Chinese Journal of Aeronautics, (2015), 28(6): 1616–1628

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

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

# Numerical investigation on the flow structures in a narrow confined channel with staggered jet array arrangement

Liu Haiyong [a,b,*], Liu Cunliang [b], Wu Wenming [a]

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

Received 1 April 2015; revised 11 June 2015; accepted 3 August 2015
Available online 20 October 2015

**KEYWORDS**

Confined channel;
Cross flow;
Flow structure;
Impingement cooling;
Turbulence model

**Abstract** A series of numerical analyses have been performed to investigate the flow structures in a narrow confined channel with 12 staggered circular impingement holes and one bigger exit hole. The flow enters the channel through the impingement holes and exits through the far end outlet. The flow fields corresponding to two jet Reynolds numbers (25000 and 65000) and three channel configurations with different ratios of the channel height to the impingement hole diameter ($Z_r$ = 1, 3, 5) are analyzed by solving the Reynolds averaged Navier–Stokes equations with the realizable $k$–$\varepsilon$ turbulence model. Detailed flow field information including the secondary flow, the interaction between the jets and the cross flow, and flow distribution along the channel has been obtained. Comparisons between the numerical and experimental results of the flow fields at the four planes along the channel are performed to validate the numerical method. The calculated impingement pattern, high velocity flow distribution, low velocity separation region and vortices are in good agreement with the experimental data, implying the validity and effectiveness of the employed numerical approach for analyzing relevant flow field.



## 1. Introduction

Jet impingement cooling has achievable potential to increase the heat transfer coefficient at the desired location, and is used in many applications such as cooling of electrical and electronic components, cooling of turbine blades and material processing. One of the main applications in gas turbines is represented by the leading edge cooling of turbine blades. A large amount of literature shows that impingement cooling has been employed extensively. Han and Goldstein[1] reviewed the works in the past decades. They discussed the influences

* Corresponding author at: School of Power and Engine, Northwestern Polytechnical University, Xi'an 710072, China. Tel.: +86 29 84359390.
E-mail address: helian_xicheng@163.com (H. Liu).

Peer review under responsibility of Editorial Committee of CJA.

Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.cja.2015.08.017
1000-9361 

of the Reynolds number of the jet as well as the geometry of the impingement cooling system. In particular, they studied the effect of nozzle pitch, the shape of target wall and jet angle on the cooling performance. Zuckerman and Lior[2] reviewed both numerical and experimental research on impingement cooling in the past. They pointed out that the Reynolds averaged Navier–Stokes equations with $k$–$\varepsilon$ model, $k$–$\omega$ model, Reynolds stress model (RSM), and algebraic stress model (ASM) could produce considerable errors in the simulation of the heat transfer of impingement cooling. Wong and Saeid[3] conducted a numerical study of mixed convection on jet impingement cooling in an open cavity filled with porous medium. The results show that the average Nusselt numbers decreased with the increase of dimensionless cavity depth, and the opposing mixed convection was demonstrated to cause deterioration in the average Nusselt number. A numerical investigation of the flow field and heat transfer characteristics of a slot turbulent jet impinging on a semi-circular concave surface with uniform heat flux was carried out by Yang et al.[4] Numerical results were shown to overestimate experimental results by 15% of the maximum Nusselt number. Kim et al.[5] carried out a numerical study on the optimal design of impinging jets in an impingement/effusion cooling system. By varying the interval space of impingement jets and effusion holes, the ratio of the height of impingement jets to the effusion surface and mass flux, the calculated results show that the patterns of thermal stress distributions were similar for all tested cases, and high stresses were observed near the film cooling holes even at low average temperature. Caggese et al.[6] presented experimental and numerical studies on a fully confined impingement round jet. The CFD results have an acceptable prediction of the local and spanwise averaged Nusselt number on both target and jet plates. However, numerical results were shown to overestimate the heat transfer level in terms of Nusselt number by 7–8% in the stagnation region. Singh et al.[7] conducted experimental and numerical studies of turbulent circular air jet impingement cooling on a circular heated cylinder. The results indicate that the effect of geometric parameters was significant only in the jet impinging region. Varol et al.[8] presented a numerical analysis of heat transfer due to slot jet impingement on cylinders with different diameters. The purpose of their study was to examine the effect of effective parameters on the heat transfer and flow fields. The diameter of the cylinders was found to have a major influence on heat transfer and local flow field features. Furthermore, the presence of a secondary cylinder in the recirculation zone was observed to have a negligible effect on the global heat transfer performance. Numerical investigation of the impingement cooling of the rotating and stationary pin–fin heat sinks was carried out by Yang et al.[9] A square heat sink with uniform $5 \times 5$ pin–fins was considered, and the influence of different turbulence models was analyzed.

