GPPS-TC-2022-0104

A new solver for turbomachinery simulations in OpenFOAM: development and application to a 1.5 axial turbine stage

Stefano Oliani
University of Ferrara
stefano.oliani@unife.it
Ferrara, Italy

Nicola Casari
University of Ferrara
nicola.casari@unife.it
Ferrara, Italy

Michele Pinelli
University of Ferrara
michele.pinelli@unife.it
Ferrara, Italy

Giove De Cosmo
University of Bath
gdc32@bath.ac.uk
Bath, Somerset, United Kingdom

Mauro Carnevale
University of Bath
mc2497@bath.ac.uk
Bath, Somerset, United Kingdom

ABSTRACT
In the last decade, a tendency in the usage of open-source softwares for CFD simulations has appeared. Among these, the C++ library OpenFOAM is widely employed in many applications and has proven a suitable framework for developing and delivering numerical tools. Unfortunately, it lacks an implicit density-based solver and its usage is therefore limited when it comes to compressible turbomachinery simulations. The implementation and application of such a new solver in the OpenFOAM environment is presented in this work. This type of solver is suitable to obtain steady and transient solutions of the compressible Navier-Stokes equations when accurate simulation of transonic/supersonic turbomachinery flows is required. The main features of the code are detailed, and a relevant test case is provided to show the solver capabilities. Specifically, a 1.5 axial turbine stage with a stator-rotor cavity is studied through an unsteady simulation in order to capture the interaction between different blade rows and the coherent structures behaviour in the rim seal region.

INTRODUCTION
In the last years, the interest towards opensource CFD softwares has grown steadily, and it is likely to increase further in the future. Among these, the finite volume C++ library OpenFOAM (OF) (Weller et al. (1998)) has recently gathered momentum, rapidly establishing as a state of the art tool for academics and industries. In addition, OF is a mature code, meaning that its off-the-shelf libraries provide the user with a wide variety of complex multi-physics solvers and well-established modeling approaches. These include, but are not limited to, many CFD solvers ranging from incompressible laminar to compressible turbulent flows. Other applications involve multiphase, lagrangian and structural solvers, to cite some. Despite its versatility, the use of OF in the context of compressible turbomachinery flows has been very limited so far. Even if a few dedicated turbomachinery libraries developed by Jasak and Beaudoin (2011) can be found in the literature, this capability has largely lagged until now. The reason for this is twofold. First, the segregated nature of the system matrix solution. Indeed, for a system of coupled PDEs, the solution of one equation at a time often leads to convergence issues, especially when a strong coupling between the physical quantities is expected (e.g. high speed flows). Second, the use of a pressure-based formulation. This method was originally devised by Patankar (1980) for incompressible flows in its SIMPLE form, and was then extended to an entire class of algorithms and also to compressible flows by Issa (1986). Indeed, historically, numerical gas dynamics has followed a different path, with the usage of density-based solvers. Very good reviews can be found in the works of Laney (2008) and Hirsch (2007). The main feature of density-based solvers is their wave detection capability. The key ingredients to achieve this behaviour are the development of upwind schemes able to isolate propagating waves directions, and the usage of high-resolution schemes to prevent the appearance of spurious oscillations in the vicinity of shock-waves. These methods therefore still represent the state-of-art for the calculation of compressible turbomachinery flows and are implemented in many commercial, opensource and in-house CFD codes.

To present, only one density-based solver is available in the official version of OF: rhoCentralFoam. Unfortunately, this is an explicit solver and therefore suffers from stability issues and strict limitations on the Courant number. Conversely, implicit solvers are unrivaled in terms of stability and performances in complex compressible flows, especially

This work is licensed under Attribution 4.0 International (CC BY 4.0)
See: https://creativecommons.org/licenses/by/4.0/legalcode
for steady-state solutions. Therefore, the main aim of the current work is to push forward the current state of the art concerning the simulation of compressible, high-speed flows in OF.

