AERODYNAMIC INTERACTIONS BETWEEN A HIGH PRESSURE TURBINE STAGE AND A SHROUD CAVITY

E. Tang\textsuperscript{a,b,c} - I. Trébinjac\textsuperscript{b} - G. Ngo Boum\textsuperscript{b} - M. Philit\textsuperscript{a}

\textsuperscript{a}Safran Helicopter Engines, 64511 Bordes, France
\textsuperscript{b}Univ Lyon, Ecole Centrale de Lyon, LMFA, UMR CNRS 5509
36 av. Guy de Collongue, 69134 Ecully Cedex, France
\textsuperscript{c}Safran Tech, Rue des Jeunes Bois, Chateaufort CS80112, 78772 Magny-Les-Hameaux, France
etienne.tang@centraliens.net
isabelle.trebinjac@ec-lyon.fr

ABSTRACT

This study focuses on a high pressure turbine stage featuring a cavity above the rotor shroud. This cavity is purged by cooling air, and connected to the main gas path by an axisymmetric slot between the stator and the rotor. A 360° URANS simulation of this configuration has been performed. Flow structures generated by an aerodynamic instability are found in the slot between the annulus and the cavity. They form cells of hot gas ingestion alternating with zones of cooling air ejection, which rotate at a lower speed than the rotor. The effect of these structures on the flow through the turbine stage has been assessed, and compared to previously published results on a restricted computational domain which did not exhibit this instability. Similar interaction mechanisms occur, but their frequency differs because of the rotation of the structures in the slot.

KEYWORDS

Turbine, Cavity flows, Aerodynamics, CFD, Unsteady flows

NOMENCLATURE

\( \beta \) Flow angle in the rotor frame of reference  
\( c_p \) Specific heat capacity at constant pressure  
\( BPF \) Blade Passing Frequency  
\( C_x \) Axial chord  
\( BPP \) Blade Passing Period  
\( f \) Frequency  
\( \dot{\omega} \) Fluctuations \((\dot{\omega} - \langle \dot{\omega} \rangle)\)  
\( N_{rev} \) Number of rotor revolutions  
\( \omega_r \) Relative to the rotor  
\( p_s \) Static pressure  
\( \omega_s \) Relative to the stator  
\( p_t \) Total pressure  
\( n_t \) Massflow passing through the turbine (including the cooling flows)  
\( t \) Time  
\( \dot{W} \) Extracted power  
\( T_t \) Total temperature  
\( \eta \) Efficiency  
\( x \) Axial coordinate  
\( \gamma \) Heat capacity ratio  
\( x_i \) Mass fraction of the flow \( i \)  
\( \langle \cdot \rangle \) Time-averaged \( \cdot \)  
\( Z \) Number of blades  
\( \Omega \) Rotation speed of the rotor  
\( \text{DFT} \) Discrete Fourier Transform  
\( \theta \) Azimuthal angle  
\( \text{RANS} \) Reynolds Averaged Navier-Stokes  
\( \text{URANS} \) Unsteady RANS

INTRODUCTION

High pressure turbines feature numerous cavities, which may be prone to ingestion of hot gases from the annulus. In order to avoid the overheating of internal parts, secondary air extracted from the compressor is usually used to cool these parts and purge the cavities. The study presented here is part of an investigation on the effect of a cavity at the shroud on the aerodynamic performances of a
turbine. Very few papers can be found about this topic. Zlatinov et al. (2012) performed steady RANS simulations of a turbine stage with a cooling air injection from an axisymmetric slot at the shroud, with varying purge massflow and swirl values. They found that the purge flow induces a blockage effect, an increase in shroud passage vortex strength, additional mixing loss, and a reduction of the tip leakage flow at the front of the blade.

