Computational and Experimental Assessment of Rim Sealing Flows in Axial Turbine Chute Rim Seals

Donato Maria Palermo¹

University of Surrey, Guildford, Surrey, GU27XH, UK

Anna Bru-Revert²

Rolls-Royce plc, Bristol, BS347QE, UK

Feng Gao³, Dario Amirante⁴ and John W. Chew⁵

University of Surrey, Guildford, Surrey, GU27XH, UK

Paul F. Beard⁶

University of Oxford, Oxford, Oxfordshire, OX20ES, UK

Hot gas ingestion in a chute seal configuration has been numerically and experimentally investigated. First, the NGVs were included to generate a swirled annulus flow. This staged approach to rig configuration is used to understand the relative contribution of rotation and NGVs induced pressure asymmetries in turbine rim sealing flow ingestion.

Computations were performed with a URANS model and a novel LES code including near wall boundary layer modelling, the wall-modelled LES (WMLES). The experimental data was obtained in the Oxford Rotor Facility (ORF). The main focus of this study was the mean cavity flow aerodynamics and the sealing performance of the chute seal under a range of operating conditions. These are studied through measurements of pressure and gas concentration within the rotor-stator disc cavity and the rim seal which are compared to CFD predictions. In addition, experimental and CFD assessments of concentration-based sealing effectiveness in the gas path for the vaned configuration are presented. Measurements were taken at a representative rotor leading edge axial position to focus on the radial diffusion and interaction between the purge flow and annulus flow. Overall the WMLES code better captured the interaction between main annulus and turbine cavity flow positioning it as a powerful tool with potential to be implemented in the design and verification process.

¹ PhD student, Department of Mechanical Engineering Sciences

² Thermal & Fluid Systems Engineer, Future Programs

³ Research Fellow, Department of Mechanical Engineering Sciences

⁴ Research Fellow II, Department of Mechanical Engineering Sciences

⁵ Full Professor, Department of Mechanical Engineering Sciences

⁶ Senior Researcher, Oxford Thermofluids Institute

I. NOMENCLATURE

Roman symbols

A_n	Roe matrix
b	Disc rim radius
C_p	Pressure coefficient = $\frac{ p - \langle p \rangle_\theta }{0.5\rho(\Omega r)^2}$
$C_{p,a}$	Pressure annulus coefficient = $\frac{p_{01} - p}{p_{01} - p_2}$
d	Diameter
D	Artificial viscosity coefficient
F	Flux
f/Ω	Flow modes' frequency normalised by disc frequency
Q	Primitive flow variables
H	Tensor Flux
K^*	Non-dimensional laminar diffusion coefficient
$\dot{m}$	Mass flow rate
Ma	Mach number
p	pressure
Pr	Prandtl number
r	Local radius
r^*	Non-dimensional radius ($=r/b$)
$r_{m,i}$	Mid radius of the inner seal inlet
Re_θ	Rotational Reynolds number ($=\rho\Omega b^2/\mu$)
Re_x	Annulus axial Reynolds number ($=\rho U_{ax} b/\mu$)
s_c	Seal clearance
S_{ct}	Turbulent Schmidt number ($=0.7$)
T	Temperature
u	velocity
u_τ	Wall friction velocity ($=\sqrt{\tau_w/\rho}$)
u^+	Non-dimensional velocity ($=u/u_\tau$)
$v_{\theta,i}$	Inner seal inlet swirl velocity ($=0.5\Omega r_{m,i}$)
U_e	Mean annulus axial velocity
U_m	Mean velocity through the seal ($=\dot{m}/(2\pi\rho b s_c)$)
x^*	Non-dimensional axial distance from the stator surface ($=(x - x_{stator})/(x_{rotor} - x_{stator})$)
y^+	Non-dimensional wall distance ($=\rho u_\tau \Delta y/\mu$)

Greek symbols

χ_a	Species concentration at main annulus inlet
χ_i	Species concentration at coolant inlet
Δ	Difference
ε	Sealing effectiveness ($=(\chi - \chi_a)/(\chi_i - \chi_a)$)
μ	Dynamic viscosity
μ_w	Effective dynamic viscosity from wall function
ν	Kinematic viscosity
ω_s/Ω	Speed of flow modes normalised with the rotor speed
Ω	Rotor disc Angular speed
ρ	Density
τ_w	Wall shear stress estimated with wall function ($=(\mu_w + \mu)u/\Delta y$)
θ	Angle or quantity evaluated in the tangential direction

Subscripts

c	convective
---	------------

L	left
n	normal direction
R	right
x	axial component
ν	viscous
0	initial condition

Abbreviations

CFD	Computational fluid dynamics
CO ₂	Carbon dioxide
FFT	Fast Fourier transform
K-H	Kelvin-Helmoltz
LES	Large eddy simulation
ORF	Oxford Rotor Facility
NGV	Nozzle guide vane
RANS	Reynolds-averaged Navier-Stokes
RBPF	Rotor blade passing frequency
SAS	Secondary air system
SGS	Sub-grid scale
TET	Turbine entry temperature
URANS	Unsteady RANS
UTC	University technology centre

