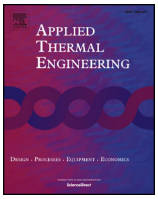
Applied Thermal Engineering 143 (2018) 935–946

Contents lists available at ScienceDirect

Applied Thermal Engineering

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



Research Paper

Effects of a pocket cavity on heat transfer and flow characteristics of the endwall with a bluff body in a gas turbine engine

Jian Liu[a], Safeer Hussain[a], Lei Wang[a], Gongnan Xie[b], Bengt Sundén[a,*]

[a] Division of Heat Transfer, Department of Energy Sciences, Lund University, P.O. Box 118, SE-22100 Lund, Sweden
[b] School of Marine Science and Technology, Northwestern Polytechnical University, Xi'an 710072, Shaanxi, China

HIGHLIGHTS

- Heat transfer effects of a pocket on an endwall with a bluff body are revealed.
- Liquid Crystal Thermography (LCT) is employed to measure heat transfer.
- Detached Eddy Simulation (DES) is applied in numerical simulations.
- Detailed flow patterns and physical phenomena are revealed.

ARTICLE INFO

Keywords:
Pocket cavity
Bluff body
LCT
DES
Flow impingement

ABSTRACT

A pocket cavity is generated at the connection of two parts, namely the transition part between the Low Pressure Turbine (LPT) and Outlet Guide Vane (OGV) in a gas turbine engine. This kind of pocket cavities, due to the high Reynolds number and the specific shapes, are hardly investigated in previous researches. In the present work, the effect of the pocket on the heat transfer of the endwall with a bluff body in the rear part of a gas turbine engine is investigated. A triangular pocket cavity is built in a rectangular channel and two kinds of bluff bodies, a cylinder or a cuboid is respectively put on the endwall. Liquid Crystal Thermography (LCT) is employed to measure the heat transfer over the endwall at Reynolds number ranging from 87,600 to 219,000. The turbulent flow details are presented by numerical calculations with the turbulence model, Detached Eddy Simulation (DES). For the pocket channel with a bluff body, the large heat transfer areas are usually found at the downstream edge of the pocket cavity and the vortices shedding regions around the bluff body. For the cuboid case, the high heat transfer regions are not at the leading edge of the cuboid, but are distributed as two banded regions upstream of the cuboid off the centerline. When a pocket cavity is placed in the upstream of the bluff bodies, cylinder or cuboid, the high heat transfer areas around the bluff bodies are decreased. The angle of the main flow attachment is changed due to the disturbances of the cavity. Accordingly, the flow impingement on the bottom wall of the bluff body is weakened and the heat transfer around the bluff body is decreased. The research displays the influence of incoming flows on the bluff body in the downstream and provides some references for placing OGV in the rear part of a gas turbine engine.

1. Introduction

As a heat transfer enhancement element, pockets or grooves are also widely used in modern heat exchangers and other cooling equipment. In the past decades, researchers have put more attention on the investigation of rectangular and triangular grooves [1–4]. Lorenz et al. [1] measured the distributions of the heat transfer coefficient along channel walls with periodic transverse grooves for thermal developing and periodic turbulent flow at $104 \le Re \le 105$. The Nusselt number augmentation ratio of the grooved surfaces was about $1.52 \le Nu/Nu_0 \le 1.75$ compared to that of a smooth channel. Adachi and Uehara [2] presented a correlation between heat transfer and pressure drop in channels with expanded grooves and contracted grooves. They found that the channels with expanded grooves performed efficiently, while the channels with contracted grooves were inefficient. The contracted grooves in that research were similar to conventional rib turbulators. Jaurker et al. [4] investigated heat transfer and friction characteristics of a rectangular solar air heater duct using rib-grooved artificial

---

[*] Corresponding author.
E-mail address: bengt.sunden@energy.lth.se (B. Sundén).

https://doi.org/10.1016/j.applthermaleng.2018.08.020
Received 7 September 2017; Accepted 5 August 2018

Nomenclature

d_P	depth of pocket cavity (cm)
d_L	distance between pocket cavity and bluff body
D_h	hydraulic diameter (cm)
D_B	length of bluff body
H	channel height (cm)
H_B	height of bluff body
h	heat transfer coefficient ($W/m^2·K$)
k	turbulent kinetic energy (m^2/s^2)
L	total length of the channel (cm)
L_{front}	length of upstream extended channel (cm)
L_{back}	length of downstream extended channel (cm)
Nu	Nusselt number
Nu_0	Nusselt number of a smooth channel
p	pressure (Pa)
q_w	wall heat flux (W/m^2)
q_{loss}	heat loss (W/m^2)
R	fillet radius (cm)
Re	Reynolds number, $Re = \rho u_m D_h/\mu$
T	temperature (K)
T_f	fluid temperature (K)
T_w	wall temperature (K)
u	flow velocity (m/s)
x	streamwise direction
y	spanwise direction
z	normal direction
W	channel width (cm)

