International Journal of Thermal Sciences xxx (2014) 1–13

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Concept development for the experimental investigation of forced convection heat transfer in circumferential cavities with variable geometry

David Spura*, Jasmin Lueckert, Sophie Schoene, Uwe Gampe

Technische Universität Dresden, Institute of Power Engineering, Chair of Thermal Power Machinery and Plants, 01062 Dresden, Germany

ARTICLE INFO

Article history:
Received 27 February 2014
Received in revised form
31 July 2014
Accepted 26 August 2014
Available online xxx

Keywords:
Forced convection
Heat transfer
Cavity
Industrial steam turbine
Nusselt correlation
Test rig

ABSTRACT

Cavities of various geometries occur in several technical applications and the knowledge about the heat transfer inside them is limited. For industrial steam turbines, this becomes critical since thermal casing distortion needs to be predicted reliably in the development process. A test rig concept is presented in this paper to investigate the forced convection heat transfer in cavities with various geometries where the enclosed fluid is driven indirectly by an external turbulent air flow. The dimensions are chosen to achieve similarity with side spaces in steam turbines. For estimating the flow conditions in the side spaces, a computational fluid dynamics analysis and experiments at a shallow water duct were conducted. Two different methods will be used to determine the heat transfer coefficient along the outer cavity wall of the compressed air test rig: the overtemperature method and the steady-state inverse method. In order to eliminate flow disturbances due to natural convection in the cavity and interference among the measurement systems, a parametrical finite element analysis was performed to determine an appropriate concept for the sensor arrangement.

1. Introduction

Surface cavities occur by nature or design in various technical applications in energy engineering, for example as slots in airplanes and spacecraft, grooves in combustion chambers and nuclear reactors as well as in finned heat exchangers, ribbed channels or solar heat pipes, but also in steam or gas turbine passages.

For applied sciences, it is fundamentally important to predict the heat transfer between the fluid in the cavity, which is driven indirectly by an external flow, and the adjacent surface. This applies particularly to the side spaces between the guide vane carriers and the casing of industrial steam turbines which result from the modular design of industrial steam turbines (Fig. 1). Their cross-section generally has an L- or a T-shape and varies in its dimensions. In addition, these side spaces can feature a steam injection, a tap or an extraction.

The heat transfer between the steam and the outer casing varies significantly. Depending on the flow structure in these side spaces, this leads to asymmetric heating and, thus, to thermal distortion of the casing. Thermal casing distortions have a negative influence on both the leak tightness of the split joint seal and the radial and axial clearance between the rotor and the stator. That is why they need to be predicted in an early phase of the development process with the help of FEM simulations. Therefore, heat transfer coefficients are required as thermal boundary conditions.

Various numerical and experimental investigations have been carried out during the past 50 years to determine flow and heat transfer in simple cavities with rectangular cross section in an external transverse flow [1–21]. However, this knowledge about the heat transfer cannot be applied directly to the more complex geometry due to different flow structures in the cavities.

This is also the case for the many various references that can be found about heat transfer [22,23] in rotating cavities, as they occur e.g. in internal cooling passages between the high pressure compressor discs, the low pressure shaft and the shroud of gas turbine rotors. Because of the rotating shroud, the swirl in the axial flow has a basically different characteristic profile (Fig. 2d) than the swirling flow after a blade wheel in a steam turbine passage with a static outer wall (Fig. 2e). In addition, centrifugal forces through cavity side wall friction have a predominant effect on the fluid structure in the cavity (see Refs. [24,25] respectively), and, thus, another considerable impact on the cavity heat transfer.

* Corresponding author. Tel.: +49 351 463 32605; fax: +49 351 463 37759.
E-mail addresses: david.spura@tu-dresden.de (D. Spura), jasmin.lueckert@tu-dresden.de (J. Lueckert), sophie.schoene@tu-dresden.de (S. Schoene), uwe.gampe@tu-dresden.de (U. Gampe).

