Lire la première partie de la thèse
Part II

LES of combustor-turbine interactions
Chapter 5

Indirect combustion noise generation in a high-pressure turbine

Contents

5.1 Motivation ........................................... 92
5.2 Indirect combustion noise state-of-the-art .................. 93
5.3 DMD test case: 2D entropy spot propagation in a periodic channel .. 95
5.4 Turbine stage simulation set-up ................................ 99
  5.4.1 Mesh ............................................. 100
  5.4.2 Numerical schemes .................................... 100
  5.4.3 Boundary Conditions ................................... 101
5.5 Numerical Results ........................................ 105
  5.5.1 Overall flow topology .................................... 105
  5.5.2 Dynamic Mode Decomposition of the LES flow field ........ 107
  5.5.3 Quantifying the indirect noise and comparisons with the compact theory ............................................ 117
5.6 Conclusions ............................................. 119

This chapter details the first application of the MISCOG method to combustion chamber/turbine interactions: the indirect combustion noise generated across a high-pressure turbine from temperature non-uniformities. As with most previous investigations of this process, a decoupled simulation of a turbine stage from the combustion chamber is performed. The consequence of this choice is that the temperature non-uniformities do not come from the unsteady combustion process. Instead, they are modeled by sinusoidal fluctuations introduced by boundary conditions in the turbine stage alone. The acoustic response of the turbine is then analyzed with a specific interest on the indirect noise generation mechanisms. A part of this work was performed during the author’s participation to the Summer Program of the Center for Turbulence Research (CTR) at Stanford.
Chapter 5: Indirect combustion noise generation in a high-pressure turbine

University. Results have been published in the proceedings of the summer program [177], the ASME Turbo Expo 2015 conference [178] and will also appear in the Journal of Turbomachinery.

5.1 Motivation

According to the International Civil Aviation Organization (ICAO), the world passenger traffic has increased on average by 5.8% per year throughout this decade and the European Union predicts that without any substantial technological improvements, the number of people in Europe that will be affected by high levels of aircraft noise will double by the year 2026 [179]. Consequently, ACARE’s Vision 2050 calls for increased research in the field to reduce the perceived noise by 65% until 2050, with respect to a new aircraft built in 2000. Engines are the main noise source in commercial and military aircraft. For helicopters, the main noise source is still and will remain for some time the rotor blades of the vehicle, hence reducing the significance of the gas turbine noise. Nonetheless, recent studies (for example in the frame of the TEENI project) clearly indicate traces of engine noise emitted from such aircraft.

The first studies on gas turbine noise, carried out by Lighthill [180], showed the importance of jet noise on the overall noise emitted by an aircraft. Lighthill showed that jet-noise scaled with the eighth power of the jet-exhaust speed, meaning that doubling the jet speed would lead to an increase of the acoustic intensity level by nearly 50 dB. The development of the turbofan engine alleviated significantly this issue thanks to the fact that the majority of the propulsion comes from a cold by-pass flow stream, with large mass flow and low speed, resulting in a drastically reduced jet noise. Jet noise has been reduced to the point that the noise emitted from the other components, notably from the fan, the compressors/turbines and the combustion chamber, is becoming more important. Figure 5.1 compares the noise contributions of the different gas turbine components and the noise directivity from an early turbojet engine and a modern turbofan engine. It is evident that in the latest designs fan noise is the prevalent source. However, as further reductions on fan noise are achieved, the relative importance of combustion noise, also called core noise, whose contribution stayed unchanged over the years, is increasing. This suggests that further research on combustion noise is essential if a continuous reduction of the overall engine noise levels is envisioned for the next years. This has led to a renewed interest in the topic with several dedicated EU projects forming the last five years, such as TEENI¹ and RECORD².

Combustion noise is principally a low-frequency noise generated in the combustion chamber of gas turbines and arising from two main mechanisms, as mentioned in Chapter 1. First, direct noise emanates from the acoustic waves created at the unsteady flame front and propagated through the rest of the engine. It is a source of noise that has received considerable attention in the past [181, 182, 183, 184]. Second, the unsteady combustion will give rise to low-frequency temperature fluctuations, or entropy waves,

¹Turboshaft Engine Exhaust Noise Identification
²REsearch on Core Noise Reduction
that are convected with the flow velocity to the combustor nozzle and turbine. There, the waves are subject to significant acceleration and distortion in the blade passages, generating acoustic waves in the process. This noise generation mechanism is called indirect and its importance is twofold: (a) it increases the noise signature of the engine and (b) the acoustic waves propagating upstream can impact the thermoacoustic stability of the combustion chamber [1, 35]. Leyko et al. [18] computed analytically the ratio between indirect and direct combustion noise for different operating conditions, modeling the flame in the combustor and the nozzle at the combustor exit as jump conditions. The results, Fig. 5.2, indicate that for the typical operating conditions of aeronautical gas turbines (red point), indirect combustion noise will likely be the dominant source of combustion noise. Yet its actual relevance remains controversial.

5.2 Indirect combustion noise state-of-the-art

Due to the complexity of a full 3D HPT, past theoretical and numerical studies have used simplified turbine-like geometries. The first in-depth analyses on turbine-like geometries focused on the propagation of entropy waves through quasi-1D nozzles. In this context, Marble and Candel [37] developed an analytical method to evaluate the transmission coefficients of acoustic and entropy waves propagating through a compact quasi-1D nozzle, its length being significantly smaller than the wavelength of the incoming waves. More recently, Duran and Moreau [34] proposed an analytical method to calculate the transmission coefficients of general quasi-1D nozzles, removing the compact nozzle assumption. These analytical methods, accompanied by numerical predictions from LES, have been evaluated on the experimental Entropy Wave Generator [40], with [18] and [42] reporting
Chapter 5: Indirect combustion noise generation in a high-pressure turbine

Figure 5.2: Estimation of the ratio $\eta$ between indirect and direct noise by an analytic approach. The ratio $\eta$ is plotted here as a function of the Mach number $M_1$ representing the Mach number in the combustion chamber and at the nozzle inlet, and of the Mach number $M_2$ representing the outlet nozzle Mach number - Typical combustor operating point indicated with a red point [18].

good agreement on both subsonic and supersonic operating conditions. The same problem was also treated theoretically by Howe [36], who used the compact hypothesis with an acoustic analogy and highlighted the impact of the entropy wave form on the generated noise. A potential coupling of indirect combustion with vortex noise from separated flow regions at the nozzle walls was revealed to be an influencing factor on the measured noise.

The theory of Marble and Candel for nozzles was extended to 2D compact blade rows by Cumpsty and Marble [38], taking into account the reorientation of the flow. This is achieved by imposing an additional constraint, the Kutta condition at the blade trailing edge. The method was originally conceived for a single blade row and tested against numerical simulations by several authors, both for compact and non-compact frequencies [185, 186]. It was extended to multi-stage turbines by Duran et al [187] and was compared against numerical simulations of a 2D high-pressure turbine stage as well as experimental results of [39] for a 3-stage turbine (NASA test case 1629).

The present work is the first numerical evaluation of the indirect combustion noise generated in a realistic, fully 3D, transonic high-pressure turbine. As it was observed in the previous chapters, the flow across a realistic turbine is much more complex than in 2D simulations, where no endwall effects are present and turbulence evolves differently. The operating point of these 2D studies was also subsonic [34, 185], instead of the typical transonic conditions encountered in real configurations. As a result, the mixing of the entropy waves as they go through the turbine and the indirect noise generation is expected to be altered. This simulation can, thus, serve as an additional validation of previous
5.3 DMD test case: 2D entropy spot propagation in a periodic channel

To illustrate the DMD capabilities, a simple test case is employed: the propagation of an entropy wave across a 2D, periodic in the transverse coordinate, channel. To remove any diffusion effects, the Euler equations are resolved. The inflow consists of a uniform axial velocity $U = 10 \, m/s$ at ambient conditions, i.e $T = 300 \, K$ and $P = 1.01298 \, bar$ respectively. At the inlet a sinusoidal entropy wave of frequency $f = 4 \, kHz$ and amplitude $T_p = 20 \, K$ is also introduced through the characteristic boundary conditions\(^2\). A schematic of the test case with the boundary conditions employed is presented in Fig. 5.3.

\(^2\)A description on the introduction of waves through the characteristic boundary conditions is presented in the next section.
Chapter 5: Indirect combustion noise generation in a high-pressure turbine

The simulation run for four periods of the wave frequency and the sampling frequency for the flow field snapshots is 10 kHz. The sampling frequency at the temporal probe is the same as for the DMD and is chosen to be superior to the Nyquist frequency. Figure 5.4 depicts the temperature of the flow field obtained from an instantaneous solution. The form of the injected sinusoidal entropy wave with amplitude 20 K is evident. The entropy pulsation is convected by the flow and its wavelength can be easily computed:

\[ \lambda = \frac{U}{f} = 0.0025 \text{ m}. \]

The maximum and minimum temperature in the domain, as a function of time, are presented in Fig. 5.5(a). The recovered values correspond perfectly to \( T \pm Tp \), highlighting that the entropy waves are injected correctly and no additional perturbations on the temperature are generated. An important requirement is that the forcing does not generate acoustic waves. If acoustic waves are generated by the boundary conditions they can pollute the indirect noise measurements. To verify that, the maximum and minimum pressure of the domain are also investigated, Fig. 5.5(b), and indicate that the difference between the two is less than 0.2 Pa, confirming that the entropy forcing is, indeed, "quiet".

Figure 5.6 depicts the temperature spectrum from the DMD analysis of the flow field snapshots along with the FFT of the temperature signal recorded at the temporal probe. It is evident that DMD captures correctly the introduced wave both in terms of frequency (4 kHz) and amplitude (20 K) with no other significant mode appearing. Note that comparatively and for this set of sampling frequency and signal duration the FFT is incapable to capture correctly neither the amplitude nor the frequency. Improving the frequency prediction would require increasing the FFT length to a size that divides
the frequency of interest closer to an integer or use of the zeropadding technique in the signal. Improving the amplitude can only be obtained by simulating longer periods of the phenomenon. These difficulties typical of FFT analysis confirm that DMD can provide reliable information with a considerably reduced number of snapshots and without additional treatments.

To confirm further that DMD captures accurately the introduced waves, the spatial form of the 4 kHz mode, in terms of temperature modulus and phase across the domain, is presented in Fig. 5.7. The depicted temperature modulus (top) is, indeed, showing the right entropy wave amplitude with minimal variation across the domain, despite the fact that only 4 periods T of the phenomenon were effectively computed. The phase of the temperature, Fig. 5.7(bottom), provides information on the propagative direction of the waves. As it changes linearly across the axial direction and there is no variation across the transverse coordinate, purely axial propagating entropy waves are observed, as anticipated for this simple test case.

From these results, it is evident that DMD can provide a wide range of information, both global and local, at an affordable cost and high precision. To investigate further the limits of DMD, a parametric analysis of the temperature spectra is performed in the following: a) for a varying sampling frequency and a total runtime of 4 periods and b) for a varying runtime and a constant sampling frequency of 10 kHz.

**Effect of sampling frequency**

The first step is to evaluate the impact of the sampling frequency. A method capable of finding the oscillatory motions with low sampling frequencies is particularly advantageous: the memory requirements for storing the necessary information indeed would reduce drastically and the CPU cost decreases if the code does not need to pass through the routines that create solution files. These aspects are particularly important in large LES where storage requirements of a single solution file can be of the order of Gigabytes.

In this simple test case there are no other oscillatory phenomena present apart from the entropy forcing so the effect of the sampling frequency can be straightforwardly evaluated. The sampling frequency can be reduced up to the Nyquist frequency, that is twice the
Chapter 5: Indirect combustion noise generation in a high-pressure turbine

The temperature spectra for sampling frequencies ranging from 8 – 40 kHz are presented in Fig. 5.8. The total runtime remains 4 periods of the entropy wave frequency. It is evident that even with the minimum possible sampling frequency of 8 kHz the pulsating mode is recovered, albeit with an overpredicted amplitude. For sampling frequencies of 10 kHz and higher the amplitude prediction improves considerably and the results reach the desired 20 K amplitude.