To validate and present the code capabilities, a typical turbomachinery configuration is analyzed. A transient calculation is employed to study 1.5 axial turbine stage with a stator-rotor cavity. This allows to capture the unsteady interaction between different blade rows and also to highlight the formation of coherent structures in the rim seal region. As pointed out by Schädler et al. (2016), the presence of these structures can potentially influence hot gas ingestion inside the cavity space. The authors also suggest a stabilising effect induced by higher purge flow rates. The most relevant parameters for the classification of these structures are their number and rotational speed. See Horwood et al. (2018) for a comprehensive summary on literature data. Although inviscid and viscous mechanisms have been proposed for the physical origin of these structures, the exact physical phenomenon behind their formation and their relation to hot gas ingestion are still not fully understood. Extensive reviews concerning the ingestion phenomenon can be found in Scobie et al. (2016) and Chew et al. (2019). Moreover, the interaction of the purge flow with the main annulus can lead to augmented losses through the blade passages. Therefore, the simulation of this configuration is of potential interest for the engineering community and can help shedding new light on the ingestion phenomenon. At the same time, it represents a suitable test case to assess the new solver performances and capabilities. In this way, the current work aims to represent a step forward for those who want an opensource robust and accurate tool for the simulation of high-speed/turbomachinery flows combined with the high flexibility of OF. The code is available in the authors repository: https://github.com/stefanoOliani/ICSFOam, and can be freely shared, edited and distributed by the users.

METHODOLOGY
Governing equations

In the present work, we solve the unsteady three-dimensional compressible Reynolds-averaged Navier-Stokes (RANS) equations in conservation form:

$$\int_V \frac{\partial Q}{\partial t} dV + \int_{\partial V} (F_c - F_v) dS = 0$$  \hspace{1cm} (1)

where $V$ and $\partial V$ denote the control volume and the related closed surface, respectively. The vector of conservative variables $Q$, the convective flux vector $F_c$ and the diffusive flux vector $F_v$ are given by

$$Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho h \end{bmatrix}, \quad F_c = \begin{bmatrix} \rho u \cdot n \\ (\rho u \otimes u) \cdot n + \rho n \\ \rho u H \cdot n \end{bmatrix}, \quad F_v = \begin{bmatrix} 0 \\ \tau \cdot n \\ (\tau \cdot u + q) \cdot n \end{bmatrix}$$  \hspace{1cm} (2)

where $n$ is the surface outward-pointing normal vector, $\rho$ is the density, $u$ is the velocity, $h$ is the total internal energy, $H$ is the total enthalpy, $\rho$ is the static pressure, $\tau$ is the viscous stress tensor and $q$ is the heat flux vector. The resultant equations represent the conservation of mass, momentum and internal energy for an arbitrary control volume $V$. By using the structure described in the previous section, the continuity, momentum and energy equations are solved together in a coupled manner. Conversely, to obtain the turbulent viscosity, the turbulence equations are solved in a segregated manner in order to exploit the built-in OF structure. In this way, the turbulent quantities can be obtained without modifying the related part of the code, and the user can freely choose the desired turbulence model among the many present in OF. Therefore, support for RANS and LES simulations is automatically included in the new solver. Finally, the ideal gas law is used to close the system of equations. In the present work, the working fluid is calorically perfect air with $\gamma = 1.4$ and $\mu = 1.8 \times 10^{-5}$ Pas.

Numerical discretization

We use a cell-centred finite volume method to carry out the spatial discretization of Eqs. (1), obtaining a set of semi-discretized equations:

$$V D_t Q = R(Q)$$  \hspace{1cm} (3)

where $D_t$ is the physical-time derivative operator and $R(Q)$ is the numerical flux residual term. The dual time-stepping (DTS) technique of Jameson et al. (1981) is typically used to perform the time integration, by adding a pseudo-time term $\tau$ in the equations:

$$V \frac{\partial Q}{\partial \tau} + V D_t Q = R(Q)$$  \hspace{1cm} (4)

