ANALYSIS OF THE SEALING FLOW RATE ON LOSS PRODUCTION MECHANISMS IN LPT STAGE

Dario Barsi¹, Davide Lengani¹, Daniele Simoni¹, Francesco Bertini²,
Matteo Giovannini³, Filippo Rubechini³

1. University of Genova – DIME, Genova16145, Italy e-mail:dario.barsi@unige.it
2. AvioAero, Torino 10040, Italy, e-mail: francesco.bertini@avioaero.it
3. Morfo Design, Sesto Fiorentino (FI) 50019, e-mail: matteo.giovannini@morfodesign.it

ABSTRACT
In the present work, numerical simulations have been carried out to model the flow behavior within an aeroengine rotor/stator cavity system of low pressure turbine, and to investigate the interaction process between the main flow and the flow entering/exiting from the cavity system. Different sealing flow rates injected into the cavity to prevent thermomechanical failure of the disks have been simulated. Experimental results acquired in a cold-flow facility reproducing one-and-half Low Pressure Turbine axial flow stage have been used to validate the simulations. Cavity characteristic mass flow rates and the total pressure loss distribution coefficient have been computed for different sealing flow rates to understand the effect that this parameter has on the stage performance and leakage flow ingested into the cavity. The time mean effect related to the rotor/stator interaction, provoking ingestion in the cavity of the wake generated by the upstream rotor bars and blockage effects related to the presence of the downstream bars have also been considered. The results confirm that unsteady calculation procedures are necessary if the aim is to correctly capture the effect induced by the leakage flow on the cavity sealing capability. Moreover, the increase of the mass flow rate injected into the cavity leads to the enhancement of the strength of the loss core in the lower 50% span observed downstream of the vane, also inducing local variation of flow angle close to the hub, associated with the flow exiting from the cavity. The results provide a clearer picture on the mechanisms responsible for producing additional losses and how this mechanism is affected by the sealing flow rate.

KEYWORDS
AXIAL FLOW TURBINE, CAVITY/Mean-FLOW INTERACTION, CFD, RIM SEAL

NOMENCLATURE

\[ A \] Area

\[ Cp \] Total pressure coefficient \[ Cp = \frac{P_{1}(r, y) - P_{2}(r, y)}{P_{1,ref} - P_{2,ref}} \]

\[ C_{x} \] Axial chord length

\[ \dot{m}_{cool} \] Cooling mass flow rate
\[ \dot{m}_{fp} \] Main flow path mass flow rate
\[ \dot{m}_{\text{leak}} \] Leakage mass flow rate
\[ \dot{m}_{\text{rim}} \] Rim mass flow rate
\[ p_t \] Total pressure
\[ p \] Static pressure
\[ r \] Radial coordinate
\[ Ro \] Rossby number \( Ro = \frac{\nu_{\text{rad}}}{\omega r} \)
\[ v \] Velocity
\[ x \] Axial coordinate
\[ \theta \] Circumferential coordinate
\[ \omega \] Rotational speed or overall total pressure loss \( \omega = \frac{1}{A} \int_{\text{hub}}^{\text{tip}} \int_{0}^{\text{pitch}} C_p(r, \theta) r \, dr \, d\theta \)

**Subscripts**
1 Vane upstream
2 Vane downstream
\text{mid} Midspan
\text{ref} Reference value (for pressure values corresponding to midspan section)
\theta Tangential