More papers focusing on the cavities at the hub between the stator and the rotor disks are publicly available. Overall, they found the same kind of interactions between the purge flow and the main stream as in the shroud cavity cases, with a blockage effect and an increase in hub passage vortex strength (e.g. Paniagua et al., 2004). In addition, they were also able to highlight some complex and unsteady flow features caused by the cavity. Indeed, Jakoby et al. (2004) and Cao et al. (2004) performed simulations of a full rotor-stator cavity and an annulus without any blade rows, and identified large scale structures rotating within the cavity, at a lower speed than the rotor. Cao et al. (2004) also reported experimental measurements on a turbine rig revealing low frequency phenomena in the cavity, thus confirming their numerical results. Boudet et al. (2006) found similar low frequency oscillations in a URANS simulation of a turbine stage taking into account the cavity over a periodic sector at low purge flow rates. Wang et al. (2013) carried out URANS computations on a full 360° turbine stage with a cavity. They identified irregular cells inside the cavity rotating at a lower speed than the rotor, and pointed out that they did not exhibit the same periodicity as the blade rows. Mirzamoghadam et al. (2014) extended these results by running a similar computation over a larger time range. They monitored carefully the convergence of their simulation and observed that they could not reach any stabilized state even after 16 rotor revolutions as the characteristics of the rotating cells were still slightly evolving, therefore much attention has to be paid to the convergence of such a computation. Town et al. (2016) evaluated experimentally the spatial periodicity and the rotation speed of these flow structures in a low speed turbine rig, and showed that a sector URANS computation is able to predict these values reasonably well. Schädel et al. (2016) performed detailed measurements in the annulus of a high pressure turbine rig, as well as 360° URANS computations. They obtained low frequency fluctuations in the cavity, which affect the flow in the annulus as these frequencies are found up to 30 % blade span. They also found that these modes account for a non-negligible amount of the sound produced by the turbine.

Therefore, the behaviour of the flow in rotor-stator hub cavities in turbines has been fairly well described. However the underlying mechanisms of instability generating the observed structures still remain unclear. Boudet et al. (2006) suggested that their oscillations were generated by a Taylor-Couette instability in the rim seal region, but they did not prove it. They might also share a common generation mechanism with the fluctuations obtained in closed rotor-stator cavities (Launder et al., 2010).

Shroud cavities differ from hub cavities as none of the walls of the cavity rotates. This might alter, or even prevent the development of any flow instability. However, to the authors’ knowledge, no study focusing on such a configuration is publicly available. The present paper addresses this issue by presenting a detailed URANS computation of the turbine, taking into account the full cavity. The test case together with previous results are first presented. Then, the numerical setup is described. Next, the convergence is carefully assessed, and the fluctuations of the flow are described and discussed. Finally, the interaction mechanisms between the cavity and the main gas path are analyzed.

**TEST CASE**

Figure 1 shows a meridional view of the stage, typical of a high pressure turbine of a helicopter turboshaft engine. It is studied at its design operating point, in a transonic regime, with a total pressure ratio of around 3. The stator has 19 vanes, and the rotor 35 unshrouded blades. The cavity is stationary, located above the rotor casing, and connected to the main gas path by an axisymmetric slot between the stator and the rotor. Secondary air is injected from holes at the outer part of the cavity in order
to cool the rotor shroud. Cold air is also introduced from an axisymmetric inlet in the slot. The total cavity purge flow amounts for around 2% of the exit mass flow.

Tang et al. (2016) carried out single-passage unsteady RANS computations on this geometry, but only took into account a small part of the cavity, up to the red solid line in Fig. 1. They found that the interaction phenomena identified by Zlatinov et al. (2012) are in fact very unsteady. This is explained by the fact that the areas of cooling air ejection are mostly fixed with respect to the stator, so the rotor meet the secondary air only intermittently. However, the accuracy of the cavity exit flow pattern predicted by these simulations is questionable as the cavity is not fully taken into account in the computational domain.

A single-passage computation of the stage with the full cavity was previously attempted by using the same numerical setup as Tang et al. (2016) with phase-lagged periodic boundary conditions, but it could not converge to a time periodic state at the blade passing frequency. This suggests the presence of an instability generating fluctuations at frequencies differing from the blade passing frequency. But phase-lagged simulations are not able to represent such oscillations, so this motivated the realization of the full 360° domain presented here.

NUMERICAL SETUP

Solver

The computation was performed with elsA (Cambier et al., 2013), a finite volume solver on multi-block structured meshes developed by ONERA. Wilcox (1988) $k – \omega$ turbulence model was used, with a low-Reynolds approach at the walls. The boundary layers are modeled as fully turbulent. The convective fluxes were discretized by Roe’s scheme with a MUSCL approach allowing a third order accuracy, and the diffusive fluxes by a second order centered scheme. Gear’s second order scheme was selected for the temporal discretization, with 10 sub-iterations per time step. Each rotor revolution was represented by 13 300 time steps, giving a time step of about $10^{-7}$ s. The hot annulus gases and the secondary air were modeled with a perfect gas assumption.