When an implicit method is employed to march the equations in pseudo-time to the iteration $n + 1$, the residual is linearized about iteration $n$ as
\[
R(Q^{n+1}) = R(Q^n) + \frac{\partial R(Q)}{\partial Q} \Delta Q^n + \mathcal{O}(\Delta Q^2)
\]
\[
\Delta Q^n = Q^{n+1} - Q^n
\]

Since at each physical step, the system of equations is solved as a steady state problem in pseudo-time, a first-order backward scheme is used for the pseudo-time term. For time-accurate simulations, a second-order backward scheme is employed in the present work. All this being considered, at each iteration one needs to solve the linear system of equations for the solution increment \(\Delta Q^n\):

\[
V\left(\frac{1}{\Delta t} - \frac{3}{2\Delta t}\right)I - \frac{\partial R(Q^n)}{\partial Q} \Delta Q^n = R(Q^n) + \left(V \frac{3Q^n - 4Q^n + Q^n-1}{2\Delta t}\right)
\]

where \(k\) is the current physical time level and \(\Delta t\) and \(\Delta\) represent the pseudo and physical time-steps, respectively. If the numerical flux Jacobian is derived from an exact linearization of the numerical flux \(R(Q)\), Eq. (7) represents a standard Newton iteration for the nonlinear system (4). Nevertheless, only approximate Jacobians are usually employed since an exact linearization of second-order inviscid fluxes requires large storage and can be excessively expensive to compute, as reported by Luo et al. (1998). Here, for a generic face between cell \(i\) and cell \(j\) we choose the approximate Jacobian as follows:

\[
\frac{\partial R(Q_i)}{\partial Q_i} = \frac{1}{2} S_f (J(Q_i) + |\lambda_{ij}|I)
\]
\[
\frac{\partial R(Q_j)}{\partial Q_j} = \frac{1}{2} S_f (J(Q_j) - |\lambda_{ij}|I)
\]

where \(S_f\) is the face area, \(J\) is the convective flux Jacobian and \(|\lambda_{ij}|\) is the sum of the spectral radii of the Roe and viscous flux matrices as described in Blazek (2015):

\[
|\lambda_{ij}| = |u_{ij} \cdot n_i| + c_{ij} + \frac{1}{|x_i - x_j|} \max \left( \frac{4}{3} p_{ij} \frac{\gamma}{Pr}, \left( \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \right) \right)
\]

where \(c_{ij}\) is the sound velocity and the subscript \(ij\) denotes quantities interpolated at the face between cell \(i\) and cell \(j\). This approximate Jacobian form has also the important advantage that the application of LUSGS preconditioner requires only scalar diagonal inversion (Yoon and Jameson (1988)). This significantly speeds up the computation with respect to the inversion of matrix blocks used in other smoothers or preconditioners.

A linear interpolation based on the MUSCL reconstruction-evolution approach of van Leer (1979) is used to achieve a second-order accuracy in space. To avoid spurious oscillations in the vicinity of shocks, the slope of the interpolation is limited to obtain TVD schemes. Several limiters are already available in OF and can be used for this purpose. After the reconstruction, left and right states are available at each internal face of the mesh and are used to compute the numerical flux. Due to variable limiting, these states need not match each other, and approximate Riemann solvers are usually employed to calculate the upwind fluxes basing on waves directions and strength. The reader is referred to Toro (2009) for a comprehensive analysis of Riemann solvers and their application. Three approximate Riemann solver have been implemented: Roe, HLLC and AUSM+Up. The integration of the fluxes (cfr. second term of Eq.1) over all faces of a cell \(i\) gives the numerical flux residual term \(R(Q_i)\). Moving to the implicit part, the upper, lower and diagonal contribution of convective fluxes are given by Luo et al. (1998):

\[
U = \frac{1}{2} S_f (J(Q_j) - |\lambda_{ij}|I)
\]
\[
L = \frac{1}{2} S_f (-J(Q_i) + |\lambda_{ij}|I)
\]
\[
D = \sum_j \frac{1}{2} S_f (J(Q_j) + |\lambda_{ij}|I)
\]
RESULTS AND DISCUSSION