**INTRODUCTION**

The request for increased efficiency in aircraft and power production turbines, have led to an increase in the turbine entry temperature. This entails on the one hand, the need to create efficient cooling technologies, able to keep the temperature of the materials below the limit necessary to guarantee the mechanical properties, and on the other hand to prevent ingestion of hot gas in areas of the machine not designed to withstand high temperatures, especially in the rotor-stator cavity disks. However, if too much cooling air is employed, it may result in a decrease of the overall system efficiency. Similar problems, concerning the leakage flow, however involving other aspects, are also relevant for different types of turbomachinery (Canepa et al. 2013 and Canepa et al. 2019). A growing attention has been devoted in recent years to the study of flows in cavities and systems to prevent the ingestion of hot gases, showing an increasing interest in design and optimization of these systems. The ingestion of hot gases is dominated by a series of main mechanisms. Johnson et al. (1994) early identified the basic phenomena, among that the most important are: disc pumping effect (rotationally induced ingestion), time dependent pressure field created by vanes and blades, or a combination of both effects. Other factors, such as the 3D geometry of the sealing system and the action of turbulence and secondary flows, also contribute to the modification of the interaction mechanisms between the main and the cavity flows. However, the ingestion mechanisms strongly depend on the conformation of the cavities themselves and on the type of sealings used to reduce the passage sections.

Moreover, the injection of secondary air, employed to cool the disks and to seal the cavity, influences the leakage flow as well. In their two-stage low-pressure turbine rig, Schrewe et al. (2013) highlighted that the efficiency reduced as the seal mass flow rate increased. Moreover, they observed that the outflow of the seal air strongly interacts with the flow passing through the vane and blade rows, also changing the main flow angle (thus the incidence angle) on the downstream blade. The
entrainment of low momentum flow exiting the cavity is also able to increase the strength of the rotor hub passage vortex. Also, studies on high pressure turbine show similar trends, as provided by the works of Ong et al. (2006), Zerobin et al. (2017), and Schuepbach et al. (2010).

It is known that the boundary layer at the turbine hub is influenced by the rim seal flow. This interaction is made complex by its unsteady nature that depends on the potential pressure field of the downstream rotor and on an unsteady Kelvin-Helmoltz type instability developing within the cavity (e.g. Chilla et al. 2013). Several experimental works have shown that the unsteady nature of these phenomena is only partially linked to the blade passage frequency affecting the flow developing in main channel. The experimental works by Cao et al. (2004), Roy et al. (2004), Beard et al. (2016), Savov et al. (2016), Schadler et al. (2016) and Town et al. (2016) identified unsteady characteristics of the cavity rim seal flow unrelated to blade passing. In fact, the natural frequencies of the flow passing through the cavity are related to pumping phenomena of the disks and to the interaction of the flow with the complex geometry and narrow sections of the cavity, which typically induce phenomena at a lower frequency than those of the blade passing.

For this reason, RANS methods allow partially capturing the effect of these phenomena on the overall behaviour of the flow in the cavity. Within the scenario of non-stationary CFD simulations, LES methods certainly offer a high level of detail in representing the flow in turbomachinery and within the cavities of axial turbines, as demonstrated in the work of Gao et al. (2018). However, these methodologies have the limit of requiring high computational efforts, especially if the aim is to represent realistic operating conditions, typical of aeronautical or power plant applications.

Within this context, URANS methods offer a good compromise between reliability of results and computational efforts and times, provided that the URANS method is based on fully unsteady simulations and does not introduce simplified assumptions on time scales, as occurs for example with Non-Linear Harmonic (NLH) methods. In NLH methods, in fact, the simulated frequencies are directly related to the blade passing frequency. In this way it is not possible to capture the effect related to phenomena at different frequencies from the ones belonging to the “chosen set”.