II. INTRODUCTION

Over the years, common practice has targeted improvements in efficiency and specific power output of gas turbines through gradual increases of the turbine entry temperature (TET). Indeed, Kyprianidis (2011) reported that the TET had been progressively raised at a rate of approximately 3K per annum in the previous 40 years. More efficient engines have been possible thanks to advances in cooling technologies as well as the development of temperature capable metal alloys for turbine stages, the most thermally stressed aircraft engine components. In fact, without adequate cooling systems, a higher TET could overheat turbine blades and discs and reduce the life of the hot turbine engine components. In order to prevent the turbine disks from overheating, cold flow is extracted from the compressor and guided through a secondary air system (SAS) to the turbine. The secondary air serves several functions – it supplies cooling flow to the turbine blades and provides the purge air that seals the turbine disc cavities isolating them from the hot turbine gas. Although purging the turbine disc cavity penalizes the overall efficiency of the system, it is required to ensure the operation of the engine. Therefore a compromise between maximising efficiency through increases in TET whilst minimizing the purge flow must be sought. Indeed, Moore (1975) estimated that saving 1% of sealing purge flow could reduce the specific fuel consumption of an engine by 0.4%. In fact, the mixing between the main annulus flow and rim seal purge flow generates high viscous shear in the interface between the hot and cold streams. This is a consequence of deficit in swirl between the two flows. Shearing also occurs in wakes, leakage jets and within corner separated regions. This is followed by entropy generation that translates into pressure losses in the fluid flow. The flow phenomena involved in these processes are generally unsteady, three-dimensional and therefore quantitative estimations of losses are difficult. A wide review on losses in turbomachinery flows and their related estimation by means of entropy generation can be found in the work of Denton (1993).

Early reports of unsteady flow structures in turbine rim seals were given by Smout *et al.* (2002) and Cao *et al.* (2004) and many recent research studies have confirmed these effects. To the authors knowledge, Rabs *et al.* (2009) were the first to attribute such flow structures to Kelvin-Helmholtz (K-H) instabilities in rim seal flows. They claimed that coherent vortex structures observed in URANS studies were due to the shear layer associated with the deficit in swirl between the mainstream and purge flow. Although the presence of vanes and blades almost suppressed these instabilities, they did occur at the highest purge rate tested. Chilla *et al.* (2013) discussed the formation of these instabilities and found from URANS calculations that they are sensitive to the sealing mass flow and its swirl. This study also reported that the unsteadiness within the rim seal region is shed into the rotor hub endwall at a frequency below the blade passing frequency, and could

lead to increased losses within the turbine stage. Schädler *et al.* (2016) employed a fast response aerodynamic probe in their experiment reporting that cavity induced instabilities migrate into the main annulus up to 30% of blade span leading to aerodynamic losses and noise generation. Spectral plots for three radial positions at two seal purge flow rates detected three distinct frequencies for the lower sealing flow rate. Two of them are related to the rotor blade passing frequency (RBPF) whilst the third frequency was found to be in the range 25-35% of the RBPF. Despite being lower in frequency, the peak energy content of the latter was found to be of the same order of magnitude. Savov and Atkins (2016), studied the performance of two rim seal geometries (single and double rim seals) varying the dimensionless seal purge flow rate, for configurations with and without blades. Spectrograms reported in this study were computed from Fast Fourier Transforms (FFT) of the unsteady pressure data recorded over 60 revolutions. The RBPF was detected at $f/\Omega \sim 96$, and in addition to it, increased unsteady activity in the rim seal for the unbladed cases was measured in both rim seal geometries at 25-35 f/Ω and 30-40 f/Ω respectively. Moreover, they found that the unsteady phenomena was enhanced at high flow rates with a widened band of excited frequencies. The authors related the band of unsteady activity in the rim seal region to a K-H type shear layer instability driven by swirl velocity difference between the purge and mainstream flows. The authors also pointed out that the spectral activity increased at higher flow coefficients, when greater tangential velocity difference (and higher shear) exists in the rim seal. Furthermore, the activity registered in the rim seal cavity generally did not appear in the inner rotor-stator cavity signal. Iranidokht *et al.* (2020) reported experimental evidence of cavity modes and their sensitivities to turbine axial gap and turbine pressure ratio. An increased rotor-stator axial gap is found to damp the peak amplitude of cavity modes by approximately a factor of seven. An increase of the turbine pressure ratio is found to increase the band power of the cavity frequency spectra. Hence, it is argued by the authors that cavity modes are linked to the variations of the flow tangential velocities in the annulus-cavity interaction zone. Further discussion of the flow mechanisms in turbine rim seals is given in a review by Chew *et al.* (2019) and an interpretation of sealing flows unsteadiness in terms of inertial waves is given by Gao *et al.* (2020).

Experimental ingress measurements using the tracer gas technique was pioneered in the 1980s. For example, Graber *et al.* (1987) reported an experimental study of ingress in a rotor-stator test rig with an external annulus with bladeless stator and rotor discs. The purge flow was seeded with CO₂ as a tracer gas and introduced into the rotor-stator cavity whilst the main annulus flow was unseeded. Gas was sampled from the stator wall to determine how much unseeded air had been ingested from the annulus into the turbine cavity. In a more recent example, Scobie *et al.* (2016) assessed the re-ingestion of purge flow in a 1.5-stage gas turbine rig. They claimed that the purge flow from the upstream cavity did not remarkably affect the fluid dynamics of the downstream seal as it partially mixed with the annulus flow near the downstream seal. However, there is no information about the pitchwise egress distribution over the annulus in this study.

Ingress modelling in axial turbine rim seals is a trending topic in turbomachinery and is currently the subject of several joint numerical and experimental campaigns. Numerical studies have been conducted with both commercial (Ansys Fluent/CFX) and also in house codes (HYDRA, DLR-TRACE) that use second order accurate discretisation schemes. Notably, Horwood *et al.* (2019) obtained good agreement between experimental and numerical results of sealing effectiveness for a 1.5 stage turbine rig with a chute seal fitted at the periphery of the cavity. They used the DLR-TRACE second-order scheme accurate solver with URANS K-Omega SST turbulence model. However, the same authors obtained dissimilar experimental and computational results in a previous radial seal configuration, Horwood *et al.* (2018). The authors hypothesised that the turbulence model used failed to capture the flow separation within the rim seal region.