Effect of runtime

Another important parameter, apart from the sampling time, is the runtime during which the flow snapshots are recorded. It is desirable for any frequency domain method to be reliable after few periods of the oscillating phenomenon of interest have been simulated by the CFD solver. This limits the computational cost and facilitates the post-processing since again less memory is required to save and process the data. As was observed in Fig. 5.6, FFT has difficulties in recovering oscillatory phenomena with as few as 4 periods. Figure 5.9 depicts the DMD temperature spectra for different runtimes (hence different total number of snapshots) and constant sampling frequency of 4 kHz. It is evident that DMD recovers the mode with only 2 periods and with approximately the correct amplitude. A slight improvement on the amplitude prediction is the only outcome of the runtime increases. These findings highlight the potential of the method on more realistic flow configurations.

These findings suggest that DMD is a very reliable and robust method. However, as the proposed test case is very simple and lacks any stochastic fluctuations, a convergence
5.4 Turbine stage simulation set-up

The objective of this chapter is to investigate the generation of indirect combustion noise across a fully 3D turbine stage. To this end, the transonic MT1 turbine stage [2] is chosen for this study as it provides a validated and realistic geometry of a high-pressure turbine. As in chapter 4 the scaled geometry is employed with 1 stator blade and 2 rotors (12 degree periodicity) to reduce the computational domain. Two simulations were performed: a) one with a steady inflow that serves as a reference case and b) one where an entropy wave train is introduced at the inlet to evaluate the indirect combustion noise generation process.

In the following, the principal flow characteristics are first identified for both the steady inflow and the forced cases. The analysis of the steady inflow reference simulation using DMD is performed and the most important natural modes are identified. The DMD global spectra of the forced case are then investigated against those of the steady inflow case to evaluate the impact of the incoming entropy waves on the noise generation of the turbine. Afterwards, the response of the flow field at the pulsation frequency for the forced case is examined in further detail on the basis of DMD and transmission coefficients are obtained for the generated acoustic waves. To finish, the results are compared to those obtained with the 2D theoretical model of [38] and 2D pseudo-LES of a similar turbine configuration [189].
5.4.1 Mesh

The mesh employed is, as for the previous MT1 simulations, a fully 3D hybrid mesh with 10 prism layers around the blades and tetrahedral elements in the passage and endwalls. Three views of the mesh are provided in Fig. 5.10. It is composed of 8.1 million cells in total for the stator domain and 10.5 million cells for the rotor domain. It is more refined than the coarse mesh used in Chapter 4 to improve the acoustic predictions and is designed to place the first nodes around the blade walls deeper in the logarithmic region. Note also that a low aspect ratio for the prisms, set to $x^+ \approx 5y^+ \approx 5z^+$, is maintained to permit a good resolution of streamwise/spanwise flow structures. In the rotor tip region, there are approximately 15 cell layers, as shown in Fig. 5.10(c), in an effort to keep the time step reasonable. In wall units, the maximum values of $y^+$ measured around the blade is approximately 50.

5.4.2 Numerical schemes

As in the previous chapters, the AVBP solver with the MISCOG method is used to perform LES of the MT1 turbine stage. The numerical integration in this chapter is handled by the two-step, finite-element TTGC \cite{119} scheme that is 3rd order accurate in time and space and explicit in time. It is chosen over the cheaper LW scheme for its performance in handling acoustics, an important parameter in this problem. This scheme is used in conjunction with the Hermite-type 3rd order interpolation for the data exchange at the overlap zone, ensuring low dissipation and low dispersion of the rotor/stator interactions, while preserving the global order of accuracy of the numerical
5.4 Turbine stage simulation set-up

![Figure 5.9: DMD temperature spectrum of the 2D test case for different runtimes. Sampling frequency 10 kHz.](image)

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Euler</th>
<th>Navier-Stokes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Subsonic Inflow</td>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td>Subsonic outflow</td>
<td>1</td>
<td>4</td>
</tr>
</tbody>
</table>

Table 5.1: Number of conditions required for a well-posed 3D simulation [25].

The computational cost for simulating the time corresponding to one 360-degree revolution of the turbine stage is 30K CPU hours.

5.4.3 Boundary Conditions

Performing numerical simulations requires from the user to specify certain boundary conditions at the domain limits. Table 5.1 presents the number of necessary conditions to ensure a well-posed problem near the boundaries [25]. In the LES context particular attention on the boundary conditions is required as the compressible LES formalism allows the resolution of acoustic waves generated by the flow, which will propagate and reach the boundaries of the domain. Imposing the necessary boundary conditions in a hard way will lead to the waves being reflected back into the domain thus modifying and polluting the flow field. As a result, treating these waves is necessary to ensure minimal reflection while respecting the user imposed flow parameters.

In AVBP, the boundary conditions follow the NSCBC formulation [118]. In this method, the flow is decomposed into characteristic waves crossing the boundaries using the Linear One-Dimensionsal Inviscid (LODI) relations. Figure 5.11 depicts the characteristic waves crossing an inlet and an outlet. Two acoustic waves (upstream propagating...
$w^-$ and downstream propagating $w^+$) are identified and are complemented by an entropy wave $w^s$ and two waves related to transverse variations of the velocity. At the inlet, four of these waves are entering the domain so 4 physical conditions need to be specified. At the outlet one condition is sufficient as only the wave $w^-$ is entering the domain at this position.

**Outlet**

At the outlet of the domain, Fig. 5.11 demonstrates that there is only one acoustic wave ($w^-$) coming in the domain that needs to be treated. With the NSCBC method, for the acoustic wave entering the domain from the outlet, $w^-$, one writes:

\[
\frac{\partial w^-}{\partial t} - L^- = 0.
\]  

(5.1)

where $L^-$ is the amplitude variation of the characteristic wave $w^-$ that would enter the computational domain. For a perfectly non-reflecting boundary condition $L^- = 0$. In practice however this is not possible. In most commonly used outlet conditions, the user wants to impose a static pressure. To recover this desired property while being acoustically nearly non-reflection, a partially non-reflecting formulation is usually used:

\[
L^- = K_p \frac{p - p_{\text{ref}}}{\gamma p_{\text{ref}}}. 
\]  

(5.2)
In Eq. (5.2) $K_p$ is the relaxation coefficient and $p_{ref}$ the reference pressure imposed by the user. With this formulation, small values of the relaxation coefficient result in a less reflecting outlet, while large values ensure that the pressure is close to the user-defined target but wave reflection is stronger [190]. For real flow problems where locally 1D flow does not apply, a modification of the NSCBC, described by Granet et al. [191] can also be used, to ensure that vortices created by the blade wakes are leaving the domain without generating noise.

**Inlet**

At the inlet, the boundary is treated in a similar way. The user-specified physical conditions in the investigated case include the total pressure and the total temperature. The incoming acoustic wave (since now it is an inlet) will be a function of these 2 variables. The formula for the incoming acoustic wave reads:

$$\frac{\partial w^+}{\partial t} - L^+ = 0. \tag{5.3}$$

with the amplitude of the incoming acoustic wave written as [192] (transverse fluctuations are ignored):

$$L^+ = -\frac{e_c}{K_+} K_{tt}(Tt - Tt_{ref}) - \frac{T}{\rho K_+} K_{pt}(Pt - Pt_{ref}). \tag{5.4}$$

where $e_c$ is the kinetic energy, $Pt$ and $Tt$ are the total pressure and temperature, $K_{pt}$ is the relaxation coefficient on the total pressure, while $K_+ = \frac{(c+u_n)T}{2} + \frac{cT}{2c}$, $u_n$ being the velocity normal to the boundary and $c$ the speed of sound.
Chapter 5: Indirect combustion noise generation in a high-pressure turbine

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational Speed (rpm)</td>
<td>9500</td>
</tr>
<tr>
<td>Inlet total pressure (Pa)</td>
<td>4.56e5</td>
</tr>
<tr>
<td>Inlet total temperature (K)</td>
<td>444</td>
</tr>
<tr>
<td>Mass flow (kg/sec)</td>
<td>17.4</td>
</tr>
<tr>
<td>Outlet static pressure (Pa)</td>
<td>1.4 \times 10^5</td>
</tr>
<tr>
<td>Wave amplitude (K)</td>
<td>20</td>
</tr>
<tr>
<td>Wave frequency (Hz)</td>
<td>2000</td>
</tr>
</tbody>
</table>

Table 5.2: Operating conditions of the MT1 turbine.

Note that no turbulent fluctuations are added at the inlet, as only pure indirect combustion noise generated in the turbine is investigated. For the forced simulations, sinusoidal entropy spots are introduced through the corresponding characteristic equation:

$$\frac{\partial w^s}{\partial t} - L^s = 0.$$  \hfill (5.5)

where $w^s$ is the entropy wave. In an unforced simulation, $L^s = \frac{\rho (c + u_n) L^+}{2 C_p T} + \frac{\varphi}{T} K_{tt}(T t - T t_{ref})$, with $K_{tt}$ being the relaxation coefficient of the total temperature. For the forced simulations, one should instead write:

$$L^s = \frac{\rho (c + u_n) L^+}{2 C_p T} + \frac{\rho}{T} K_{tt}(T t - T t_{ref} - T t_f^*) + \frac{\partial w_f^s}{\partial t},$$  \hfill (5.6)

where $w_f^s = A \sin(\omega t)$ is the entropy temporal signal of amplitude $A$ and frequency $\omega$ injected in the domain and $T t_f^*$ is the fluctuation of the total temperature due to this wave.

For the forced LES, the frequency of the imposed waves is fixed at 2 kHz and the amplitude is 4.8% of the inlet total temperature. This value has been shown to generate acoustic waves of linear dynamics [185]. The reduced frequency of the forcing is $\Omega = f L_n/c_0 = 0.1$, with $L_n$ being the rotor chord length, $f$ the forcing frequency and $c_0$ the speed of sound at the turbine inlet. While combustion noise is usually associated to lower frequencies, 2 kHz was found to be approximately the limit of validity of the compact theory in 2D configurations [189] and renders the simulations more affordable. Additionally, due to the complexity of this high Reynolds transonic 3D turbine, a monochromatic pulsation is preferred over a more realistic broadband pulsation in an effort to distinguish pure indirect noise from other sources of noise more easily. Note that no acoustic waves are introduced into the computational domain by this approach, as was shown in section 5.3. The operating and boundary conditions employed in this work are summarized in Table 5.2.
5.5 Numerical Results

5.5.1 Overall flow topology

The overall flow topology is analyzed for the two cases. First and in an attempt to validate the mean flow predictions with this mesh (of an intermediate resolution compared to MESH1 and MESH2 of Chapter 4), the isentropic Mach number and the static pressure across the stator and rotor blades are respectively shown on Fig. 5.12(a) and 5.12(b) at mid span for the steady inflow case. The agreement with the experimental measurements is fair and the plots correspond well with the results obtained in the previous chapters for this configuration.

Looking at the full 3D field, as before, a particularly complex flow field is revealed. Figure 5.13 depicts density gradient contours (in logarithmic scales) of the flow across a cylindrical cut at mid-span of the turbine for the steady inflow and pulsed cases, Figs. 5.13(a) and 5.13(b) respectively, complemented by a view in an x-normal plane near the rotor trailing edge for the steady inflow case, Fig 5.13(c). Some of the phenomena highlighted in Fig. 5.13 are the shock/boundary layer interaction on the suction side of both the stator and the rotor (positions A and B), vortex shedding from the trailing edge of the blades and the accompanying acoustic waves emitted (position C), as well as strong secondary flows developing at the endwalls (positions D and E), as observed in Chapter 4. For the pulsed case, Fig. 5.13(b), in addition to the previously highlighted phenomena, the planar entropy waves approaching the stator are also evidenced (position F). As these waves go through the stator passages they get distorted and partially mixed by the blade wakes before being cut by the passing rotors. The mixing and the developing turbulence clearly make the entropy waves less visible in the rotor domain.

