International Journal of Heat and Mass Transfer 120 (2018) 895–908

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Flow structure and surface heat transfer from a turbine component endwall contoured using the ice formation method

Sven Winkler [a], Bernhard Weigand [a], Phil Ligrani [b,*]

[a] *Institute of Aerospace Thermodynamics, University of Stuttgart, 70569 Stuttgart, Germany*
[b] *University of Alabama in Huntsville, Huntsville, AL 35899, USA*

ARTICLE INFO

Article history:
Received 13 July 2017
Received in revised form 5 December 2017
Accepted 6 December 2017

Keywords:
Aerodynamic losses
Gas turbine engines
Turbine components
Airfoil/endwall interactions
Secondary flows
Vorticity
Endwall contouring

ABSTRACT

Steady-state flow structure and surface heat transfer characteristics are numerically predicted for a turbine vane passage, which is contoured using the ice formation method, using ANSYS® Fluent version 12.1. Utilized are an SST turbulence closure model with a low-Reynolds formulation, along with a pressure-based approach to solve the governing equations with the SIMPLE algorithm. The solution domain is spatially discretized with a hybrid grid, created with the commercial grid generator CENTAUR™. The endwall contour shape is obtained using an IFM or ice formation method, which relies upon natural processes wherein energy dissipation and entropy production are minimized. This ice-contouring is imposed only within the vane passage, with endwall transition regions just upstream and just downstream. Particular attention is devoted to the intricate and detailed flow structural variations which are present near the endwall, both through the vane passage and downstream of the vanes. Considered are surface oil flow visualization distributions, vortex distributions, flow pathline distributions, and variations of the z-component of velocity, including their effects of ice-contoured flow structure on surface heat transfer coefficients, relative to a flat, baseline endwall for a turbine flow passage with air flow for a vane chord Reynolds number of 49,900. Results show that contouring significantly alters the endwall heat transfer coefficient distribution, with local values which are both lower and higher compared to baseline results. Such changes are strongly linked to the altered flow field due to the contouring, including changes to passage vortex advection and development.

1. Introduction

Design and development of turbine components of gas turbine engines consider a variety of subject areas. Two of the most important of these are surface heat transfer loading, and aerodynamic loss generation. Aerodynamic losses are generally categorized as either profile losses, tip clearance losses, or end-wall losses. Profile losses are caused by the blade or vane "profile" and are generated on the airfoil surface due to the growth of boundary layers. Tip leakage losses mostly occur in rotors and are due to the pressure difference that is formed over the blade tip between the pressure and suction sides of the blade. The third major type of loss, known as end-wall loss or secondary flow loss, is due to viscous effects from the presence of the end-wall, and interactions of end-wall boundary layers with the airfoils. The primary flow that is created by blades and vanes is diverted due to viscous effects and gives rise to secondary flows. The most important resulting secondary flows are passage vortices, counter vortices, and corner vortices [1]. For a turbine blade row with low aspect ratio, such secondary losses (from flows near the endwall) can be as high as 30–50 percent of the total aerodynamic loss in a single blade row [1]. Management of such secondary flows using endwall contouring, along with the associated endwall surface heat transfer variations, is the subject of the present investigation.

Such contouring of endwall platforms is generally employed to reduce the detrimental influences of associated secondary flows. As such, the objective is generally either: (a) reduction of local and overall surface heat transfer magnitudes, or (b) reduction of local and overall aerodynamic loss magnitudes. Note that the achievement of one objective generally does not necessarily always lead to achievement of the other objective. Early investigations of the development and consequences of turbine passage endwall contouring are described by Deich et al. [2], Ewen at al. [3], Duden et al. [4], Dossena et al. [5] and Burd and Simon [6]. More recently, Shih et al. [7] examine flow within a nozzle guide vane passage with one flat endwall and with one contoured endwall. Other investigations by Lin and Shih [8], Shih and Lin [9],

---

* Corresponding author.
E-mail address: pml0006@uah.edu (P. Ligrani).

https://doi.org/10.1016/j.ijheatmasstransfer.2017.12.028

Nomenclature

Latin symbols

C	true chord length (m)
C_{ax}	axial chord length (m)
htc	heat transfer coefficient (W m^{-2} K^{-1})
k	thermal conductivity (W m^{-1} K^{-1})
$\dot{m}$	mass flow rate (kg s^{-1})
N	number of numerical cells (–)
p	static pressure, (N m^{-2})
q''	surface heat flux (W m^{-2})
r	refinement ratio (–)
Re_C	chord length Reynolds number (–)
S	vane span (m)
T	static temperature (K)
u; v; w	velocity components (m s^{-1})
u_{exit}	spatially-averaged streamwise velocity at the exit cross section of the vane passage (m s^{-1})
y_1^+	non-dimensional distance from endwall (–)
x; y; z	spatial directions (m)

