improved methodology to quantify the endwall blockage is proposed in Section 4. Section 5 shows the numerical analysis. The last section draws the conclusions.

2. Numerical method

The axial compressor rotor is simulated in a rotating coordinate system in this paper. Here we assume the rotational axis coincides with the z-coordinate axis. The angular velocity is written as $\boldsymbol{\omega} = (0, 0, \omega)$, and the absolute flow velocity $\boldsymbol{U}^a = (U^a, V^a, W^a)$ results from the sum of the relative velocity $\boldsymbol{U} = (U, V, W)$ and the entrainment velocity $\boldsymbol{U}^e = \boldsymbol{\omega} \times \boldsymbol{x}$ as

$$\boldsymbol{U}^a = \boldsymbol{U} + \boldsymbol{U}^e = (U - \omega_z y, V + \omega_z x, W), \tag{1}$$

where $\boldsymbol{x} = (x, y, z)$ is the space coordinates in the rotating frame of reference. The three-dimensional governing equations, including the Reynolds-averaged Navier–Stokes (NS) equations and Spalart–Allmaras (SA) turbulence equation, are written as

$$\partial \boldsymbol{Q}(\boldsymbol{x}, t)/\partial t + \nabla \bullet \boldsymbol{F} - \boldsymbol{S} = 0, \ \boldsymbol{x} \in \Omega, \tag{2}$$

where t is the physical time, and $\boldsymbol{Q} = (\rho, \rho U, \rho V, \rho W, \rho E, \rho\tilde{\nu})^T$ is the conservative variables vector with the density ρ, the relative velocity $\boldsymbol{U} = (U, V, W) = (U_x, U_y, U_z)$, the relative total energy E, and the variable of SA turbulence model $\tilde{\nu}$. Fluxes and source terms are defined as

$$\boldsymbol{F} = \boldsymbol{F}^I - \boldsymbol{F}^V = (\boldsymbol{F}^I_x - \boldsymbol{F}^V_x, \boldsymbol{F}^I_y - \boldsymbol{F}^V_y, \boldsymbol{F}^I_z - \boldsymbol{F}^V_z), \tag{3}$$

$$\boldsymbol{F}^I_i = \begin{pmatrix} \rho U_i \\ \rho U_x U_i + \delta_{xi} p \\ \rho U_y U_i + \delta_{yi} p \\ \rho U_z U_i + \delta_{zi} p \\ U_i(\rho E + p) \\ \rho U_i \tilde{\nu} \end{pmatrix}, \ i = x, y, z, \tag{4}$$

$$\boldsymbol{F}^V_i = \begin{pmatrix} 0 \\ \tau_{xi} \\ \tau_{yi} \\ \tau_{zi} \\ U\tau_{xi} + V\tau_{yi} + W\tau_{zi} - q_i \\ \frac{\mu}{\sigma}\frac{\partial \tilde{\nu}}{\partial i} + \frac{\sqrt{\rho}\tilde{\nu}}{\sigma}\frac{\partial(\sqrt{\rho}\tilde{\nu})}{\partial i} \end{pmatrix}, \ i = x, y, z, \tag{5}$$

$$\boldsymbol{S} = \begin{pmatrix} 0 \\ \rho(\omega_z^2 x + 2\omega_z V) \\ \rho(\omega_z^2 y - 2\omega_z U) \\ 0 \\ \rho\omega_z^2(Ux + Vy) \\ c_{b1}\tilde{S}\rho\tilde{\nu} - \frac{c_{b2}}{\sigma}\nabla(\sqrt{\rho}\tilde{\nu}) \bullet \nabla(\sqrt{\rho}\tilde{\nu}) + c_{w1} f_w \rho\left(\frac{\tilde{\nu}}{d}\right)^2 \end{pmatrix}, \tag{6}$$

in which δ is Kronecker delta.