Mesh

The generation of good-quality multi-block structured meshes with classical matching connectivities is hard and time-consuming for such complex geometries. Therefore, a Chimera overset mesh technique is used (Castillon et al., 2014). The mesh is split in several classical multi-block structured meshes sharing overlapping zones where the flow variables are exchanged by interpolation during the computation. The Chimera setup was performed by the Cassiopée pre-processing package (Benoit et al., 2015). Three different Chimera domains were generated, as illustrated in Fig. 2:

1. the main gas path, a standard turbine stage mesh generated by NUMECA’s AutoGrid5, refined
in the axial direction at the location of the slot. This mesh is the same as the one used in (Tang et al., 2016);

2) the cavity, with an axisymmetric mesh featuring an overlapping zone in the annulus;

3) the cooling air injection holes, overlapping with the cavity mesh.

The grid density is similar to the values used in previous successful studies using the same code (e.g. Wlassow, 2012). The complete mesh over the 360° domain has 230 million cells. The dimensionless wall distance $y^+$ of the first cell layer at the walls is kept below 1. As the slot is located close to the rotor leading edge, the rotor-stator interface is positioned between the stator trailing edge and the annulus-cavity overlapping zone. Therefore, the mesh of the cavity has to be modeled in the rotor frame of reference. This is a problem as the part of the cavity containing the injection holes is not axisymmetric and has to be represented in the fixed frame of reference, because these holes do not rotate. Therefore, an additional rotor-stator interface plane has been setup within the cavity, represented by a dashed red line in Fig. 1. Even if some of the cavity walls are simulated in a rotating frame of reference, these walls are made stationary in the fixed frame of reference by defining the wall velocity in the relative frame as the opposite of the velocity of the rotating frame.

**Boundary conditions**

The boundary conditions at the turbine inlet prescribe stagnation pressure and temperature radial profiles representing realistic combustion chamber exit conditions. The stagnation pressure and temperature are imposed at the cooling holes injection inlets, and a uniform massflow injection boundary condition with no swirl is used at the secondary air inlet in the slot. Its stagnation temperature is the same as the air injected from the holes. The outlet is modeled with a radial equilibrium boundary condition. A sliding mesh approach is used at the rotor-stator interfaces. All the walls (blades, hub, shroud and cavity walls) are modeled as adiabatic.

**RESULTS**

**Convergence**

As shown by Mirzamoghadam et al. (2014), it may be hard to reach a reasonably stabilized state in turbine computations taking into account a cavity. It is therefore important to carefully monitor the convergence of the computation. Fig. 3 shows the temporal evolution of the static pressure field over a point at the junction of the slot with the annulus over the last two rotor revolutions, seven revolutions after the beginning of the computation. This point is modeled in the rotor frame of reference, therefore the relative movement of the stator vanes generates fluctuations at their Blade Passing Frequency ($BPF_s$). It is clear that the flow is not periodic, neither with the stator Blade Passing Period ($BPP_s$) nor with the rotor revolution period.

The amplitude of the discrete Fourier transform of this signal is represented in Fig. 4. There is a clear peak at the blade passing frequency, but the spectrum is dominated by several peaks over a range at low frequencies between 0.35 and 0.8 times $BPF_s$. Thus, the passing of the blades is not the only source of oscillations. However, part of the low frequency content may also be explained by an unsatisfactory convergence of the flow, which still evolves over large time scales.

As no temporal periodicity is expected, it is not possible to define the convergence criterion as a measure of the periodic nature of the flow as it is commonly done in classical URANS simulations of turbomachinery stages (e.g. Clark and Grover, 2006). It was therefore chosen to evaluate the convergence by monitoring the stabilization of a value of interest: the stage efficiency, computed with a moving average of the total pressure and temperature. The evolution of this quantity over time is represented in Fig. 5. It shows a very slow convergence behaviour as the efficiency changed by only 0.35 % over four revolutions. Additional computation runs would help to confirm the convergence of the efficiency, but for computational costs reasons, the simulation was stopped at this point. It has to be noted that the convergence of the efficiency does not mean that the full three-dimensional flow
Flow oscillations in the cavity

In order to identify the flow structures formed in the cavity and at the junction between the slot and the annulus, the flow variables were extracted over a cavity mid-span surface. It is defined as a surface of revolution generated by the meridional curve located at equal distance from the inner and the outer surfaces of the cavity, as illustrated by the dashed red curve in Fig. 7.

