Chinese Journal of Aeronautics, (2016), **29**(3): 639–652

CSAA

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

cja@buaa.edu.cn
www.sciencedirect.com

Numerical study of corner separation in a linear compressor cascade using various turbulence models

Liu Yangwei, Yan Hao, Liu Yingjie *, Lu Lipeng, Li Qiushi

Collaborative Innovation Center of Advanced Aero-Engine, Beihang University, Beijing 100083, China
National Key Laboratory of Science and Technology on Aero-Engine Aero-Thermodynamics, School of Energy and Power Engineering, Beihang University, Beijing 100083, China

Received 14 May 2015; revised 30 June 2015; accepted 6 January 2016
Available online 10 May 2016

KEYWORDS

Compressor cascade;
Corner separation;
Turbomachinery CFD;
Turbulence anisotropy;
Turbulence models

Abstract Three-dimensional corner separation is a common phenomenon that significantly affects compressor performance. Turbulence model is still a weakness for RANS method on predicting corner separation flow accurately. In the present study, numerical study of corner separation in a linear highly loaded prescribed velocity distribution (PVD) compressor cascade has been investigated using seven frequently used turbulence models. The seven turbulence models include Spalart–Allmaras model, standard k–ε model, realizable k–ε model, standard k–ω model, shear stress transport k–ω model, v^2–f model and Reynolds stress model. The results of these turbulence models have been compared and analyzed in detail with available experimental data. It is found the standard k–ε model, realizable k–ε model, v^2–f model and Reynolds stress model can provide reasonable results for predicting three dimensional corner separation in the compressor cascade. The Spalart–Allmaras model, standard k–ω model and shear stress transport k–ω model overestimate corner separation region at incidence of 0°. The turbulence characteristics are discussed and turbulence anisotropy is observed to be stronger in the corner separating region.

1. Introduction

Three-dimensional (3D) corner separation in compressor cascade has been considered as an inherent flow phenomenon in compressor cascade.[1] It has great impact on compressor performance, such as higher total pressure loss, static pressure rising limitation, compressor efficiency reduction, passage blockage, and stall and surge especially for highly loaded compressor.[2] Many efforts have been made on studying the corner separation by experimental investigations[3–5] and numerical simulations[6,7] under various conditions over the past few years. Several flow controlling techniques have been

* Corresponding author at: Collaborative Innovation Center of Advanced Aero-Engine, Beihang University, Beijing 100083, China. Tel.: +86 10 82316455.
E-mail addresses: liuyangwei@126.com (Y. Liu), yanhaovsdami@163.com (H. Yan), liuyingjie@buaa.edu.cn (Y. Liu), lulp@buaa.edu.cn (L. Lu), liqs@buaa.edu.cn (Q. Li).

Peer review under responsibility of Editorial Committee of CJA.

ELSEVIER | Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.cja.2016.04.013

utilized to improve compressor performance recently.[8–11] The basic structures and characteristics of 3D corner separation are well-summarized[12], thus their primary effects on compressor performance have been considered in designing process and have improved compressor performance greatly. In compressor routine design, the corner separation should be accurately predicted by employing computational fluid dynamics (CFD).

The CFD technique has been widely used and has played an increasingly important role in the aerodynamic design routine of compressor. The Reynolds-averaged Navier–Stokes equations (RANS) method is still the most widely used approach in industrial CFD due to the computational cost and design schedule, though direct numerical simulation (DNS), large eddy simulation (LES) and hybrid LES/RANS (such as detached eddy simulation (DES)[13], scale adaptive simulation (SAS)[14]) have been used to investigate flow mechanisms in relative low Reynolds number with simple boundary.[15–18] According to Spalart[19,20], while it has been assumed that limitless increases in computing power will someday remove the need for turbulence modeling, the estimates for this milestone have been close to the year 2080, which is a long time away.

Turbulence model is one of the key elements and is currently a weakness for RANS approach. Far less precision has been achieved in turbulence modeling since a mathematical model has been created to approximate the complicated physical behavior of turbulent flows. It is generally acknowledged not any turbulence model is suitable for all kinds of complex flow problems.[21] Hence, the performance of various widely used turbulence models should be assessed for a certain kind flow and then be improved.[22–27] Then we could build up the scope of application for various turbulence models.