In the frame of the European E-Break Project, the Aerodynamics and Turbomachinery Laboratory of the University of Genoa, in collaboration with AvioAero, has designed and developed an experimental test rig capable of simulating the flow behaviour in the cavity of a typical Low Pressure Turbine (LPT) stage geometry. The rig, described in detail below, simulates the entire geometry of the system, and therefore can represent the actual flow rate in the cavity. With respect to other similar experimental test rigs, here the pressure difference induced between the inlet and the outlet sections of the cavity, which determines the leakage flow rate, is naturally imposed by the presence of a real vane row, and not by externally imposed boundary conditions. In the recent past, the rig has been used to conduct a series of experimental campaigns aimed at describing and characterizing the behaviour of the flow into and outside the cavity in terms of both global and local parameters. These data have been obtained for different operating conditions, by varying the rotational speed, the flow rate in the main channel and the cooling flow rate injected into the cavity to obtain the sealing condition (see Simoni et al. 2017 and Guida et al. 2018). Some of these experimental results were used to validate a series of numerical investigations, carried out under similar test conditions, but in the absence of cooling flow inside the cavity (Barsi et al. 2021 and Barsi et al. 2022). In this work, the CFD analysis has been extended, also considering the injection of cooling flows within the cavity. In addition, the calculations were carried out to highlight the need to consider non-stationary models if the purpose is to determine the effective interaction between the flows ingested/ejected from the cavity and the main flow. Computations have been carried out for different sealing flow rates, clearly
highlighting the effects due to the coolant flow on the sealing properties of the cavity and the corresponding loss generation mechanisms due to the interaction with the main flow.

**EXPERIMENTAL TEST RIG**

The experimental apparatus is the annular low-pressure axial flow turbine of the Aerodynamics and Turbomachinery Laboratory of the University of Genova. The rig consists of a main channel, and an axi-symmetric cavity, located below the vane. The rig is representative of the geometry of an aircraft-type low-pressure turbine stage. In particular, the apparatus reproduces a scaled geometry, with a scale factor of 1.5:1 compared to the real geometry, to facilitate experimental measurements and increase spatial resolution. The main channel consists in 4 rows: the first row is a vane allowing to set the desired inflow angle. This pre-distributor vane is followed by a row consisting of small diameter rotor bars, used to reproduce the wakes induced by the rotor blades placed upstream of the vane. The vane row reproduces the actual geometry of a low pressure turbine vane. Downstream of this row large diameter bars have been used to reproduce the potential effect induced by the presence of the rotor row downstream of the vane. The upstream bars have been designed following a similar procedure of what presented and validated in Simoni et al. (2015), obtaining a diameter of 2 mm, while rear bars have been designed following the procedure provided by Opoka and Hodson (2008), to well reproduce stagnation pressure field on the leading edge, thus adequately apply a periodic pressure distribution on the cavity exit plane. Thus, a 12 mm bar diameters have been chosen. The axisymmetric cavity system reproduces the actual geometry of a cavity used for aeronautical applications. The vane chord is 59 mm with an aspect ratio equal to 1 and 0.66 pitch to chord ratio at the hub. The rig was designed using a number vanes and blades (or better rotating bars) per row, which allowed for a reduced angular sector for geometric periodicity, thereby reducing the computational burden of CFD calculations. In particular, the stationary rows consist of 84 vanes, while the rotor rows consist of 63 bars, allowing to obtain a rotor/stator blade count ratio of 4:3. Figure 1 shows the domain used for the numerical simulations.

Figure 1 - 3D representation of the simulated geometry. Green sections correspond to inlets, while the red section correspond to test rig outlet.

The cavity is equipped with two different series of periodic slots used to inject the cooling flow into the cavity. They are staggered tangentially and arranged with a periodicity equal to that of the rotor rows. Figure 2 shows a simplified representation of the test section, and a detail of the surface of the cavity on which the cooling slots are positioned.
Focusing on the main flow path, the blue lines, indicated as "Kiel probe planes", placed 0.4 \( C_x \) upstream of the leading edge of the vane and 0.4 \( C_x \) downstream of the trailing edge of the vane, respectively, highlight the position of the measuring planes for total pressure measurements in the experiments. Both sections are axially located in the centerline of the inlet and outlet interface between cavity and main flow. Red lines highlight the position of control sections for overall total pressure loss evaluation, as successively presented in the results paragraph.