Strong 3D secondary flows are highlighted by Q-criterion isosurfaces for an instantaneous solution of the unpulsed case, Fig. 5.14. It can be observed that on the stator suction side streaky structures are developing as the trailing edge is approached, which result in a flow boundary layer laminar-to-turbulent transition at the trailing edge and a
Figure 5.13: Contours $\frac{\nabla \rho}{\rho}$ of an instantaneous solution at mid-span for the steady inflow (a) and pulsed cases (b). Contours of the same variable and at an x-normal plane near the rotor exit for the steady inflow case (c).
fully turbulent vortex shedding. These streaky structures differ from the low-resolution simulations of Chapter 4, Fig. 4.15, highlighting the effect of the mesh refinement on the near-wall predictions. Significant activity is also present at the rotor tip, where the tip leakage vortex dominates other unsteady activities.

5.5.2 Dynamic Mode Decomposition of the LES flow field

A frequency domain analysis is performed by applying DMD to a set of instantaneous flow fields to identify the most important noise-generating modes and their origins. It can also be effectively used to investigate both qualitatively and quantitatively the generated combustion noise.

To obtain converged and accurate statistics for the flow, especially in the highly turbulent rotor-blade wake, both the steady inflow and the pulsed simulations ran for a total of 10 periods of the pulsation frequency. As will be shown later in this section, the presence of turbulence necessitates this increase in runtime, in contrast to the findings in the simple 2D test case. To avoid aliasing, the sampling frequency needs to be high enough to include all the important high-frequency phenomena. In this case the vortex shedding from the stator trailing edge is the most significant and resolving it, as well as its first harmonic, is necessary. The necessary sampling frequency was determined to be 120 kHz using a simple FFT of a temporal signal recorded at a probe in the stator wake. Lower sampling frequencies were attempted (60 kHz and 30 kHz) but aliasing errors were present. Since DMD is memory consuming, the decomposition is performed at cylindrical blade-to-blade planes at mid-span with the signal including the six principal primitive variables: pressure, temperature, the three velocity components and density. For the pulsed case, a set of x-normal planes at the inlet and outlet of the turbine stage is also employed to measure the incoming/outgoing acoustic and entropy waves as well as the

Figure 5.14: Q-criterion of an instantaneous solution across the turbine stage.
associated transmission and reflection coefficients.

DMD of the steady inflow case

Before analyzing combustion noise generated in the forced case, DMD is performed for the steady inflow case to evaluate the principal sources of activity in the MT1 turbine. Figures 5.15 and 5.16 show the DMD temperature and pressure spectra of the flow in the stator (left) and rotor (right) domains (azimuthal cuts) for the unforced LES. Note that for this specific configuration, the BPF is 9.5 kHz and 4.75 kHz respectively in each domain. The depicted frequency range is 0-60 kHz, which corresponds to the sixth harmonic of the BPF for the stator and the twelfth for the rotor. It is evident both in the temperature and pressure spectra that the rotor/stator interactions are dominant, with the highest peaks located at the BPF of each domain and its harmonics. The second frequency band characterized by increased amplitudes is observed around the sixth harmonic of the BPF for the stator domain. These modes are related to the vortex shedding and the acoustic waves generated from the stator trailing edge. The corresponding mode (12 BPF) in the rotor domain is much weaker. The spatial structure of the BPF mode for each domain (9.5 kHz and 4.75 kHz), along with the common 57 kHz mode (6 BPF for the stator and 12 BPF for the rotor domain) can be visualized to identify the areas of highest amplitudes.

Figure 5.17 shows the temperature and pressure modulus and phase of the BPF mode at mid-span for each domain. In the stator domain, the highest amplitude both for pressure and temperature occurs near the suction side and close to the trailing edge (position 2 in Figs. 5.17(a) and 5.17(c)). This area of the blade is the closest to the passing rotors and thus experiences the largest fluctuations. The fluctuations originating in this area do not stay confined but also propagate upstream, principally through the stator suction side (position 1). In the rotor domain, the BPF corresponds to the rotor blades encountering the passing wakes from the stators. From the temperature and pressure phases, Figs. 5.17(b) and 5.17(d), it can be seen that at the rotor inlet (position 3) the
5.5 Numerical Results

The temperature as well as pressure modulus and phase of the 57 kHz mode (6 BPF and 12 BPF for the stator and rotor respectively) are depicted in Fig. 5.18. The principal area of activity is the stator trailing edge, where strong acoustic waves are generated. These waves are linked to the oscillating shear layers issued by vortex shedding from the stator wake. The waves generated on the pressure side propagate towards the suction side of the neighboring blade (position 1), while the ones formed from the suction side shear layer tend to move upstream (position 2). It is worth noting though that this activity appears to be largely confined between the stator blades. These observations obtained from DMD seem to match well the phenomena observed in Fig. 5.13(a), position C. In the rotor domain, as seen in the spectra of Figs. 5.15 and 5.16, the effect of the waves is largely attenuated with only the trailing edge showing noticeable amplitude.

DMD results of the forced case

Figures 5.19 and 5.20 show respectively the DMD temperature and pressure spectra of the flow in the stator (left) and rotor (right) domains (azimuthal cuts) for the stationary and forced LES. The depicted frequency ranges of Figs. 5.19 and 5.20 cover up to a frequency equal to the BPF (as seen in each domain) plus the forced Entropy Wave Frequency (EWF) 2 kHz. For the steady inflow case, Figs. 5.19 and 5.20 reveal that there is no mode at the pulsation frequency. For the forced LES, pure entropy waves are injected which create a distinctive peak in Fig. 5.19, seen both in the stator and rotor domains. Furthermore and although no acoustic forcing is imposed by the entropy waves, Fig. 5.20 reveals that a pressure mode with a distinctive peak appears at the forcing frequency. This indicates that acoustic waves have been generated, confirming the

Figure 5.16: DMD pressure spectrums of the stator (left) and rotor domain (right) at mid-span - Steady inflow case.
Figure 5.17: DMD BPF mode at mid-span - Modulus and phase of the temperature (a and b) and pressure (c and d) respectively.
Figure 5.18: DMD 57 kHz mode at mid-span - Modulus and phase of the temperature (a and b) and pressure (c and d) respectively.
indirect noise generation mechanism. The imposed EWF also leads to the appearance of interaction modes between the BPF and this forcing with noticeable pressure peaks arising at $BPF \pm EWF$. This type of interaction between combustion noise and rotor/stator tones, yielding scattered tones, has also been measured on full scale engine tests [193].

The mode of primary interest obtained by DMD corresponds to the one at the EWF. Its spatial form can be visualized to identify the spatial activity at the origin of the EWF pressure peak present in Figs. 5.19 and 5.20. The modulus and phase of temperature, as well as pressure of the DMD mode are depicted in Fig. 5.21 at mid-span. The temperature modulus at the inlet, Fig. 5.21(a), is almost uniform and equal to 20 K, corresponding to the plane entropy waves injected in the domain. The phase at the same position, Fig. 5.21(b), indicates that the waves in this area are simply convected by the flow and remain planar. Further downstream in the blade passage, the modulus gets distorted with a reducing maximum value as found in previous 2D propagation studies in a stator [18] and in a turbine stage [189]. The phase also reveals an asymmetric distortion of the planar waves. This distortion is caused by the strong flow acceleration and turning.
imposed by the blades. An azimuthal component of the velocity vector is created, with the higher velocity near the suction side resulting in an asymmetric propagation velocity across the azimuthal coordinate. In the rotor domain, due to the rotation the blades see rather uniform entropy waves, with the phase at the rotor inlet being practically planar and perpendicular to the axial direction. As these waves pass through the rotors, they get deformed in a similar fashion as in the first blade row. Such strong distortions of the injected entropy wave at both the stator and the rotor lead to scattering in additional azimuthal modes [18]. This energy redistribution mechanism can explain the additional peaks observed in the pressure and temperature spectra of Figs 5.19 and 5.20.

As anticipated in the discussion based on Figs. 3 and 4, convected temperature spots produce pressure waves in both blade rows at the forcing frequency. The pressure modulus and the phase of the DMD mode at EWF, pictured in Figs. 5.21(c) and 5.21(d), reveal a complex pressure field. A significant peak of the modulus exists between the suction side at 20% chord length and the trailing edge on the pressure side, as the domain is periodic in the azimuthal direction (position 1). In this area the phase hardly changes, Fig. 5.21(d), suggesting an excited cavity mode that stays confined between the blades, making it irrelevant to combustion noise where only propagating waves are of interest. The second area of high pressure modulus can be observed on the suction side close to the trailing edge (position 2), with the sharpest peak corresponding to a shock. In the rotor domain, both the pressure modulus and phase appear to simply follow the flow, with a smooth change of phase throughout indicating simple wave propagation. To finish, a large peak in the pressure and temperature modulus at the trailing edge of the blade corresponds to another trailing edge shock (position 3). At the outlet, the acceleration of the temperature spots through the rotor as well as the acoustic waves generated in the stator and transmitted in the rotor are strong enough to yield a significant pressure trace (non-zero modulus) that sticks above the broadband level. All these features identified in the stator and rotor domains are at the root of the indirect combustion noise emitted and will be quantified later in this work.

Convergence of the DMD

One of the advantages of DMD is the quick convergence of the method, particularly when dealing with oscillatory motions [188, 194] as shown in the 2D test case. The case of 3D turbine stage, however, is much more complex. While the phenomenon of interest consists of oscillating acoustic and entropy waves of known frequency, it coexists with broadband turbulence, shocks, blade wakes, boundary layers and secondary flows that might alter the convergence of the DMD in terms of temporal resolution and overall length of the treated simulations. To evaluate this potential source of uncertainties, DMD on the pulsed case at mid-span is performed with a varying number of snapshots and the same constant sampling frequency, i.e the length of the simulation is modified.

Figures 5.22 and 5.23 depict the DMD temperature and pressure spectra of the pulsed case for five different simulation runtimes, each equal to a multiple of the period $T = \frac{1}{EWF}$, which relates to the primary frequency of interest in this work. The sampling frequency for the snapshots is constant and equal to 120 kHz, as in the previous section. The first
Figure 5.21: DMD 2 kHz mode at mid-span - Modulus and phase of the temperature (a and b) and pressure (c and d) respectively
5.5 Numerical Results

Figure 5.22: DMD temperature spectrums of the pulsed case with different number for different runtimes - stator (left) and rotor domain (right) at mid-span.

Figure 5.23: DMD pressure spectrums of the pulsed case with different number different runtimes - stator (left) and rotor domain (right) at mid-span.
conclusion that can be drawn is that for the EWF amplitude there is good agreement for all runs with a duration above $3T$. For a run time of $1T$, EWF in the stator is found to be shifted to slightly above 2 kHz, while in the rotor domain no mode at 2kHz is present. Regarding the BPF mode, a relatively good agreement is also observed, particularly for runtimes above $6T$. Most differences appear for the interaction modes $BPF \pm EWF$, where a trend of reduced pressure amplitudes appears as the run-time increases. Regarding the overall spectra, it can be observed that, as more snapshots are added to the signal, the amplitudes of the modes with irrelevant frequencies drop. This indicates that non-coherent broadband phenomena, such as turbulent fluctuations, are present and should not be interpreted as coherent or significant modes. For the cases with $1T$ and $3T$ of total runtime, for example, there are several notable peaks that either disappear or are largely reduced when more snapshots are added. Note that for the EWF, where combustion noise will occur, 6 periods $T$ of run-time and above appear adequate for the method to converge.

These results highlight that DMD is more robust when treating purely coherent periodic phenomena. Its use in fully turbulent flows, with stochastic fluctuations and broadband noise present, makes the method prone to reveal more coherent modes than actually present. These flows, therefore, require longer runtimes compared to basic test cases to eliminate such discrepancies. To alleviate this shortcome, a modified version of DMD, called Sparsity-Promoting DMD has been recently developed [195].

