Applied Thermal Engineering xxx (2014) 1–12

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Numerical simulation on impingement and film composite cooling of blade leading edge model for gas turbine

Zhao Liu, Lv Ye, Changyee Wang, Zhenping Feng*

Institute of Turbomachinery, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, PR China

HIGHLIGHTS

- The performances of different turbulence model are validated.
- Effects of blowing ratio and spanwise angle on composite cooling are investigated.
- Spanwise angle is more effective to improve film cooling than impingement cooling.
- Proper blowing ratio and lower spanwise angle is suitable for cooling design.

ARTICLE INFO

Article history:
Received 15 January 2014
Received in revised form
28 April 2014
Accepted 7 May 2014
Available online xxx

Keywords:
Blade leading edge
Impingement cooling
Film cooling
Composite cooling

ABSTRACT

In this paper numerical simulation is performed to simulate the impingement and film composite cooling on blade leading edge region. The relative performances of turbulence models are compared with available experimental data, and the results show that SST $k-\omega$ model is the best one based on simulation accuracy. Then the SST $k-\omega$ model is adopted for the simulation. The grid independence study is also carried out by using the Richardson extrapolation method. A single array of circle jets and three rows of film holes are arranged to investigate the impingement and film composite cooling. Five different blowing ratios and five different film hole spanwise angles are studied in detail. The results indicate that the heat transfer coefficient on the internal surface of turbine blade leading edge increases with the blowing ratio, and slightly changes with film hole spanwise angle. And the external film cooling effectiveness distribution would vary rapidly with the blowing ratio and the film hole spanwise angle.

1. Introduction

In order to enhance the turbine efficiency and power, modern gas turbine systems are required to operate at higher and higher temperature, and thus turbine components are suffering an increasing heat loads. As a result, turbine inlet temperature has already been far beyond the material acceptable level. It is worth to mention that the leading edge area of gas turbine blade will have a higher heat flows as it is upwind to high temperature inflows. A series of cooling technologies such as impingement cooling, film cooling and other forced convection heat transfer methods have been applied to the turbine blade leading edge, among which impingement cooling is one of the most effective internal cooling methods but its arrangement would weaken the blade structure strength [1], and also film cooling is the only method for external cooling. Thus impingement and film composite cooling are used extensively in gas turbine blades in recent years.

Two decades ago, experiment was the only way to obtain detailed and reliable heat transfer information about the turbulent, 3-dimensional flows in the complex impinging cooling flows. The early investigation on concave surface impingement cooling has been summarized by Chupp et al. [2], Metzger et al. [3,4], and then Bunker and Metzger [5] continued their works in 1990 to suit the need of turbine cooling design. The cooling performance are influenced by many parameters, such as, Reynolds number [2,3,5,6], jet spacing [2,4,6], the distance between jet exit and target surface [2,5,6], target surface shape [4,5], jet nozzle shape [7], rotation [8], jet direction [9] and so on.

The first numerical investigation was conducted by Kayansayan and Küçüka [10], and then many works compared the numerical results with experimental data [10–19]. Further studies on the influence factors were continued, such as wall roughness [11], horseshoe ribs [12], cross flow and exit flow schemes [13], curvature target surface [18], and the arrangement of jet nozzle [19]. And some of them deduced dimensionless correlations between the

* Corresponding author. Tel.: +86 29 82663195; fax: +86 29 82668704.
E-mail address: zpfeng@mail.xjtu.edu.cn (Z. Feng).

