International Journal of Heat and Mass Transfer 61 (2013) 554–564

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Computational analysis of mist/air cooling in a two-pass rectangular rotating channel with 45-deg angled rib turbulators

T.S. Dhanasekaran, Ting Wang *

Energy Conversion and Conservation Center, University of New Orleans, New Orleans, LA 70148-2220, USA

ARTICLE INFO

Article history:
Received 26 August 2012
Received in revised form 31 January 2013
Accepted 1 February 2013
Available online 15 March 2013

Keywords:
Gas turbine heat transfer
Gas turbine blade cooling
Mist cooling
Two-phase heat transfer
Heat transfer enhancement

ABSTRACT

Increasing the turbine inlet temperature can increase the gas turbine cycle efficiency. In order to increase the turbine inlet temperature significantly, an advanced cooling system has to be essentially developed. Injection of mist to the coolant fluid is considered a promising technique to protect the hot components such as combustor liners, combustor transition pieces, and turbine vanes and blades. A series of experiments conducted in the past proved the success of mist cooling technology in the laboratory environment. Favorable results from the numerical simulation further encourage continuous exploration of employing mist-cooling technology in the actual gas turbine working environment in various applications. The present study focuses on applying mist cooling to the rotating mist/air internal cooling passage with rib turbulators using numerical simulation.

In the first part, the computational fluid dynamics (CFD) models of smooth and ribbed channels without mist and rotation are validated with the experimental results available in literature. The agreement between the predicted and experimental values in the lower Reynolds number (Re) range is within 3% deviation, and, at higher Re range, the deviation is about 10%. For the smooth channel, the agreement with experimental result is good for the entire range of Re values. In the second part, the rotational effect on the smooth and ribbed channels is predicted and analyzed. In the last part, the mist cooling enhancement on the ribbed channel with rotation is simulated. The secondary flows created due to channel bend and rotation are specifically analyzed. The results show that the mist cooling enhancement is about 30% at the trailing surface and about 20% at the leading surface of the first passage with 2% mist injection. In the second passage, 20% enhancement is predicted for both the surfaces.

1. Introduction

Increasing the turbine inlet temperature is one of the major means to increase the gas turbine (GT) efficiency. The increased hot gas temperature typically exceeds the allowable material limit for the blades and vanes. Hence, there is always demand for continuously developing new advanced cooling technologies to cool the hot components in high-performance gas turbines. One of the promising technologies to significantly enhance the heat transfer is to inject water mist into the coolant flow. Each droplet acts as a cooling sink and flies over a distance before it completely vaporizes. This “distributed cooling” characteristic allows controlled cooling by manipulating different sizes of injected water droplets. The mist/steam cooling scheme applicable to an Advanced Turbine System (ATS) was introduced and verified with extensive basic experiments under laboratory working conditions, in a horizontal tube [1,2], a 180-degree curved tube [3], impingement jets on a flat surface [4], and impingement jets on a curved surface [5]. Typically, an average cooling enhancement of 50–100% was achieved by injecting 1–3% (wt.) mist into the steam flow. A very high local cooling enhancement of 200–300% was observed in the tube and on a flat surface, and local cooling enhancement above 500% was observed when the steam flow passed the 180-degree bend.

The mist/air film cooling system applied to modern gas turbines was simulated by Li and Wang [6,7] and showed that a small amount of mist injection (2% of the coolant mass flow rate) could increase the adiabatic film cooling effectiveness by about 30% ~ 50% under low temperature, velocity and pressure conditions similar to those in the laboratory. They also investigated the effects of different flow parameters, injection hole configuration, and coolant supply plenum on the cooling effectiveness. In order to simulate the actual GT operating conditions more closely, Li and Wang [8] presented the mist/air film cooling heat transfer coefficient under a conjugate wall condition by employing internal channel cooling beneath the blade surface. The result of conjugated 2-D cases indicated that reverse heat conduction from downstream

---

* Corresponding author. Tel.: +1 504 280 7183.
E-mail addresses: tsdhana@gmail.com (T.S. Dhanasekaran), twang@uno.edu (T. Wang).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.02.006

Nomenclature

A_p	surface area of droplet
b	slot width (m)
C	concentration (kg/m^3)
D_h	channel hydraulic diameter
GT	gas turbine
h	convective heat transfer coefficient (W/m^2-K)
H	height of channel (m)
k	turbulent kinetic energy (m^2/s^2)
K	thermophoretic coefficient
Nu	local Nusselt number, hD_h/kc
m	mass (kg)
q″	wall heat flux (W/m^2)
Re	Reynolds number (ρ V_j d/ μ)
Ro	rotational number, Ω D_h/ V
T_w	wall temperature (°C)
V	inlet bulk velocity (m/s)
W	width of channel (m)

Greek

ε	turbulence dissipation (m^2/s^3)
λ	thermal conductivity (W/m-K)
Ω	rotational speed (rad/s)

Subscript

p, d	particle or droplet
r	riblet
s	smooth.

to upstream along the solid wall was strong within a distance of 5 slot widths. Li and Wang [9] studied the curvature effect on mist film cooling, as well. They found that the magnitude of the mist cooling enhancement was ordered as follows: flat surface > pressure surface > suction surface > leading edge.