http://dx.doi.org/10.1016/j.ijthermalsci.2014.08.018
1290-0729/

Nomenclature

Latin symbols

A	cross sectional area, [m^2]
B	width of the channel, [mm]
Cl	Clausius number $Cl = \rho u^3 l/(\lambda \Delta T)$, [−]
D	major/outer diameter, [mm]
Eu	Euler number $Eu = p/(\rho u^2)$, [−]
Fr	Froude number $Fr = u^2/(lg)$, [−]
H	water level, [mm]
Ho	homochronity number $Ho = ut/l$, [−]
I	electric current, [A]
L	length, [mm]
Ma	Mach number $Ma = u/w$, [−]
Nu	Nusselt number $Nu = hl/\lambda$, [−]
P	power output $P = VI$, [W]
Pe	Péclet number $Pe = \rho c_p ul/\lambda$, [−]
R	electric resistance $R = V/I$, [Ω]
Re	Reynolds number $Re = \rho ul/\eta$, [−]
T	temperature, [°C, K]
U	wetted perimeter, [mm]
V	voltage, [V]
a	distance between two measuring points, [mm]
b	width of the cavity, [mm]
c	specific heat capacity, [J/(kg K)]
d	minor/inner diameter, [mm]
g	gravitational acceleration $g = 9.81$ m/s^2, [m/s^2]
h	heat transfer coefficient, [W/(m^{2}K)]
h_1	height of the inlet, [mm]
h_2	height of the cavity, [mm]
l	characteristic length $l = 4A/U$, [mm]
$\dot{m}$	mass flow rate, [kg/s]
n	rotational speed, [1/min]
p	pressure, [bar]
r	radius, [mm]
s	width of the inlet, [mm]
t	time, [s]
u	flow velocity, [m/s]
w	speed of sound, [m/s]
$\vec{x}$	point at inside of outer wall $\vec{x} = (r, \varphi, z)$, [−]
y^+	dimensionless wall distance $y^+ = \sqrt{\tau_W/\rho} \cdot \Delta y_{nW}/\nu$, [−]
z	axial coordinate, [mm]
z^*	axial coordinate starting from centre line of cavity inlet, [mm]

Greek symbols

Λ	scaling, [−]
η	dynamic viscosity, [Pa s]
λ	thermal conductivity, [W/(m K)]
$\vec{\xi}$	vector of aspect ratios of side space, [mm]
ρ	volumetric mass density, [kg/m^3]
ϱ	relative radial gap $\varrho = \Delta r/r_i$, [−]
τ	shear stress, [Pa]
φ	rotation angle, [°]

Abbreviations

3D	three-dimensional
CFD	computational fluid dynamics
FE	finite element
FEM	finite element methods
HTC	heat transfer coefficient
NTC	negative temperature coefficient
PMMA	poly(methyl methacrylate)

Subscripts

av	average
bulk	bulk
F	fluid
f	forced convection
h	hydraulic
i	inner
loc	local
max	maximum
n	natural convection
nW	near-wall
ori	original
p	isobaric
TR	compressed air test rig
W	wall
WT	wall temperature

For stationary cavities with more complex geometry (L- or T-shaped), literature only provides little information about the flow profiles [26–32] and no information about the internal heat transfer. State-of-the-art CFD calculations are able to predict heat transfer conditions in any flow domain, but the results are often too imprecise and only cover one geometrical set and one operating point at a time. Hence, CFD calculations are too expensive and too unreliable to determine the heat transfer for every case.

In order to improve knowledge about heat transfer in side spaces, a test rig was designed for the systematic study of the heat transfer in these more complex cavities. The influence of the main flow conditions (Reynolds number, swirl) and aspect ratios of the cavity (ranging from I- over L-to T-shape) are subjects of further investigations.

2. Test rig concept

2.1. Requirements and general conditions

In the first stage of development, the test rig is designed to achieve representative test conditions using compressed air from the existing infrastructure for operation. Two parallel working screw compressors with air dryers deliver a maximum air flow of 1.28 kg/s at a constant temperature of 26 °C, which is then throttled to a constant absolute pressure between 2 and 3 bar. The mass flow rate through the test rig is adjusted via a bypass (10 to 100% of $\dot{m}_{max}$) and measured with an orifice plate.

The air is forced to flow through an annular gap between an inner and an outer tube forming the flow channel of the test rig (Fig. 7). The cavity is situated along the outer circumference of the main flow passage. Because of the rotational symmetry, it is possible to eliminate side effects on the cavity flow and, thus, on the determination of the heat transfer coefficients along the inner surface of the outer wall (as reported in Refs. [10,32]). Furthermore, the real annular flow conditions can be reproduced, in contrast to planar test sections.

The dimensioning of the compressed air test rig has to meet the requirements of the similarity theory to ensure that the results and the knowledge can be transferred to the original steam turbine scaling (see Section 2.2).

In order to investigate as many different geometry cases as possible, the side space geometry has to be variable and adjustable.