The design of turbine rim seal systems strongly relies on empirical models and extrapolation of experimental rig data to engine conditions. Computational codes can provide predictions at a fraction of the cost of an experimental campaign positioning CFD as a very valuable design tool. No established CFD methodologies able to accurately predict the sealing performance of axial turbine rim seals are currently available. Unless robust computational codes are developed, empirical correlations based on steady state data will continue to be the main design tool. The challenge with the numerical approach is to achieve the right balance between accuracy and practicability. RANS-legacy models consistently underpredict ingress in axial turbines and do not capture the unsteady effects. Unsteady RANS (URANS) studies have improved the understanding of the unsteady phenomena developing in the rim seal although still often overestimated the sealing effectiveness. Explicit wall resolved LES are still unfeasible from the modelling and computational

points of view for commercial use at engine realistic conditions. Therefore, the development of a methodology with the right balance between predictive capability and affordability is required to speed up the verification and exploitation of numerical tools in the future gas turbine engine designs. The Wall-Modelled LES (WMLES) introduced by Palermo *et al.* (2021) showed encouraging results that improved the rim seal performance predictions. Utilisation of the WMLES with the implementation of the passive scalar equation is considered by the authors to be a promising avenue not only for capturing the unsteady flow phenomena but also to quantify ingress in axial turbine rim sealing flows at representative engine test conditions.

In this paper we present further computational and experimental assessments of chute rim sealing flows to evaluate the mean cavity flow aerodynamics and highlight the interaction between the main annulus and cavity flows. A brief introduction of the state-of-the-art in computational tools used in the field of turbine rim seals research has been given above. The methodology followed in this study is detailed next, covering both the numerical and experimental approaches. Section 3 describes the mean flow and sealing performance in the cavity volume followed by an assessment of the flow field in the annulus in section 4. The interaction between the purge and the annulus flows is evaluated through the instantaneous flow field, the mean flow field and sealing effectiveness.

III. METHODOLOGY

A. NUMERICAL APPROACH

The Rolls-Royce proprietary HYDRA CFD solver is based on a finite volume method and cell-vertex based scheme as seen in Fig 1a) The control volume is defined by the median dual cell approach described by Moinier (1999). The application of boundary conditions is straightforward for cell vertex schemes as the flow variables are known along the boundaries, as opposed to cell-centred schemes where an extrapolation is involved. In addition, cell-vertex schemes allow better resolution of the flow field close to the wall. The initial HYDRA code has been modified at the University of Surrey to increase the accuracy of the computed flow variables through an improved second order accuracy discretisation scheme.

Fig 1. Two-dimensional geometry representation of the median-dual around each mesh node and corresponding Gauss integration points. (a) Production version of HYDRA and (b) Modified Surrey UTC version of HYDRA

In the modified HYDRA solver, known as HYDRA5, the primitive flow variables Q are linearly reconstructed with second order accuracy, through Taylor expansion series including the first derivatives only. This is illustrated in Fig 1b) The gradients are computed with the Green-Gauss methodology described through Eq. 1-4. As the reconstruction of the primitive variables on both sides of the cell interface is not equal, the flux tensor F on each Gauss integration point is computed twice, making the code slower compared to the standard second order implementation. The convective flux F^c is approximated using the

Riemann solver of Roe (1981) and consists of an inviscid and a dissipative contribution. The viscous fluxes are obtained averaging the two reconstructed gradients at the interface.

$$\mathbf{F} \cdot \vec{n} = (\mathbf{F}^c + \mathbf{F}^v) \cdot \vec{n} \sim \mathbf{H}^I \cdot \vec{n} + \mathbf{H}_n^D + \mathbf{H}^v \cdot \vec{n} \quad 1)$$

where

$$\mathbf{H}^I \cdot \vec{n} = \frac{1}{2} (\mathbf{F}_L^c + \mathbf{F}_R^c) \cdot \vec{n} \quad 2)$$

$$\mathbf{H}_n^D = -D \frac{1}{2} |A_n| (Q_R - Q_L) \cdot \vec{n} \quad 3)$$

$$\mathbf{H}^v \cdot \vec{n} = -\mathbf{F}^v (\nabla Q_{RL}) \cdot \vec{n} \quad 4)$$

in which R, L denote the right and left position of the integration point considered ; A_n is the Roe matrix associated with the normal direction $\vec{n}$; and $\mathbf{H}^I$, $\mathbf{H}_n^D$, $\mathbf{H}^v$, represent the inviscid, dissipative and viscous flux tensors respectively. The resulting scheme has a leading term of the truncation error proportional to the third spatial derivative which makes it predominantly dispersive. The numerical dissipation corresponds to an error with a third-order leading term. In Eq. 3), the upwinding parameter D can be used to calibrate the dissipative flux, depending on the mesh resolution adopted. Interestingly, a second order discretisation without numerical dissipation (on a uniform mesh) can be obtained imposing $D = 0$. However, an upwind contribution is always required to resolve the discontinuity at the cell interface and to preserve the stability. Thus, the upwind parameter D is reduced to the minimum value necessary to preserve the stability of the computation.

