International Journal of Heat and Mass Transfer 91 (2015) 390–400

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Effects of coolants on the flow and heat transfer characteristics in a non-rotating and rotating two-pass rectangular channel

Feng Zhang, Xinjun Wang [*], Jun Li

Institute of Turbomachinery, Xi'an Jiaotong University, Xi'an 710049, China

A R T I C L E I N F O

Article history:
Received 30 January 2015
Received in revised form 1 August 2015
Accepted 2 August 2015

Keywords:
Rectangular channel
Heat transfer enhancement
Coolant
Mist cooling

A B S T R A C T

The effects of coolants on the flow and heat transfer characteristics in a non-rotating and rotating two-pass rectangular channel with 45-deg angled rib turbulators are numerically investigated using the CFD software ANSYS-CFX. The three-dimensional Reynolds-Averaged Navier–Stokes (RANS) solution with SST turbulence model is applied in this paper. Four kinds of coolants, such as air, steam, mist/air and mist/steam, are used to analyze the effects of coolant types on the heat transfer performance in the channel at six different Reynolds number flow conditions. The channel averaged Nusselt number for air-only are numerically studied and the results are in good agreement with the experimental data. The internal flow pattern and heat transfer characteristics in the channel with air-only are conducted at two kinds of temperature difference. The results indicate that the 39 K temperature difference provides a slightly better heat transfer performance than that of the 60 K temperature difference. In addition, the obtained results show that the mist/steam gives the best heat transfer enhancement and thermal performance in the channel. The detailed flow pattern and Nusselt number distribution for four coolants are also illustrated and discussed.

1. Introduction

The efficiency of a gas turbine can usually be improved by increasing the temperature at entry to the turbines. This temperature rise however is limited by the melting point of component materials. Hence, there is a steady strive for designing effective methods to cool the blades and other thermally loaded components to prevent a failure. Many cooling technologies are used to remove heat from the blades (Fig. 1), and one of the ways is using serpentine passage as channel for coolant. To improve the heat transfer effectiveness in channels, roughened surfaces and different coolants are adopted. Ribs are commonly used to enhance the heat transfer in the channels by enlarging the wetted (cooled) surface area, tripping the boundary layer, inducing reattachment flow and enhancing the fluid mixing.

Researchers have investigated the flow and heat transfer characteristics in a two-pass channel with 180-deg turns. Al-Qahtani et al. [2], Al-Hadhrami et al. [3], Huh et al. [4] and Li et al. [5] experimentally and numerically studied the effect of channel orientation on the flow and heat transfer characteristics in channels. Liu et al. [6], Lei et al. [7] and Hagari et al. [8] considered the effect of rib spacing on the heat transfer and pressure drop in the two-pass channels. Su et al. [9], Fu et al. [10] and Siddique et al. [11] investigated the Nusselt number distribution for different aspect ratio channels. The effect of the different entrance geometry on heat transfer coefficients in rotating two-pass narrow channels has been discussed by Liu et al. [12] and Saha and Acharya [13]. Liou et al. [14] experimentally measured the Nusselt number distribution and thermal performance in the parallelogram channel.

It is noted that, even though cooling the turbine thermally loaded components with air extracted from the compressor effectively increases the components life, however, it pays a penalty of considerable decreased overall efficiency of the engine. Moreover, along with the turbine inlet temperature further rising, the air cooling may not prevent the hot components from failure. Hence, it is necessary to continuously develop new advanced cooling technologies to improve cooling effectiveness of the hot components and decrease extracting air in high-performance gas turbines. One of the promising methods is adopting other coolants, such as steam, mist/air and mist/steam, to cool hot components. Gao et al. [15,16] performed both numerical and experimental analyses on the flow and heat transfer characteristics of steam cooling in a ribbed channel. The results indicated that the steam coolant increased the heat transfer rates compared with the air coolant. To obtain better heat transfer performance, Wang and Ragab [17] began to consider the feasibility of water mist cooling for blades,

---

[*] Corresponding author.

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.08.004
0017-9310/© 2015 Elsevier Ltd. All rights reserved.

