Experimental Thermal and Fluid Science 81 (2017) 314–326

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

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

Experimental investigation of the influence of freestream turbulence on an anti-vortex film cooling hole

Stephen A. Hayes, Andrew C. Nix *, Christopher M. Nestor, David T. Billups, Shane M. Haught

Department of Mechanical and Aerospace Engineering, West Virginia University, 271 ESB Annex, Morgantown, WV 26506-6106, USA

ARTICLE INFO

Article history:
Received 16 March 2016
Received in revised form 18 October 2016
Accepted 20 October 2016
Available online 25 October 2016

Keywords:
Gas turbine
Film cooling
Heat transfer
Infrared thermography
Anti-vortex hole

ABSTRACT

Film cooling is used in gas turbines to thermally protect combustor and turbine hot section components by creating a layer of relatively cooler air to insulate the components from the hot freestream gases. This relatively cooler air is taken from upstream in the compressor section at a loss to the engine efficiency and therefore must be used as effectively as possible. A novel anti-vortex hole (AVH) geometry has been investigated experimentally through a transient infrared thermography technique to study the film cooling effectiveness and heat transfer coefficient by varying blowing ratio and freestream turbulence intensity. The AVH geometry is designed with two secondary holes stemming from a main cooling hole that attempts to diffuse the coolant jet and mitigate the vorticity produced by conventional straight holes. The AVH geometry data collected showed improved cooling performance at low freestream turbulence intensities compared to conventional straight holes. Three turbulence intensities of 1, 7.5, and 11.7% were investigated for the AVH geometry at blowing ratios of 0.5, 1.0, 1.5, and 2.0 for a total of twelve different conditions. Results showed that higher freestream turbulence conditions were beneficial to the performance of the AVH. Increasing the blowing ratio at all turbulence levels also improved film cooling effectiveness, both span-averaged and on the centerline. The highest performing case was at a turbulence intensity of 7.5% and a blowing ratio of 2.0. The 11.7% cases outperformed the 1% cases, but it appears that at the 11.7% cases the higher freestream turbulence reduces the performance of the secondary holes compared to the 7.5% case. Increasing the blowing ratio and turbulence intensity will result in a higher heat transfer coefficient, which must be taken into account for future designs and overall reduction of heat load.

1. Introduction

One primary way to increase the efficiency and power output of a gas turbine engine is to raise the combustor exit temperature, and therefore, the turbine inlet temperature. The melting point of the materials in the combustor and turbine sections are often exceeded and the components can and will fail due to thermal loading if not cooled. As a result, relatively cooler air from the compressor section upstream must be extracted and used to cool the components in the turbine section. Cooling of components occurs through internal cooling passages or externally via film cooling holes. Film cooling is an external cooling method in which the coolant is ejected through discrete holes in the airfoil surface creating a layer of insulation over the surface, protecting it from the hot freestream gas in the region directly downstream of the holes. For areas of low curvature on turbine airfoils, cooling of these regions can be modeled by flat plate studies. Multiple types of cooling hole geometries have been studied including, but not limited to the conventional straight hole, shaped holes, and multi-hole concepts. Kadotani and Goldstein [1,2] determined that freestream turbulence intensity, blowing ratio (BR), and the ratios of turbulence length scale to cooling hole diameter and length scale to boundary layer thickness were of greatest importance when studying the variation of film cooling effectiveness. Blowing ratio is defined in Eq. (1) below where ρ_c is the coolant density, ρ_∞ is the freestream density, U_c is the coolant velocity exiting the holes, U_∞ is the freestream velocity.

$$BR = \frac{\rho_c U_c}{\rho_\infty U_\infty} \tag{1}$$

Han and Mehendale [3] found for a single row of conventional straight holes that a blowing ratio between 0.5 and 0.7 was best for coverage of film cooling. Two significant issues with standard hole configurations are the lift off that occurs as the coolant jet exits the hole at higher blowing ratios and the formation of

* Corresponding author.
E-mail address: andrew.nix@mail.wvu.edu (A.C. Nix).

http://dx.doi.org/10.1016/j.expthermflusci.2016.10.025
0894-1777/

Nomenclature

BR	blowing ratio or mass flux ratio $(\rho V)_c/(\rho V)_\infty$
C_p	specific heat at constant pressure
D	main film cooling hole diameter
h	heat transfer coefficient
k	thermal conductivity
p	hole pitch
Re	Reynolds number
t	time
T	temperature
Tu	turbulence intensity
U	velocity
x	streamwise direction from hole leading edge
z	span wise direction from the hole centerline

Greek

α	thermal diffusivity, coolant injection angle
δ	boundary layer thickness
η	film cooling effectiveness
η_{aw}	adiabatic film cooling effectiveness
ρ	density

Subscripts

aw	adiabatic wall
c	coolant conditions
i	coordinate direction (parallel with x), pixel number
∞	freestream inlet conditions
f	film
RMS	root mean square
w	wall condition
X	streamwise distance

Abbreviations

AVH	anti-vortex hole
CFD	computational fluid dynamics
CRV	counter rotating vortex
IR	infrared
LFE	laminar flow element
RMS	root mean square
WVU	West Virginia University

vortices that entrain the hot freestream gas along the airfoil surface. Haven et al. [4] described and illustrated in detail, this phenomenon known as a counter-rotating vortex (CRV) pair as seen below in Fig. 1.