Figure 8 represents the instantaneous static temperature field on this surface, viewed from upstream of the turbine (view (a) in Fig. 7) and from outside (view (b)). It shows that cells of ingested hot gases, separated by zones of secondary air ejection, develop within the slot. The view from outside shows that most of the hot gases remain in the slot and do not penetrate further into the cavity. The temperature at the back of the cavity is higher than in the rest of it. This may be caused by an ingress of part of the ingested gas to this location, but this may also be explained by the bad convergence of the flow inside the cavity. It is interesting to note that the number of cells is identical to the number of stator vanes (19). But some irregularities are present over the circumference: the spatial periodicity of the pattern is not perfect, especially at the locations marked (1) and (2) on the figure where the cooling air ejection zone is particularly large and the ingestion cells are more widely spaced.
Figure 7: Schematic meridional view of the cavity mid-span surface in a dashed red line

(a) Seen from upstream of the turbine. The black circle marks the boundary between the annulus and the slot.

(b) Seen from outside of the turbine

Figure 8: Instantaneous static temperature field on the cavity mid-span surface

Figure 9: Temporal evolution of the static temperature field on the cavity mid-span surface, seen from upstream of the turbine in the fixed frame of reference (zoom on the top of Fig. 8a). Short red ticks are printed at the azimuthal location of the stator trailing edges and long blue ticks at the rotor leading edges.

Figure 9 represents a zoomed view of Fig. 8a for different time steps over five rotor blade passing periods. Two consecutive frames are separated by half of $BPP_r$. Ticks have been added to mark the azimuthal location of the stator trailing edges (in red, shortest ticks) and the rotor leading edges (in blue, longest ticks). It shows that the flow pattern inside the cavity rotates in the same direction of rotation as the rotor, but at a much lower speed. Indeed, the pattern moved by only around one stator vane pitch between the first and the last frame of the figure while in the same time, the rotor blades travelled around 2.5 times the stator vane pitch. The rotation speed of the ingestion cells in the fixed frame of reference can then be deduced: it is approximately 0.4 times the rotor speed. It is possible to define a cell passing frequency in the same way as the blade passing frequency by using this rotation speed and the number of cells. In the fixed frame of reference, this generates a low frequency oscillation at around 0.2 times $BPF_r$. In the rotor frame of reference, the cell passing frequency is 0.6 times $BPF_r$, which is consistent with the frequencies of the amplitude peaks in the Fourier transform in Fig. 4. The fact that amplitude peaks were found on a relatively large range of
frequencies can be explained by the fact that the ingestion cells are not evenly spaced: closer cells generate higher frequency oscillations while more spaced ones cause lower frequency peaks.

In order to determine the evolution of these structures, the flow variables were extracted on a circumferential line located at the junction between the slot and the annulus. The location of this line is marked (c) on the schematic meridional view in Fig. 7. A space-time diagram of the static temperature along this line during the last four rotor revolutions is represented in Fig. 10. As a reference, the rotor rotation speed is shown with an oblique dashed line. It confirms that the flow pattern rotates at a lower speed than the rotor. The rotation speed could be accurately determined by extracting the dominant rotating modes of this signal with a spatio-temporal Fourier transform. It corresponds to 0.37 times the rotor speed. Figure 11 represents the same data, as well as a space-time diagram of static pressure zoomed on 3 stator vane pitches and on the last 15 rotor blade passing periods. This time and space frame is indicated by a black rectangle on the top of Fig. 10. The temperature is highest when and where the pressure is lowest, which indicates that the ingestion pattern is driven by the pressure gradient between the annulus and the cavity: low pressures in the cavity generate hot gas ingestion while high pressures lead to cold cavity air ejection. A modulation of the flow pattern also appears because of the pressure field generated by the stator vanes: the cooling air ejection zones shrink when they pass through high pressure areas in the annulus, thus generating larger ingestion cells. The passing of the rotor also has a significant influence on the pressure, as shown by the large oblique streaks with the same slope as the dashed line, but its only effect on the temperature is a small modulation in the ingestion cells.