Available experimental data includes pressure distributions in the cavity, total pressure distributions upstream and downstream of the vane and estimation of the leakage flow of the cavity for different operating conditions. These data have been used for a proper validation of CFD results. All the simulated cases correspond to an inlet flow rate in the main channel of 3.02 kg/s, corresponding to a Reynolds number based on the axial velocity at the vane exit and the axial length of the cavity equal to about 100000. The rotational speed has been set to 300 rpm, corresponding to a Rossby number of 0.7. A complete description of the experimental test rig can be found in the works of Simoni et al. (2017) and Guida et al. (2018).

**Figure 2 - Simplified meridional view of the test section (left) and detail of the cooling slots (section A-A on the right)**

**NUMERICAL SIMULATIONS**

Numerical simulations, after the validation against experiments, are here exploited for a deeper insight in the complex flow field coming from the cavity-main flow interaction. Both steady and unsteady simulations were carried out. In particular, the steady simulations were used for the preliminary analysis and to identify the computational models suitable for conducting the subsequent unsteady simulations.

**Simulated domain and meshing**

As previously mentioned, the test rig is characterized by a bar/stator count ratio of 4:3. This allows the adoption of a reduced computational domain, avoiding approximations typically adopted with
domain scaling technique. The RANS calculation, instead employs a single blade passage for each row, adopting the mixing plane method to transmit information between the different rows. For both RANS and URANS simulations, the HexpressOPEN module by Numeca was used to create the meshes into the blade passage and into the cavity system. More specifically, the mesh of the main channel was first realized through Autogrid, the structured mesh creator of Numeca, to exploit the advantage introduced by the meshing tool specifically developed for turbomachinery rows. An O-4-H mesh topology was used to generate the mesh around the four rows constituting the main channel. The mesh of the cavity and of the auxiliary domains necessary for the correct imposition of the cooling slots inlet boundary conditions was instead created by means of HexpressOPEN, the unstructured mesh generator by Numeca. To determine the height of the first cell of the main channel and of the cavity domains, a value of $y^+$ equal to 1 has been imposed along all the surfaces. Inside the cavity, the number of viscous layers is variable, depending on the mesh sizing. The number of layers for each surface surrounding the cavity domain has been obtained by using optimization process contained in HexpressOPEN, controlling the growth factor (below 1.3). The complete mesh of the domain, consisting of the main channel, the cavity, and the cooling slot domains, was subsequently assembled within the HexpressOPEN environment. The interface surfaces between the main channel and the cavity have been identified as full non-matching boundaries and subsequently modified in rotor/stator interfaces to allow the imposition of the rotation of the calculation domain of the cavity.

The mesh size for the main channel was chosen after a grid independence analysis of the solution conducted in the previous authors’ work (Barsi et al. 2022). Additional mesh dependence analysis has been carried out since the cavity domain is meshed through unstructured grids. The mesh of the cavity has been created by imposing constrains on the cells maximum aspect ratio and expansion ratio. The final mesh presents a maximum aspect ratio of about 200 and a maximum expansion ratio of about 7. Both values are below the critical ones suggested by HexpressOPEN best practise (respectively 2000 and 18). An image showing the grid for the cavity domain is presented in Figure 3.

![Figure 3 – Cavity domain mesh](image-url)

The final meshes consist of about 10 million cells for the RANS case and up to about 35 million cells for the URANS case. More specifically, for the 10 million cell mesh, about 4 millions are used for the main channel and 6 millions are used to mesh the cavity. For the URANS calculations, 15 million cells are used for
the main channel, while about 20 million cells are used for the cavity domain. The mesh, for both configurations, allows a multiblock approach employing two coarser meshes in order to fasten the calculation during the convergence process.

**Numerical models and boundary conditions**

The identification of the most appropriate turbulence model was investigated during the preliminary RANS calculations. Different turbulence models have been tested: the simple one-equation model proposed by Spalart & Allmaras (1992), the standard k-ε model by Launder and Sharma (1974) and the k-ω SST model proposed by Menter (1994). The leakage mass flow rate passing through the cavity and the rim mass flow rate, defined as:

\[
\dot{m}_{rim} = \dot{m}_{leak} - \dot{m}_{cool}
\]

has been employed as test variable for a first evaluation of the turbulence model reliability.