Computational model

The geometry analyzed is the 1.5 axial turbine stage rig of the University of Bath designed by Siemens. The experimental facility was designed to study ingress into the wheel-space cavities of an axial turbine and is discussed in details by Scobie et al. (2016). The rig operates at fluid-dynamically scaled conditions at relatively low Reynolds numbers, and is composed of 32 upstream vanes, 48 rotor blades and 32 downstream vanes. A double radial overlap seal is placed in the wheel-space upstream of the rotor and included also in the computational domain. Thanks to the vane/blade count ratio, a reduced $22.5^\circ$ sector with two vanes and three blades could be employed by exploiting periodic conditions at the pitchwise boundaries. Although the usage of reduced sectors can potentially have an impact on the unsteady structures development inside the wheel-space cavity, a preliminary analysis revealed that the $22.5^\circ$ sector is adequate to capture the relevant flow features. In addition, the computational domain was further simplified by not including the entire wheel-space of the cavity and placing the purge flow inlet at the radius of the first radial seal. A previous analysis with full wheel-space modeling by De Cosmo et al. (2022) showed that at this location the flow typically has also a tangential components. Therefore a swirl angle of $57^\circ$ was prescribed for the purge inlet. This stub domain is the best compromise between computational cost and accuracy.

The boundary conditions used for the simulations are reported in Table 1 and correspond to the experimental test operating conditions.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational Reynolds number, $Re_\phi$</td>
<td>$7.2 \times 10^5$</td>
</tr>
<tr>
<td>Flow coefficient, $C_F$</td>
<td>0.41</td>
</tr>
<tr>
<td>Nondimensional sealing parameter, $\Phi_0$</td>
<td>0.018;0.104</td>
</tr>
<tr>
<td>Inlet total temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Purge flow total temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Outlet static pressure</td>
<td>99300 Pa</td>
</tr>
<tr>
<td>Disc rotational speed</td>
<td>3000 rpm</td>
</tr>
<tr>
<td>Purge inlet swirl angle</td>
<td>$57^\circ$</td>
</tr>
</tbody>
</table>

The rotational Reynolds number, the flow coefficient and the nondimensional sealing parameter are respectively defined as

$$Re_\phi = \frac{\rho \Omega_d b^2}{\mu}; \quad C_F = \frac{W}{\Omega_d b}; \quad \Phi_0 = \frac{U}{\Omega_d b}$$

where $\Omega_d$ is the angular speed of the disc, $b$ is the outer seal radius, $W$ is the inlet axial velocity in the annulus and $U$ is the mean velocity at the purge inlet. To match the operating conditions, appropriate mass flow rates were imposed at the annulus and purge flow inlets, while at the outlet a static pressure boundary condition was set. As shown in Table 1, two different purge flow rates were investigated.

A structured multi-block grid composed of 6.3 million elements was generated using ANSYS ICEM CFD. The tip clearance region of the rotor was fully meshed using six grid points in the radial direction. Nonmatching grid interfaces were employed between stationary and rotating domains. The dynamic mesh support was employed to impose a rotating motion to the rotor domain. A new type of coupled interface was implemented to transfer the required information in non overlapping areas of the domain appearing during the relative motion of different zones. To this purpose, information at both sides of domain interfaces was replicated an integer number of times to cover the full 360 sector and then exchanged. Second-order accuracy in space was achieved by limited linear variables reconstruction from cell centers to faces employing the Van Leer limiter. The parameters for the dual time stepping algorithm were chosen as follows. The physical timestep was selected to resolve 100 instants per blade passing period employing a second-order accurate backward scheme. For the pseudo-time convergence, 20 subiterations with a Courant number of 50 were selected to guarantee a drop in the residuals of at least three orders of magnitude during the inner loops. The two-equations $k-\omega$ SST model of Menter et al. (2003) was used for the turbulence closure of URANS equations. The average $y^+$ values on the airfoils and endwalls are about 14 and 36 respectively, allowing the usage of wall-functions on every no-slip wall.