Their simulation showed that the film cooling effectiveness increases approximately 40% at the leading edge, 60% on the concave surface, and 30% on the convex surface with 2% mist concentration. Their studies [6–9] on mist/air film cooling were conducted with the turbine in a stationary condition. Recently, Dhanasekaran and Wang [10] simulated the mist/air film cooling enhancement over a rotating blade under gas turbine working conditions with elevated pressure, heat flux, and Reynolds number. They predicted an average of 35% mist cooling enhancement with an equivalent blade surface temperature reduction of 100–125 K.

As a continuation of mist cooling technology developments, the present study employs computational fluid dynamics (CFD) simulations to investigate the cooling performance of applying mist in a gas turbine internal blade with riblets under both stationary and rotating conditions. The early investigations of enhancing turbine airfoil internal cooling under rotating conditions were performed with straight, smooth, circular tubes [11–13]. Han et al. (for example Ref. [14,15]) have conducted several experimental studies on straight rib-roughened, non-rotating and rotating channels to investigate the effect of turbulator configurations (such as rib height, spacing and angle), the flow channel aspect ratio, and the flow Reynolds number on the distributions of the local heat transfer and pressure drop. Recently, as CFD schemes became more sophisticated and powerful, numerical studies of heat transfer in ribbed channels have become popular. Chang and Mills [16] employed a low Reynolds number turbulence model for a two-dimensional situation involving flow in a stationary circular tube with repeated rectangular ribs. Arman and Rabas [17] subsequently predicted the flow field and heat transfer in a stationary circular tube with repeated ribs using a two-layer model. Prakash and Zerkle [18] have used the standard k-ε model with wall functions to predict the heat transfer in rotating rib ducts with the assumption of periodic, fully developed flow situations.

In the present study, a mist/air computational model is developed to predict the mist cooling enhancement on the rotating, ribbed, rectangular channel. Initially, the air-only computational model is validated with the experimental results available in open literature. The effects of rotational force on the flow physics and heat transfer in the channel with riblets are investigated. Finally, the mist cooling enhancement is simulated using the validated computational model.

2. Numerical method

A feasible method to simulate the air/mist flow is to consider the droplets as a discrete phase since the volume fraction of the liquid is small (less than 0.1%) in this study. The trajectories of the dispersed phase (droplets) are calculated by the Lagrangian method. The impacts of the droplets on the continuous phase are considered as source terms to the governing equations of mass, momentum, energy, and species. The continuous phase, including air and water vapor, is formulated with the Eulerian method. The mist cooling computational model has been established and validated within 3% to 15% deviation from the experimental data for various configurations including: mist/steam impingement on a flat surface [19], on a curved surface [20], in a horizontal tube [21], and in a 180 degree bend tube [22]. The detailed description of CFD mist flow modeling is referred to Dhanasekaran and Wang's other studies [19,21] and is not repeated here. A summary of the CFD model is presented below.

2.1. Continuous Phase

The time-averaged governing equations of mass, momentum, energy, and species are:

$$\frac{\partial}{\partial x_i}(\rho u_i) = S_m \tag{1}$$

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = \rho \vec{g}_j - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\left(\tau_{ij} - \rho\,\overline{u_i' u_j'}\right) + F_j \tag{2}$$

$$\frac{\partial}{\partial x_i}(\rho c_p u_i T) = \frac{\partial}{\partial x_i}\left(\lambda\frac{\partial T}{\partial x_i} - \rho c_p \overline{u_i' T'}\right) + \mu\Phi + S_h \tag{3}$$

$$\frac{\partial}{\partial x_i}(\rho u_i C_j) = \frac{\partial}{\partial x_i}\left(\rho D_j \frac{\partial C_j}{\partial x_i} - \rho\overline{u_i' C_j'}\right) + S_j \tag{4}$$

where τ_{ij} is the symmetric stress tensor. The source terms (S_m, F_j and S_h) are used to include the contributions from the dispersed phase. $\mu\Phi$ is the viscous dissipation and λ is the thermal conductivity. C_j is the mass fraction of species j in the mixture, and S_j is the source term for this species. D_j is the diffusion coefficient of species j. The diffusion term is used for bi-diffusion between the water vapor and air mass. When the liquid evaporates, the vapor produced surrounds the liquid droplet. Then, this vapor will be transported away through convection and mass diffusion. Three species (oxygen, nitrogen and water vapor) are simulated in the paper.

The terms of $\rho\overline{u_i' u'}_j$, $\rho\ c_p \overline{u_i'\ T'}$ and $\rho\overline{u_i' C'}_j$ in the equations above represent the Reynolds stresses, turbulent heat fluxes, and turbulent concentration (or mass) fluxes, which should be modeled