International Journal of Heat and Mass Transfer 69 (2014) 300–311

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Large-eddy simulation of an inclined round jet issuing into a crossflow

E. Sakai *, T. Takahashi, H. Watanabe

Central Research Institute of Electric Power Industry, Energy Research Laboratory, 2-6-1, Nagasaka, Yokosuka-shi, Kanagawa-ken 240-0196, Japan

ARTICLE INFO

Article history:
Received 11 July 2012
Received in revised form 5 February 2013
Accepted 13 October 2013
Available online 8 November 2013

Keywords:
Crossflow jet
Vortex
Large-eddy simulation
Film cooling

ABSTRACT

For understanding of film cooling flow fields on gas turbine blades, this paper reports on a series of large-eddy simulations of an inclined round jet issuing into a crossflow. Simulations were performed at four blowing ratios, BR = 0.1, 0.5, 0.7 and 1.0, and the Reynolds number, Re = 15,300, based on the crossflow velocity and film cooling hole diameter. Results showed that the cooling jet flow structure drastically changed with the blowing ratio. A pair of rear vortices and hairpin vortices were observed for BR = 0.1. A horseshoe vortex periodically ejected, a pair of hanging vortices, a pair of rear vortices and hairpin vortices were observed for BR = 0.5. Similar vortical structures to BR = 0.5 were observed for BR = 0.7 although horseshoe vortex was not ejected periodically and stayed at a leading edge of the hole exit. For BR = 1.0, in addition to the former mentioned vortices, shear layer vortices and vertical streaks were observed on an upstream edge of the jet. It was consequently understood that the ubiquitous counter-rotating vortex pair which can be observed in the time-averaged flow field was actually originated in the different vortical structures with varying BR conditions. Temperature fields were also investigated to clarify how these different vortical structures affect the film cooling effectiveness.

1. Introduction

A jet in crossflow is a highly complex turbulent flow, with applications in a wide variety of technological problems, including plume dispersion, control of missiles, fuel injection for burners, and film cooling for turbines and combustors [1–4]. In the film cooling used in gas turbines, coolant is ejected into the hot crossflow as crossflow jets. The coolant forms a thin layer over the turbine blades, protecting the surface from direct exposure to the hot crossflow. It has been recognized that when the jet is discharged to the crossflow, there is a complex interaction between the jet and the crossflow, leading to the formation of complicated vortical structures that have strong influence on the film cooling effectiveness.

From the flow visualization of the round jet issued vertically into the crossflow, Fric et al. [1] categorized the vortical structures into four groups: a horseshoe vortex, jet shear layer vortices, wakes, and counter rotating vortex pair (CRVP). Among the four vortices, the most important vortex is arguably the CRVP, which is often observed in time averaged flow fields in the far region aligned with the trajectory of the jet. The CRVP is widely known to significantly degrade the film cooling effectiveness by promoting both the jet lift-off and entrainment of the hot crossflow towards the wall. Thus, for developing highly efficient film cooling techniques for gas turbine blades, it is important to clarify how the CRVP is formed in flow fields.

Although a lot of experimental and numerical studies have been placed to understand the formation process of the CRVP, its origins are still subject of much debate. Yuan et al. [3] reported a series of large-eddy simulations of a round jet issuing normally into a crossflow at comparatively high blowing ratios (BR $\sim$ 3.3) and low Reynolds numbers ($Re \sim 2010$), and they concluded that the CRVP originates from a pair of hanging vortices in the skewed mixing layer that develops between the jet and the crossflow. Whereas, Tyagi et al. [5] performed large-eddy simulations of an inclined round jet issuing into a crossflow at comparatively low blowing ratios (BR = 0.5) and high Reynolds numbers (Re = 15,000). They clearly showed a formation of hairpin vortices in the downstream interface of the crossflow and the jet, and concluded that the hairpin vortices are the origin of the CRVP. Recently, Fawcett et al. [6] experimentally visualized the ejected jet structures and showed that the jet structure changes with the blowing ratio. According to these studies, the formation of vortices, as well as the origins of the CRVP could somehow be changed with the blowing ratio.

This paper reports on the series of large-eddy simulations of an inclined round jet issuing into a crossflow. Simulations were performed at four blowing ratios. The main focus of this paper is to examine the development of large scale unsteady vortical structures in the flow fields and to understand how these structures affect the formation of the CRVP at different blowing ratios. In addition, the effects of these vortical structures on the film cooling effectiveness are investigated.