To improve rim sealing ingress estimation, the WMLES and the convection-diffusion equation for a passive scalar have been implemented in the modified version of HYDRA5 as documented by Palermo *et al.* (2021). With WMLES an explicit time-stepping approach is computationally feasible. Further, the WMLES allows time-step and data communication savings owing to utilisation of coarser meshes compared to wall resolved LES. Moreover, this method is highly attractive for mixing flow problems, making it suitable to study effect of sealing flows in the main annulus gas path. Conversely, WMLES inherently models the near wall flow behaviour, thus a full resolution of the flow close to the wall is neglected. Large grid cells near wall may delay turbulence formation close to the wall with potential spurious oscillation in the near wall area as reported by Sagaut (2005).

B. CODE VERIFICATION

The Taylor-Green vortex, described by Taylor and Green (1937), is an unsteady flow of a decaying vortex and is often used to test the accuracy of CFD codes. The flow is governed by the Navier-Stokes equations and bounded in a 2π cubic periodic domain as shown in Fig. 2. Here the reference Mach number is $Ma = 0.1$ and the reference Reynolds number is 1600. In addition, the Prandtl number Pr is set to 0.71, the ratio of the specific heats $\gamma = 1.4$ and the overall flow field is initialized with a static temperature $T = 300$ K. and density $\rho = 1 \text{ kg/m}^3$.

Fig. 2 Taylor-Green Vortex, u-velocity initialization component

The evolution of the total kinetic energy E_k and total enstrophy ζ (see Eq. 5 and Eq. 6) obtained with the second order version of HYDRA5 was compared against a benchmark solution on a 64^3 and 256^3 hexa-elements.

Total kinetic energy

$$E_k = \frac{1}{\rho V} \int_V \frac{1}{2} \rho \vec{u} \cdot \vec{u} dV \quad (5)$$

where $\vec{u}$ is the velocity vector and V the control volume of integration.

Total enstrophy

$$\zeta = \frac{1}{\rho V} \int_V \frac{1}{2} \rho (\nabla \times \vec{u}) \cdot (\nabla \times \vec{u}) dV \quad (6)$$

Fig. 3 Total Kinetic Energy a) and enstrophy b) comparison between HYDRA and other CFD codes. ASTR data available from Fang *et al.* (2019).

Using the results of a high-order code ASTR as reference, the results of HYDRA5 (modified version available in Surrey UTC) are compared with those of several 2nd order codes, in Fig. 3. HYDRA5 results with the 256³ element mesh offer an overall good agreement .

C. EXPERIMENTAL APPROACH

A brief overview of the experimental set up used in the Oxford Rotor Facility (ORF) is presented in this section. The ORF was initially designed for research of the transonic high pressure turbines blading flows at engine representative conditions by Ainsworth *et al.* (1988). Beard *et al.* (2017) upgraded its capability to study the secondary air system with focus on turbine stator-rotor cavities. Recently, Bru-Revert *et al.* (2021) redesigned the air feed supply system to allow for steady state long duration tests in a reduced span gas path to investigate the sealing performance of the chute rim seal arrangement of the original ORF. The modularity of the modified ORF and the refinement of the cavity instrumentation enabled the study of different mechanisms of ingestion through vaneless and vaned gas path configurations. The vaned ORF set up included 36 vanes with an axial chord of 34 mm and radial span of 7.5 mm at the trailing edge. The axial distance between the vane trailing edge and the rim seal was 9.4% of the vane axial chord. NGVs were mounted on the stator side of main gas path through 36 slots present in the casing ring. The geometry and boundary conditions were agreed between the numerical and experimental team beforehand to allow for comparison.

Static steady pressure measurements in the casing wall above the seal ($r/b = 1.03$) as depicted in Fig. 4, were obtained by First Sensor BTEM5000 pneumatic unidirectional differential pressure transducers with a nominal uncertainty of $\pm 0.2\%$ of the full-scale output. Note that pressure measurements were referenced to the lowest radial data point in the cavity. Gas concentration measurements were taken inside the stator-rotor cavity at 9 radial positions, at the entry to the test section and in the main gas path in order to derive the radial profile of sealing effectiveness. A more detailed description of the cavity radial measurements for the ORF is given by Bru-Revert *et al.* (2021).

Fig. 4 a) Pressure and temperature measurement points in the working section of the vaneless reduced annulus, b) CAD assembly of the annulus measurement probe installed in the working section and c) downstream view of the annulus measurement probe installed in the ORF

A further test campaign aimed at understanding the behaviour of the sealing flow once it has left the cavity was also conducted without rotor blades but with the dummy vanes installed. The extent of the sealing flow migration in the main gas path was studied at an axial position representative of the rotor blade leading edge. The annulus gas concentration measurements were taken with a probe formed by three $d = 1$ mm scanivalve tubes radially spaced by 2 mm to cover the reduced annulus height, see Fig. 4b). The annulus probe was

positioned in five different circumferential positions in the annulus extending over one NGV pitch as depicted in Fig. 4c). As a result, a traverse of sealing effectiveness data was obtained at 15 points (3 radial positions at 5 circumferential locations) so that the effect of the pressure asymmetries in the radial migration of the purge flow into the annulus could be investigated.

IV. CAVITY FLOW

The flow behavior inside the rotor-stator cavity is studied in this section based on radial profiles of cavity pressure coefficient, swirl velocity ratio and sealing effectiveness. First, the mean cavity flow is studied for the rig configuration without annulus flow and without vanes to study the effect of rotationally-induced ingestion at various purge flow rates. Secondly, the vaned gas path configuration is studied to report the effect of pressure-induced ingestion at various rotor disc speeds. Results from the numerical investigations and experimental tests are compared to discuss the ability to predict the flow field of several URANS and WMLES.

