International Journal of Heat and Fluid Flow 70 (2018) 160–170

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Large-eddy simulation of unsteady turbine rim sealing flows

Feng Gao*,[a], John W. Chew[a], Paul F. Beard[b], Dario Amirante[a], Nicholas J. Hills[a]

[a] Faculty of Engineering and Physical Sciences, University of Surrey, Guildford GU2 7XH, UK
[b] Osney Thermo-Fluids Lab., Dept. of Eng. Sci., University of Oxford, Oxford OX2 0ES, UK

ARTICLE INFO

Keywords:
Turbine rim seal
Chute seal
Unsteady flow structures
Large-eddy simulation
Taylor–Couette instability

PACS:
47.20.Qr
47.27.Ep

ABSTRACT

Unsteady flow phenomena unrelated to the main gas-path blading have been identified in a number of turbine rim seal investigations. This unsteadiness has significant influence on the sealing effectiveness predicted by the conventional steady RANS (Reynolds-averaged Navier–Stokes) method, thus it is important for turbine stage design and optimisation. This paper presents CFD (computational fluid dynamics) modelling of a chute type rim seal that has been previously experimentally investigated. The study focuses on inherent large-scale unsteadiness rather than that imposed by vanes and blades or external flow. A large-eddy simulation (LES) solver is validated for a pipe flow test case and then applied to the chute rim seal rotor/stator cavity. LES, RANS and unsteady RANS (URANS) models all showed reasonable agreement with steady measurements within the disc cavity, but only the LES shows unsteadiness at a similar distinct peak frequency to that found in the experiment, at 23 times the rotational frequency. The boundary layer profile within the chute rim seal clearance has been scrutinised, which may explain the improvement of LES over RANS predictions for the pressure drop across the seal. LES results show a clockwise mean flow vortex. A more detailed sketch of the rim sealing flow unsteady flow structures is established with the help of the LES results. However, there are some significant differences between unsteadiness predicted and the measurements, and possible causes of these are discussed.

1. Introduction

Turbine rim sealing flows are a key issue in turbomachinery design, affecting both turbine aerodynamic losses and turbine rotor disc lifetime. A typical rim seal arrangement includes a narrow azimuthal clearance between rotor and stator disc platform, connecting the rotor/stator disc cavity and the main gas path. Additional cooling air is required to prevent ingestion of the hot main annulus gas into the turbine disc cavity which, if allowed, may lead to thermal failure of the rotor disc. In addition, pumping too much cooling air into the main gas path may result in aerodynamic losses. Therefore, correct modelling of sealing effectiveness is desired for turbomachinery design and its optimisation. However, modelling of such flows with Reynolds-averaged Navier–Stokes (RANS) has proved difficult and several experimental and numerical studies have indicated that the rim seal gap and disc cavity flows can contain large scale unsteady flow structures with frequencies unrelated to those associated with the rotating blades.

Basic mechanisms involved in rim sealing flow phenomena were classified by Johnson et al. (1994). Among these are (1) disc pumping effect, (2) three-dimensional (3-D) and time dependent periodic pressure field created by vanes and blades, (3) 3-D geometry within rim seal region, (4) asymmetries in the rim seal geometry, (5) turbulent transport in the platform overlapping region, and (6) flow entrainment.

In addition to these basic mechanisms mentioned above, a number of experimental and numerical studies have reported large-scale low-frequency unsteady flow structures in the rim seal gap and disc cavity which are not attributed to the main gas-path vanes and blades. The first evidence of this kind of unsteadiness emerged in the early 2000s. In 2002, Smout et al. (2002) mentioned the possible presence of large-scale low-frequency pressure fluctuations having larger wave length than that associated with the pitch of the vanes or blades, in the Aachen University 1.5 stage turbine rim sealing test rig designed by Bohn et al. (2003). The first published experimental evidence of the large-scale unsteady flow features was reported by Cao et al. (2004) in 2003, who also showed some agreement with unsteady RANS (URANS) solutions. Prior to this, in 2002, Autef (2002) reported URANS solutions showing unsteady flow structures of a rim seal configuration without vanes and blades, as described by Chew et al. (2003). In 2004 Jakoby et al. (2004) reported URANS studies showing large-scale unsteady flow structures unrelated to the blade passing, and supported by measurements from Bohn et al.'s rig.

---

* Corresponding author.
E-mail addresses: f.gao@surrey.ac.uk (F. Gao), j.chew@surrey.ac.uk (J.W. Chew), paul.beard@eng.ox.ac.uk (P.F. Beard), d.amirante@surrey.ac.uk (D. Amirante), n.hills@surrey.ac.uk (N.J. Hills).

https://doi.org/10.1016/j.ijheatfluidflow.2018.02.002
Received 14 July 2017; Received in revised form 4 December 2017; Accepted 7 February 2018