The next high-power-density generation aeroengines must be supported by more advanced compressors in the future and this requires that the RANS method applied in designing process can predict main flow structures in compressor cascade to decrease designing risk as much as possible. It is very practical and critical to simulate the corner separation accurately by using RANS approach. However, it is a big challenge for RANS method to precisely predict such complicated turbulent flow field, because the 3D corner separation contains complex vortices and large separation in cascade corner region.[28]

In this paper, seven turbulence models which are frequently applied in engineering have been used in the detailed numerical investigations using software FLUENT for a linear highly loaded compressor cascade experimentally studied by Gbadebo et al.[29] In order to minimize grid impacts on numerical results, great efforts for the grid generation have been made till grid independence was obtained. The numerical results of seven turbulence models have been compared with experimental data carefully. The turbulence characteristics are discussed for analyzing the mechanisms for discrepancies. This study provides a worthy reference for using turbulence models properly and helps designers understand numerical results further. It provides valuable reference on modifying turbulence models for 3D corner separation.

2. Numerical method

In this study, the same numerical platform and grid have been used to minimize other factors' effects on numerical results when assessing different turbulence models. The commercial flow solver package FLUENT[30] is applied to conduct numerical simulation in the cascade.

2.1. Governing equations and turbulence models

The governing equations of steady RANS method for incompressible fluid field are as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$u_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\cdot\frac{\partial p}{\partial x_i} + \upsilon\frac{\partial^2 u_i}{\partial x_j^2} + \frac{1}{\rho}\cdot\frac{\partial(-\rho\overline{u_i'u_j'})}{\partial x_j} \tag{2}$$

where ρ is density of the fluid, p is pressure of the fluid, u_i is a mean component of velocity in the direction x_i. υ is kinematic viscosity. The additional fluctuation quantities $\overline{u_i'u_j'}$ are unknown Reynolds-stress tensors, while u_i' represents the velocity fluctuation in i-direction, the bar above represents the time-averaged result of the invariant.

In the momentum equation, the Reynolds stress tensors $\overline{u_i'u_j'}$ are unknown which makes this equation unclosed. The Reynolds stress tensors are modeled by turbulence model. Turbulence models utilized in the study except for the Reynolds stress model are based on the Boussinesq hypothesis, which is expressed as follows:

$$-\rho\overline{u_i'u_j'} = \mu_\mathrm{t}\left(2S_{ij} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right) - \frac{2}{3}\delta_{ij}\rho k \tag{3}$$

where μ_t is turbulent viscosity, δ_{ij} is Kronecker delta function ($\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$), k is the turbulent kinetic energy. The repeated index implies summation from 1 to 3. S_{ij} is the mean strain-rate tensor which is calculated as

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{4}$$

Turbulent viscosity μ_t are modeled by various turbulence models. The transport equations are different from one turbulence model to another, so the turbulence viscosity is quite different which leads to discrepancies on the results of complicated turbulent flow field.

(1) Spalart–Allmaras model (SA)
As a simple one-equation model, the SA model solves a transport equation for the kinematic eddy viscosity. It has been proposed and developed by Spalart and Allmaras[31,32] since 1992. The transportation equation of this model in FLUENT is as follows:

$$\frac{\partial}{\partial t}(\rho\tilde{\upsilon}) + \frac{\partial}{\partial x_j}(\rho\tilde{\upsilon}u_j) = c_{\mathrm{b}1}\rho\tilde{S}\tilde{\upsilon} + \frac{1}{\sigma_{\tilde{\upsilon}}}\left\{\frac{\partial}{\partial x_j}\left[(\mu+\rho\tilde{\upsilon})\frac{\partial\tilde{\upsilon}}{\partial x_j}\right] + c_{\mathrm{b}2}\rho\left(\frac{\partial\tilde{\upsilon}}{\partial x_j}\right)^2\right\} - c_{\mathrm{w}1}\rho f_\mathrm{w}\left(\frac{\tilde{\upsilon}}{d}\right)^2 \tag{5}$$

The turbulent viscosity is computed as

$$\mu_\mathrm{t} = \rho\tilde{\upsilon}f_{\upsilon 1} \tag{6}$$

and