Nomenclature

A	surface area, m^2
c	concentration, kg/m^3
d	droplet diameter, μm
D_h	channel hydraulic diameter, m
f	friction factor, –
h	heat transfer coefficient, W/m^2 K
k	thermal conductivity, W/m K
L	heated section length, m
m	mass, kg
Nu	Nusselt number, –
p	pressure, Pa
Pr	Prandtl number, –
T	temperature, K
u	mean flow velocity, m/s

Greek

ρ	density, kg/m^3
Δ	difference, –
μ	dynamic viscosity, kg/s m
η	thermal performance, –
β	channel orientation angel, °

Subscripts

0	inlet
b	bulk flow
p,d	particle or droplet

and presented a numerical study to investigate the feasibility of transporting water mist to the rotating blades of a high pressure turbine. The results showed that it was feasible to transport the mist from the turbine casing to the blade through the air system, and the droplets fed to the blades had sufficient particle diameter and mist loading. Moreover, Zhao and Wang [18,19] experimentally studied the mist/air film cooling on a flat plate with a phase Doppler particle analyzer system and measured the mist flow velocity field, droplet size distribution, droplet dynamics, turbulence characteristics, and temperatures. They found that compared to air-only film cooling, adding mist performed a higher cooling effectiveness, cooling coverage and more uniform surface temperature. Dhanasekaran and Wang [20] and Elwekeel et al. [21,22] numerically investigated the heat transfer enhancement of mist/air cooling in a rotating two-pass square channel with two opposite ribbed side wall and a stationary one-pass rectangular channel with only one ribbed side wall, respectively. The results indicated that the Nusselt numbers of the mist cooling in channels are significantly higher than that of air-only cooling. In addition, Wang et al. [23,24] compared experimental work of mist/steam cooling in a two-pass smooth channel with 3D CFD, and Elwekeel et al. [21] numerically studied the mist/steam cooling in a one-pass ribbed channel. These results showed that the heat transfer performance of steam could be significantly improved by adding mist into the main flow.

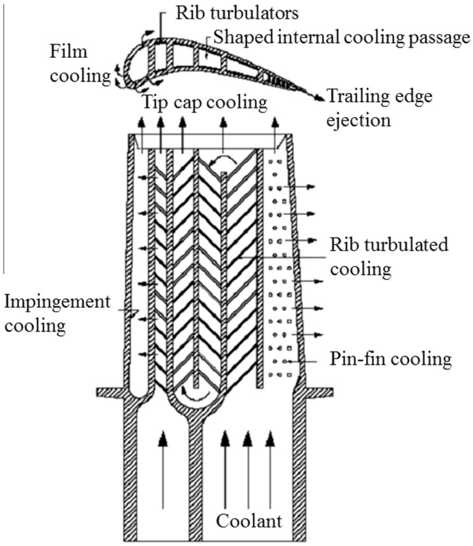

Fig. 1. Typical turbine blade internal cooling passages [1].

This paper conducts numerical studies on the effects of coolant types on the flow and heat transfer characteristics in a non-rotating and rotating two-pass rectangular channel with 45-deg angled ribturbulators. The channel averaged Nusselt number for air-only are numerically studied to validate the computational model with the experimental results, and the accuracy of the utilized numerical approach is verified. The internal flow pattern and heat transfer characteristics with air-only are then conducted at two kinds of temperature difference. In addition, four kinds of coolants, such as air, steam, mist/air and mist/steam, are used to analyze the effects of coolant types on the heat transfer performance in the channel at six different Reynolds numbers ranging from 5000 to 80,000.

2. Computational model and numerical method

2.1. Computational model

In this study, the computational configuration is adopted from Fu et al. [10]. Fig. 2a and b shows the geometry containing a two-pass rectangular ribbed channel with aspect ratio of 1:2 considered for this study. The dimensions of geometry are exactly the same as the apparatus used in the experimental study [10].

Fig. 3 shows the structural computational grids on the 180-deg turn of the channel which are generated using ANSYS ICEM-CFD.

2.2. Boundary conditions

Boundary conditions are summarized in Table 1. The uniform velocity profiles corresponding to the inflow Reynolds number and total temperature are applied as inlet conditions. All of walls in the heated portion are considered as no-slip walls with a constant wall temperature, while the other walls are modeled as adiabatic no-slip walls. At the outlet, a pressure of 1 atm is defined. In the rotating channel, the rotational speed of 550 rpm is also considered. For mist injection, a uniform mist size of 5 μm is considered and the mass ratio of mist over the main fluid is 1%. The mist injection points are equally spaced at the inlet. The boundary condition of droplets at the walls is assigned as "reflect," which means the mist would elastically rebound off once reaching the wall.

To validate the computational model, case 1, which is the same as the experimental condition in Fu et al. [10], is employed to