International Journal of Heat and Mass Transfer 103 (2016) 572–581

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Effects of approaching main flow boundary layer on flow and cooling performance of an inclined jet in cross flow

Lingxu Zhong, Chao Zhou *,1, Shiyi Chen

College of Engineering, Peking University, Beijing, China

ARTICLE INFO

Article history:
Received 3 April 2016
Received in revised form 30 June 2016
Accepted 24 July 2016

ABSTRACT

In high pressure turbines, film cooling methods are used to reduce the metal temperatures of the blade. Understanding the mixing of the main flow and the coolant is key for the development of better cooling designs. It was found that surface film cooling effectiveness distribution were different with a laminar or a turbulent approaching main flow boundary layer, but the physical mechanism was not fully understood. In this paper, large eddy simulation (LES) method is used to investigate the effects of the approaching main flow boundary layer on the flow physics and cooling performance of inclined jets in cross flows. The different mixing processes of the coolant with different boundary layers status of approaching main flow are investigated with the instantaneous flow field provided by LES method. With a laminar approaching boundary layer, the horseshoe vortex develops upstream of the cooling hole. The coolant mixes with this horseshoe vortex and cools the area underneath the horseshoe vortex near the cooling hole. With a turbulent approaching boundary layer, the horseshoe vortex is not evident, and the distribution of film cooling effectiveness changes near the cooling hole. Although the distributions of time-averaged film cooling effectiveness become similar for both approaching boundary layers on the area 5d downstream of the cooling hole, the current study found that the coolant mixing process are different.

1. Introduction

Film cooling methods are widely used in hot sections of gas turbines to reduce the metal temperatures of the blade surfaces. In many cases, the cooling air injects into the main flow via coolant holes at inclined angles. After the cooling air exits the cooling hole, it mixes with the main flow near the surface of the blade. A thermal protection layer forms on the surface, thus reducing the metal temperature of the blade [1]. There are many studies about film cooling in the past several decades. Reviews of the film cooling research can be found in [2–4].

Many previous studies on the film cooling focus on the time-averaged flow field and time-averaged thermal performance by both experimental (Goldstein et al. [1], Pietrzyk et al. [5] and Sinha et al. [6], Yuen and Martinez-Botas [7]) and numerical methods (Leylek and Zerkle [8]). One of the most frequently investigated film-cooling configurations consists of a plenum and a film cooling hole with a streamwise inclination angle of 30–35 degrees. The results showed that after the coolant exits from the cooling hole, the CRVP appeared as the coolant mixed with the main flow. It is found that the CRVP is a key flow structure that determines the mixing of flow, and thus temperature distribution of the flow and the film cooling effectiveness (Liu and Malak et al. [9], Hunley et al. [10]).

To improve the thermal performance, different cooling hole geometries are used. Based on the time-averaged results, it is found that the CRVP generated by certain cooling hole geometries, e.g. the fan-shaped hole, is less tend to lift from the surface than the cylindrical hole, so the coolant can achieve better protection of the surface(Thole et al. [11] and Gritsch et al. [12]). The cooling performance is often explained with the time-averaged flow field, e.g. the location of CRVP. However, the instantaneous flow field that forms the CRVP is seldom discussed.

In the leading edge region and for some distance downstream, the boundary layer flow along the blade may be laminar (Goldstein and Yoshida [28]). Because boundary-layer characteristics range from laminar to turbulent, and from relatively thin to thick along an airfoil, film-cooling injection is subjected to a range of approach flow conditions (Bogard and Thole [4]). As the mixing of the coolant and the main flow occurs near the blade surface, the approaching boundary layer of the main flow should have an effect on the mixing of the cooling air near the surface. Liess [13] indicates that for an approaching boundary layer with $\delta^*/d < 0.2$, the laterally averaged film cooling effectiveness remained almost unchanged

---

* Corresponding author at: State Key Laboratory for Turbulence and Complex Systems, College of Engineering, 100871, China.

E-mail address: czhou@pku.edu.cn (C. Zhou).

[1] Collaborative Innovation, Center of Advanced Aero-Engine, 100191 Beijing, China.

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.07.089
0017-9310/

Nomenclature

BR	blowing ratio = $(\rho_j U_j)/(\rho_\infty U_\infty)$
d	film hole diameter
DR	density ratio = ρ_j/ρ_∞
H	shape factor = δ^*/θ
I	momentum ratio = $(\rho_j U_j^2)/(\rho_\infty U_\infty^2)$
L	length of film hole
Ma	mach number
P	pitch of film holes
$\mathrm{Re_d}$	Reynolds number based on d, ρ_∞ and U_∞
Re_θ	Reynolds number based on θ, ρ_∞ and U_∞
T	temperature
TKE	non-dimensional turbulent kinetic energy = $(u'u' + v'v' + w'w')/U_\infty^2$
TR	temperature ratio = T_j/T_∞
$\lvert U \rvert$	velocity magnitude
U,V,W	velocity in x, y, z direction
$u_i'u_j'$	Reynolds-stress tensor
VR	velocity ratio = U_j/U_∞
x, y, z	Cartesian coordinate system
Δx_i^+	wall unit, $\Delta x_i^+ = x_i u_\tau/\nu$
δ	boundary layer thickness
δ^*	boundary layer displacement thickness
θ	boundary layer momentum thickness
ρ	density
η	local film cooling effectiveness $\eta = (T - T_\infty)/(T_j - T_\infty)$
$\bar{\eta}$	laterally averaged film cooling effectiveness $\bar{\eta}(x) = \frac{1}{P}\int_{-0.5P}^{0.5P} \eta(x,y)dy$