The results of the analysis of the influence of the turbulence model on the normalized leakage and rim flow rates are shown in Figure 4a. Except for the standard k-ε model results, which present an overestimation of the leakage flow rate for all the tested conditions, the effect of the different turbulence model used on leakage and rim mass flow rate estimation is quite limited. Although it presents slightly different trends for the experimental data and the CFD results, the leakage mass flow remains almost constant as the cooling flow varies. For all tested models, the rim mass flow rate trend is quite well captured for low values of non-dimensional cooling flow rate, while raising the flow rate injected into the cavity to values larger than the 1% of the main flow rate, appreciable difference with respect to experimental results can be observed. In any case, as highlighted in Figure 4b and discussed in detail in the next paragraph, such errors are reduced in URANS calculations.

![Figure 4](image-url)

**Figure 4** - Non-dimensional leakage and rim mass flow rates for different values of cooling mass flow rate: turbulence model effect (a) and RANS vs URANS results (b)

Moreover, the sealing mass flow rate, which is estimated in the surround of 2.7% by the experimental results, is predicted for values higher than 3% for all the RANS simulations. This is probably related to the inability of RANS simulations to correctly evaluate the time mean effect of
the phenomena induced by the intrinsic low frequency unsteadiness of the flow inside the cavity. However, to avoid that the adoption of a simple turbulence model, such as the Spalart & Allmaras, could lead to excessive approximations on subsequent URANS calculations, the two equations k-ω SST model has been adopted. The discretization in space is based on a second order cell centred finite volume approach with second and fourth order artificial dissipation (Jameson et al. 1984). For the URANS calculations, a four-stage Runge-Kutta scheme with local time stepping is adopted for time discretization associated with several convergence acceleration techniques, such as implicit residual smoothing, dual time stepping and full multigrid approach. For boundary condition settings, the mass flow rate is imposed at the inlet section, located about 6 Cx upstream of the main vane. The flow direction is imposed with zero pitch and yaw angles. At the outlet section, located about 12 Cx downstream of the vane trailing edge, the ambient pressure condition is imposed. For all solid walls the no-slip condition is imposed. For the shroud and for the stator surfaces, a static wall boundary condition is imposed. For the rotating parts, as the hub, the bar surfaces, and the external surface of the cavity, rotating boundary conditions are imposed based on the nominal rotational speed. For the cooling domains, the cooling mass flow rate and turbulence parameters are imposed at the inlet sections (highlighted in green in Figure 1).

For RANS initialization, uniform flow field condition has been imposed, with constant ambient pressure and zero velocity values in all directions. For each of the coarser mesh from the multiblock approach, 500 iterations are employed before passing to the finer grid. For the finest mesh, 2000 iterations are employed to reach convergence. For URANS calculation, the initial condition is imposed from the final solution of the corresponding RANS calculation. For the considered domain, consisting of three blade passages and four vane passages, 80 angular positions were simulated, using 30 inner sub-iteration in the dual time-stepping approach.

Ten bar passing periods (corresponding to 800 time steps) were computed in order to reach a periodic solution before proceeding to extract and storage samples for further post processing and analysis. This allow reaching, for all cases, a periodic trends for at least three periods for monitored flow rates. The sampling has been carried out for an entire period, covering a complete passage of the rotating domain over the pitch direction.

RESULTS

Figure 4b shows the trends of the normalized leakage and rim mass flow rates as the percentage of the cooling flow rate varies with respect to the flow rate passing through the flow path. Values from RANS and from URANS time averaged results are presented and compared with those obtained from the experiment.

Both the leakage and the rim mass flow rates provided by URANS results are closer to the experimental data than those obtained from RANS simulations. Furthermore, the trend of the rim flow rate obtained from the URANS solver identifies a value of the cooling flow rate capable of sealing the cavity equal to approximately 2.85%, significantly closer to the value experimentally measured (2.7%) with respect to RANS results.

