International Journal of Heat and Mass Transfer 55 (2012) 6757–6765

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Performance of 2D scheme and different models in predicting flow turbulence and heat transfer through a supersonic turbine nozzle cascade

Liang Guo [a,b], Yuying Yan [a,*], John D. Maltson [c]

[a] Faculty of Engineering, The University of Nottingham, Nottingham NG7 2RD, UK
[b] Department of Internal Combustion Engine, Jilin University, Changchun 130022, China
[c] Siemens Industrial Turbomachinery Limited, Lincoln LN5 7FD, UK

ARTICLE INFO

Article history:
Received 23 January 2011
Accepted 18 June 2012
Available online 26 July 2012

Keywords:
Turbine cascade
Unsteady flow
Wake
CFD
Turbulence model

ABSTRACT

A number of turbulence models were employed to investigate the heat transfer and aerodynamic characteristics through a nozzle cascade of a high-pressure gas turbine. Isentropic Mach number and Nusselt number around the vane were predicted and compared to existing experimental data obtained at a supersonic flow condition. According to the result presented by different models, possible source of the prediction error was identified; and the performance of different turbulence closures in predicting the heat transfer characteristics around the vane was discussed. It shows that the calculated heat transfer result was affected directly by the predicted turbulence transportation throughout the boundary layer. Finally considering the computational cost and the performance of the models, the suitable model(s) are recommended for the further 3D applications.

1. Introduction

The flow in the turbine passage between the upstream vanes and the rotor blades has extremely high unsteadiness. The unsteady flow gives significant impact on not only the performance of individual components but also the efficiency of the whole system.

Over the past decades, many studies have been conducted either experimentally or numerically to investigate the turbulent flow through the turbine cascade. Theoretical studies on the behavior of wake-induced boundary layer separation were carried out by Mayle and Dullenkopf [1], Walker [2,3], Gostelow [3] and Addison and Hodson [4,5], respectively. The effects of the unsteady wake and other flow unsteadiness on laminar-to-turbulent transition were discussed deeply with corresponding theories being developed significantly. With a view towards design, unsteady flow and wake-induced losses which accompanied by the various passage vortices were investigated experimentally by Payne et al. [6,7], Ubaldi and Zunino [8] and numerically by Kulisa and Dano [9]. In the studies of Payne and Kulisa, the entropy was qualified for evaluating the unsteady losses across the stage; by this means the effects of pressure fluctuation, vortex and the second flow on efficiency of the stage can be analyzed. The effects of the unsteadiness on boundary layer separation and surface heat transfer were studied recently by Rhee and Cho [10], Schobeiri et al. [11] and Gao et al. [12]. Rhee reported that the relative blade position determines the behavior of the incoming flow and may further lead to variations in velocity and turbulence field throughout the stage. For the surface heat transfer, according to Schobeiri et al. the behaviors of the velocity boundary layer are completely decoupled from performances of the thermal boundary layer, but the behaviors of the thermal layer are directly coupled with the boundary layer aerodynamics.

Due to the limitations of the testing methods in the current stage, experimental studies were still imperfect in capturing all the detailed behaviors of flow unsteadiness. For example, it is not easy to point out the weak fluctuations carried by the wakes or distinguish the shock-induced transition from the natural one. Numerical prediction, as an alternative has therefore been applied and developed to provide additional information for flow transport and heat transfer through the blade rings. Selection of the turbulence model is one of the crucial issues for the above problems. In the earlier stage, RANS-based closures were wildly employed for such simulations. According to Pouagare and Delaney [13], Chen and Kim [14], Moroz et al. [15] and Garg [16], the RANS-based turbulence models are capable to provide accurate solutions for the general problems. However, for the problems like supersonic flow, blade rotation [17], wake passing and their effects on the surface heat transfer; the accuracies of RANS-based closures were still debatable.

Nowadays with a rapid development of CFD technique and boosting of the computational resource, turbulence models such as LES and DES have more often been applied to the studies

* Corresponding author. Address: Faculty of Engineering, The University of Nottingham, B22 Lenton Firs Building, University Park, Nottingham NG7 2RD, UK. Tel.: +44 0 115 951 3168.

E-mail address: yuying.yan@nottingham.ac.uk (Y. Yan).

