Journal of Environmental Management xxx (2017) 1–8

Contents lists available at ScienceDirect

Journal of Environmental Management

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

Research article

Effect of an upstream bulge configuration on film cooling with and without mist injection

Jin Wang [a], Qianqian Li [a], Bengt Sundén [b], Ting Ma [c, *], Pei Cui [a]

[a] School of Energy and Environmental Engineering, Hebei University of Technology, Tianjin 300401, China
[b] Department of Energy Sciences, Division of Heat Transfer, Lund University, Lund SE-22100, Sweden
[c] Key Laboratory of Thermo-Fluid Science and Engineering, MOE, Xi'an Jiaotong University, Xi'an, Shaanxi, 710049, China

ARTICLE INFO

Article history:
Received 23 December 2016
Received in revised form
16 June 2017
Accepted 23 June 2017
Available online xxx

Presented at the 2nd South East European Conference on Sustainable Development of Energy, Water and Environment Systems, June 15–18, 2016, Piran, Slovenia.

Keywords:
Film cooling
Bulge
Temperature non-uniformity
Mist
Gas turbine

ABSTRACT

To meet the economic requirements of power output, the increased inlet temperature of modern gas turbines is above the melting point of the material. Therefore, high-efficient cooling technology is needed to protect the blades from the hot mainstream. In this study, film cooling was investigated in a simplified channel. A bulge located upstream of the film hole was numerically investigated by analysis of the film cooling effectiveness distribution downstream of the wall. The flow distribution in the plate channel is first presented. Comparing with a case without bulge, different cases with bulge heights of 0.1d, 0.3d and 0.5d were examined with blowing ratios of 0.5 and 1.0. Cases with 1% mist injection were also included in order to obtain better cooling performance. Results show that the bulge configuration located upstream the film hole makes the cooling film more uniform, and enhanceslateral cooling effectiveness. Unlike other cases, the configuration with a 0.3d-height bulge shows a good balance in improving the downstream and lateral cooling effectiveness. Compared with the case without mist at M = 0.5, the 0.3d-height bulge with 1% mist injection increases lateral average effectiveness by 559% at x/d = 55. In addition, a reduction of the thermal stress concentration can be obtained by increasing the height of the bulge configuration.

1. Introduction

To meet the economic requirements of power output, the inlet temperature of modern gas turbines, which is always above the melting point of the material, is much higher than before. Therefore, it is necessary to develop high-efficient cooling technology in order to protect the blades from high-temperature hot flue gas. However, experimental studies on cooling technology of blades are very expensive and time-consuming. Over the past 20 years, numerical simulation has been widely used in various studies due to its low cost and high efficiency, particularly in the complex flow, heat and mass transfer process, including temperature and species concentrations analysis inside a cement calciner (Mikulčić et al., 2016), multicomponent evaporation analysis (Baleta et al., 2017), pollutant prediction in internal combustion engines (Petranović et al., 2017), prediction of the influence of vegetation on atmospheric processes (Wania et al., 2012), two-stage NOx removal strategy analysis (Javed et al., 2007), pressure drop analysis in two down-flow wood bark-based biofilters (Kafle et al., 2014), sedimentation analysis with parallel retrofit baffles (He et al., 2015) and the two-phase flow analysis of a rising branch (Diogo and Oliveira, 2016).

Moreover, numerical simulation has also been used to predict the effects of various cooling techniques on the cooling behaviour of gas turbine blades. A literature review was conducted using research from 2001 to 2008 focusing on cooling techniques with blade tip leakage flow and heat transfer (Sunden and Xie, 2010). The film-cooling effectiveness on different curved surfaces was investigated numerically, and the results showed that the film cooling effectiveness on a given curved surface depends on the optimum selection of the blowing ratio and the injection angle (Koç et al., 2006). The local heat/mass transfer characteristics and friction losses were investigated using 60° V-shaped ribs and 45° V-shaped ribs (Lee et al., 2009). An optimal design of transverse ribs in tubes was detected, and it was found that Nu ratios and friction losses decrease with increasing rib width. The effect of the aspect ratio was significant with 60° V-shaped ribs (Kim et al., 2010). A

---

* Corresponding author.
E-mail address: mating715@mail.xjtu.edu.cn (T. Ma).

http://dx.doi.org/10.1016/j.jenvman.2017.06.055
0301-4797/© 2017 Elsevier Ltd. All rights reserved.

Nomenclature

c	Deposition location (distance), m
d	Film hole throat diameter, m
k	Turbulence kinetic energy, m^2/s^2
M	Blowing ratio, $= \rho_j V_j / \rho_\infty V_\infty$
P	Pressure, N/m^2
F	Force, N
T	Temperature, K
u	Streamwise velocity component, m/s
V	Velocity magnitude, m/s
w	Width of deposition, m
x, y, z	Coordinates, m
Greek symbols	
α	Inclination angle, deg
ε	Turbulence dissipation rate, m^2/s^3
η	Adiabatic film cooling effectiveness, $=(T_{aw}\text{-}T_i)/(T_j\text{-}T_i)$
λ	Thermal conductivity, W/(m·K)
ρ	Density, kg/m^3
τ	Stress tensor, N/m^2
Subscripts	
i, j	Indices

