Chinese Journal of Aeronautics, (2018), **31**(5): 1095–1108

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

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

Effect of inlet rotating swirl on endwall film cooling for two representative hole arrangements

Yang ZHOU, Yang ZHANG, Xinrong SU *, Xin YUAN

Key Laboratory for Thermal Science and Power Engineering of Ministry of Education, Tsinghua University, Beijing 100084, China

Received 9 January 2017; revised 23 October 2017; accepted 9 January 2018
Available online 22 March 2018

KEYWORDS

Films-cooling;
Holes locating;
Isobaric;
Swirling flow;
Turbine design

Abstract The swirl generator is widely used in lean-burn combustor to guarantee flame stability and reduce NO_x emissions. Thus, the non-uniformities induced by the swirler would affect and damage film-cooling effectiveness on the turbine components, even blow-off coolant coverage, to some certain extent. The arrangement of film-cooling holes was normally designed to be perpendicular to the axial direction and in standard straight row. In this work we experimentally studied the effects of inlet swirl and mass flow ratio on traditional film cooling holes arrangement and a new arrangement pattern whose holes are located along the isobars. Results indicated that the swirl perturbation would damage film coverage. The film-cooling effect of endwall on which the holes are along the isobars, is invariably more promising than that of endwall on which hole arrangement is perpendicular to axial with the same mass flow of coolant, whether the inlet conditions is uniform or not.

1. Introduction

In modern turbomachinery, the constantly increasing turbine inlet temperature proposes stringent challenge to the design, and advanced cooling technology is heavily employed to protect the high temperature turbine components.[1] To reduce the NO_x emission and preserve the combustion stability, swirl generator is widely used in lean-burn combustor.[2] Inside the combustor, the swirling flow is not fully smeared and contributes to the non-uniform flow field at the combustor-turbine interface. As a result, the real inlet flow condition of turbine is non-uniform and the non-uniformity would have negative impact on the aerodynamics and heat transfer features of the turbine. Consequently, the investigation on inlet swirl and Inlet Guide Vane (IGV) endwall film-cooling[3] is exceedingly indispensable.

In the last decade, several related studies concerning the combustor-turbine interactions were made. Shih and Lin[4] studied the interaction mechanism between inlet swirl and leading-edge airfoil fillet. They controlled secondary flow using these two methods to reduce aerodynamic loss and vane surface heat transfer and found that the swirls and fillets increased the size of the stagnation region on the endwall around the airfoil's leading edge. Fillets accomplished this by geometry

* Corresponding author.
E-mail address: suxr@mail.tsinghua.edu.cn (X. SU).
Peer review under responsibility of Editorial Committee of CJA.

ELSEVIER Production and hosting by Elsevier

https://doi.org/10.1016/j.cja.2018.03.008
1000-9361

Nomenclature

C	actual chord length of scaled-up blade profile
D	film-hole diameter, mm
i	incidence angle
I	light intensity
L	length of film hole, mm
M	blowing ratio
Ma	Mach number
PS	pressure side
P	partial pressure
PSP	pressure-sensitive paint
Re	Reynolds number
SS	suction side
TE	Trailing edge
V	velocity, m/s
X,Z	Cartesian coordinate system
η	film-cooling effectiveness

Subscripts

aw	adiabatic wall
air	air condition
ax	axial chord
blk	background value
c	coolant fluid
in	inlet
mix	mixture condition
O_2	pure oxygen
ratio	partial pressure of oxygen
ref	reference value
∞	freestream condition

contouring and swirl flow achieved this by more kinetic energy. But they did not discuss the effect of inlet swirl on the film covering on the endwall. Researchers in Oxford University proposed the Integrated Combustor Vane (ICV) concept, Rosic et al.[5] investigated numerically the interaction between the combustor and the first vane in an industrial turbine inside which 16 can-combustors and 32 vanes were installed. It was claimed that the combustor walls had dreadful effects on the first vane film cooling. Promising combustor-vane integration minimized the axial distance between the combustor and vane. In addition, the cooling and aerodynamics performance of the turbo system was not detrimentally affected. Beneficial in reducing secondary flows in the ICV vane, the integrated design also saved up to 25% of the total coolant. Jacobi and Rosic[6] compared aerothermal experimental data and their in-house Computational Fluid Dynamics (CFD) code TBLOCK simulation to analyse the decrease in total pressure loss coefficient and heat transfer coefficient levels of integrated vanes. Their method halved the number of vanes, eliminated the coolant need of leading edge and considerably reduced manufacturing and development costs. Furthermore, Khanal et al. studied the effect of combustor hot streak and swirl on the unsteady aerothermal performance of high pressure turbine stage based on MT1,[7] then proposed a combined performance parameter which enabled aerodynamics and heat transfer behaviour to be quantified and marked in their solver HYDRA. According to He et al.[8], the combined interplay of hot-streak and swirl resulted in crucial variations in heat transfer and aerodynamic performance. Qureshi et al.[9] investigated the effect of residual swirl from combustor exit on thermodynamics by comparing with and without inlet swirl. Inside their research, experimental part presented time-averaged heat transfer and static pressure measurement in an unshrouded turbine, and numerical simulation part based on Rolls-Royce in-house code HYDRA also were proceeded. In view of Oxford colleagues' research, Beard et al.[10] dig deeper to study the relationship between inlet swirl and stage efficiency. They conducted the experiment about the effect of swirl on the turbine stage efficiency with an efficiency measurement system capable of resolving efficiency changes within $\pm$ 0.16%. However, their current research hardly focused on endwall film cooling which is very significant in modern gas turbine system. Yang et al.[11] predicted film cooling and heat transfer characteristic by numerical simulations with consideration of stator-rotor purge flow and discrete hole flows. The calculation was carried out based on Reynolds stress turbulence model with a non-equilibrium wall function. Laskowski et al.[12] conducted several research cases about turbine vanes with leading edge film cooling in the conjugate calculation method which can be in analogy with endwall film cooling. Dong et al.[13] constructed a CFD model to study the impact of vane wakes and leading edge bow wakes on ingestion, meanwhile, predicted turbine wheelspace cooling flow interactions with transonic hot gas path. These simulation research was meaningful in the study of industrial complicated calculations, especially for actual turbine working conditions, such as inlet swirl.

As for experimental investigation, Cha et al.[14] measured temperature and flow fields of full-annual, rich-burn combustors and high pressure turbines using passive scalar tracing methods which were isothermal and non-reacting. The complementary results generally provided a datasets benchmark for conventional turbines design in the future. Koupper et al.[15] developed a kind of engine representative combustor simulator to generate hot streak profile. This trisector rig was equipped with an automatic control system which allowed the facility to operate at stable situations for long time runs. The research results mentioned above were conducted on the industrial sized rigs, it is more difficult to monitor the exact mechanism about the flow phenomenon, than the same experiments on simple cascades. Especially for catalytic and dry low NO_x combustor, Ames et al.[16] measured endwall heat transfer distributions under turbulence from the combustor exit in a large-scale, low speed linear cascade facility. They made the assumption that the individual vortex had less evident influences. With consideration of a large range of Re numbers and turbulence conditions, the database of the exercise could be an assessment of endwall thermal simulation capabilities. Hedlund et al.[17] observed heat transfer and flow phenomena in a swirl combustor and relative turbine blade passage, and found that surface Nusselt numbers and time-averaged flow characteristics had important variations because of arrays of passage vortex pairs. Papa et al.[18] used naphthalene-saturated air and oil-dot visual-