The two larger ejection zones (1) and (2), already identified in Fig. 8a, are also found in Fig. 10. A slow growth of their extent is obtained. Therefore, it is likely that this irregularity is not an artifact caused by bad convergence (if it was the case, it would be vanishing instead of growing), but rather a physical feature of the flow. It is interesting to note that the irregularity moves at the same speed as the rest of the flow structures, so that it always affects the same cells. In these zones, the amplitude of the cell size fluctuations caused by the interaction with the stator vanes is larger than in the rest of the circumference. At a particular time step, labelled (A) in the figure, the ejection zone is split in two, and an additional ingestion cell appears. But around half a revolution later (B), this new cell disappears and the two ejection zones are merged again. The computation was stopped less than one revolution after the end of this event, so it was not possible to determine whether further evolutions of this irregularity would be obtained or not. Putting this phenomenon aside, the number of ingestion cells did not change during all the computation runs over which this extraction was performed.

Discussion on the cavity flow oscillations

This is the first time to the authors’ knowledge that rotating structures are found in shroud cavities. One of the most surprising observations is that the number of cells formed within the slot matches the number of stator vanes while these oscillations are obviously not generated by rotor-stator interactions. According to Tyler and Sofrin (1962), the interaction modes are written as \( \cos (m\theta - nZr\Omega t) \) where \( m = nZr + kZs \) and \( n \) and \( k \) two integers. If the structures were caused by such a mode, \( m = Zs \) as the number of cells equals the number of stator vanes. The rotation speed of this mode would be \( nZr\Omega / m = nZr\Omega / Zs \). This mode would then rotate faster than the rotor (for any strictly positive integer \( n \), \( nZr/Zs > 1 \)), unlike what is obtained here. The first mode rotating at around the same speed as the cells is the mode 92 (\( n = 1 \) and \( k = 3 \)). But spatial Fourier transforms of the pressure and temperature signals in Fig. 11 did not identify the presence of this mode.

The instability causing this unsteadiness may be similar to the one which generates oscillations in hub cavities. Indeed, they share the following characteristics.

- They form ingestion cells in the cavity, rotating in the same direction as the rotor.
- These cells remain close to the annulus (in this case, in the axisymmetric slot).
- The cavity flow structures are not perfectly periodic.
Figure 10: Space-time diagram of static temperature in the fixed frame of reference on a circumferential line at the junction of the slot and the annulus (line (c) in Fig. 7)

Figure 11: Space-time diagrams in the fixed frame of reference on a circumferential line at the junction of the slot and the annulus (line (c) in Fig. 7), zoomed on 3 stator pitches and 15 rotor blade passing periods
The convergence is very slow and a stabilized state is hard to reach with CFD computations as found by Mirzamoghadam et al. (2014). Thus, the data analyzed in this paper might not represent the fully converged flow but only a transient state.

However, the rotation speed obtained (37% of the rotor speed) is lower than the values commonly measured in hub cavities, usually around 80-90% of the rotor speed. Also, the number of cells match the number of stator vanes, which was never observed in hub cavity studies. It is possible that this equality is a pure coincidence. Another hypothesis would be that the flow instability that generates these oscillations has a natural length scale close to the vane pitch, and the disturbance generated by the stator vanes triggers to the development of the instability with this periodicity.

Effect of the cavity on the main gas path

A previous paper by Tang et al. (2016) focused on the effect of the cooling air ejection from the slot on the flow in the annulus. Two computations were presented: a baseline simulation without any cavity, and a simulation modelling a simplified cavity featuring only a small part of the slot. The flow at the cavity exit did not exhibit oscillations like in the current case, but developed a pattern which remained fixed with respect to the stator, with one zone of secondary air ejection and one zone of recirculating hot gases per stator pitch. Therefore, each rotor blade tip intermittently encountered cooling air at a frequency of one BPF. As this cooling air was ejected from the cavity with a low tangential velocity, it entered the rotor row with under-incidence. That caused an intermittent increase of the shroud passage vortex intensity and a decrease of the tip leakage flow. In the present case, zones of cooling air ejection and pockets of ingested hot gas also form in the cavity, but they rotate slowly instead of remaining fixed. It was therefore checked whether or not the mechanisms described in this previous study still occur in the present computation by using the results of the embedded post-processing scripts. It has however to be noted that in order to limit the computation cost, these extractions were only applied for an amount of time limited to two cell passing periods in the fixed frame of reference. Therefore, the time-averaged data presented in this section were only averaged over this limited time frame.