To quantify the effects induced by the variation of the cooling flow in terms of the vane performance, time averaged total pressure and entropy distributions into and outside the cavity have been evaluated. Figure 5 shows the total pressure loss coefficient distributions for different flow rates predicted by the URANS simulations. Experimental results are also shown for comparison. The diagrams on the right hand side correspond to the case of complete sealing.

Data are extracted in the plane identified in Figure 2 by the blue lines, corresponding to the sampling plane during the experiments.
As a general consideration, URANS results exhibit higher peak values (red loss cores in the figures) and lower $C_p$ values in the potential flow regions (blue areas). However, URANS and experimental results show similar trends as the cooling flow rate increase. In particular:
- there is a progressive thinning of the area characterized by high $C_p$ values located near the hub;
- the loss core at lower radius, mainly due to the action of secondary flows close to hub, moves to higher radius and intensifies in magnitude. Interestingly, also an enlargement on the right side of the loss core in the bottom half of the channel can be observed;
- the loss core in the upper part of the channel seems not influenced by the action of the cooling flow rate.

Figure 6 shows the circumferentially averaged normalized entropy fields characterizing both the vane and the underlying cavity. The higher entropy flux related to the presence of the boundary layer of the hub is partially ingested by the cavity in the absence of cooling flow supplied into the cavity (the flow contoured by black ellipse highlight a low entropy region at vane inlet). As the cooling flow rate increases and consequently with the progressive increase of the sealing process, the flow associated with the boundary layer remains in the main channel since it is not ingested by the cavity (red ellipse regions). Consequently, passing through the vane, this high entropy flow region is transported in the main channel, and contributes in intensifying the entropy production downstream of the vane (white ellipse regions). However, it’s worth noting that entropy flux imposed at the entrance of the cooling slot is not controlled. In fact, the coolant flow rate, the static temperature and the turbulence parameters are imposed while inlet pressure results from imposing the desired flow rate, and consequently the inlet entropy comes from this static pressure value. In this context, the
entropy value observed into the cavity is not of interest, while it is the entropy rise from the inlet to the outlet section of the cavity.

<table>
<thead>
<tr>
<th>(\frac{m_{\text{cool}}}{m_{FP}})</th>
<th>Normalized entropy [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0%</td>
<td></td>
</tr>
<tr>
<td>0.6%</td>
<td></td>
</tr>
<tr>
<td>2.7%</td>
<td></td>
</tr>
</tbody>
</table>

![Figure 6 - Circumferential averaged entropy fields for different cooling mass flow rates. The diagrams on the right hand side correspond to the case of complete sealing](image)

In order to better identify the effect that the presence of the cavity flow induces on the radial distributions of the quantities leaving the vane row and successively entering the downstream rotor, Figure 7a shows the normalized circumferentially averaged \(C_{p_t}\), while Figure 7b shows the flow angle deviation distributions along the blade span.

![Figure 7 - Spanwise distributions of normalized \(C_{p_t}\) (a) and flow angle deviation (b)](image)

For the latter, the spanwise distributions are reported in two different axial positions: the first one (UP) is placed in the middle of cavity outlet section, while the second one (DOWN) is placed at the
end of the cavity slot. The values are referred to the midspan value measured in the upstream section with positive values corresponding to flow under turning.

As previously observed also from the $C_p$ maps, increasing the coolant flow rate induces a progressive stronger loss core in the bottom half of the channel, together with a progressive shift towards greater radius of the peak loss, while the values in the upper 50% of the channel remain almost unchanged. This intensification of the loss core related to the secondary flows in the lower half of the channel is coherent with what observed in Figure 6 in terms of the increase of the hub boundary layer flow transported through the vane as the cooling flow rate increases. However, while highlighting trends similar to those obtained from experimental results, the $C_p$ value predicted by CFD is generally higher than the experimental one, as also observed from the maps in Figure 5. Table 1 presents the overall total pressure loss for the different tested conditions. Data are made non-dimensional with the value computed without coolant flow. The calculation considers as upstream and downstream section those denoted with red lines in Figure 2, to correctly capture the upstream boundary layer and the downstream coolant flow exiting the cavity. In addition, the loss coefficient takes into account the energy contributions introduced by means of the cooling flow rates.