A. MEAN FLOW

Radial profiles of cavity pressure coefficient C_p obtained on the stator wall and swirl velocity ratio in the middle of the cavity are shown for the vaneless configuration in and for the vaned gas path in Fig. 6. Experimental and computational results of cavity pressure coefficient are compared at different seal purge flow rates normalized by rotor disc speeds with WMLES data in a) and b) and Fig. 6a) and Fig. 6b) whilst URANS results are shown in c) and d) and Fig. 6c) and Fig. 6d).

The cavity pressure coefficient is obtained from time-averaged and circumferentially-averaged pressures referenced to the measurement point located furthest inboard in the cavity. The rotor disc dynamic pressure at the hub was used to non-dimensionalise the pressure. CFD results are compared with the experimental data available from Bru-Revert *et al.* (2021). Experimental results reported for the vaneless configuration in were obtained for a non-dimensional sealing flow ratio of $U_m/(\Omega b) = 0.04$, and rotational Reynolds number $Re_\theta \sim 3.3E+06$. The numerical data are well aligned with the experimental results at these conditions with interpretation of the experimental data indicating the presence of two coexisting vortices: a core vortex rotating at $\sim 43\%$ of the rotor disc speed in the outer part of the cavity and an inner cavity vortex rotating at $\sim 34\%$ of the disc rotor speed. Uncertainties in inlet swirl could account for some differences between CFD and measurements. As the purge flow rate increases, the strength of the cavity vortices decreases, suggesting that vortex flow features are principally influenced by purge flow rate and disk rotor speed, as might be expected for a cavity with weak rotationally-induced ingress.

The experimental conditions for the vaned configuration shown in Fig.5c) correspond to a sealing-to-annulus velocity ratio $U_m/U_e = 0.08$ and $Re_\theta \sim 3.3E+06$. In this case, a core vortex rotating at $\sim 50\%$ of the rotor disc speed is observed in the outer part of the cavity. The inner cavity vortex rotates at about $\sim 40\%$ of the rotor disc speed. The CFD indicates that the strength of the inner cavity vortex is invariant with the rotor disc speed, in agreement with Bru-Revert *et al.*'s findings (2021). However, according to the WMLES, as the rotation increases, the strength of the upper cavity vortex changes in a non-monotonic manner, suggesting that vortex flow features in the upper part of the cavity are influenced by ingestion and rotation (as will be discussed later). This may be associated with stronger ingestion than for the vaneless cases and the combined effects of rotationally-induced and pressure-induced ingress.

In the inner part of the cavity, C_p and swirl velocity predictions by WMLES and URANS are relatively similar for both the vaneless and vaned configuration. In contrast, in the outer part of the cavity, remarkable differences in the computations of the velocity ratio by WMLES and URANS can be observed. This is linked to the fact that WMLES predict more ingress than the URANS simulations, as will be shown later, and therefore the additional momentum coming from the ingress of swirled annulus flow in the vaned configuration makes the flow within the outer part of the cavity rotate faster.

Fig. 5 Radial profiles of the vaneless configuration showing a) C_p with WMLES, b) swirl velocity ratio with WMLES, c) C_p with URANS and d) swirl velocity ratio with URANS

Fig. 6 Radial profiles of the vaned configuration showing a) C_p with WMLES, b) swirl velocity ratio with WMLES, c) C_p with URANS and d) swirl velocity ratio with URANS

B. SEALING PERFORMANCE

Numerical and experimental results for sealing effectiveness are compared in Fig. 7 for the configuration without vanes. Data obtained at $r/b = 0.96$ with the passive scalar technique with URANS and the WMLES are compared to the ORF experimental values and the disc pumping orifice model derived by Chew (1991) in Fig. 7a). The sealing effectiveness is given by Equation 6, where the parameter k indicates the ability of each specific rim seal geometry to prevent ingestion. Bru Revert *et al.* (2021) showed that for the chute seal under investigation, the orifice model best fitted the experimental data with $k = 1$.

$$\varepsilon = \frac{1}{0.8 + 0.02428 k \frac{\Omega b}{U_m}} \quad (6)$$

This equation for the estimation of the sealing effectiveness ε is valid for $0 < U_m < U_{m,min}$ where $U_{m,min}$ is the minimum seal purge flow velocity required to seal the cavity. The sealing effectiveness values obtained with the URANS model show a significant quantitative mismatch with the experimental results, although they capture the trend that Eq. 6 would provide for a parameter $k = 0.4$. This indicates that the ability to seal the cavity is overpredicted, with an increasing departure from the experimental data as the purge flow ratio, $U_m/(\Omega b)$, increases. The results of the WMLES, given as red data points in Fig. 7a), reveal a much improved agreement with the ORF measurements. This is especially the case at $U_m/(\Omega b) = 0.04$, although the misalignment increases with higher purge supply. The WMLES data shown by black triangles represent a pre-swirled purge flow in which the inlet swirl velocity is equal to $0.5\Omega r_{m,i}$, with $r_{m,i}$ denoting the inner purge flow radius location. Fig. 7a) reveals closer agreement with the experimental data for the pre-swirled purge flow.

Fig. 7 URANS and WMLES passive scalar results compared against experimental data at $Re_\theta \sim 3E+06$ for (a) the vaneless configuration at various sealing purge flow rate and (b) the vaneless configuration at $U_m/(\Omega b)=0.04$, $U_m/U_e=0.08$

The cavity radial profile of sealing effectiveness for the vaned gas path configuration is shown in Fig. 7b) where similar observations to those described above still hold. The URANS model overpredicts the sealing effectiveness across the span of the cavity, unable to reproduce the variations in the radial direction. Both pre-swirled and unswirled purge flows simulated with WMLES show significantly improved estimations which capture the radial variations of sealing effectiveness detected in the experimental data. The increased level of ingress predicted by WMLES for the pre-swirled purge flow configuration compared to the unswirled purge flow configuration is directly linked to the presence of flow unsteadiness in the pre-swirled purge flow configuration. These are shown in the frequency spectra presented in Fig. 8 in the form of spikes. It is conjectured that the pre-swirled purge flow configuration increases the strength of the shear layer originated by the interaction of main annulus and cavity flow.