For both purge flow rates, the unsteady calculations were initialized using a steady simulation with frozen-rotor interfaces and ran for at least 10 complete rotor revolutions. This was necessary to achieve a periodic behaviour of flow quantities in the cavity due to large difference in convective time scales between the main flow and the cavity flow itself. Calculations were run on 1,024 cores and required approximately 100,000 core hours each to achieve the convergence.
Fig. 1a shows the time-average pressure field on solid walls of the domain for the three rows considered in the calculation. To validate the solution, Fig.1b shows the comparison between CFD and experimental data for the time-averaged pressure coefficient on a probe located on the hub downstream of the first vane over two passages. The silhouette on the right shows the location of the probe in the meridional passage. The transient analysis allows also to identify accurately the secondary flows across the blade passages. These are inherently unsteady and their intensity is often underpredicted by steady-state analysis. Fig.2 shows the instantaneous entropy contours in three axial planes located downstream of each row for the low purge flow condition. Dotted white lines indicate the extension of one vane passage. Although the low aspect ratio of the turbine geometry does not give rise to a high degree of secondary flows, some typical vortex structures are found to develop inside the passage. Hub passage vortices (HPV) can be clearly identified at the exit of the two vane rows in Figs.2a and 2c. Downstream of the rotor, tip leakage vortices (TLV) are present and interact with the shroud passage vortices (SPV). As can be noticed from the strong difference from one passage to the next in Fig.2b, these structures are highly unsteady. Moreover, the vorticity generated is maintained through the successive vane row. The tip leakage vortex is the structure showing the highest vorticity and is stretched and pushed against the shroud during the deviation undergone by the flow through the successive row (Fig.2c). These observations are consistent with the experimental findings of Behr et al. (2006) for a similar 1.5 axial turbine stage. Finally, at low spans the downstream vane operates at off-design conditions since boundary layer separation (SB) occurs on the suction side of the vane. This is evident from the big separation bubble in Fig.3a which shows entropy contours at 25% span for the low purge condition. The high entropy zone around mid-chord of the blade passage represents the interaction area between the sealing flow and the incoming flow from the main passage. For the high purge flow condition (Fig.3b) this interaction is clearly enhanced due to the increased mass flow exiting from the cavity. It is also noticed that the interaction area continuously shifts upstream and downstream inside the blade passage according to the relative position of the upstream vane trailing edge. This effect is more evident for the high purge flow case. Clearly, the interaction between the annulus and the rim gap flows influences the amount of gas swallowed into the cavity. In high-temperature gas turbine applications, hot gas ingestion can jeopardize the rotating discs integrity.

**Numerical results**

Fig. 1a shows the time-average pressure field on solid walls of the domain for the three rows considered in the calculation. To validate the solution, Fig.1b shows the comparison between CFD and experimental data for the time-averaged pressure coefficient on a probe located on the hub downstream of the first vane over two passages. The silhouette on the right shows the location of the probe in the meridional passage. The transient analysis allows also to identify accurately the secondary flows across the blade passages. These are inherently unsteady and their intensity is often underpredicted by steady-state analysis. Fig.2 shows the instantaneous entropy contours in three axial planes located downstream of each row for the low purge flow condition. Dotted white lines indicate the extension of one vane passage. Although the low aspect ratio of the turbine geometry does not give rise to a high degree of secondary flows, some typical vortex structures are found to develop inside the passage. Hub passage vortices (HPV) can be clearly identified at the exit of the two vane rows in Figs.2a and 2c. Downstream of the rotor, tip leakage vortices (TLV) are present and interact with the shroud passage vortices (SPV). As can be noticed from the strong difference from one passage to the next in Fig.2b, these structures are highly unsteady. Moreover, the vorticity generated is maintained through the successive vane row. The tip leakage vortex is the structure showing the highest vorticity and is stretched and pushed against the shroud during the deviation undergone by the flow through the successive row (Fig.2c). These observations are consistent with the experimental findings of Behr et al. (2006) for a similar 1.5 axial turbine stage. Finally, at low spans the downstream vane operates at off-design conditions since boundary layer separation (SB) occurs on the suction side of the vane. This is evident from the big separation bubble in Fig.3a which shows entropy contours at 25% span for the low purge condition. The high entropy zone around mid-chord of the blade passage represents the interaction area between the sealing flow and the incoming flow from the main passage. For the high purge flow condition (Fig.3b) this interaction is clearly enhanced due to the increased mass flow exiting from the cavity. It is also noticed that the interaction area continuously shifts upstream and downstream inside the blade passage according to the relative position of the upstream vane trailing edge. This effect is more evident for the high purge flow case. Clearly, the interaction between the annulus and the rim gap flows influences the amount of gas swallowed into the cavity. In high-temperature gas turbine applications, hot gas ingestion can jeopardize the rotating discs integrity.
Figure 3 Instantaneous entropy contours at 25% span for the low and high purge flow cases.