* Corresponding author. Tel.: +81 46 856 2121; fax: +81 46 856 3346.
E-mail addresses: e-sakai@criepi.denken.or.jp (E. Sakai), tosihiko@criepi.denken.or.jp (T. Takahashi), whiroaki@criepi.denken.or.jp (H. Watanabe).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.10.027

Nomenclature

a	thermal diffusivity
BR	blowing ratio, =$\rho_j U_j/(\rho_c U_c)$
DR	density ratio, =ρ_j/ρ_c
d	diameter of film cooling hole
h	enthalpy
L	length of film cooling hole
p	pressure
Pr_t	turbulent Prandtl number
Re	Reynolds number, =Ud/ν
S	rate-of-strain tensor
t	time
T	temperature
u	x-velocity
U	cross sectional average velocity
v	y-velocity
w	z-velocity
x	streamwise length
y	lateral length
z	vertical length
δ^*	boundary layer displacement thickness
η	film cooling effectiveness, =$(T_c - T_w)/(T_c - T_j)$
θ	non dimensional temperature, =$(T_c - T_f)/(T_c - T_j)$, or boundary layer momentum thickness
μ	viscosity
ν	kinematic viscosity
ρ	density
τ	stress tensor
ω	vorticity

Subscripts

c	crossflow
f	mixing air of main and coolant flows
j	jet
lat	laterally averaged
rms	rms, $\phi_{rms} = \sum \sqrt{(\bar{\phi} - \phi')^2}$
w	wall
x	x-direction
y	y-direction
$'$	fluctuation

2. Computational methods

2.1. Governing equations

A computational code, NuFD/FrontFlowRed-extended by CRIEPI, was used in this study. Governing equations are the grid-filtered, conservative equation, compressible Navier–Stokes equations, energy equation and equation of state for perfect gas:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j) = 0 \quad (1)$$

$$\frac{\partial(\bar{\rho}\tilde{u}_j)}{\partial t} + \frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_i\tilde{u}_j + \bar{p}\delta_{ij} - \tilde{\tau}_{ij} - \tau_{u_iu_j}\right) = 0 \quad (2)$$

$$\frac{\partial(\bar{\rho}\tilde{h})}{\partial t} + \frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\tilde{h} - \bar{\rho}a\frac{\partial \tilde{h}}{\partial x_j} - \tau_{u_jh}\right) = 0 \quad (3)$$

$$\bar{\rho} = \frac{p_0}{R\tilde{T}} \quad (4)$$

where, a line over a variable, $\bar{\phi}$, indicates a grid-filtered quantity, a tilde over a variable, $\tilde{\phi}$, indicates a Favre averaged quantity.

τ_{ij} in Eq. (2) is the shear stress tensor, given for a Newtonian fluid by:

$$\tilde{\tau}_{ij} = 2\mu\tilde{S}_{ij} - \frac{2}{3}\mu\delta_{ij}\tilde{S}_{kk}. \quad (5)$$

$\tau_{u_iu_j}$ and τ_{u_jh} in Eqs. (2) and (3) repsent the effects of subgrid-scale, and are modeled in the simulations using the dynamic subgrid-scale model by Germano [7] as follows:

$$\tau_{u_iu_j} = -\bar{\rho}(\widetilde{u_iu_j} - \tilde{u}_i\tilde{u}_j) \quad (6)$$

$$\tau_{u_iu_j} - \frac{1}{3}\delta_{ij}\tau_{u_ku_k} = 2C\bar{\rho}\Delta^2|\tilde{S}|\tilde{S}_{ij} \quad (7)$$

$$\tau_{u_jh} = -\bar{\rho}\left(\widetilde{u_jh} - \tilde{u}_j\tilde{h}\right) = \bar{\rho}\frac{C\Delta^2}{Pr_t}|\tilde{S}|\left(\frac{\partial \tilde{h}}{\partial x_j}\right) \quad (8)$$

$$C = C_s^2 \quad (9)$$

where, Δ is a band of the filter. The Smagolinsky constant, C_s is determined by the method proposed by Lilly [8]. For the stability of the simulations, C is clipped at 0. Turbulent Prandtl number, Pr_t, is set to be 0.4 [9].

S_{ij} is the rate-of-strain tensor and can be written as:

$$\tilde{S}_{ij} = \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right). \quad (10)$$

Computation was performed using the low-Mach-number approximation with temperature dependent density. The convective terms of the Navier–Stokes and the energy equations were discretized by the 2nd order central-differencing scheme. For the stability of the simulations, 1st order upwind scheme was blended 2% for the Navier–Stokes equations, and 10% for the energy equation. The solution is advanced in time using a 1st order Euler implicit scheme. Based on the previous studies [10,11] using the same code, it could be said that the discretization scheme used in this study is adequate for the prediction of vortices. The discretized equations were solved by SIMPLE algorism. Statistics were calculated by averaging the results between the time steps of 80,001 and 150,000. Non dimensional time step, $\delta t^* = \delta t U_{c/d}$, is 1.6×10^{-3}.

2.2. Computational domains and boundary conditions

Fig. 1 shows computational domains. Fig. 1 (a) and (b) represent the domains for main simulations and for validations, respectively. Closeup view of film cooling hole exit is shown in Fig. 1 (c). The computational domain for the main simulations, Domain A, consists of a crossflow channel and a film cooling hole. The hole diameter, d, is 12.5 mm and the hole is inclined at 35° with respect to the crossflow. The hole length, L is 5.23d. The crossflow channel is 25d long, 3d wide, and 5d high. The x, y, z axes are taken to be streamwise, lateral, and vertical to the bottom wall of the crossflow channel. The origin of the axes is the trailing edge of the hole exit. The numbers of grids are about 7.6×10^6 in the crossflow channel, and 2.0×10^6 in the cooling hole, respectively.

Because the simulations run for this study required significant amounts of computer time (about 80,000,000 CPU seconds for a run), grid resolution studies were impractical. Instead, we compared the computational results to the previous precise