Subscripts

j	coolant jet quantity
∞	main flow quantity

Abbreviation

CFD	computational fluid dynamics
CRVP	counter-rotating vortex pair
LES	large eddy simulation
RANS	Reynolds-averaged Navier-Stokes equations
SGS	sub-grid scale
WALE	wall-adapting local eddy-viscosity

for cylindrical holes. Drost and Bölcs [14] also found that the displace thickness of the approaching boundary layer has a small effect on the distribution of the laterally averaged film cooling effectiveness. Nevertheless, they observed that a laminar and a turbulent approaching boundary layers resulted in different film cooling effectiveness on the blade surface, especially for the area near the cylindrical cooling hole exit. The cooling performance is a result of coolant mixing. But most previous studies explained the cooling performance based on the time-averaged flow field. The unsteady flow physics due to different approaching boundary layers and its effect on the cooling performance are still not well understood.

The mixing of the jet in cross flow is inherently unsteady (Fawcett et al. [15]). The accuracy of the film cooling prediction based on RANS method often depends on the choice of turbulence models and the empirical parameters. (Acharya et al. [16] and Silieti et al. [17]). Recently, the unsteady flow features were studied using LES method. The high resolution of the spatial and temporal flow field provided by LES made it possible for a more detailed analysis of the time-averaged and instantaneous flow fields. Tyagi and Acharya [18] is among the first that studied an inclined jet with a cylindrical hole with LES method. They found that the hairpin vortices formed downstream of the hole close to the wall. The counter-rotating vortex pair was associated with legs of hairpin vortices. The unsteady flow field of the cylindrical hole was also studied by Renze et al. [19] and Peet & Lele [20] with LES. Sakai et al. [21] is among the first who studied not only the flow field, but also the temperature field of film cooling using LES. They showed that the unsteady vortices could affect the temperature distribution.

The thermal performance of film cooling is determined by the mixing process of the coolant and the main flow. The difference in the status of the approaching main flow boundary layer will result in different coolant mixing processes, which are inherently unsteady. Previous studies observed that the distributions of film cooling effectiveness are different with a laminar and a turbulent approaching main flow boundary layer. It was not understood how different the unsteady mixing process of coolant due to different approaching boundary layer status.

In this paper, LES method is used to investigate an inclined jet in cross flow with a cylindrical hole. The study aims to understand the flow physics of the coolant mixing with a laminar and a turbulent approaching main flow boundary layer. The cooling performance will be discussed based on both the time-averaged and the instantaneous flow fields. The understanding of the physical mechanism can help to improve the future film cooling configurations.

2. Numerical methods

2.1. Solver

An in-house CFD code developed by Peking University is further developed for the current LES calculations. This code is a compressible solver, and can be used to parallelly calculate multi-block structured meshes. The basic governing equations for LES method are the three-dimensional Favre-filtered non-dimensional Navier-Stokes equations, which are normalized by the diameter of film hole d and the mainstream variables, such as the density ρ_∞, temperature T_∞ and velocity U_∞ [22]:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial(\bar{\rho}\tilde{u}_i)}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial(\bar{\rho}\tilde{u}_i)}{\partial t} + \frac{\partial(\bar{\rho}\tilde{u}_i\tilde{u}_j + \bar{p}\delta_{ij})}{\partial x_i} = \frac{1}{\mathrm{Re}}\frac{\partial \tilde{\sigma}_{ij}}{\partial x_j} + \frac{\partial \tau_{ij}^{\mathrm{LES}}}{\partial x_j} \tag{2}$$

$$\frac{\partial(\bar{\rho}\tilde{e})}{\partial t} + \frac{\partial[\tilde{u}_i(\bar{\rho}\tilde{e} + \bar{p})]}{\partial x_i} = \frac{\partial \tilde{q}_i}{\partial x_i} + \frac{1}{\mathrm{Re}}\frac{\partial(\tilde{\sigma}_{ij}\tilde{u}_j)}{\partial x_i} + \frac{\partial q_i^{\mathrm{LES}}}{\partial x_i} + \frac{\partial J_i^{\mathrm{LES}}}{\partial x_i} \tag{3}$$

The governing equations are numerically solved by a finite-volume method. The Cartesian variables are calculated and stored at the cell center. The convective terms are discretized by the ECUSP scheme (Zha and Bilgen [23]). The viscous terms are discretized by the second-order center difference scheme. More details can be found in [22].

With LES method, large-scale turbulence is resolved and small-scale turbulence is modelled by sub-grid scale (SGS) model. The SGS model used in this paper is WALE (wall-adapting local eddy-viscosity) model (Nicoud and Ducros [24]), which is based on tensor invariant and has proper scaling near the wall.