The internal cooling channels of the integrally cast turbine blades are narrow confined channels formed by the orifice plate and the inner surface of the blade envelope, as shown in Fig. 1. The coolant system feeds the air into the narrow channels through the holes. The air impinges the walls of the channels by means of impingement jets. The air spent through the jets is either sent to the new channels or discharged through the film cooling holes. This configuration has a significant effect on the cooling performance of the impingement jet array. Generally, the cross flow disrupts and deflects the downstream jets and leads to a decrease in the heat transfer intensity, when it enhances the convective cooling in the channel as well. Chyu and Alvin[10] pointed out that this kind of cooling structure significantly reduced the blade metal temperature, with the reduction being highly dependent on the internal convection coefficients. Florschuetz and Su[11] studied the effect of cross flow on the Nusselt number of jet array. They pointed out that when the velocity of the cross flow is less than 10% of that of the jet, it will have little effect on the cross flow structure. In addition, the heat transfer through the channel walls was observed to increase, due to cross flow-jet interaction. However, for cross flow velocities higher than the threshold value, the heat transfer through the channel was reduced. Studies on impingement cooling with narrow channels applied to turbine airfoils were carried out by Chambers et al.[12,13] They found that the cross flow has a major effect on the impingement cooling performance in the narrow confined channel. When the cross flow grew stronger, the jet potential core was no longer able to traverse the channel, and the heat transfer enhancement occurred at the location where the mixed out jet wakes stroke the target surface. Their experiment results also show that the average Nusselt number enhancement in the early part of the channel lied between 28% and 77%. The average Nusselt number enhancement in the cross flow dominated region was 16%. Recent investigation on the impingement cooling in a narrow channel was conducted by Fechter et al.[14] They used a transient liquid crystal technique for the experiment and a commercial CFD package for the numerical simulation. The results show that the overall heat transfer coefficient patterns of CFD predictions agreed well with the measurements. However, the numerical results significantly over-predicted the heat transfer level at the stagnation point. Yang et al.[15] presented an experimental and numerical investigation of unsteady impingement cooling within a blade leading edge channel. The results show that the intensity, unsteadiness and skewness of the vortex structure increased as the channel cross-flows became more non-uniform. Chi et al.[16] presented a novel multi-row impingement cooling configuration anti-crossflows (ACF) cooling structure to reduce the negative effect of cross flows. The results show that the novel ACF impingement cooling structure can achieve a more uniform heat transfer on the target surface, compared with the prototype with the same Reynolds number and jet array arrangement. The novel impingement cooling configuration showed potential benefits if applied to cooled vanes. Alzwayi and Paul[17] numerically studied the convective flow inside an inclined parallel walled channel. They selected the $k$–$\varepsilon$ model for the turbulence closure. The distributions of velocity, turbulent kinetic energy and local heat flux were presented. The results indicate that the average of heat transfer coefficient was dependent on both the width and angle of the channel.

Most studies mentioned above focus on the effect of cross flow on impingement heat transfer. However, a complete analysis of heat transfer characteristics of impingement cooling affected by the cross flow requires detailed knowledge of flow field within both the impinging cavity and the jet holes. In addition, the entire flow field of the discharged air plays an important role in the heat transfer of the downstream channels and needs to be investigated. Iacovides et al.[18] studied the flow field and the surface heat transfer for internal rotating cooling flows in gas turbine blades. They pointed out the importance of simultaneous flow and thermal measurements to pursue a