Experimental Thermal and Fluid Science 48 (2013) 184–197

Contents lists available at SciVerse ScienceDirect

Experimental Thermal and Fluid Science

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

# Aerodynamic investigation of impingement cooling in a confined channel with staggered jet array arrangement

Liu Hai-yong [a,b,*], Liu Song-ling [b], Qiang Hong-fu [a], Liu Cun-liang [b]

[a] School of Engine and Energy, Xi'an High-Tech Institute, Xi'an 710025, China
[b] School of Engine and Energy, Northwestern Polytechnical University, Xi'an 710072, China

ARTICLE INFO

*Article history:*
Received 25 March 2012
Received in revised form 13 November 2012
Accepted 27 February 2013
Available online 21 March 2013

*Keywords:*
Impingement cooling
Staggered jet array arrangement
Cross flow
Mass flux
Flow characteristics

ABSTRACT

An enlarged model of a confined channel with staggered circular impingement jet holes and an exit hole was built. The cross flow was induced in the passage by the outflow of the exit hole. Experiments were performed to measure the flow field in the passage. The variations of mass flux distribution, static pressure and discharge coefficient were also studied. Jet Reynolds number of 10,000, 25,000 and 65,000 with *Zr* (ratio of passage height to diameter of impingement hole) of 1, 3 and 5 were considered. Experimental results showed that the jets impinged the target wall effectively along the entire passage of *Zr* = 1. But they were deflected by the cross flow in the downstream region of passage of *Zr* = 3 and 5. The impingement induced strong secondary flows in the passage that caused high velocity flow moving to the side and top walls. The flow structure in the passage changed distinctly with the increases of *Zr*. Mass flux distribution, static pressure and discharge coefficient were mostly dominated by *Zr* under the present test condition.



## 1. Introduction

The cooling techniques are important for modern gas turbine since it is designed to be operated at high turbine inlet temperature. The impingement cooling, which has the most significant potential to increase the local heat-transfer coefficient [1], is developed for the components that need highly localized cooling. In the internal cooling passage of the blade, the continual injection of coolant forms the cross flow, which is constrained to flow along the channel before releases from the film cooling holes or enters other chambers. The impingement cooling and convection cooling are combined in this cooling scheme, which is broadly utilized in the turbine vanes, as shown in Fig. 1.

Many studies of impingement cooling with cross flow have been performed extensively. Goldstein et al. [2] examined the Nusselt number distribution due to a circular jet with cross flow; his results showed that the Nusselt number was significantly asymmetric and that the peak Nusselt number shifted to a downstream location due to the jet deflection by the cross flow. Florschuetz et al. [3–6] carried out a series of experiments to study the aerodynamic parameters and the heat transfer characteristics for a jet array impingement at different cross flow conditions. Some results showed that although the cooling by jet impingement was decreased due to the jet deflection, convection heat transfer seemed to be enhanced by the cross flow. The effect of cross flow on the flow distribution and on the aerodynamic parameters for two-dimensional impingement jet arrays were particularly studied in Ref. [4]. Huang et al. [7] examined the effect of cross flow direction on impingement heat transfer.

Some researches on impingement cooling focused on an impingement/effusion cooling system with or without cross flow were reported recently. Rhee et al. [8] used a naphthalene sublimation method to measure the local heat transfer coefficient on an effusion plate. Their results showed that a high transfer region was formed at the stagnation region and at the mid-line of the adjacent impinging jets due to secondary vortices and flow acceleration to the effusion hole. The experiment conducted by Rhee et al. [9] studied the effect of cross flow on heat transfer characteristics in an impingement/effusion cooling system. The results showed that the cross flow would disturb the jet flows and weaken the interaction between the adjacent wall jets. It also induced locally low transfer regions located at the mid-way region between the effusion holes.

Recent development in casting technology has allowed very intricate internal passage to be manufactured. This has opened up the possibility of casting small diameters with different aspect ratio impingement passage into turbine blades. It is expected that the impingement cooling in such passages would be significantly different from the traditional one applied in vanes. Part of the study results of this cooling structure was provided by Chambers

* Corresponding author at: School of Engine and Energy, Xi'an High-Tech Institute, Xi'an 710025, China. Tel.: +86 29 88486075.
*E-mail address:* helian_xicheng@163.com (H.-y. Liu).