Nomenclature

Roman symbols

b	Rotor disc rim radius	0.2367 [m]
C_p	Nondimensional pressure	$\frac{\langle p\rangle - \langle p_1\rangle}{0.5\rho\Omega^2 b^2}$ [-]
C_s	Smagorinsky model coefficient	0.08 [-]
C_w	Nondimensional flow rate	$\frac{\dot{m}_c}{\mu b}$ [-]
f	Frequency	[Hz]
L	Length of pipe	$15R$ [m]
$\dot{m}_c$	Coolant mass flow rate	[kg/s]
N	Number of unsteady flow structures in the entire annulus	[-]
p	Static pressure	[Pa]
R	Radius of pipe	4.17×10^{-3} [m]
r	Radius	[m]
Re_b	Reynolds number based on pipe diameter and bulk velocity	$\frac{\rho u_b 2R}{\mu}$ [-]
Re_ϕ	Rotational Reynolds number	$\frac{\rho\Omega b^2}{\mu}$ [-]
Re_τ	Reynolds number based on pipe radius and friction velocity	$\frac{\rho u_\tau R}{\mu}$ [-]
u_b	Pipe flow bulk velocity	[m/s]
u_s, u_n, u_θ	Streamwise, wall-normal and tangential velocity components in rim seal coordinates	[m/s]
u_τ	Friction velocity	[m/s]
u_z	Pipe flow streamwise velocity	[m/s]
x	Axial coordinate	[m]

Greek symbols

α	Angle between two pressure sensors	[rad]
β	Angle between two adjacent flow structures	[-]
$\Delta(r\theta)^+$	Nondimensional distance in the azimuthal direction in wall units	[-]
Δt_α	Time lag of pressure signals between two azimuthal pressure probes	[s]
Δt_β	Time lag between two adjacent flow structures	[s]
Δx^+	Nondimensional distance in x direction in wall units	[-]
Δy^+	Nondimensional wall distance in wall units	[-]
ϵ	Weighting coefficient of numerical viscosity in Roe scheme	0.005 [-]
γ	Angle of flow structure to radial direction	[rad]
μ	Molecular viscosity	1.81×10^{-5} [kg/(ms)]
Ω	Rotor's angular speed	$\frac{2\pi \times 7000}{60}$ [rad/s]
ω_s	Speed of unsteady flow structures	[rad/s]
ω_θ	Circumferential vorticity component	[s^{-1}]
ρ	Density	[kg/m^3]
θ	Circumferential angle	[rad]
ξ	Random number	$\xi \in [-1, 1]$

Superscripts

$+$	Nondimensional quantity
$'$	Fluctuating component

Other symbols

$\langle\cdot\rangle$	Ensemble average

Acronyms

CFD	Computational fluid dynamics
CFL	Courant–Friedrichs–Lewy condition
LES	Large-eddy simulation
OPLUS	Oxford parallel library for unstructured solver
ORF	Oxford rotor facility
PSD	Power spectral density
RANS	Reynolds-averaged Navier–Stokes
SGS	Subgrid-scale
URANS	Unsteady RANS

The above studies relate to axial clearance seals. Boudet et al. (2005) reported URANS solutions for a chute seal geometry. These revealed that the flows in this configuration were also inherently 3-D and unsteady. The authors attributed this phenomenon to the possible Taylor–Couette instability in the seal. O'Mahoney et al. (2011) extended the turbine stage URANS study of Boudet et al. to LES, and showed closer agreement of sealing effectiveness with the experiment of Gentilhomme (2004).

Other researchers have also confirmed the presence of non-blade passing related unsteadiness associated with rim seals, using both computational fluid dynamics (CFD) and through experiment. Schuepbach et al. (2010) claimed that the asymmetric pressure field induced by the large-scale flow features can significantly reduce engine performance. Chilla et al. (2013) reported strong unsteady flow interaction between the rim seal and the main gas path at nominal sealing flow conditions, and periodically vortex shedding from rim seal into the main annulus. Rabs et al. (2009) identified similar vortex structures and conjectured that they could be induced by the Kelvin–Helmholtz instabilities. The latest investigations have all experimentally confirmed the existence of rim seal cavity modes which are unattributed to the blade passing. Amongst recent experimental studies are papers by Beard et al. (2016), Savov et al. (2016), and Schädler et al. (2016). The study by Beard et al. revealed, for the first time, the speed and number of flow structures independently of CFD solutions.

A number of researchers have developed "orifice models" for rim sealing flows. These models estimate the inflow and the outflow through the seal, taking account of the pressure asymmetry in the main annulus. A recent example of this type of model is that developed at the University of Bath (Scobie et al., 2016). With appropriate choice of model parameters it shows a good scaling capability in correlating measured sealing effectiveness for various seal configurations. Hills et al. (2001) developed an orifice model to consider the pressure asymmetries due to both blades and vanes, with inclusion of a term to account for inertial effects. This showed the significance of unsteady flow effects but does not represent the effects of the low frequency unsteadiness discussed above. The unsteady RANS model of Boudet et al. (2005), which shows inherent unsteady flow features of the rim seal flow, achieves significant improvement in agreement with measured sealing effectiveness to the steady RANS. Thus, it can be conjectured that correct modelling of unsteady rim sealing flow structures is essential for the accurate prediction of sealing effectiveness, and that current design methods do not capture some important flow physics.

From recent publications it is clear that the detailed flow physics in rim seals is of considerable interest, with a need for better understanding of the underlying flow mechanisms. The present study focuses on the inherent unsteadiness involved in the rim seal, and considers CFD modelling of the chute rim seal geometry published in