http://dx.doi.org/10.1016/j.applthermaleng.2014.05.060
1359-4311/

Nomenclature

B	width of equivalent slot jet used in Ref. [5] [mm]
C	hole spacing of each row [mm]
D	diameter [mm]
H	jet from target surface [mm]
h	heat transfer coefficient [W/(m^2 K)]
I	turbulent intensity
L	length of flat portion [mm]
M	average blowing ratio
Nu	Nusselt number
P	pressure [Pa]
q	heat flux [W/m^2]
r^*	ratio of curvature radius to of semi-cylindrical airfoil radius used in Ref. [5]
Re	Reynolds number
S	streamwise coordinate [mm]
V	velocity [m/s]
T	temperature [K]
Y	transverse coordinates from wall (m)
y^+	non-dimensional distance, $=Yu_\tau/\nu$
Greek symbols	
η	cooling effectiveness
ρ	density [kg/m^3]
α	angle of the second row of film holes from the stagnation line [°]
β	angle of film holes from the surface in spanwise [°]
γ	attack angle of main flow [°]
λ	thermal conductivity of fluid [W/(m K)]
u_τ	shear velocity [m/s]
ν	kinematic viscosity [m^2/s]
subscripts	
av	average
aw	adiabatic wall
c	coolant
f	film hole
i	inlet
ip	impingement hole
sp	spanwise average
w	wall
o	outlet
x, y, z	the direction of the vector
∞	mainstream
1,2	film hole number

average or the maximum Nusselt number and the influence parameters [15,19].

Concern on film cooling, most work focus on the effect of film hole exit shape and geometry [20–22], effect of film cooling combined with inclined ribs [23] and so forth.

All papers cited above are about the investigation on new shape film holes, or the effect of jet parameters on impingement cooling without film coolant extraction. The addition of film cooling is expected to affect the heat transfer in the internal region of the impingement cooling since it changes the flow path of the coolant and reduces the cross flow. Only few of pervious work focused on impingement/film composite cooling. The first work was done by Hollworth and Dagan in 1980 [24], and also Ekkad et al. [25] investigated impingement cooling with film coolant extraction on a plate. Then Metzger and Bunker [26] experimentally studied impingement cooling of concave surface through lines of circular air jets with and without film coolant extraction. Taslim et al. [27,28], and Taslim and Khanicheh [29] investigated heat transfer of impingement cooling on concave surface with film holes under the influence of roughness target wall, jet nozzle geometry, shower-head and gill film holes. Mouzon et al. [30], Ravelli et al. [31], Maikell et al. [32] and Mathew et al. [33] investigated the effect of internal impingement on film cooling of blade leading edge model.

Obviously, the previous work focused on the influence of film extraction on impingement cooling or the effect of flow condition on external film cooling. But almost no attentions were paid to the effects of film hole geometries on the internal heat transfer and the external film cooling effectiveness. In this study, the flow structure and heat transfer of the impingement and film composite cooling on turbine leading edge model are studied, and the effects of blowing ratio and spanwise angle of film holes are investigated.

2. Physical and mathematical model

2.1. Geometrical details

This numerical study adopted the blade leading edge model which was used in the experimental study of Maikell et al. [32] and the computational study of Mathew et al. [33]. Fig. 1 shows the leading edge model geometry structure. The film cooling configuration included three rows of holes positioned staggered along the stagnation line and at ±25° from the stagnation line, the impingement jets were directed to impact the internal surface in each stagnation film hole. The geometrical dimensions of different jet configurations are listed in Table 1.

2.2. Numerical method

2.2.1. Boundary conditions and solution procedure

The boundary conditions are matched with those in Ref. [32]. As shown in Fig.1(b), periodic and symmetry conditions were applied to minimize the computational effort. The mainstream inlet air total temperature is 300 K; the velocity is 15 m/s and the inlet turbulent intensity I is 5%, and its definition can be found in Equation (1). The coolant inlet total temperature is 200 K, and the inlet turbulent intensity is 5% as well. The average static pressure at the outlet is 0.106 MPa. The impingement target wall temperature is 235 K, other walls are adiabatic and nonslip. The fluid is nitrogen (ideal gas). The convergence of simulation is achieved when the root mean square residuals of the momentum equations, mass equation, energy equation and turbulent equations are lower than 10^{-5} and remain steady.

$$I = \frac{\sqrt{V_x^2 + V_y^2 + V_z^2}}{V_{av}} \tag{1}$$

2.2.2. Mesh procedure

The numerical simulations are performed by using a commercial CFD software CFX11.0. The solutions are obtained by solving the steady compressible Reynolds averaged Navier–Stokes equations, in which the finite control volume method is applied to discretize these equations, and a second order form with high

Please cite this article in press as: Z. Liu, et al., Numerical simulation on impingement and film composite cooling of blade leading edge model for gas turbine, Applied Thermal Engineering (2014), http://dx.doi.org/10.1016/j.applthermaleng.2014.05.060