**Sparsity-Promoting DMD**

The spectra of Figs. 5.19 and 5.20 reveal that several other modes are also present around the EWF. Combining this with the fact that the amplitudes of irrelevant modes can require a large amount of snapshots for convergence, it is desirable to be able to evaluate the most important contributions in terms of noise generation and eventually clean up the spectra. To do so automatically a modified version of the DMD has been developed [195] and called the Sparsity-Promoting DMD (SPDMD). This advanced version of DMD aims at selecting the long-standing coherent modes that generate noise while removing the fast decaying ones, typically present because of turbulence, by employing a user-defined regularization parameter that controls the balance between accuracy and a dataset with a reduced set of modes.

In the following, the SPDMD is performed on pressure using the same set of instantaneous flow fields as in the previous sections, to identify the most important noise-generating modes in the flow. The intention of this analysis is to verify that this optimized method will recover the pulsation mode and confirm its significance. Figure 5.24 depicts the original pressure DMD spectrum with all the modes present complemented by the sparsity-promoting spectrum superimposed for the turbine inlet and outlet respectively. Both diagnostics provided in Fig. 5.24 are measured at the x-normal inlet and outlet planes for the pulsed case, as it is where the combustion noise will be measured. It can be seen that at the stator inlet the algorithm keeps only the pulsation mode, as expected. At the rotor exit, even though many more modes exist (caused by the local high turbulence levels), the mode corresponding to the BPF and the pulsation frequency are chosen
as the most coherent ones. It can further be noted that the algorithm retains the EWF mode despite its weak amplitude.

These findings confirm the importance of the indirect combustion noise with respect to other flow phenomena, as well as the ability of DMD to extract it. It also shows that SPDMD can be an appealing method for the analysis of real combustors. With realistic entropy waves generated at flame fronts being broadband and not monochromatic, such a method has the potential of quickly identifying the entropy modes that are most probable to generate indirect noise and thus provide more guidance for the design. However, in this monochromatic study the frequency of the indirect noise is known a priori, as it corresponds to the user-defined EWF so the standard DMD method is sufficient.

5.5.3 Quantifying the indirect noise and comparisons with the compact theory

The noise that is measured in this study is the result of a pulsated, realistic 3D turbine with several flow features present (notably the secondary flows at the hub and casing of the stator, the tip leakage flow at the rotor, the complete 3D shock structures and the shock-boundary layer interactions). In terms of noise generation, it can be compared with the 2D compact theory of Cumpsty and Marble [38]. Numerical results from 2D pseudo-LES (using the MISCOG approach and a simple mesh deformation technique) of a simplified turbine stage described in detail in [189] serve as an additional complement to the theory and the full 3D simulations. It is worth noting that Duran et al. [34] commented that for his configuration (a modified version of the MT1 turbine stage at mid-span with a 30:30 blade count) 2 kHz is approximately the limit after which the compact assumption is not valid.

To measure the transmission of the generated acoustic waves, DMD is performed at the inlet and outlet x-normal planes. Assuming that at these locations the waves are 1D plane waves, the downstream propagating acoustic wave can be calculated as
\[ w^+ = u_p^t + u_p^s \] the upstream propagating acoustic wave as \( w^- = u_p^t - u_p^s \) and the entropy wave \( w^s = u_p^t - u_p^s \). The overline in these expressions indicates time averaged quantities, the prime indicates fluctuations and the heat capacity ratio \( \gamma \) is assumed to be constant throughout, while \( u \) indicates the axial component of the velocity. The formulation of these waves is dimensionless. The transmission coefficients of interest are the entropy wave attenuation \( T_s = \frac{w_s^2}{w_s^1} \), the acoustic wave reflection \( R_a = \frac{w_1}{w_1^2} \) and the acoustic wave transmission \( T_r = \frac{w_2}{w_1^2} \), with the subscript 1 indicating the turbine inlet. The subscript 2 refers to the turbine outlet and \( w_s^1 \) is the forced entropy wave imposed at the inlet.

The procedure to construct the characteristic waves and measure the transmission coefficients at the inlet and outlet of the turbine stage can be decomposed into 5 steps:

1. Perform DMD of the principal flow variables at an x-normal plane both at the inlet and outlet of the turbine.

2. Isolate the mode of interest (EWF in this case) and form the temporal fluctuations of the variables.

3. For each point in the plane construct the 1D plane waves using the reconstructed fluctuations and a time-averaged solution.

4. Perform surface averaging and calculate the transmission coefficients.

Applying this procedure at the inlet of the turbine stage is straightforward, since there is no free-stream turbulence imposed. However, as the flow goes through the turbine it generates broadband fluctuations. While DMD allows an easy filtering of all irrelevant frequencies, turbulence or hydrodynamic phenomena whose frequency coincides with the pulsation frequency will be present in the signal and can therefore modify the evaluation of the transmission coefficients. As a result, at the rotor outlet an extra step is added before step (4): a hydrodynamic filtering based on the Characteristics Based Filtering (CBF) method [196] is applied to separate hydrodynamics from acoustics knowing their different propagation velocities. To apply this filtering, the waves are measured in 3 outlet x-normal planes in close proximity (instead of just 1). The Taylor hypothesis and the known wave speed are then used to correlate the data between the 3 planes at different physical times following the formula:

\[
   w_a = \frac{1}{3} \sum_{i=0}^{2} f(x - i\Delta x, t - \frac{i\Delta x}{u_p})
\]  \hspace{1cm} (5.7)

In Eq. (5.7), \( f \) is the wave of interest, \( w_a \) is the filtered wave, \( \Delta x \) is the distance between the planes and \( u_p \) is the wave speed, i.e \( \bar{u} + c \) for \( w^+ \) and \( \bar{u} \) for \( w^s \).

Results, obtained with the procedure described above, are summarized in Fig. 5.25, where they are also compared to the theory and 2D numerical predictions. For the compact theory and 2D simulations, predictions across a broad frequency range are available. The 3D predictions are close to the 2D ones, while the compact theory predicts stronger upstream propagating generated noise and slightly lower transmitted noise. Regarding
the entropy wave transmission, the results of the 3D simulation suggest that at the turbine outlet the injected wave has been dissipated more than in the 2D simulations, while the theoretical approach neglects the entropy wave attenuation process. Concerning the acoustic waves generated at the forcing frequency, for the downstream propagating acoustic wave, the two numerical simulations are in reasonable agreement. For the upstream propagating wave, the 3D simulation predicts a small decrease in strength compared to the 2D prediction, probably because of the choked operating condition that prevents acoustic waves generated downstream the sonic line to travel towards the turbine inlet.

5.6 Conclusions

In this chapter, the indirect combustion noise generation mechanism across a high-pressure turbine has been investigated. It is achieved by performing LES of a 3D high-pressure turbine stage subject to a constant-frequency entropy wave train pulsation. To simplify the data processing, the flow field and the generated noise are analyzed through the Dynamic Mode Decomposition of instantaneous snapshots at several positions across the turbine and the results are compared with a steady inflow case. When no wave forcing takes place, the strongest noise-generating mechanisms are revealed to be the rotor/stator interactions occurring at the BPF and its harmonics, followed by weaker activity due to the stator vortex shedding and trailing edge acoustic wave generation. When entropy wave injection is activated, a distinctive high-amplitude mode at the pulsation frequency is generated, as well as interaction modes with the blade passing frequency. The influence of the entropy waves is also captured by the sparsity-promoting DMD, a modified DMD algorithm that provides an accurate reconstruction of the flow field with few well-selected modes. Despite the presence of broadband turbulence and non-linear interactions, the blade passing frequency and pulsation modes are shown to be the most important ones. For the forced frequency, a detailed analysis of the 3D LES predictions is performed and the results are compared with the compact theory [38] as well as 2D simulations of a similar turbine configuration. While the theory overpredicts the noise levels, the 3D LES of the choked transonic HP turbine reveals that the entropy waves get highly distorted and weaker as they are transmitted to the following stages if compared to 2D results or the compact theory (unlikely to generate any additional indirect noise). The transmitted acoustic waves to the consequent stages remain strong, and will equally contribute to the indirect noise as in the 2D simulations. The reflected acoustic waves are slightly weaker than in 2D predictions and much more attenuated than in the compact theory.

This chapter also serves as the first application of MISCOG on a combustion chamber-turbine interaction problem. While the problem is treated in a decoupled fashion from the combustor, the method proves to be capable of capturing the complex generation mechanisms of indirect combustion noise in a fully 3D high-pressure turbine stage where rotor/stator interactions are important. These results, thus, can provide some degree of confidence that the method is capable of treating a fully coupled combustion chamber-turbine problem.
Chapter 5: Indirect combustion noise generation in a high-pressure turbine

Figure 5.25: Transmission coefficients - a) Ra b) Tr and c) Ts - Compact theory (solid line), 2D simulations (+ and ●) and LES (●)
Chapter 6

LES of an industrial combustion chamber-turbine system

Contents

6.1 Multicomponent simulations of gas turbines .......................... 122
6.2 Hot-streak migration across turbines ................................. 125
   6.2.1 Segregation effect ........................................... 125
   6.2.2 Other parameters influencing the hot-streak migration ....... 126
6.3 LES of an industrial high-pressure turbine stage .................... 129
   6.3.1 Standalone turbine geometry .................................. 129
   6.3.2 Mesh .......................................................... 130
   6.3.3 Boundary conditions .......................................... 132
   6.3.4 Numerical setup and initialization ............................ 133
   6.3.5 Results ...................................................... 133
6.4 Fully coupled combustion chamber-turbine simulation ............. 141
   6.4.1 Geometry .................................................... 141
   6.4.2 Mesh .......................................................... 142
   6.4.3 Combustion modelling ....................................... 142
   6.4.4 Initialization and numerical set-up .......................... 144
   6.4.5 Results ...................................................... 146
6.5 Conclusions ........................................................ 152
In this chapter, the MISCOG method is used to perform combustor-turbine LES of a helicopter engine, the focus being placed on the aerothermal interactions in such systems. Before presenting the numerical simulations, the literature on previous multi-component simulations of gas turbines and on the migration of combustor-generated non-uniformities across turbines is reviewed. Although the main objective here is to illustrate the capacity of MISCOG to treat the full combustor-turbine LES flow of real industrial configurations, the turbine is first investigated alone. Comparatively to the simulations of Chapter 4, increased fidelity is introduced by imposing realistic time-averaged temperature profiles at the inlet of this standalone turbine LES, provided by an existing LES of the combustion chamber. Comparisons of the aerodynamic flow field with standard steady-state RANS simulations that employ the cheaper mixing plane method for the rotor/stator interface (provided by Turbomeca) permits the comparison of the results of this first complex turbine stage LES with the typical industrial simulation method.

The second part of this chapter is dedicated to a fully coupled, multi-species and reactive LES of the entire combustion chamber-HPT system. With this approach, all the heterogeneities at the combustor outlet can be propagated in real time through the turbine, thus taking into account all combustor-turbine interactions in time and space. This last simulation highlights the potential of the developed methods for future multi-component simulations of gas turbines and is compared to the standalone turbine LES. Although such predictions remain at this stage preliminary and clearly require further efforts, both simulations are compared with a focus on the migration of temperature non-uniformities across the entire turbine stage.

Note that a large part of the presented simulations was performed during a 3-month secondment at Turbomeca, in the frame of the project COPA-GT. For confidentiality reasons, figures that include temperature and pressure have been normalized by the total temperature and total pressure at the turbine inlet issued by the thermodynamic cycle of the engine.

6.1 Multicomponent simulations of gas turbines

The advantages of coupled multi-component simulations, complemented with the increasing computational resources, have sparked considerable interest since the beginning of 2000’s. A first full engine simulation was produced for the General Electric GE90 engine by Turner et al. [19] within the NASA Numerical Propulsion System Simulation (NPSS) program. For this simulation, dedicated solvers for each component were used, the unstructured National Combustion Code for the combustion chamber [197] and the structured multi-block APNASA code for the turbomachinery stages. Both codes are steady-state RANS solvers employing the $k - \epsilon$ turbulence model [198]. This work relied on a turbomachinery-combustor coupling method developed earlier [199, 200] which ensured mass and total enthalpy conservation and resulting efforts were oriented towards the capacity of such a tool to recover the main cycle parameters of the engine. Despite the success of the simulation in capturing such engine data (Fig. 6.1 presents the error on the prediction of the main thermodynamic parameters across the engine compared
to experimentally measured values), the RANS approach and the modeling of significant technological effects, such as the tip clearances of the rotating machineries or coolant injections, were found to impact the predictions significantly [201].