It is evident that both URANS and WMLES underpredict the ingestion of mainstream flow into the cavity, thereby estimating higher sealing capability than revealed by the experiments. Nonetheless, the improved predictive capability observed with the WMLES positions this as a powerful tool to provide more accurate estimations of hot gas ingestion during the design process. Despite the quantitative mismatch observed with the URANS model, it may be noted that the model gave similar qualitative trends to the experimental and analytical results.

A. FLOW UNSTEADINESS

The FFT pressure plots recorded in the upper part of the cavity are shown in Fig. 8 and discussed below. The frequency spectra from WMLES data with an inlet swirl reveal the presence of unsteady flow structures in the rim seal. For these purge flow rates, $U_m/(\Omega b) > 0.06$, the results without inlet swirl are not reported as they show no flow unsteadiness in the seal. This confirms the presence of a strong link between

Fig. 8 WMLES Frequency spectra recorded on the upper part of cavity at 5 different radial positions for the configuration without vanes, with purge inlet swirl at a) $\frac{U_m}{\Omega b} = 0.06$ and b) $\frac{U_m}{\Omega b} = 0.08$

unsteadiness and ingestion for the chute seal with higher rim seal unsteadiness peak magnitude generally indicating worse sealing performance. Interestingly, in Fig. 7a) the WMLES test case with inlet swirl at $U_m/(\Omega b) = 0.08$ predicts a slightly better rim sealing performance compared to the WMLES test case with no purge inlet swirl at $U_m/(\Omega b) = 0.06$ despite the presence of a subtle unsteadiness in Fig. 8. It is possible that the addition of inlet swirl in the purge flows increases the strength of the velocity difference and shear between the main annulus and cavity flow and this generates the unsteadiness within the rim seal. This is in agreement with the findings by Savov and Atkins (2016) for their unbladed rotor-stator test case configuration. Conversely, for the vaned configuration, the addition of the inlet swirl does not have an effect on sealing effectiveness performance and in the overall unsteadiness content within the rim sealing region as found by Palermo (2020). This may be because the swirling the purge flow on the vaned configuration does not increase the velocity differential and shear within the rim seal region as the annulus flow is also having a swirl velocity component produced by the NGVs.

V. GAS PATH FLOW

The flow field in the main gas path is investigated in this section by examining the instantaneous and mean flow contours from the WMLES. In addition, sealing effectiveness values from the WMLES and the experiments in the ORF are compared to examine the ability of the WMLES to capture the interaction of the purge flow with the annulus flow once it has left the rim seal.

A. INSTANTANEOUS FLOW

The flow structure around the NGV can be discussed through visualization of flow vortices. A full review on the flow vortex identification methods, with their relative strengths and weaknesses, is given by Jeong

and Hussain (1995) and more recently by Tucker (2016). Methods using gradients of the velocity tensor such as the Q-criterion are suitable. The contours shown in Fig. correspond to a simulation with the WMLES where a positive instantaneous Q-criterion iso-surface around the suction side of the NGV obtained for $U_m/U_e=0.08$, $U_m/(\Omega b)=0.09$, $Re_\theta \sim 3E+06$ can be observed. The contours are coloured based on values of Mach number.

Fig. 9 Contours of Q-criterion = 1 iso-surface around the NGV suction side obtained with the WMLES at $U_m/U_e=0.04$, $U_m/(\Omega b)=0.09$, $Re_\theta \sim 3E+06$ coloured by Mach number

The area-weighted average annulus flow speed at the main gas path inlet far upstream of the CFD domain for the vaneless and vaned configurations are approximately 1.22 m/s and 7.80 m/s respectively. No turbulence at the inlet is added in the WMLES simulations, hence the flow would start laminar. A clear laminar to turbulent transition can be seen around the suction side of the NGV. This is triggered near mid-chord around the suction side of the NGV hub. Indeed, near the platform, the flow experiences the highest pressure coefficient which leads the annulus flow to sonic conditions. The mean flow remains attached to the NGV mid-chord, becoming fully turbulent close to the NGV trailing edge. It is believed that the development of the boundary layer downstream of the NGV and the presence of turbulent fluctuations in the wake affect the flow structures present in the rim seal region. This would be reflected in the species concentration of the purge and annulus flows mixing in the rim seal region as well as through the radial migration and diffusion at the rotor blade leading edge position (with no blades).

B. MEAN FLOW

Time-averaged annulus pressure coefficient $C_{p,a}$ around the suction side of the NGV and flow Mach number at midspan for the WMLES at $U_m/U_e=0.08$, $U_m/(\Omega b)=0.04$ and $Re_\theta \sim 3.3E+06$ is reported in **Fig. .** Here $C_{p,a}$ is based on the pressures in the main gas path and defined as in Eq. 5

$$C_{p,a} = \frac{p_{01} - p}{p_{01} - p_2} \quad 5)$$

where p_{01} is the circumferentially-averaged total pressure evaluated one axial chord upstream of the NGV leading edge, p_2 is the circumferentially-averaged static pressure evaluated at 0.25 axial chords downstream of the NGV trailing edge and p is the static pressure evaluated on the NGV surface.