Table 1 – Loss level percentage variation for different coolant flow rates

<table>
<thead>
<tr>
<th>$\dot{m}<em>{cool}/\dot{m}</em>{FP}$</th>
<th>$\Delta\omega/\omega_{ref}$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 %</td>
<td>reference</td>
</tr>
<tr>
<td>0.6 %</td>
<td>+ 0.92</td>
</tr>
<tr>
<td>2.7 %</td>
<td>+ 10.62</td>
</tr>
</tbody>
</table>

As can be observed, the increase of coolant flow rate induces a slight increase of overall total pressure loss for the partially sealed condition, while almost 11 % loss increase is observed for the sealed cavity condition. This highlights how the sealing condition of the cavity is obtained at the expense of the energy injected into the cavity through the cooling flow rate. In Figure 7b, on the upstream section (continuous lines) it is possible to notice how the increase in the cooling flow rate induces an increase of flow deviation in the lower 50% span of the channel, while keeping similar trends in the upper part of the channel. Interestingly, observing the downstream section distributions (dotted lines) the flow angle deviation close to the hub is reduced and becomes negative (overturning) with respect to the one in the upstream section, since the flow exiting the cavity has a different swirl with respect to the main flow. This leads to a local flow deviation that may induce a non-nominal incidence angle on the downstream rotor near the hub section.

**CONCLUSIONS**

In this work, the flow field developing into a test rig reproducing one and a half stage of low-pressure turbine for aeronautical applications, equipped with a cavity system, was investigated by means of CFD methods for several cooling flow rates conditions. The comparison of the leakage flow rate and the rim flow rate curves for the different tested conditions with the experimental results confirms that unsteady calculation procedures are necessary if the aim is to correctly capture the effect induced by the leakage flow on the cavity sealing capability. Detailed comparisons of local total pressure coefficient distributions downstream of the vane allow a better evaluation of the cooling flow rate effects on the vane losses. The increase of the mass flow rate injected into the cavity leads to the enhancement of the strength of the loss core in the lower 50% span observed downstream of
the vane, which also moves to higher radius, while the loss core in the upper part of the channel seems not influenced by the action of the cooling flow rate.

The analysis of the circumferentially averaged entropy field allows highlighting how the increase of coolant flow rate leads to a reduction of ingestion into the cavity of the boundary layer developing at the hub upstream of the cavity, thus leading to a strong interaction with the secondary flows of the vane.

The analysis of the circumferentially averaged total pressure loss coefficient and of the flow angle deviation makes it possible to better evaluate the local effect induced by the action of the flow exiting the cavity near the hub. The computed overall total pressure loss level downstream of the vane shows an increase of about 11% for the sealed cavity case, highlighting how this condition is obtained at the expense of the energy injected within the cavity associated with coolant flow. Flow deviation distributions show a local variation close to the hub, associated with the flow exiting the cavity.

All these results provide useful information for future developments in the design of blade row optimized for flow conditions downstream of cavities. However, future analysis based on URANS unsteady results obtained for several significant time instants, instead as time mean values as studied in the present work, can be used to gain more insight into the additional loss mechanisms and secondary flows alterations related to main flow and cavity flow interaction.

ACKNOWLEDGEMENTS

The research leading to these results has received funding from the European Union Seventh Framework Program FP7/2007-2013 under grant agreement n° ACP2-GA-2012-314366-E-BREAK. The authors also wish to acknowledge MIUR for funding the rig realization and the industrial partners that collaborated through the entire activity, Blue Engineering s.r.l. for the rig design, Progesa s.r.l. for the manufacturing and GE AvioAero for supporting the test program.

REFERENCES