The combination of the CRV pair and blowing ratios in gas turbine engines more commonly being 1.5 and above led to research into mitigating the CRV pair at higher blowing ratios with the use of different cooling hole geometries. The main idea behind shaped and multi-hole designs is derived from the need to decrease the jet velocity out of the holes by increasing the hole exit area relative to the hole entrance area therefore reducing the jet lift off. Bunker [5] presented a review of shaped holes that included a laidback (allowing expansion downstream along the surface) and a fan shaped hole (allowing lateral expansion) as well as combinations of the two.

The focus of the current work is to experimentally investigate the effectiveness of a multi-hole cooling concept, the anti-vortex hole (AVH) geometry, and to analyze the effects of freestream turbulence on cooling performance at varying blowing ratios. The AVH configuration, shown in Fig. 2, consists of a main cooling hole with two sister holes that branch out from the main hole. As shown in the figure, the hole geometry is defined by several angles and spacings. Details of the actual geometry tested in the current study are shown later in Table 1. The AVH geometry is an attempt to reduce both lift off and vortex generation. The benefits of the

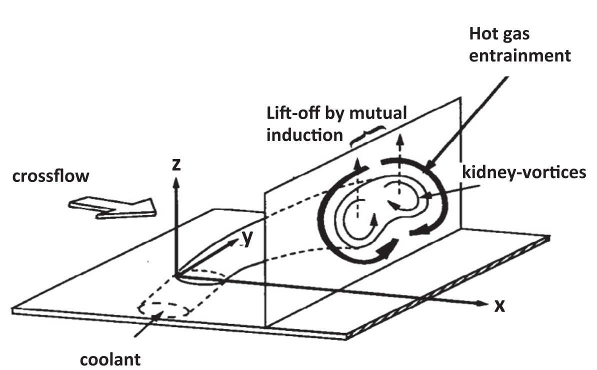

Fig. 1. Counter-rotating vortex pair and jet lift-off from Haven et al. [4].

AVH geometry were realized through CFD simulations prior to this investigation [6,7] and for low freestream turbulence intensities through experimentation [8]. This geometry was selected because it was the optimal performing configuration of this geometry for low freestream turbulence [8]. Previous studies to measure and evaluate the effects of freestream turbulence intensity on this AVH geometry were performed solely computationally [9–12]. These studies showed that the AVH geometry has improved cooling performance as turbulence intensity increases. The current paper will report on experimental measurements of the effect of turbulence intensity on cooling performance of the AVH geometry.

2. Experimental facility

The experimental facility at West Virginia University (WVU) shown in Figs. 3 and 4 is an open loop tunnel that has been designed and fabricated to non-dimensionally simulate the aerothermal environment experienced by the first stage of a gas turbine engine as a flat plate representation. The motor at full load is capable of 3625 RPM (380 rad/s) resulting in tunnel freestream velocities of approximately 25 m/s (82 ft/s). The cross-sectional area ratio through the diffuser is 2:1 and is designed with an expansion angle as small as possible in an attempt to prevent boundary layer separation [13]. The mesh heaters are capable of a step temperature increase of approximately 10 °C (18 °F) above ambient at the current freestream flow Reynolds number (*Re*) based on hole diameter (D) of approximately 10,000. These resistance heaters are made from 304 stainless steel wire mesh screens connected in series so as to reduce the current requirements on the welder used to provide current. The mesh is a 200 × 200 wire per square inch arrangement with wire diameter of 0.0016 in. and a spacing of 0.0034 in. between wires resulting in an open area of 46% for each screen. The three current options for turbulence intensities are 1% (no grid), and 7.5%, and 11.7% using passive style grids, measured at the leading edge of the coolant holes. Turbulence intensity was measured using a Dantec Dynamics 54T42 MiniCTA hotwire anemometer survey in the streamwise direction downstream of the grids to insure that heat transfer measurements were performed beyond the region of decaying turbulence. The hotwire has a frequency bandwidth up to 10 kHz. The test section is 15.24 cm (6 in.) tall by 25.4 cm (10 in.) wide and is 91.55 cm