It is therefore interesting to analyse more in detail the fluid dynamics occurring inside the cavity and the rim gap regions. First of all, we compare the results obtained with the present solver with the numerical data from Horwood et al. (2018). The authors employed the well-know and validated TRACE solver developed at DLR Becker et al. (2010). Fig. 4a shows the radial traverses of time-averaged effectiveness for $\Phi_0 = 0.104$ upstream and downstream of the blade row, as sketched in the silhouette. In practice, the effectiveness can be obtained by solving a passive scalar transport equation where a unitary concentration is uniformly injected at the purge flow inlet and transported inside the domain. Available experimental data is also reported for comparison. It can be seen that the two numerical solutions are in good agreement with each other, especially upstream of the blade. Comparing also the results to experimental data, there is a large overprediction at lower spans, while for $r/b > 1.04$ the trend is well reproduced. For the downstream radial traverse, CFD from Horwood et al. appears to better predict the effectiveness peak location related to the migration of purge flow towards higher spans. On the other hand, the present solver better captures the peak entity and the profile decay at lower spans. Fig. 4b reports the radial variation of the swirl profile along the cavity wheel space for the low and high purge flow conditions obtained numerically. Available experimental data at three radial locations is also reported. Swirl is defined as $U_\theta / \Omega d r$, where $U_\theta$ is the tangential component of the absolute velocity. As can be expected, increasing the purge flow rate reduces the corresponding swirl level. Excellent agreement is found between the two CFD solvers for both purge flow conditions. Furthermore, for $\Phi_0 = 0.104$, the experimental data are accurately reproduced throughout the cavity. Conversely, for $\Phi_0 = 0.018$, at lower radii the numerical and experimental results closely match to each other, while near the rim gap bottom the swirl intensity tends to be underpredicted.

To highlight the flow structures developing in these areas, it is useful to study the effectiveness of the rim sealing for the two purge flow conditions. Fig. 5 shows meridional and axial sections of the effectiveness for $\Phi_0 = 0.018$ and $\Phi_0 = 0.104$. The white and black arrows represent the flow swirl direction in the cavity and the annulus, respectively. It can be noticed that for the high purge condition, the cavity is fully sealed, while for $\Phi_0 = 0.018$, main flow ingestion occurs (effectiveness < 1). This is in agreement with the experimental data reported by Scobie et al. (2016). The position of the leading edges of the three blades is denoted by the three grey rectangles. It is worth noticing that effectiveness contours enables to distinguish three coherent structures which form within the rim gap area at both flow rates. According to Horwood et al. (2018), these structures are relatively stable and rotate at the disc speed but, at the low sealing flow rate, they coalesce into a larger structure. This larger unstable structure appears to rotate slower than the disc speed and drives ingress deeper into the wheel-space. On the other hand, at the higher purge flow rate, these structures are suppressed, while smaller ones result more stretched and pushed outside the cavity (Fig. 5b). To calculate the rotational speed of these structures, we follow a simplified approach recently devised by De Cosmo et al. (2022) based on the simultaneous sampling of two probes. These are placed close to each other, but one in the synchronous frame and one in the stationary frame. Basing on the Doppler shift of the signal sampled by the two probes, one can obtain the number and the rotational speed of the structures. Fig. 6 shows the one-sided FFT of the signals of the stationary and rotating probes for the two sealing flow rates. The frequencies are normalized by the disc frequency $f_d$. Probes positions in a meridional plane are depicted in the silhouette enclosed in Fig. 6a. The spectrum of the low purge flow rate reveals, as expected, marked peaks at the blade passing frequency (BPF) in the stationary frame, and at the vane passing frequency (VPF) in the rotating frame. This tonal content is represented respectively by the peaks at $48f_d$ and $32, 64f_d$ in Fig. 6a. The pair [13, 3], instead, corresponds to 16 large scale structures rotating
(a) Radial traverses of time-averaged effectiveness upstream and downstream of the blade row ($\Phi_0 = 0.104$).