The perfect gas equation of state is employed in this paper as

$$p = \rho RT = (\gamma - 1)\rho\left(E - \frac{1}{2}\boldsymbol{U} \bullet \boldsymbol{U}\right), \tag{7}$$

in which R = 287.096 is the gas constant, γ = 1.4 is the specific heat ratio, and p and T are the pressure and temperature, respectively. The viscous stress and heat flux in the flux terms can be calculated by $\tau_{xx} = 2(\mu + \mu_t)(\frac{\partial \boldsymbol{U}}{\partial x} - \frac{2}{3}\nabla \bullet U)$, $\tau_{xy} = \tau_{yx} = (\mu + \mu_t)\left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x}\right)$, $\tau_{xz} = \tau_{zx} = (\mu + \mu_t)(\frac{\partial \boldsymbol{U}}{\partial z} + \frac{\partial W}{\partial x})$, $\tau_{yy} = 2(\mu + \mu_t)\left(\frac{\partial V}{\partial y} - \frac{2}{3}\nabla \bullet U\right)$, $\tau_{yz} = \tau_{zy} = (\mu + \mu_t)\left(\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y}\right)$, $\tau_{zz} = 2(\mu + \mu_t)(\frac{\partial W}{\partial z} - \frac{2}{3}\nabla \bullet U)$, $\boldsymbol{q} = (q_x, q_y, q_z)^T = -\left(\frac{\mu}{\mathrm{Pr}} + \frac{\mu_t}{\mathrm{Pr}_t}\right) c_p \nabla T$, and Pr equals 0.71, and Pr_t equals 0.9.

Sutherland formula $\mu = \mu_0(T - T_0)^{\frac{3}{2}}(T_0 + T_s)/(T + T_s)$ is used to calculate the molecular dynamic viscosity μ. These constants in this formula are $\mu_0 = 1.711 \times 10^{-5}$, $T_0 = 273.16$ K, and $T_s = 110.4$ K. SA turbulence model is employed to compute the turbulent eddy viscosity μ_t

$$\mu_t = \rho\tilde{\nu} f_{v1}, \tag{8}$$

in which $f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}$, $\chi = \frac{\rho\tilde{\nu}}{\mu}$. Other parameters are given as follows:

$$\tilde{S} = S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}, \tag{9}$$

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}, \tag{10}$$

$$f_w = g\left(\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6}\right)^{\frac{1}{6}}, \tag{11}$$

$$g = r + c_{w2}(r^6 - r), \tag{12}$$

$$r = \min\left(\frac{\tilde{\nu}}{\tilde{S}\kappa^2 d^2}, 10\right), \tag{13}$$

where $S = \sqrt{2S_{ij}S_{ij}}$, $i, j = x, y, z$ is the vorticity magnitude, and d is the wall distance for each flow field point. It should be noted that the vorticity is the absolute vorticity as follow

$$\begin{bmatrix} S_{xx} & S_{xy} & S_{xz} \\ S_{yx} & S_{yy} & S_{yz} \\ S_{zx} & S_{zy} & S_{zz} \end{bmatrix} = \begin{bmatrix} 0 & \frac{1}{2}\left(\frac{\partial U}{\partial y} - \frac{\partial V}{\partial x}\right) - \omega_z & \frac{1}{2}\left(\frac{\partial U}{\partial z} - \frac{\partial W}{\partial x}\right) \\ \frac{1}{2}\left(\frac{\partial V}{\partial x} - \frac{\partial U}{\partial y}\right) + \omega_z & 0 & \frac{1}{2}\left(\frac{\partial V}{\partial z} - \frac{\partial W}{\partial y}\right) \\ \frac{1}{2}\left(\frac{\partial W}{\partial x} - \frac{\partial U}{\partial z}\right) & \frac{1}{2}\left(\frac{\partial W}{\partial y} - \frac{\partial V}{\partial z}\right) & 0 \end{bmatrix}. \tag{14}$$

These constants in SA turbulence model are $c_{b1} = 0.1355$, $c_{b2} = 0.622$, $\sigma = \frac{2}{3}$, $\kappa = 0.41$, $c_{w2} = 0.3$, $c_{w1} = \frac{c_{b1}}{\kappa^2} + \frac{1 + c_{b2}}{\sigma}$, $c_{w3} = 2$, and $c_{v1} = 7.1$.