In order to determine the effect of the cavity on the incidence at the rotor inlet, a radial profile of the flow angle in the rotor frame of reference was extracted upstream the blades on the plane marked (A) in Fig. 1 and plotted in Figure 12. The data from the previous study are also plotted on the same graph. Similarly to the previous study, a large flow angle deficit is found close to the shroud, down to 80% of blade span. This means that the flow comes with under-incidence at this location. The magnitude of the predicted under-incidence is smaller very close to the shroud than in the computation with the simplified cavity, but this may be due to errors caused by the bad convergence of the computation and the small size of the time-averaging window.

The unsteady relative flow angle was extracted on the same plane over the circumferential line at 95% span, marked (B) in Fig. 1. It is plotted in the rotor frame of reference (as "seen" by the rotor blades) on a space-time diagram in Fig. 13. The diagram is zoomed on 5 rotor blade pitches and around 5 stator blade passing periods. Black dashed lines show the relative stator rotation speed, and solid green lines the relative speed of the flow structures inside the cavity obtained from Fig. 10. The flow angle oscillates between nominal and very low values. Small patches of low flow angle (I) are found rotating at the same speed as the stator: they are caused by the negative jet of the stator wakes. Larger and more intense patches (II) are aligned with the green lines indicating the rotation speed of the cavity flow structures. They are caused by the cooling air ejected from the cavity with low tangential velocity. Their strength depends on their location with respect to the stator vanes. For example, let us follow the ejection zone close to the lowest solid green line. A large low angle zone is first formed (II), and then disappears until a smaller under-incidence area (III) is found. After that, the size of this patch increases again (IV). The disappearance of these zones always happen immediately after they crossed the locations of the stator wakes. This is consistent with Fig. 11 which showed that
the size of the ejection zones depends on their location relative to the stator vanes. There is therefore, in addition to the rotation of the ejection zones, a modulation of the under-incidence that they generate because of an interaction with the stator vanes. Thus, the rotor blades encounter low incidence cold air when it passes in front of the ejection zone, and the strength of this zone depends on its relative location with respect to the stator vanes.

Figure 14 represents radial profiles of relative flow angle downstream the rotor at plane (C) in Fig. 1, for the present computation and for the two previously published simulation results. It helps to identify the secondary flows: the over-deviation (low angle) at the hub and the shroud followed by under-deviation zones (high angle) are caused by the hub and the shroud passage vortices. Here, the two vortices meet at around mid-span, thus generating a unique under-deviation zone at this location. Close to the shroud, this effect is combined with the tip leakage vortex, which rotates in the opposite direction of the shroud passage vortex. Similarly to the previously published simplified computation, the intensity of the shroud passage vortex is increased when compared to the case without the cavity: there is an increase of the magnitude of the under-deviation at mid-span and of the over-deviation at the shroud. Moreover, the vortex is larger as the location of the under-deviation peak is shifted towards the hub. The over-deviation at the hub is not affected by the cavity, thus the hub passage vortex strength is not affected. Small differences are obtained between the present results and the simplified computation: the shroud passage vortex seems to be slightly more intense as the magnitude of the under and over-deviations is larger, but again, this difference may be artificially generated by errors caused by bad convergence and a small time-averaging window.