Greek symbols

α	attack angle of pocket (degree)
Δp	pressure drop (Pa)
λ	thermal conductivity (W/m·K)
μ	fluid dynamic viscosity (Pa·s)
ρ	fluid density (kg/m^3)

Subscripts

b	bottom surface
in	inlet
m	average/overall
max	maximum
out	outlet
w	wall

roughness. The experimental investigation showed that Nusselt number can be further enhanced beyond that of a ribbed duct with a low friction factor. Groove can combine with other heat transfer elements to augment heat transfer [5–8]. Eiamsa-ard and Promvonge [5] investigated the characteristics of turbulent flows in a rib-grooved channel. The rib-groove combined structures significantly enhanced the heat transfer rate compared with a smooth duct. Al-Shamani et al. [6] investigated heat transfer enhancement characteristics in a channel with trapezoidal rib-grooves using nano fluids. The study showed that these trapezoidal rib-grooves with nano fluids had the potential to dramatically increase heat transfer characteristics and thus might be very useful for the development of efficient heat exchanger devices. The grooves were combined with other structures to enhance the cooling efficiency. Gutierrez et al. [7] conducted a numerical and experimental analysis of heat transfer enhancement in a grooved channel with curved flow deflectors. The main function of the deflectors is to divert a portion of the flow into the space between blocks in order to increase the fluid motion in this area and to remove the trapped fluid. Skullong et al. [8] investigated the thermal performance in a solar air heater channel with combined wavy-groove and perforated-delta Wing Vortex Generators (WVG). The combined devices gave a thermal performance augmentation of about 37.7–46.3% higher than the groove alone and also at about 1.5–12.5% above the combined groove and non-perforated WVG.

Three-dimensional vortical structures produced by a bluff body greatly affect the heat transfer on the endwall. A huge amount of literature is available for heat transfer devices related to placing a bluff body in the mainstream flow [9–15]. Goldstein et al. [9] experimentally investigated mass transfer in cross flow of a square bluff body and on its endwall. They found that there were two high mass transfer peaks upstream of the square bluff body generated by the corner vortex and horse-shoe vortex pairs. Chyu and Natarajan [10] used a liquid crystal technique to study the base plate heat transfer for different obstacle geometries at different Reynolds numbers. Cylindrical, square and diamond shaped bluff bodies with different heights and different arrangements in tandem arrays were investigated in their research. Using simultaneous PIV and liquid crystal thermography, Praisner and Smith [11,12] investigated the endwall heat transfer and vortices formed upstream of an airfoil. Different cross-sections were selected to study the vortical behavior and corresponding endwall heat transfer. Two high heat transfer zones were found upstream of the symmetric airfoil. Wang et al. [13] investigated the endwall heat transfer characteristics of forced flow past bluff bodies using Liquid Crystal Thermography (LCT). The results indicated that due to the formation of horseshoe vortices, the heat transfer of a single bluff body and two bluff bodies arranged in tandem were both enhanced and the power index of Reynolds number was flow dependent. The film cooling effectiveness in the region of the blade-endwall corner junction with injection was studied by Milidonis and Georgiou [14]. The experimental results showed a very encouraging cooling effectiveness by using the recirculating vortex flow with a tiny amount of coolant flow. Kumar et al. [15] investigated fluid flow and heat transfer around a confined semi-circular cylinder. The two-dimensional simulations were carried out at various values of the controlling parameters: Reynolds number = 50–200 and Prandtl number = 0.7, 10 and 100 at a fixed blockage ratio of 25% for a Newtonian constant-properties fluid. They used the calculation results to develop drag coefficient, Strouhal number and Nusselt number correlations.

In a gas turbine engine, a pocket cavity is generated at the junction position of the Low Pressure Turbine (LPT) and the Outlet Guide Vane (OGV) when they are assembled together. The triangular pocket cavities are not designed for heat transfer enhancement purposes but they are difficult to avoid in the mechanical assembly. The OGV controls the flow leaving the gas turbine and then affects the propulsion of the engine. In the design of advanced aero engines, the requirements of flow manipulation of the OGV become significantly higher and the heat transfer and flow pattern of pocket cavities have significant effects on the incoming flow of the OGV. The effects of the incoming flow characteristics on the endwall heat transfer and turbulent flow of turbine blades have received some interest [16–18]. Qureshi et al. [16] investigated the impact of swirl on the aerodynamic and heat transfer characteristics of an HP turbine stage using a combustor swirl simulator. Zhang et al. [17] studied the effects of inlet turbulence and endwall boundary layer on aero thermal performance of a transonic turbine blade. They observed noticeable changes in heat transfer are observed for the suction side near-tip surface due to the different inlet end wall boundary layer profiles. Lynch and Thole [18] compared the three-dimensional boundary layer on flat versus contoured turbine endwalls. They found elevated turbulence levels in the contouring endwall compared to the flat endwall.

Based on the above discussion, the pocket cavity and the bluff body