computational analysis of film cooling was made using different rows of novel semi-circular holes. Results indicated that the mean height of the coolant jet for semi-circular holes is smaller than for circular holes (Asghar and Hyder, 2011). A three-dimensional analysis of temperature distributions was made at different streamwise (downstream of the wall) and horizontal (along the lateral direction) positions, and film cooling effectiveness was assessed at a wide range of inclination angles (Bayraktar and Yılmaz, 2011). For a multi-slot system, the effect of a flame holder on the film cooling effectiveness was investigated through the use of CFD software. The ejected secondary flow was suppressed by the accelerated flow, resulting in better cooling performance with the use of the flame holder. The difference in cooling performance was shown to be greater with an increase in the blowing ratio (Song et al., 2012). One study found that an improvement in film cooling performance cannot be provided by increasing the tangential angle of the coolant jet from 30° to 45° (Shine et al., 2013). The film cooling effectiveness on the wall was investigated with various deposition heights and widths. Results showed that the film cooling effectiveness deteriorates with increasing deposition height and width, and that the film cooling effectiveness is improved by decreasing the blowing ratio (Wang et al., 2016a). Effects of slot injection configuration and endwall film cooling were investigated by using the shear stress transport (SST) k-ω turbulence model. The comparison of numerical and experimental results showed that the SST k-ω turbulence model was suitable for this film cooling prediction. The results indicated that the coolant coverage area extends with decreasing the slot width (Du and Li, 2016). Using the same SST k-ω turbulence model, it was also found that a larger coolant coverage area is achieved with the use of a leading edge injection slot than without (Du et al., 2016).

Some research has investigated the effects of the upstream configurations and mist injection on the characteristics of film cooling. The effect of a step configuration located upstream of the film-cooling holes was studied using a film-cooling plate. The results indicated that the presence of the upstream step increases the film cooling effectiveness in the region near the hole (Rallabandi et al., 2011). The improvements of novel upstream steps in film cooling performance were predicted using computational simulations (ANSYS CFX). The results showed that the curved step with lower width (W/8) results in higher lateral film cooling effectiveness and a lower heat transfer coefficient compared to the normal step and rectangular film holes without the step (Abdala and Elwekeel, 2016). Mist injection is also used to improve the film cooling effectiveness. Film cooling with mist injection was numerically investigated using slot configurations with a round hole and a diffusion hole. The results revealed that air film cooling performance is significantly improved by injecting mist into the coolant air (Li and Wang, 2006). Simulations on rotating turbine blades were conducted via the CFD method, and a cooling enhancement of 35% was obtained under elevated conditions (Dhanasekaran and Wang, 2012). The improvement of the cooling performance was analyzed by investigating the effects of mist concentration, diameters and interaction conditions, and the results indicated that the temperature of the boundary layer is decreased by injecting mist (Jiang et al., 2014). The effects of deposition configurations and the mist injection were investigated by using numerical simulations, and the results showed that the deposition formation for the mist injection obviously enhances cooling performance (Wang et al., 2016b). In addition, the evaporation of water droplets was found to improve the film cooling effectiveness through the injection of 2% mist. Furthermore, higher deposition was shown to cause a greater spread of water droplets in a lateral direction (Wang et al., 2016c).

Most similar recent studies show either a mist injection study without bulge, or an effect of a bulge configuration without mist injection. The objective of the present study is to understand the usefulness of placing a bulge configuration upstream of the film hole on improving the adiabatic film cooling effectiveness. In other words, the gas-liquid two-phase flow is investigated based on the bulge configuration. Different bulge configurations with heights of 0.1d, 0.3d and 0.5d are considered in investigating the fluid flow field and the cooling performance in the plate channel. The film cooling effectiveness in both streamwise and lateral directions is compared to obtain an optimal bulge configuration. Effects of the blowing ratio and mist injection are also considered. All the simulations are carried out using ANSYS FLUENT 16.0 software.

2. Numerical model and validation

Three-dimensional (3D) models are used to investigate the effect of bulge configurations on the film cooling effectiveness as shown in Fig. 1. The computational domain is 80d × 20d × 4d, and the cases have a round hole diameter (d) of 5 mm. The hole is set to 60d from the outlet of the mainstream, and the vertical height of the jet hole is 2d. An inclination angle (α) of 35° is adopted, and the bulge configuration has a streamwise length (w) of 0.2d.

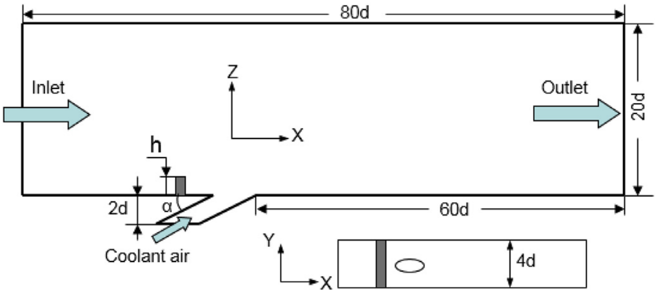

Fig. 1. Computational domain.

Please cite this article in press as: Wang, J., et al., Effect of an upstream bulge configuration on film cooling with and without mist injection, Journal of Environmental Management (2017), http://dx.doi.org/10.1016/j.jenvman.2017.06.055