(b) Swirl profiles along the cavity wheel space for low and high purge flow conditions.

Figure 4 Comparison between experimental data, CFD results from Horwood et al. (2018) and results from the present solver.

Figure 5 Meridional and axial views of instantaneous effectiveness across the cavity and the annulus.

at $0.81\Omega_{disc}$. This corroborates the remarks relative to Fig. 5a. Finally, the peak at $35f_d$ represents a non linear interaction between the tonal content and the rotating structures. Conversely, as can be seen in Fig. 6b, for high
purge flow rate only the tonal content is present with the first two harmonics of the BPF and the first harmonic of the VPF. Apart from some broad frequency activity, especially at low frequencies, no other evident peaks appear in the spectrum. Therefore, for this case no clear coherent structure pattern appears in the cavity region, further confirming the observations relative to Fig.5b. All the aforementioned findings are in agreement with the numerical data obtained by Horwood et al. (2018) for the same geometry and similar flow conditions, further testifying the goodness of the obtained results and the capability of the new solver of capturing three-dimensional, unsteady flow features in a complex turbomachinery environment.

CONCLUSIONS

A new library for compressible turbomachinery simulations has been developed in the opensource software OpenFOAM and is freely accessible in the authors repository (https://github.com/stefanoOliani/ICSFoam). A novel structure for implicit coupled solutions has been used as the backbone for the implementation of a density-based solver for the solution of complex turbomachinery flows. The simulation of these type of flows has been very limited so far in OF since pressure-based solver typically diverge. To validate and test the code performances an unsteady simulation of 1.5 axial turbine stage was performed. It is shown that the new solver combines robustness and accuracy compared to experimental data and previous CFD calculations. Three dimensional flow features were correctly identified, including endwall secondary flows, tip leakage vortices and interaction between the rim seal flow and the main flow. Furthermore, unsteady structures formation and interaction in the cavity space for two different purge flow rates was investigated. A frequency analysis revealed a slipping behaviour with respect to rotor blades and found good agreement with other literature data. This highlights the capabilities of the new solver of capturing complex three-dimensional flow features evolving in time. For this reason, the present work aims to represent a step forward in the growing application of open-source codes to industrial problems. Given also the wide diffusion of the C++ library OpenFOAM, the research community could benefit from this work as a starting point to enrich the current capabilities of compressible turbomachinery simulations.

ACKNOWLEDGMENTS

The work has been performed under the Project HPC-EUROPA3 (INFRAIA-2016-1-730897), with the support of the EC Research Innovation Action under the H2020 Programme; in particular, the author gratefully acknowledges the support of Mauro Carnevale of University of Bath and the computer resources and technical support provided by EPCC.

References

Becker, K., Heitkamp, K. and Kuegeler, E. (2010), Recent progress in a hybrid-grid cfd solver for turbomachinery flows, Fifth European Conference on Computational Fluid Dynamics ECOMMAS CFD.