Figure 6.1: Full GE90 simulation comparison to engine cycle data. Percent difference in total pressure (P), total temperature (T) and flow rate (W) from experimentally measured values [19].

Figure 6.2: LES/RANS interface at the compressor exit with turbulent fluctuations superimposed at the time averaged RANS solution [20].

To increase the fidelity of such multicomponent tools, combustor simulations switched to solvers based on the LES formalism, considered more accurate than RANS for this component [202]. For the wall-bounded high-Reynolds turbomachinery parts, the (U)RANS approach was retained. The most notable work using this type of coupling was the full engine simulation performed at Stanford University in the framework of the ASCI project [20, 21]. The principal challenge with this different approach is in the handling of the interfaces between the different solvers and formalisms: a) at the RANS/LES interface, located at the compressor exit, turbulent fluctuations need to be reconstructed and
injected in the LES domain and b) the LES/RANS interface placed at the turbine inlet needs a specific treatment to ensure a smooth transition and conservativity. Difficulties were also added due to the fact that the combustion chamber solver was incompressible compared to the compressible turbomachinery solver.

For the reconstruction of turbulent fluctuations at the RANS/LES interface, Medic et al. [20, 203, 204] proposed a recycling technique. In this approach the turbulent fluctuations are calculated in parallel ”on the fly” by a periodic duct LES with similar conditions as those prevailing at the combustor inlet. Obtained fluctuating fields were then superposed to the time averaged RANS profile and used at the LES inlet, Fig. 6.2. On the other side of the combustion chamber, to overcome the difficulties between the incompressible-compressible solver coupling, Schlüter et al. [205, 206] proposed the use of the body force method: the LES domain has an overlapping region with the RANS domain and within this region the mean velocity of the LES solution is driven towards the RANS solution with the addition of body forces. The final full engine simulation, combining LES and RANS, depicted very realistic flow fields and the existing measurements for the pressure radial profiles at two different engine locations (low-pressure/high-pressure
6.2 Hot-streak migration across turbines

As mentioned in Chapter 1, the combustion chamber generates temperature heterogeneities, turbulence and swirl that propagate and migrate across the turbine stage. Figure 6.4 depicts a typical time-averaged temperature profile at a combustor exit highlighting the large temperature differences across the sector. The thermal load of the blade rows, already constrained by the high mean operating temperatures, is altered considerably by these heterogeneities. This necessitates consequently a good prediction of the migration of the non-uniformities if an optimal design of the blade and its cooling systems is to be obtained. The importance of the combustor-turbine interactions can be better highlighted if one considers that an under-prediction of the blade temperature by just 15 degrees (when the inlet temperatures are well over 1500 K) can reduce the expected life duration of the turbine by half. Note that different physical effects and features are important on the migration of temperature heterogeneities across turbine stages, as detailed below.

6.2.1 Segregation effect

An important consideration when trying to predict the migration of temperature non-uniformities in the rotor blades of a turbine stage is the segregation effect first described in [207]. If the Mach number and flow angles are considered constant, fluid of higher temperature will have an increased absolute velocity at the stator exit due to the increased speed of sound, while cold gases will be slower. As a result, when the rotational velocity

![Figure 6.4: Typical combustor outlet temperature profile [22].](image)
is added at the stator/rotor interface, the relative velocity angle seen by the rotor blades is different as illustrated in Fig. 6.5. While for both hot and cold gases the flow angle of the absolute velocity is the same, the higher magnitude of the hot gas velocity creates a relative velocity with a higher angle and magnitude. In real turbines with hot streaks, this leads to a preferential migration of the hot gases towards the pressure side of the rotors as confirmed experimentally by Butler et al. [208]. An interesting thing to note is that this phenomenon is fully unsteady. As a result, predictions from steady-state numerical simulations, which are prevalent at the design stage of a turbine, cannot take such effects into account.

6.2.2 Other parameters influencing the hot-streak migration

Figure 6.4 depicts a hot spot located in the center of the sector and contained between 40 and 60% of the span. This specific alignment of the non-uniformities can be however altered by the relative placement of the fuel injectors and the dilution holes present in the combustion chamber. As a consequence, a lot of research has been performed to evaluate the effect of the azimuthal and radial placement of hot-streaks on the resulting temperature profiles across the blade passages. Additional flow characteristics at the turbine inlet are also know to affect the heat load on the blades. Among others, the free-stream turbulence and length scales along with the residual swirl from the combustor are of crucial importance.

- Effect of the fuel injector and vane clocking

The principal parameter in hot-streak migration is considered to be the azimuthal placement, also called clocking, of the hot spots with respect to the blade leading edge.
Povey et al. [15] performed a numerical and experimental investigation of the effect. A significant increase in the heat transfer on the suction side of the stator (also called Nozzle Guide Vane, NGV) was observed when the hot part was aligned with the blade leading edge, while aligning the hot streak with the blade passage was reducing it, as shown in Fig. 6.6. Although the latter alignment can be beneficial for the stators, it can prove to be damaging for the rotor if the segregation effect is taken into account. In realistic geometries, the distance between blade rows is small and the flow at the outlet of the stator is not uniform. The wake region has a velocity deficit compared to the free-stream. Aligning the hot flow with the leading edge would direct the hotter, higher velocity gas in the wake region, reduce the velocity deficit and help cancel the segregation effect. As a result, the heat load on the rotor pressure side can be reduced. This has been confirmed by He et al. [26], in a thorough study of different hot-streak counts in the heat loads across the MT1 turbine stage.

- Impact of the hot-streak radial position

Besides the azimuthal placement of the non-uniformities, there is also evidence of the importance of the radial position of hot-streaks. In fact, small contained hot-streaks around mid-span are shown to have little impact on the blade heat load close to the endwalls. More specifically, Povey et al. [15] observed a decrease of the heat load as the temperature is decreased locally near the endwalls. On the contrary, when hot-streaks are enlarged they can interact with the secondary flows, the passage vortices being able to transport hot, high-energy fluid from the free-stream towards the hub and casing boundary layers, thus increasing the heat load at these locations [29]. Similar conclusions were drawn by Roback and Dring [28], when the position of the hot streak is moved radially towards either the hub or the casing. Finally, note that additional radial
migration of the non-uniformities can exist because of the radial pressure gradients and buoyancy effects for example [30].

- Influence of the free-stream turbulence

Apart from the mean temperature nonuniformities, the flow at the exit of the combustion chamber is also highly turbulent. Colban et al. [160] measured in experimental combustor simulators turbulence intensities up to 30% at the leading edge of the first turbine blade row. Similar results were measured by Barringer et al. [209], who also found a correlation between the turbulent length scales and the size of the dilution holes in the combustion chamber. Such levels of turbulence are known to impact the aerodynamic and thermal flow field of the turbine, as well as the hot-streak mixing.

A number of experimental and numerical investigations have shown a significant increase of the heat transfer of blades when high levels of turbulence are present at the inflow [23, 99]. Indeed, the incoming eddies interact with the blades and get stretched, forming long longitudinal structures, Fig. 6.7, that enhance the heat transfer, particularly on the pressure side. On the suction side, the strong flow acceleration alleviates this effect in the first half of the blade [23, 210]. However, as the flow decelerates when it approaches the trailing edge, laminar-to-turbulent transition of the boundary layer can be observed and is affected by the vane turbulent flow. A turbulent boundary layer is known to enhance the mixing of high momentum, high temperature fluid from the outer layer with the low momentum fluid of the inner layer, resulting in a considerable increase of the heat transfer to the blade walls. Free-stream turbulence is one of the principal ways transition can be triggered and is shown to influence the position where it occurs, thus impacting the overall heat load of the blade [9, 31]. It has also been shown that free stream turbulence does not decay across the turbine and that additional turbulent kinetic energy is produced around the stagnation points [210]. Interactions of turbulence with hot-streaks have also been investigated. Jenkins et al. [32] reported that medium levels of free-stream turbulence render the hot streak more compact and enhance temperature gradients.

- Influence of swirl

Recent experimental and computational work from Qureshi et al. [4, 33] indicate a considerable impact of the residual flow swirl at the turbine inlet, generated in the combustion chamber, on the aerothermal flow field across a high-pressure turbine stage. At the NGV, the swirl is altering the incidence angle of the flow, thus modifying the aerodynamic flow field and secondary flows. These changes naturally lead to significant local alterations of the heat transfer, particularly near the endwalls. Across the rotor the changes are less significant [33]. Nonetheless, the overall heat transfer should be increased as swirl is likely to enhance the free-stream turbulence levels and hence the near-wall mixing.
6.3 LES of an industrial high-pressure turbine stage

The findings of previous research efforts on the migration of temperature non-uniformities underline that there is a strong coupling of the HPT and the combustor during the operation of a gas turbine. Important phenomena, such as the turbulence levels and length scales, cannot be accurately modeled in standalone HPT simulations, thus highlighting the potential gains in prediction accuracy issued by a fully coupled combustion chamber-turbine simulation. However, before illustrating the MISCOG ability to treat such complex industrial flows, the industrial HPT stage LES is first simulated to validate the proposed methodologies in an industrial turbine. To increase the fidelity of these simulations, a temperature hot-spot is imposed at the inlet and its migration across the turbine stage is studied. Comparisons with RANS, the preferred method for quick numerical predictions at the design phase, are also shown at this occasion for key performance figures.

6.3.1 Standalone turbine geometry

The high-pressure turbine geometry is selected from an existing engine of Turbomeca. It is an unshrouded design, similar to the MT1 turbine analyzed in the previous chapters. An advantage of the selected engine is that, parallel to this work and in the frame of the European project COPA-GT, Turbomeca performed LES of the corresponding combustion chamber, used here to determine the inlet profiles of this HPT simulation.

As with the MT1 geometry, the reduced blade count technique is employed in the rotor to reduce the computational domain to an azimuthally periodic extension of the combustor simulation (considered in the next section) with 1 stator and 2 rotors. However, the geometry set-up used in this chapter demanded the scaling of the rotors to be done in the cartesian coordinates instead of the cylindrical ones as proposed by Mayorca et al. [60] and used in the simulations of Chapter 4. As a result the blade angles and the
mean flow will be significantly altered compared to the true geometry. It is also worth noting that the geometry used corresponds to a cold geometry, meaning that the thermal expansion of the blades due to the high operating temperature is not taken into account. The CAD source file uses a multi-zone approach where the combustor-turbine system is segmented in 3 zones to facilitate the coupled simulations: the combustion chamber, the stator and the rotor domains. For this case, i.e the standalone turbine LES, only the latter two parts are employed.

Besides the blades, a real HP turbine has complex technological effects that weren’t present in the experimental MT1 turbine: fillets, internal blade cooling channels and holes that inject fluid in the main passage, as well as cavities in the hub and casing between the blade rows. Additionally, it is not uncommon for rotors to have squealers, which are obtained by removing material from the central area of the blade tip, Fig. 6.8. Such a process reduces the weight and mechanical forces applied to the blade. It also permits a reduced tip clearance to be achieved by the manufacturing [211]. Frequently, most or all of these specificities are omitted in CFD simulations as typical turbomachinery solvers use structured meshes and therefore require considerable effort to take them into account. The chosen configuration for LES retains only a squealer of depth approximately the one of the real blade and with a flat bottom, as it was assumed to be the most significant technological effect [5]. It is worth highlighting that here the unstructured approach allows for easier inclusion and meshing of even the most intricate technological aspects of the turbine geometry.