0017-9310/$ - see front matter
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2012.06.083

Nomenclature

C	chord length
C_p	constant pressure specific heat
C_μ	coefficient for k–ε model
C_{des}	calibration coefficient for DES model
D_{trg}	trailing edge tip diameter
h	heat transfer coefficient
k	turbulence kinetic energy
L	turbulence length scale
Ma	Mach number
Nu	Nusselt number
NGV	nozzle guide vane
p	pressure
Re	Reynolds number
T	temperature
Tu	turbulence intensity
TKE	turbulence kinetic energy
u'	fluctuating velocity
x,y	x and y coordinate
Y	dissipation term for turbulence model
α^*	damping coefficient
β^*	constant of the SST model
γ	specific heat capacity ratio
κ	thermal conductivity
ρ	density
ε	dissipation rate of turbulence kinetic energy
ω	specific dissipation rate of turbulence kinetic energy k
μ_t	eddy viscosity

Subscripts

1	nozzle cascade inlet
2	nozzle cascade outlet/rotor cascade inlet
3	rotor cascade outlet
b	parameter relates to rotor blade
gas	parameter relates to freestream condition
s	local static
t	total
v	parameter relates to nozzle guide vane
wall	parameter relates to wall condition
x	in x direction
y	in y direction

relating to highly turbulent flow. $\bar{v}^2 - f$ model, developed from the conventional RANS model has also be qualified as an effective one to give satisfied solution for rotating machinery. Among them, the LES model was performed for both 2D and 3D evaluations in previous studies [18–21]. It shows that the complex turbulent flows relevance in turbomachinery can be predicted realistically by LES models, especially for the problems related to vortex-shedding and unsteady wakes. The performance of the D-DES model (Delayed Detached Eddy Simulation) in predicting the aerodynamic characteristics of a turbine cascade was examined by Gu et al. [22]. They concluded that with a properly calibrated coefficient C_{des}, the D-DES model performed better than the RANS models, and more detailed information of the unsteadiness could be obtainable from the result. The $\bar{v}^2 - f$ model developed by Durbin [23] uses an elliptic relaxation methodology in which the near-wall turbulence anisotropy and non-local pressure–strain effects were incorporated. It has been proved to have higher capacity in dealing with flow separation, heat transfer and wall friction over two-equation models [24,25]. Although each turbulence model showed its predominance on specified areas of the modeling, further investigations are still desired to evaluate the overall performance of the above models on all-sided prediction of the cascade flow.

Over the years 2D modeling has been validated competent to give reasonable solutions for general problems relating to the turbulence and heat transfer with a much cheaper computational cost. But according to the open literature in the current stage, few references are available for the performance of 2D modeling upon the supersonic rotating cascade.

In order to improve the understanding of performance of the 2D modeling in dealing with the supersonic flow; investigation was carried out in the present work using a sliding mesh scheme applied with various turbulence models. Numerical results were then validated using existing experimental data, and the most suitable model(s) were recommended for the future study.

2. Problem definitions

2.1. Geometry and boundary conditions

The blade geometry was firstly extracted from a 3D geometry with a curve-fitted sheet at 50% blade height and then projected onto an orthogonal plane. As shown in Fig. 1, the domain includes 3 nozzle guide vanes and 5 rotor blades, pressure-controlled inlet and outlet conditions are adopted for the modeling. A sliding interface condition similar to that employed by Yang et al. [26] was applied to link the nozzle cascade with the blade row of the rotors. By this means, the interaction between the upstream cascade and the downstream row can be modeled properly. No-slip boundaries were used for all blade walls of a constant temperature. The top and bottom borders along each cascade were enforced periodic. Additional information about the blades' parameters and flow configurations are shown in Table 1.

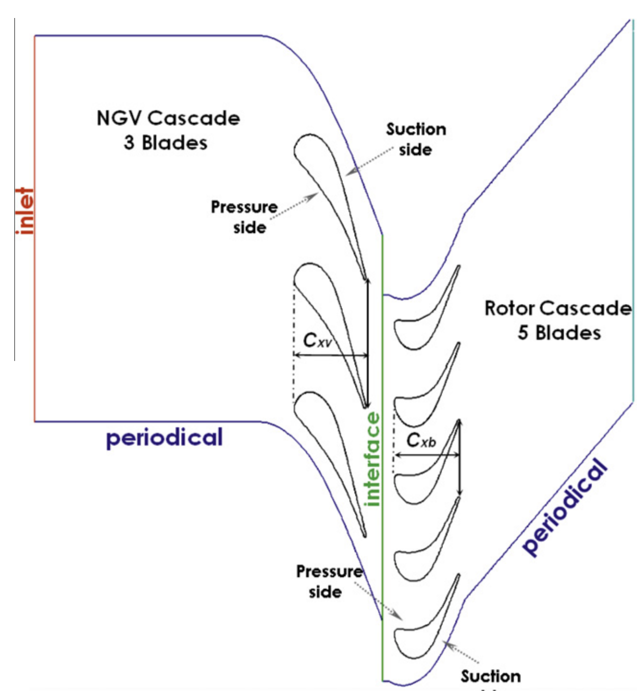

Fig. 1. Computational domain and boundary conditions.