0894-1777/$ - see front matter © 2013 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.expthermflusci.2013.02.023

**Nomenclature**

| | |
|---|---|
| $A$ | cross section area |
| $C$ | mass flux ratio |
| $L_D$ | diameter of impingement holes |
| $Lsp$ | spanwise interval of impingement holes |
| $L_{str}$ | streamwise interval of impingement holes |
| $L_x$ | length of passage |
| $L_y$ | width of passage |
| $L_z$ | height of passage |
| $m$ | mass flow rate |
| $P$ | pressure |
| $Re$ | Reynolds number |
| $V$ | velocity |
| $Zr$ | ratio of passage height to $L_D$ |
| $x, y, z$ | coordinates of passage |

*Greek symbols*

| | |
|---|---|
| $\rho$ | density of air |
| $\mu$ | viscosity |

*Subscript*

| | |
|---|---|
| $avg$ | average value |
| $c$ | cross flow |
| $i$ | sequence number of inlet plenums |
| $j$ | jet |
| $n$ | sequence number of static pressure taps |
| $x, y, z$ | coordinates of passage |

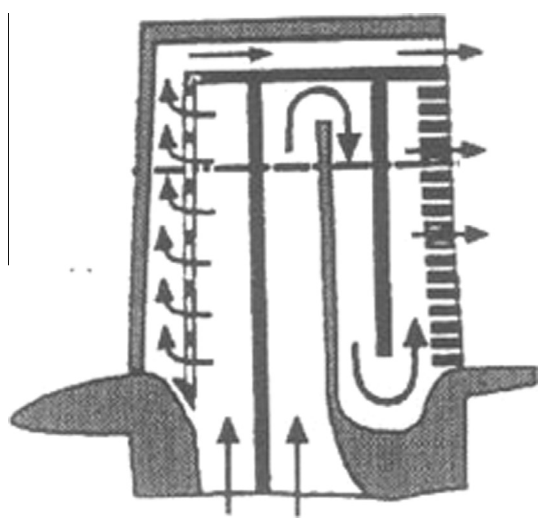

**Fig. 1.** Impingement cooling utilized in the turbine airfoil.

and Gillespie [10]. They found that the cross flow had a great effect on the impingement cooling performance in the confined channel. When the cross flow grew stronger, the jet potential core was no longer able to traverse the channel, and heat transfer enhancement occurred at the location where the mixed out jet wakes stroke the target surface. A similar experiment was also conducted by Chambers et al. [11], in which they studied enhancement of impingement cooling in a high cross flow channel using shaped impingement cooling holes. Their experimental results showed that the average Nusselt number enhancement in the early part of the channel lied between 28% and 77%, and was a weak function of the jet Reynolds number being driven by a drop in the hole discharge coefficient. The average Nusselt number enhancement in the cross flow dominated region was 16% and independent of Re within the limited number of Reynolds numbers tested.

Most studies mentioned above focused on the effect of cross flow on impingement heat transfer. However, the complete analysis of heat transfer characteristics of impingement cooling affected by cross flow requires a detailed flow field knowledge within both the impinging cavity and the jet holes. The flow pattern of the spent air, which could have significant influence on the flow and heat transfer characteristics of downstream chamber, also needs to be determined. Iacovides et al. [12] studied the flow field and the surface heat-transfer for internal rotating cooling flows in gas turbine blades. They declared that the combination of local thermal and flow measurements was essential for the advancement of a deep understanding of the physics of the heat convection process, and also for the provision of suitable data for the validation of CFD codes for heat transfer applications. The flow field and heat transfer characteristics under periodically pulsating impinging air jets were studied by Janetzke et al. [13]. The flow fields were measured by particle image velocimetry (PIV), while surface visualization was performed with the oil film method. The experimental results confirmed the formation of large toroidal vortices due to jet pulsation at the nozzle aperture, and the circular vortex structures were triggered with the pulsation frequency appearing close to the nozzle exit and impinged periodically on the wall. It would renew the velocity and thermal boundary layers periodically. The flow fields within the configuration of a compact jet impingement array with local extraction of spent fluid were measured with magnetic resonance velocimetry (MRV) by Onstad et al. [14]. The experimental data showed that Reynolds number had little influence on the structure of the flow field within their experimental conditions. The interaction between the impingement jet and the cross flow induced counter-rotating vortices and the cross flow displaced the center of the primary stagnation region from the jet centerline. A trapezoidal cross-sectional model simulating a trailing edge cooling cavity was built up by Alessandro et al. [15]. The flow field inside it showed that adjacent jets interacted strongly after their impingement and were deflected to the upper wall, and the insertion of the ribs produced a complex interaction with the crossing-jets. The flow field in a trapezoidal duct with swirl flow induced by impingement jets was measured with a seven-hole pressure probe by Liu et al. [16]. They found a large anticlockwise vortex dominating the upper part of the passage and aroused flow swirling in the passage. The cross flow played a dominating role on the flow characteristics in the passage and side exit holes, and the outflow locations and ratios of film cooling holes had little effect on the main flow structures of the passage within their test conditions.

In order to gain a greater in-sight into the structure of the impingement jet flow within a confined passage, experimental and numerical investigations of aerodynamic aspects were carried out in a large scale test model. A primary objective of this paper was to reveal the flow field of a confined channel, determine the characteristic distribution of static pressure, mass flux ratio and discharge coefficient, and emphasize the effects of jet Reynolds number and passage height.