6.3.2 Mesh
As with the other turbine cases, a hybrid mesh approach is used, Fig. 6.9. Around the blades 5 layers of prisms are added to improve the near wall resolution, while tetrahedral
elements are employed in the passages and near the endwalls. On the rotor blade, the prism layers follow not only the form of the blade but also the squealer, Fig. 6.9 (right). The tip clearance is smaller than 0.5 mm. To keep the timestep acceptable, 7 cell layers are placed in that area. The total cell count is 25 million cells (10 million cells in the stator and 15 in the rotor domain). The maximum $y^+$ across the blades is approximately 30, Fig. 6.10. While this value is not within the guidelines for wall-resolved LES, it is significantly improved compared to the coarse MT1 mesh despite the limited cell count. This is due to the smaller power of helicopter engines, which translates to reduced turbine sizes and Reynolds number compared to the large airplane engines (the MT1 turbine is a representative configuration of an airplane engine). This improved resolution is followed however by a small time-step, $\Delta t = 8 \cdot 10^{-9}$ s.
6.3.3 Boundary conditions

As with most turbine stage simulations, the total temperature and pressure are prescribed at the inlet. Instead of imposing a homogeneous total pressure and temperature at the turbine inlet, realistic profiles constructed from a time-averaged solution of previous standalone combustor LES of the same engine were used. To this end, the principal flow variables are interpolated from the combustion chamber simulation exit section to the turbine inlet surface. Then using this information, the Mach number and total temperature are calculated as well as the total pressure using the isentropic relation. Note that the solver uses a variable heat capacity ratio $\gamma$. However, as RANS simulations of turbines often use a constant $\gamma$, an average value of $\gamma$ is used to construct the inlet conditions in order to stay as close as possible to the RANS simulations of this configuration.

Another characteristic of this industrial turbine is that the real geometry has coolant air injected through cavities located between the stator and rotor hub, casing as well as through the stator trailing edge. These injections lead to an increase of the overall mass flow across the turbine stage. Taking into account these injections is costly, as that demands meshing the small cavities, penalizing the computational cost both in terms of mesh size and timestep. The typical approach for simulating their impact on the mass flow is by scaling the turbine inlet total temperature profile accordingly. Note that the
temperature variation across the inlet plane is preserved. The imposed total pressure is calculated from the total temperature using the isentropic relationship before applying the total temperature scaling.

The imposed profiles of total temperature and pressure, divided by the mean total temperature and pressure, are shown in Fig. 6.11 with the leading edge of the NGV depicted by the dashed line, along with the pressure and suction sides (PS and SS respectively) of the blade. As observed here, the hot-streak is centered at approximately 50-60% radius and towards the pressure side of the blade. This alignment indicates that the pressure side of the blade is likely to experience a higher thermal load. Note that the segregation effect can be exacerbated by such an alignment as described in section 6.2.1. Finally and contrarily to the fully integrated combustor-turbine LES, all standalone turbine stage computations use air to stay closer to the single-species formulation of most turbomachinery CFD solvers.

At the outlet of the turbine a static pressure is imposed. All walls are considered to be adiabatic.

6.3.4 Numerical setup and initialization

The Lax-Wendroff numerical scheme, 2nd order in space and time, is employed [111]. For SGS closure, following the findings and conclusions established in Chapter 4, the WALE model is employed in an effort to improve the near-wall predictions.

Initialization of the simulation is performed as before, through the use of a coarse mesh with 7 million cells in total. For these simulations, the 2D inlet profiles were not yet available so homogeneous inlet total pressure and temperature were applied. The initial flow field was simply uniform velocities throughout. After 3 rotations on the coarse mesh, the solution resembles the flow field of a turbine and is then interpolated onto the finer mesh. An additional 2 rotations with the realistic inlet profiles are performed to converge the flow on the fine mesh and a full rotation follows to obtain time-averaged data. RANS simulations are performed using the multi-block structured elsA code [212] using the same inlet profiles as the LES along with the $k - \ell$ two equation turbulence model of Smith [213] for closure. The rotor/stator interface of the RANS simulations is treated with the mixing-plane model.

6.3.5 Results

Performances

As mentioned in section 6.3.1, the rotor geometry of the turbine has been considerably modified compared to the real machine. This is expected to impact the aerodynamic flow field significantly. To quantify the effect of this modification some typical 0D performance figures measured from the LES of the scaled turbine are compared to RANS predictions of the correct geometry (without any scaling). RANS predictions of the scaled geometry are also presented to establish any similarities with the LES. Note that no squealer is included in the RANS domain which might have an impact on these figures. The
performance metrics of interest are (the subscripts \textit{in} and \textit{out} indicate turbine inlet and outlet respectively):

- **Turbine total pressure ratio** $\pi_T$
  It allows to evaluate the work extraction performed by the turbine and associated aerodynamic losses through the ratio of the total pressure at the inlet to the total pressure at the turbine exit:
  \[
  \pi_T = \frac{P_{t in}}{P_{t out}}
  \]  
  (6.1)

- **Mass flow** $\dot{m}$

- **Degree of reaction** $R_d$
  The formula for the degree of reaction reads:
  \[
  R_d = \frac{h_{rin} - h_{rout}}{h_{tin} - h_{tout}}
  \]  
  (6.2)
  where $h$ and $ht$ are the static and total enthalpy respectively, the subscript \textit{rin} indicating the rotor inlet and \textit{rout} the rotor exit.

  $R$ is a convenient metric to determine how the static pressure drop is distributed across the turbine blades. Values larger than 0.5 indicate that most of the drop occurs in the rotor blades, while values lower than 0.5 indicate that most of the flow acceleration takes place in the stator blades (impulse turbine).

- **Isentropic efficiency** $\eta_{is}$
  It is one of the best indicators of the quality of an aerodynamic design. It evaluates how close the expansion across the turbine stage is to the ideal isentropic expansion. For a perfect gas with constant heat capacity ratio, it is straightforwardly evaluated as:
  \[
  \eta_{is} = \frac{1 - \frac{T_{t out}}{T_{t in}}}{1 - \left(\frac{P_{t out}}{P_{t in}}\right)^{\frac{\gamma-1}{\gamma}}}
  \]  
  (6.3)

Dedicated turbomachinery CFD solvers usually work with a constant heat capacity ratio $\gamma$ and a mean value is applied in Eq. (6.3). In our case however, a reactive variable heat capacity ratio solver is used. As a result, calculating the isentropic efficiency with a variable $\gamma$, based on tabulated gas properties and described in [214], is preferred. Note that in this process the necessary surface averages needed at the different measuring stations (to render the result 0D) are performed using internal Turbomeca standards.
6.3 LES of an industrial high-pressure turbine stage

<table>
<thead>
<tr>
<th></th>
<th>LES</th>
<th>RANS (modified geometry)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\pi_T$</td>
<td>-1.5%</td>
<td>-2.5%</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>+1%</td>
<td>-0.1%</td>
</tr>
<tr>
<td>$R_d$</td>
<td>+65%</td>
<td>+75%</td>
</tr>
<tr>
<td>$\eta_{ls}$</td>
<td>-0.4%</td>
<td>-4%</td>
</tr>
</tbody>
</table>

Table 6.1: Summary of 0D performance differences between LES and RANS with the modified geometry to RANS with the standard geometry.

Results are summarized in Table 6.1 where the differences between LES and RANS of the scaled turbine to the RANS of the unscaled geometry are presented. From Table 6.1 it is evident that the rotor blade scaling results in a small reduction of the total pressure drop across the turbine. Likewise, the isentropic efficiency is also reduced, especially for the RANS simulations where a drop of 4% is observed. The biggest change appears in the degree of reaction, where both scaled simulations yield a large increase of $R_d$. RANS simulations in particular, show a very large increase of the degree of reaction pointing to a significant reduction of the pressure drop across the NGV. In fact, since only the rotor blades have been modified, it becomes evident that the rotor/stator interactions impact directly the NGV aerodynamic loading and the flow is not comparable to the real engine flow. This is a typical problem encountered in real applications of CFD for which full annulus simulations are required if a high-fidelity rotor/stator interface, capable of transporting broadband phenomena, is demanded.

Mean flow topology

In the following, as the geometry modifications have been shown to be considerable and to have effectively altered the flow field, only the flow organization and predictions from the LES, which is the focus of this dissertation, are analyzed.
Isosurfaces of the Q-criterion computed from time-averaged velocities, colored by the mean local temperature, are shown in Fig 6.12. A typical flow field appears and secondary flows are recovered. The horseshoe and passage vortices are visible in the NGV and rotor hub, as well as the tip leakage and induced vortices developing in the tip clearance. One can also note that the horseshoe vortex at the NGV casing is developing further upstream compared to its counterpart at the hub. This behavior has also been observed on previous LES of the combustor+NGV system (internal Turbomeca simulations) or URANS of this turbine [5] (with the unscaled geometry). Another observation is that at around 50% of the NGV chord, streaks develop in the boundary layer, indicating the beginning of laminar-to-turbulent transition, although the near-wall mesh resolution is not sufficient to capture accurately such a process ($\Delta z^+$ and $\Delta y^+$ do not respect the criteria for accurate near-wall LES). These streaks are not horizontal and instead appear to migrate towards the hub. This orientation can be linked to the radial pressure gradients in this region. The temperature coloring allows to evaluate the influence of the secondary flows in the turbine heat load. The horseshoe vortices appear to be transporting colder flow from the endwalls to mid-passage, a phenomenon already observed before by Wlassow [5].

**Hot streak migration**

The hot-streak migration across the turbine is analyzed in detail in an attempt to qualify this standalone simulation in light of the physics discussed in section 6.2. Figure 6.13 depicts the temperature distribution across the NGV blade row at different axial positions. At the turbine inlet, Fig. 6.13(a), the hot-streak is located between the blades and at approximately 60% radius. At 10% of the chord length, Fig. 6.13(b), the distribution has already been altered around the blades: on the suction side there is a radial migration towards the casing (position B) and the horseshoe vortices from the hub and the casing (positions A and C) transport colder fluid from the endwalls towards the mid-passage. Further down the passage, Figs. 6.13(c,d), the secondary flows grow considerably. This leads to a further confinement of the hot-streak with comparatively cooler endwall temperatures. At these positions, the radial pressure gradient is now seen pushing the hot-streak towards the hub (position B). At the stator exit, Fig. 6.13(d), an additional area of higher temperature appears, near the casing on the blade suction side (position D). The casing horseshoe vortex is indeed located further away from the blade and the gap is filled by hot fluid is transported by the passage vortex instead as in [5].
Figure 6.13: Temperature across the NGV passage - Turbine inlet (a), 10% of the chord (b), 60% (c) and at the NGV exit (d).
Figure 6.14: Temperature across the rotor passage - Rotor inlet (a), 10% of the chord (b), 50% (c) and at the rotor exit (d).

In the rotor domain, Fig. 6.14, at the inlet a more uniform flow with mainly radial temperature gradients is seen by the blades due to the high-speed rotation, Fig. 6.14(a). The peak of the hot-streak appears now closer to the hub than at the NGV inlet, at approximately 40% height. Further down the passage, Fig. 6.14(b,c), four different phenomena that impact the migration of the hot-streak are observed. First, the hub secondary flows create a zone of colder fluid (position H), similar to the stator hub. Another region of interest is indicated by position E, which locates near the pressure side of the blade and where high temperatures are encountered. This feature can be explained by the segregation effect, which in this case is emphasized further by the mid-passage clocking of
the hot-streak as shown in [26], suggesting a preferential migration of the hot-streak to the rotor pressure side. Note that a radial migration of the streak towards the casing is also observed. The last interesting area relates to the tip clearance and the squealer. As already described, between 20 and 50% of the blade chord, the tip clearance flow passes from the pressure side to the suction side. Due to the higher temperature at the pressure side, this flow would lead to a high temperature tip leakage vortex region (position F). However, because of the presence of the squealer, the tip leakage flow is altered with a large part of the thermal load being transferred to the squealer (position G). It results in a lower temperature tip leakage region compared to the pressure side. Finally, at the rotor exit, the temperature is further decreased by the strong flow acceleration near the rotor trailing edge.