Fig. 10 Time-averaged contours from the WMLES for the vaned configuration at $U_m/U_e=0.08$, $U_m/(\Omega b)=0.04$, $Re_0 \sim 3.3E+06$ showing a) pressure coefficient on the NGV suction side and b) absolute Mach number at a midspan cross-section of an NGV passage

Interestingly, the $C_{p,a}$ contours around the suction side of the NGV show a non-monotonic trend that highlights two pressure peaks corresponding to two distinct acceleration phases of the annulus flow around the NGV. Fig. 10b) reveals that these peaks of $C_{p,a}$ correspond to the transition of the annulus flow to sonic condition around mid-chord of the NGV. Fig. 11 reports time-averaged streak-lines showing that the flow around the NGV is attached at the trailing edge, which hints that the sonic annulus flow after the mid-chord is followed by a smooth compression without a remarkable flow separation. Therefore, in this test case, the presence of strong shockwaves within the main annulus is excluded. In addition, it is conjectured that the first NGV flow acceleration to sonic conditions is mainly due to the abrupt change of NGV suction side curvature.

Fig. 11 Time-averaged streak-lines for the vaned configuration at midspan, WMLES, $U_m/U_e=0.08$, $U_m/(\Omega b)=0.04$, $Re_0 \sim 3.3E+06$

C. CONCENTRATION MEASUREMENTS

An engine-representative purge-to-annulus mass flow ratio of 1.2% of the flow for the reduced annulus was selected to study the radial migration of the purge flow in the annulus at the rotor leading edge position. Contours of sealing effectiveness on the measurement plane are shown in Fig. 12. At the closest measurement position to the hub, $r/b = 1.01$, traces of CO_2 from the sealing flow are clearly detected whilst there is negligible penetration of the purge flow into the casing wall. The concentration of tracer gas strongly depends on the position of the probe respect to the NGV trailing edge, indicating strong dependency on the circumferential pressure variations induced by the stator vanes. The effect of the rotor disc speed can also be observed - at higher rotor disc speeds, the disc pumping imposes higher momentum onto the ejected purge flow which then reduces mixing with the unseeded annulus flow. This leads to higher content of CO_2 at the lowest annulus radial position.

The peak of tracer gas content observed near the rim in Fig. 12 is easily identified here as the yellow region in $\theta/\Delta\theta_{\text{NGV}} = -0.1$. As expected from the analyses above, URANS clearly underpredicts purge flow radial migration in the annulus compared to WMLES and experimental data. In particular, the URANS simulations predicted the egress flow is entrained into the rotor hub boundary layer. A remarkable qualitative agreement of the WMLES with experimental data is found at 6000 rpm where radial migration of the purge flow up to 60% of the annulus span is observed. A peak of the species concentration was found centred approximately in alignment with the NGV trailing edge circumferential position suggesting it was under the wake influence. Hence the sealing purge flow is not fully mixed within the annulus, in agreement with the findings by Scobie *et al.* (2017) at similar non-dimensional sealing purge flow rates.

Fig. 12. Sealing effectiveness contour comparison for URANS and WMLES and Bru-Revert's (2020) measurements for the vaned configuration, at 0.43 NGV axial chords downstream of the vane trailing edge, $U_m/U_e = 0.08$.

VI. CONCLUSIONS

This study discusses the predictive capability of two numerical approaches to model and estimate the levels of ingestion of annulus flow through a chute rim seal into a stator-rotor turbine cavity. Results from an

experimental campaign have been compared to numerical estimations obtained with URANS and WMLES. The WMLES uses a modified second order version of the LES HYDRA solver, which allows a more accurate flux computation compared to the standard second order scheme. Furthermore, it provides an order of magnitude computational cost saving through an approach which uses wall functions in the near-wall domain.

The effect of the annulus flow on seal performance has proven to be complex, computationally challenging and dependent on test conditions. Although the CFD time-averaged cavity pressure predictions are consistent with experiments, rim sealing ingestion is generally underestimated by CFD. WMLES data are much better aligned with ingestion measurements than URANS results but some discrepancies are still apparent at higher purge flow rates. It is speculated that this is associated with interactions of the purge and annulus flows, and the associated scatter in concentration measurements observed when annulus flow is introduced. For the vaneless configuration, the quantitative mismatch is reduced to 10% at low rates of purge flow. This predictive improvement positions the WMLES as a powerful tool with potential to be implemented in the design process based on the saving in computational cost (compared to LES) and increased accuracy in the predictions (compared to URANS). Therefore, WMLES is effective in improving on URANS modelling for turbine rim sealing, and offers a computationally-affordable alternative to wall resolved LES.

For all cases considered here WMLES predicted greater ingestion than URANS and aligned better with the experimental results. In the vaned configuration, WMLES overpredicted the measured sealing effectiveness, particularly in the inner part of the cavity. Time-averaged concentration-based sealing effectiveness contours show the radial migration of the purge flow on an axial plane corresponding to the leading edge of the rotor blades (if these were fitted). URANS results remarkably underpredict purge flow radial migration in the annulus compared to WMLES and experimental data. An excellent qualitative agreement of the WMLES with experimental data is observed for the $Re_\theta = 2.0E + 06$ vaned test case configuration, where radial migration of the purge flow to 60% of the (reduced) annulus span is observed, with a peak in the species concentration centred approximately on the NGV trailing edge circumferential position.

ACKNOWLEDGMENT

Funding for this research from EPSRC, Innovate UK and Rolls-Royce plc is gratefully acknowledged. Support from colleagues from the Universities of Surrey and Oxford and Rolls-Royce plc is also much appreciated. In particular, we would like to thank Sebastiaan Bottenheim, Matthew Miller and Peter Smout from Rolls-Royce. Dr. Jian Fang and Dr. Charles Moulinec are also acknowledged for having provided reference data for the Taylor-Green vortex test case.