In the previously published study on this configuration, large oscillations of the tip leakage flow were found at the front of the blade because of the periodic encounter of low-incidence cold air. A massflow per unit length entering the tip gap was defined at different axial locations over this border. It is computed as the integral of the surfacic massflow entering the gap over the radial line at the intersection between the tip gap border (over the pressure or the suction side) and an axial plane. Figure 15 represents this quantity at the suction side in a space-time diagram: the horizontal axis represents the axial coordinate, and the vertical axis the time. Negative values mean that the flow comes out of the gap, which is the normal behaviour of the tip leakage flow, and positive values represent flow entering the gap. The black line shows the separation between these two zones. Again, the behaviour observed in the previous study is confirmed. At three time steps, a large reduction
of the mass flow occur up to 50% of axial chord, with even a reversal of the flow direction. This matches the number of large zones of cold air with low incidence encountered by each rotor blade over this time span in Fig. 13. Two smaller reductions of the mass flow are also found, because of the under-incidence caused by the passing of the stator wakes. But unlike in the previous study, the mass flow reduction events are not periodic: they are not evenly spaced in time, and their duration and spatial extent do not remain the same over successive events. This is explained by the fact that the low incidence zones are not periodic because of the interactions of the cooling air with the stator vanes.

Finally, the stage efficiency computed with the Weighted Pressure (WP) definition of Young and Horlock (2006) is shown in Tab. 1. This definition of the efficiency takes into account the work potential of all the cooling flows. Similarly to the isentropic definition, the WP efficiency is the ratio between the real extracted power and the power extracted from an ideal process. Unlike the isentropic ideal process, the WP ideal process includes losses generated by the unavoidable heat transfer between the cold and the hot flows. This definition is therefore more suitable to compare the efficiencies obtained in cases with different cooling inputs, as for example between a simplified simulation without any cooling and a more complex one including the secondary air injections. In this definition, the power extracted by the ideal process is computed with the following formula, with the indices $i$ indicating the different flow inlets:

$$
\dot{W}_{ideal} = c_p \dot{m} \left( \sum x_i T_{i} \right) \left( 1 - \left( \frac{P_{t, out}}{P_{t, in}} \right)^{\frac{\gamma - 1}{\gamma}} \right)
$$

Unlike in the previous study, a significant efficiency decrease is obtained. The efficiency of a virtual domain reduced to the main gas path has also been computed from the current results. This

Table 1: Stage efficiencies (Weighted Pressure definition by Young and Horlock (2006))

<table>
<thead>
<tr>
<th>Study</th>
<th>$\eta - \eta_{no cavity}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simplified cavity (Tang et al., 2016)</td>
<td>+0.1 %</td>
</tr>
<tr>
<td>Full cavity (current)</td>
<td>-0.7 %</td>
</tr>
<tr>
<td>Full cavity (current), efficiency of the main gas path</td>
<td>+0.6 %</td>
</tr>
</tbody>
</table>
definition does not take into account the inlets in the cavity, but only a virtual inlet located at the boundary between the annulus and the slot. Therefore, this efficiency only accounts for the losses generated in the main gas path. The result is shown in the last row of the table. This efficiency is significantly larger than the efficiency of the whole domain. This means that a large amount of loss is generated in the cavity, probably because of the large velocity and temperature gradients caused by the ingestion cells.

CONCLUSION

A 360° URANS simulation of a turbine stage featuring a purged cavity at the shroud has been performed and analyzed. Oscillations not linked to rotor-stator interaction modes were found to develop in the cavity. They are caused by the development of cells composed of hot gases ingested in the cavity, rotating at 37% of the rotor speed. The number of these cells matches the number of stator vanes. This is the first time that such structures are found in a shroud cavity. Their origin remain unknown, but they might share some common generation mechanisms with hub cavity oscillations. These structures interact with the main gas path through mechanisms similar to the ones found in a simplified computation which did not exhibit such oscillations (Tang et al., 2016). The main difference with these previous results is that the rotation of the cavity flow and its interaction with the stator vanes generate a more irregular pattern, thus modifying the frequencies of the interactions of the ejected cooling air with the rotor row.

The accurate prediction of this flow by CFD computations is a challenge. First, the rotating cells generate fluctuations at specific frequencies that can not be taken into account by common techniques used to reduce the size of the simulated domain, like the phase-lagged method. Simulations over large sectors are therefore mandatory to model accurately these flow structures. Furthermore, the time range needed to reach a stabilized state is large, and may even be unreachable with current computation resources.

Finally, additional work has to be done on this topic. In particular, experiments are needed to confirm the existence and the characteristics of these oscillations. Simplified configurations may also be useful in order to isolate their source and characterize the instability. Indeed, a removal of the ingested cells could be beneficial to the cooling of the cavity and might also increase the efficiency.

REFERENCES