To evaluate the radial position of the hot-streak in the turbine, azimuthally averaged temperature profiles are plotted as a function of the radial coordinate. Figure 6.15 depicts such profiles for three different positions: the turbine inlet, the rotor/stator interface and the turbine outlet. First, at the turbine inlet the hot-streak is centered around 50-60% height with a temperature profile reducing smoothly towards the endwalls. Second, at the stator/rotor interface, the radial migration of the hot-streak towards the hub issued by the flow going through the stator blade is clearly seen: the maximum temperature drops from an initial location approximately 60% height to 40%. Heating near the endwalls is also present. Finally, at the rotor exit the hot-streak has moved towards the casing. Note that at this section the flow expansion has considerably decreased the static temperature levels and while significant turbulent mixing has taken place, a heterogeneous profile is still observed.

Blade wall temperature distributions across the stator and rotor are finally presented in Fig. 6.16. For clarity, the blade surface is opened around the leading edge (LE) and the previously highlighted features of Figs. 6.13 and 6.14 are also reported here. On the stator suction side, Fig. 6.16(a), the radial migration of the hot-streak can be observed, first towards the casing and then towards the hub (position B). The hub endwall flow effect is also evident on this view. On the rotor blade, Fig. 6.16(b), the preferential migration of the hot-streak to the pressure side is also seen with a large area of high temperature dominating the surface. On the suction side, high temperatures are confined to a narrow area near the LE, while the secondary flow regions (positions F and H) show decreased heat loads. Overall, although not detailed here, qualitative agreement with older URANS predictions [5] is found for the stator vane with some differences in the rotor domain most likely due to the blade modification.

While the aerodynamic flow field is shown to be strongly impacted by the rotor scaling, this LES of the HPT allows to conclude that a) the MISCOG method can handle complex industrial configurations and that b) generally established trends for the hot-streak migration are recovered (segregation effect, secondary flow effects etc). Based on these findings, a coupled reactive combustor-turbine LES was attempted and is detailed in the following section. The interest of this specific simulation, other than demonstrating its feasibility, is principally in the aerothermal response of such flows, i.e to evaluate the changes issued by the now fully unsteady, turbulent hot-streak arriving at the turbine when compared to the standalone HPT LES that employed a time-averaged hot-streak.
at the inflow boundary condition.

![Graph](image)

**Figure 6.15:** Azimuthally averaged temperature profiles at three different axial positions as a function of the height.

![Blades](image)

**Figure 6.16:** Blade wall temperature of the stator (a) and rotor (b) blades - Blade surface is opened around the Leading Edge (LE) for clarity.
6.4 Fully coupled combustion chamber-turbine simulation

After using the MISCOG method to evaluate the hot-streak migration on a standalone HPT computation, the full combustion chamber-turbine configuration is now simulated. This computation is the first of its kind with no physical interface between combustor and turbine thus allowing for unparalleled fidelity on the combustor-turbine interactions. Note that we now consider a multi-species simulation with reactions activated because of the combustion chamber. The rotor domain is multi-species as well but no reaction is taking place as the combustion process is assumed to be finished at this point.

6.4.1 Geometry

The geometry used for the turbine is exactly the same as in the previous section. The difference is that the combustion chamber geometry previously ignored is now connected to the computational domain, forming with the NGV a single static domain that will be simulated by one AVBP instance while the rotor is handled by a second AVBP instance. The combustion chamber is a reverse flow chamber. Air coming from the compressor arrives in a large plenum surrounding the main chamber and enters the combustor through the swirler, dilution holes and cooling devices. Although simulations with the combustor plenum have been reported in the literature [215], the less costly and simpler approach of using mass flow injections through the combustor liner and main flow paths is preferred.
Chapter 6: LES of an industrial combustion chamber-turbine system

This chamber includes all the typical features found in modern RQL combustors: a swirler to help stabilizing the flame, multi-perforated plates, film cooling and dilution holes. The general geometry and some of the combustor features are depicted on Fig. 6.17.

6.4.2 Mesh

The mesh of the combustion chamber was generated using ANSYS Mesher and is a fully tetrahedral grid. The size of the chamber mesh is approximately 30 million cells, bringing the total cell count of the coupled simulation to 55 million cells. A view of the combustor mesh is provided in Fig. 6.18. The swirler and fuel injector regions are particularly refined before being gradually coarsened through three different user-defined zones. The coarsest zone corresponds to the beginning of the re-orientating elbow located in front of the turbine. As the turbine is approached, mesh refinement takes place again to adapt the grid to the wall-bounded turbine geometry and yield a mesh resolution adequate with the use of a law-of-the-wall as tested in Chapter 4. Note that all film cooling slots are explicitly meshed while the holes of the multi-perforated plates are not. Instead, a homogeneous model [216] is used to take into account the mass flow injected through this effusion cooling system.

6.4.3 Combustion modelling

A major difference introduced in this simulation compared to HPT simulations is the combustion process of kerosene taking place in the chamber. This process is characterized by several chemical reactions that convert the fresh gases (kerosene and air) into combustion products through a flame front interacting with the turbulence generated in
the chamber. Both these aspects need therefore to be taken into account.

The chemical process of hydrocarbon combustion typically involves hundreds of species and thousands of reactions with detailed models involving between 200-250 species and over 1500 reactions [217]. However, the cost of resolving all the species transport equations and computing the reaction rates of such chemical models is prohibitive. To reduce the cost, simplified chemical schemes have been developed where only a certain number of important reactions and species are retained. Such models attempt to fit the principal flame properties of the reduced scheme to the detailed reference chemistry. In this case, a 2-step/6 species model, presented in Eq.(6.4) and described in detail in [218], is employed. It has been shown to provide good accuracy on the burnt gas temperature as well as laminar flame speeds over a large range of equivalence ratios [217].

\[
\begin{align*}
Kerosene + 10O_2 & \Rightarrow 10CO + 10H_2O \quad (6.4a) \\
CO + 0.5O_2 & \leftrightarrow CO_2 \quad (6.4b)
\end{align*}
\]

For the simulation, the fuel is injected directly in a gaseous form, ignoring the process of atomization and evaporation. This approach was chosen as previous LES of the combustion chamber performed by Turbomeca, using both gaseous and liquid fuel, showed minimal influence on the hot-streaks.

The flame/turbulence interaction is modeled using the Dynamic Thickened Flame (DTF) model [219]. In typical aeronautical combustor LES the mesh resolution is insufficient to accurately resolve the thin flame front and its wrinkling created by its interactions with turbulence. The DTF model is introduced to provide this information, as detailed hereafter.

Following the theory of laminar premixed flames [25] the flame speed \( S^0_L \) and the flame thickness \( \delta^0_L \) of a premixed front may be expressed as:

\[
S^0_L \propto \sqrt{\lambda_t A} \quad \text{and} \quad \delta^0_L \propto \frac{\lambda}{S^0_L} \quad (6.5)
\]

where \( \lambda_t \) is the thermal diffusivity and \( A \) the pre-exponential constant. Increasing the thermal diffusivity by a factor \( F \), the flame speed is kept unchanged if the pre-exponential factor is decreased by the same factor. This operation leads to a flame thickness which is multiplied by \( F \) and more easily resolved on a coarser mesh. While in reacting zones diffusion and source terms issued from the thickened reaction are well resolved, the effect of turbulence is solely represented by the efficiency function \( E \) [219, 25]. Note also that the molecular and thermal diffusion should not be over-estimated by a factor \( F \) in mixing zones where no combustion occurs (it would yield over-estimated mixing and wrong flame positions). Dynamic thickening is thus introduced to account for these points [219]. The baseline idea of the DTF model is to detect reaction zones using a sensor \( S \) and to thicken only these reaction zones, leaving the rest of the flow unmodified. In terms of implementation, thickening depends on the local grid resolution and therefore locally
adapts the combustion process to reach a numerically resolved flame front. Concerning the modeling, the flame sub-grid scale wrinkling and interactions at the SGS level are supplied by the efficiency function \cite{219, 25}. This type of approach is commonly used in similar configurations \cite{202, 215, 220}. More details on the formulation of the DTF model are provided in Appendix B.

### 6.4.4 Initialization and numerical set-up

For this multi-species case, appropriate initialization of the full configuration is more complex, the reason being that the high-pressure turbine simulation presented in the previous section was single-species. The followed initialization process can be summarized in 4 steps and makes use of a combustor+NGV LES of the same engine performed at Turbomeca:

1. **Multi-species conversion of the high-pressure turbine simulation**
   
   An instantaneous solution of the single-species turbine domain is converted to a multi-species field (with the 6 species present in the combustor simulation), with the help of the AVBP toolbox that is well-adapted for such operations. The mass fraction of each species specified in this new solution is considered constant throughout the domains. Inlet boundary conditions (including now the injection of different mass fractions for each species) are obtained from the combustor LES used for the initialization of the standalone HPT simulation. This HPT multi-species simulation is then run until converged.

2. **Extraction of the rotor domain.**

3. **Attachment of the rotor domain to the combustion chamber + NGV LES converged solution using MISCOG.**

4. **Run until convergence.**

As previously, the Lax-Wendroff scheme is used \cite{111}. Converging the fully integrated combustor-turbine simulation required 4 rotations and an additional 3 rotations (5 ms) were performed to obtain averaged solutions for a total computational cost of 600K CPU hours. Note that while the averaging time is increased compared to the simulations of Chapter 4, it is smaller than the usual averaging time employed in combustor simulations (typically larger than 10 ms). It is also important to note that in this coupled simulation there is no possibility of scaling the total temperature at the turbine inlet as no boundary condition is present there. As a result, the mass flow across the turbine is reduced by approximately 8% compared to the previous standalone HPT simulation. Correcting this flow rate would require either additional coolant injection through the film cooling slots or injecting the coolant flow realistically across the turbine at the designed positions and slots. However, as the main objective of this study is principally to demonstrate the capacity of the developed tools to handle such complex industrial configurations,
the turbine coolant flows have not been included in the simulation. It is, nonetheless, envisioned and easily applicable.

Figure 6.19: Instantaneous flow field of the coupled combustor-turbine simulation. The central image depicts temperature isosurfaces and the blade surface temperature. The top images present the heat release and temperature around the flame front (zoomed) and at the bottom the $\frac{\nabla \rho}{\rho}$ and temperature at mid-span across the turbine are depicted.
6.4.5 Results

In the first part of the results the overall flow field of the fully coupled combustor-turbine simulation is presented. It is followed by a more in-depth analysis of the hot-streak generation and migration across the turbine, where comparisons with the standalone HPT LES are also performed. Besides demonstrating the capacities of the developed methodologies in real aeronautical configurations, the emphasis is put on the aerothermal flow, i.e., evaluate the changes on the blade temperatures issued by the unsteady inflow heterogeneities.

Overall flow field

Figure 6.19 presents a view of the aerothermal flow field across the combustor and the HPT obtained from an instantaneous solution of the fully integrated LES. The isosurfaces of temperature in the chamber highlight the generation of the hot-streak which will then impact the turbine, Fig. 6.19(a). The top images, Figs. 6.19(b,c), correspond to a zoomed view around the flame front and depict the heat release or temperature flow distribution, while zoomed views in the turbine are provided at the bottom, Figs. 6.19(d,e).

In the combustion chamber a primary high-temperature zone is clearly visible and spans the entire sector width, Fig. 6.19(a). This area is characterized by significant heat release, Fig. 6.19(b), and is usually designed so that most of the combustion process takes place there. This zone is usually delimited downstream by fresh air injected through the dilution holes (Positions A in Figs. 6.19(b,c)). These jets insert fresh air that locally quenches the flame, constraining the combustion to the primary region. It can be noted here that the dilution from the bottom holes appears here to be more effective penetrating deeper in the chamber. Overall, the mass flow, angle, and the number of these dilution jets are seen here to contribute significantly in shaping the hot-streaks which for this burner is localized near the upper liner. Film cooling (Positions B in Figs. 6.19(b,c)) prevents the hot gases from touching the liners. The other noticeable feature is that by the time the flow starts turning towards the turbine, the hot-streak becomes more compact. From Fig. 6.19(b), it is also evident that some combustion continues past the primary zone (downstream of positions A). It however remains highly localized confirming an adequate design for this combustor.