REFERENCES

- Beard PF, Gao F, Chana KS and Chew J, (2016), “Unsteady flow phenomena in turbine rim seals”, (2016), *ASME J Eng Gas Turbines Power*; 139: 032501–032501-10.
- Bru-Revert A, Beard PF, Chew JW and Bottenheim S, (2021), “Performance of a turbine rim seal subject to rotationally-driven and pressure-driven ingestion” in *ASME J Eng Gas Turbines Power*. August 2021; 143(8), doi: [10.1115/1.4049858](https://doi.org/10.1115/1.4049858)
- Chew JW, (1991), “A Theoretical Study of Ingress for Shrouded Rotating Disc Systems With Radial Outflow.” *ASME. J.Turbomach.*, January 1991; 113(1): 91-97, doi: [10.1115/1.2927742](https://doi.org/10.1115/1.2927742).
- Chew JW, Gao F and Palermo DM, (2019), “Flow mechanisms in axial turbine rim sealing”, *Journal of Mechanical Engineering Science (Part C, Proc. IMechE)*, 233, 7637-7657 .
- Chilla M, Hodson H and Newman D, (2013), “Unsteady interaction between annulus and turbine rim seal flows,” *ASME J. Turbomach.*, 135(5), p. 051024.
- Denton JD, (1993), “Loss Mechanisms in Turbomachines”, *ASME 1993 International Gas Turbine and Aeroengine Congress and Exposition*, 93-GT-435, V002T14A001

- Fang J, Gao F, Moulinec C, and Emerson DR, (2019). “An improved parallel compact scheme for domain-decoupled simulation of turbulence“. *INT J NUMER METH FL*. 90,479-500.
- Gao F, Chew J. W. and Marxen, O. (2020). “Inertial waves in turbine rim seal flows.” *Physical Review Fluids*, 5(2), p. 024802.
- Horwood JTM, Hualca FP, Scobie JA, Wilson M, Sangan CM and Lock G, (2018), “Experimental and Computational Investigation of Flow Instabilities in Turbine Rim Seals”, *ASME J. Eng. Gas Turb. Power*, 141(1), pp. 011028-011028-12
- Horwood JTM, Hualca FP, Scobie, JA, Wilson M, Sangan CM, Lock GD, Dahlqvist J and Fridh J, (2019). “Flow instabilities in gas turbine chute seals”. *ASME J Eng Gas Turbines Power*.
- Iranidokht V, Kalfas AI, Abhari RS, Senoo S, Momma K, (2020). “Sensitivities analysis on the impact of geometrical and operational variations on turbine hub cavity modes and practical methods to control them”. *Proceedings of Global Power and Propulsion Society, GPPS Chania20, Paper no: GPPS-CH-2020-143*
- Kyprianidis KG, (2011). “Future Aero Engine Designs: An Evolving Visions”, *Advances in Gas Turbine Technology*, InTech, ISBN 978-953-307-611-9, DOI: 10.5772/664.
- Moinier P, (1999), “Algorithm developments for an unstructured viscous flow solver”. *Ph.D. thesis*, University of Oxford.
- Palermo DM, Gao F, Chew JW and Beard PF, (2019). “Effect of annulus flow conditions on turbine rim seal ingestion”, *In Proceedings of ASME Turbo Expo 2019: Turbomachinery Technical Conference and Exposition*. Paper No. GT2019-90489.
- Palermo DM, (2020). “Advanced unsteady aerodynamics and numerical modelling of axial turbine rim sealing flows”, *Phd Thesis*, University of Surrey.
- Palermo DM, Gao F, Amirante D, Chew JW, Bru-Revert A and Beard PF, (2021), “Wall-modelled large eddy simulations of axial turbine rim sealing”, *ASME J Eng Gas Turbines Power*, Paper No. GTP-1400.
- Pogorelov A, Meinke M and Schröder W, (2018), “Large-eddy simulation of the unsteady full 3D rim seal flow in a one-stage axial-flow turbine”. *Flow Turbulence and Combustion*, July 2018, pp. 1-32.
- Roe PL, (1981), “Approximate Riemann solvers, parameter vectors, and difference schemes”, *Journal of Computational Physics*, 43(2), pp. 357–372.
- Sagaut P, (2005), “Large-eddy simulation for incompressible flows: An introduction”, *Springer*.
- Savov SS and Atkins NR, (2017), “A Rim Seal Ingress Model Based on Turbulent Transport,” *ASME Paper No. GT2017- 63531*.
- Schadler R, Kalfas AI, Abhari RS, et al., (2016) “Modulation and radial migration of turbine hub cavity modes by the rim seal purge flow”. *ASME J Turbomach*; 139: 011011–011011-10.
- Scobie JA, Sangan CM, Owen JM, Lock G, (2016), “Review of Ingress in Gas Turbines”, *ASME J Eng Gas Turbines Power* ,2016; 138: 120801.
- Scobie JA, Hualca FP, Patinios M, Sangan CM, Owen, JM and Lock GD, (2017), “Re-Ingestion of Upstream Egress in a 1.5-Stage Gas Turbine Rig,” *ASME Paper No. GT2017-64620*.
- Taylor GI, Green AE, (1937), “Mechanisms of the production of small eddies from large ones”. *Proceedings of the Royal Society of London. Series A, Mathematical and Physical Sciences*, Vol. 158, No. 895 (Feb. 3, 1937), pp. 499-521.