Greek symbols

Θ	non-dimensional temperature difference ratio, $(T_F - T_{CI})/(T_\infty - T_F)$ (–)
γ	vane turning angle (°)
ν	kinematic viscosity (m^2 s^{-1})
ρ	density (kg m^3)
ψ	non-dimensional width (–)
ψ_1, ψ_2, ψ_3, ψ_4, ψ_5	streamwise planar location (–)

Subscripts/superscripts

bulk	bulk flow value
CI	copper inlay value
EW	endwall value
F	freezing value
in	inlet value
out	outlet value
turb	turbulent value
water	water value
—	mean value
∞	free stream value

and Lin et al. [10] address effects of inlet Mach number, and the use of inlet swirl and leading-edge airfoil fillets, including the effects of inlet swirl angle. Moser et al. [11,12] develop shroud contour designs which are defined by an axisymmetric shroud contour, and particular stagger angle and blade contour distributions. With such arrangements, the passage vortex is degraded and dislocated relative to the endwall. Taremi et al. [13] address the effects of endwall contouring on secondary flow losses. According to these investigators, the implementation of endwall contouring results in less intense vortical structures and a reduction in secondary kinetic energy. Two linear cascade configurations are considered with flat endwalls, and two configurations are considered with contoured endwalls. One pair of configurations is more highly loaded, which is associated with the larger blade spacing and larger pitch-to-span ratio. The more highly loaded cascade displays stronger secondary flow structures, wherein the use of endwall contouring decreases the sizes and strengths of the passage and counter vortices.

A variety of different approaches are employed to develop contouring arrangements for endwalls. For example, Shih et al. [7] utilize a contoured endwall which is comprised of a 45° slope straight line with a certain projected length in the streamwise direction, and a 45° arc to connect the sloped endwall portion with the flat portion located just downstream. Two positions of this contoured profile are considered. With the first, all of the contouring is upstream of the airfoil, which is the more traditional arrangement wherin contouring is not utilized to modify the flow within the airfoil passage. With the second configuration, contouring begins upstream of the airfoil, continues through the airfoil passage, and then, ends at the airfoil trailing edge. With this second design, contouring affects flow approaching the airfoil and throughout the endwall region by providing increased amounts of mainstream flow acceleration. The contoured endwall employed by Moser et al. [11] is determined based upon flowpath profiling of the shroud. With this approach, the shroud contour is varied in radial direction within specified restrictions by an algorithm, which is directly connected to a mesh generator and a CFD solver. Optimization is based upon reduction of total pressure losses over the guide vanes. In a later investigation, Moser et al. [12] employ similar optimization criteria. However, within this study, flowpath profiling is rotationally symmetric. The combination of two-dimensional shroud contour, and flow variations within the guide vane passage, gives a fully three-dimensional end wall contour, which alters secondary flow features within the turbine passage in a significant manner. Taremi et al. [13] employ a non-axisymmetric contoured endwall geometry which is based upon a gradient-based optimization algorithm, coupled with compressible CFD simulations. Employed constraints include: (a) contouring is confined to one blade passage with one degree-of-freedom in the wall-normal direction, (b) flat endwall conditions are matched including midspan loading distributions, exit flow angles, and variations of axial velocity density ratio, and (c) no modifications are made to blade shape and coordinates.

Within the present investigation, the endwall contour shape is obtained using an IFM or ice formation method, which relies upon natural processes wherein energy dissipation and entropy production are minimized. Early investigations of this approach are described by LaFleur [14,15], who describes the use of an Ice Formation Method for a flat plate Couette flow arrangement. According to this investigator, the resulting steady-state ice layers represent geometries with minimum energy dissipation. Carlson [16] minimize the drag coefficient of a cylinder in cross flow using an Ice Formation Method approach. The most optimal arrangement is ellipsoidal in shape, which results in delayed flow detachment and significant reductions in drag magnitudes. LaFleur and Langston [17] applied the Ice Formation Method to a cylinder/endwall junction and describe configurations with drag coefficient reductions by as much as 18 percent, compared to baseline arrangements. LaFleur [18] later employed the Ice Formation Method for drag coefficient reductions for a vane endwall. Optimized configurations are described which are determined by melting an initially grown ice layer, using air flow with engine realistic Reynolds numbers. More recently, Winkler et al. [19] employed the Ice Formation Method to optimize a vane endwall with respect to surface heat transfer distributions. Within this investigation, ice-contoured endwalls are first created within a water channel test facility, which are then analyzed numerically with air as the flow medium for application to gas turbine environments. Reductions to overall