When the flow is examined at the turbine entry, Figs. 6.19(d,e), the higher temperature fluid appears aligned with the NGV pressure side (Position C in Figs. 6.19(d,e)), as was the case in the standalone HPT simulation, with strong fluctuations superposed to it.

Time-averaged predictions

Turbine inlet

The principal interest of a coupled combustor-HPT simulation lies in its capacity to naturally introduce all the heterogeneities in the HPT domain. The principal non-uniformity of interest at the turbine inlet is usually the time-averaged temperature profile. Figure 6.20 depicts the mean static temperature at the turbine inlet plane of the standalone
HPT simulation compared to the one issued by the coupled combustor-HPT simulation. Both simulations reveal a similar hot spot with some differences observed near the endwalls, particularly near the hub. This is explained mainly by the fact that as the inlet temperature profile of the HPT simulation comes from a combustor-only LES which cannot take into account the interactions of the endwall cooling flows with the turbine blades and secondary flows. Note also that other perturbations observed in Fig. 6.20(b) can also be attributed to the relatively short averaging time of the coupled simulation (5 ms).

To obtain a more quantitative view of the turbine inlet characteristics, Fig. 6.21 presents various azimuthally averaged quantities, computed at the turbine inlet from the combustor/HPT and the HPT simulation (for the mean temperature only as no temporal fluctuations were imposed) and plotted as a function of the height. Figure 6.21(a) presents mean temperature profiles while Fig. 6.21(b) and Fig. 6.21(c) respectively correspond to the temperature rms (divided by the local mean temperature, contrary to other figures that are normalized by the cycle total temperature at the turbine inlet) and the turbulence intensity. The temperature of the coupled simulation has been scaled to agree with the temperature imposed at the inlet of the HPT computation to make the comparisons reliable. Based on Fig. 6.21(a), the mean temperature profiles are qualitatively and quantitatively similar, with a small difference observed on the maximum level of the hot-streak and at around 20% height which is explained by the hub flow differences observed in Fig. 6.20. As for as the unsteady profiles at the turbine inlet, the coupled simulation reveals considerable and inhomogeneous activity. The temperature
fluctuations, Fig. 6.21(b), are over 20% near the endwalls. These levels are explained by the strong mixing of the hot-streak with the cooling flows injected just upstream of the NGV. At mid-height, where less perturbations from technological effects are present, fluctuations are around 10% of the average temperature of the main stream.

The turbulence intensity distribution depicts similar trends as the temperature fluctuations. Higher levels of turbulence occur near the endwalls, where turbulent flow mixing between free-stream and cooling flows occurs while lower unsteadiness is encountered at mid-height. The latter one results principally from the convection of combustor-generated turbulence to the turbine inlet. It is worth noting though that the intensity does not drop under 10% at any radial position, in agreement with the significant turbulence levels observed previously in the literature [160]. These findings confirm that the coupled simulation is indeed capable of transporting all combustor-generated heterogeneities to the turbine. Furthermore, these extracted profiles can provide information that can prove valuable to improve the boundary conditions imposed in standalone turbine simulations.
6.4 Fully coupled combustion chamber-turbine simulation

Hot-streak migration

Having evaluated the turbine inlet flow, the hot-streak migration across the turbine for the coupled simulation is analyzed in more details and is compared to Figs. 6.13-6.14, obtained from the standalone HPT LES. Figure 6.22 depicts the temperature distribution across the NGV blade row at different axial positions. The overall trends observed in the standalone HPT simulation are recovered. As observed before, at the turbine inlet the hot-streak is still oriented between the blades and at approximately 60% radius. At 10%
of the chord length similar trends are observed as with the HPT simulation, notably the radial migration towards the casing at the suction side (position B). However, positions A and C are now hotter, probably because of the improved mixing from the high turbulence levels, a trend that continues further down the passage, Figs. 6.13(c,d). Additionally, position C appears slightly further away from the blade, highlighting a slightly modified horseshoe vortex evolution. Further downstream, the hot-streak is migrating towards the hub, only less aggressively than in the standalone HPT simulation. At the stator exit, the hot spot is found to be more homogeneous and mixed out and present at a higher radius, if compared to the elongated one of Fig. 6.13. The endwall temperatures are also noticeably higher.

![Temperature across the rotor passage for the coupled simulation - Rotor inlet (a), 10% of the chord (b), 50% (c) and at the rotor exit (d).](image)

**Figure 6.23:** Temperature across the rotor passage for the coupled simulation - Rotor inlet (a), 10% of the chord (b), 50% (c) and at the rotor exit (d).

In the rotor domain, Fig.6.23, the hot-streak enters the domain at a radial position...
that is higher than the HPT simulation, approximately 50-60% height. Further down the passage the different phenomena that impact the migration of the hot-streak are still evident. In position E, the pressure side of the blade, the segregation effect is particularly pronounced, the rotor pressure side being impinged by very hot fluid. The radial migration towards the casing appears also enhanced compared to the HPT simulation. The cooler positions F and H present because of the effect of the hub secondary flows, are still encountered but with increased temperature values. The same temperature increase appears in the squealer (position G), with the flow entering this cavity from the pressure side being slightly hotter. At the rotor exit, the cooler tip leakage flows move towards a lower span and hot fluid, previously on the pressure side of the blade and obstructed by the blade and squealer, is now moving towards the casing (position I).

Comparing the azimuthally-averaged temperature at the stator/rotor interface and at the rotor exit can be a more convenient way to quantify the differences observed in the hot-streak migration across the turbine for the two simulations. Such results are presented in Fig. 6.24. At the interface, there is only a fair agreement between the two simulations and for 20-80% of the span. The coupled simulation shows that the endwalls have considerably higher temperatures, the peak at 40% height is now less pronounced and a flatter profile is observed, pointing to a better mixed hot-streak. This trend is recovered at the turbine exit as well, along with an overall increase of the temperature levels by approximately 50 K. As observed in Fig 6.23, the tip leakage flow, that transports hot fluid from the pressure side to the casing, is also more pronounced. These findings highlight that the coupled combustor-turbine simulation does show important differences in the aerothermal predictions, particularly near the endwalls, and that the unsteady

Figure 6.24: Azimuthally averaged temperature profiles at the stator/rotor interface at the the turbine exit as a function of the height.
inflow alters the migration of the non-uniformities.

The temperature contours across the stator and rotor blades are finally presented in Fig. 6.25. As before, the blade surface is opened around the leading edge (LE) for clarity and the observed features of Figs. 6.22 and 6.23 are depicted. On the stator suction side the radial migration of the hot-streak can be seen, first near the casing and then towards the hub (position B). At the trailing edge a spread of the high-temperature region is observed. The hub endwall flow effect is now slightly altered with the cold fluid staying nearer to the hub and arriving closer to the trailing edge compared to Fig. 6.13. On the rotor blade, the preferential migration of the hot-streak to the pressure side is now more pronounced, with a large area of high temperature dominating the pressure side. It is also shifted towards the casing compared to Fig. 6.16, explaining the higher temperatures observed near the casing in Fig. 6.24. On the suction side, the high temperatures observed near the leading edge at mid span are reduced, while the cooling effect of the secondary flows (positions F and H) is now less pronounced (the endwalls experience higher temperatures so cold air transport is not as efficient).

6.5 Conclusions

In this chapter an industrial application of the MISCOG methodology was presented: a HPT LES and a fully coupled reactive combustion chamber-HPT LES of a real helicopter engine. The objective of these simulations was to study the migration of combustion-
generated heterogeneities in the turbine. The HPT simulation had a time-averaged hot-streak imposed at the inlet and was used to provide a first evaluation of the solver’s capacity to handle industrial configurations and predict the migration of time-averaged temperature non-uniformities across a turbine stage. While a necessary modification of the rotor geometry was shown to be significant, changing considerably the aerodynamic flow field of the turbine, the results highlight the potential of the developed methodologies for this type of simulations. The combustor-HPT simulation, allowing the propagation of all heterogeneities generated in the chamber, confirmed that the turbine inflow is very rich in turbulent structures and contains strong temperature fluctuations. Both simulations recover established trends, such as the segregation effect or the impact of secondary flows on the aerothermal flow field of the turbine or the migration of the non-uniformities. However, the coupled combustor-HPT LES indicates that while the time-averaged temperature field at the NGV entry is not changing significantly, the migration across the turbine and the thermal load on the blades can be impacted by the highly unsteady nature of the incoming flow. The relatively limited wall resolution and the geometry modification do not allow for a more detailed evaluation of the turbine losses and performance with accuracy. However, with the necessary tools and guidelines established, such a study can be straightforwardly undertaken in the near future.
Chapter 6: LES of an industrial combustion chamber-turbine system
General conclusions and perspectives

The constant growth of the aviation sector, along with the increasingly restrictive environmental and noise regulations, are driving considerable research in the field of gas turbines. Improving the numerical predictions of the aerothermal flow field of the high-pressure turbine and the combustion chamber/turbine interactions are at the forefront of this effort. Increasing the fidelity of numerical simulations for these components can help unlock considerable gains:

- In efficiency, as optimized or less over-engineered designs with respect to cooling flows and heat loads can be targeted.
- In development costs, as less time on test benches will be required to confirm the numerical computations and finalize the designs.

Current industrial state-of-the-art predictions treat HPT and combustors in a completely decoupled way. With this approach large errors on the critical blade thermal loads are not uncommon and the acoustic interactions between the two components are lost. This dissertation is proposing a coupled approach to the combustor-HPT interactions problem using the high-fidelity LES formalism.

Part I of the thesis is focusing on the numerical treatment of rotor/stator interfaces in an LES context, an essential part for any turbomachinery stage simulation. An overset grid method is proposed to treat such problems in a rigorous fashion that is compatible with the reactive LES solver AVBP. The properties of the interface are shown not to impact the characteristics of the numerical schemes on a series of academic test cases of varying complexity. The approach is then validated on a realistic HPT stage against experimental measurements. As this type of simulation has not been undertaken before, a sensitivity analysis with respect to SGS models and mesh resolution is performed. The results are in good agreement with the measurements and information on the time-averaged and unsteady content of turbine flows is extracted and analyzed.

Part II focuses on applications of the developed methodologies for the prediction of two different types of combustor/turbine interactions.

- The first is the indirect combustion noise generation across a HPT stage, occurring due to the acceleration of combustor-generated temperature non-uniformities (entropy waves) as they propagate through the turbine. The investigation does not
consider the combustor. Instead entropy waves are injected through the boundary conditions at the turbine inlet to simulate the effect. DMD is employed to post-process the results and analyze both the overall turbine noise mechanisms and the indirect noise generation. The measured acoustic waves are compared to predictions from an analytical theory or 2D simulations and are found to be of similar quality. The entropy waves on the other hand are found to be more attenuated as they pass through the turbine.

- The second application is a fully-coupled combustor-HPT simulation that investigates the interactions between the two components from an aerothermal point of view. The unsteady characteristics of flow at the turbine inlet are analyzed along with the migration of the temperature heterogeneities. A standalone HPT simulation serves as a benchmark to compare the impact of the fully coupled approach. Results show that strong turbulence and temperature fluctuations arrive from the combustion chamber with a large radial variation and they potentially have a considerable impact on the blade surface temperatures as they go through the blade passages, as modified migration patterns are observed and enhanced mixing.

Following this dissertation, several directions of further work and improvements can be identified on the topics discussed in this work:

- Different methods to computationally accelerate the rotor/stator interface can be evaluated, such as tabulating the interpolation coefficients or recalculating the interpolation coefficients less frequently. Methods that reduce the computational domain to one blade passage per blade row while avoiding the modification of the blade counts can also be evaluated, such as the time-inclined method [221] or the phase-lag method [66]. However, such methods have not been tested in an LES context and need to be able to conform with the strict requirements for high-fidelity LES and transport correctly broadband phenomena.

- Perform LES of the MT1 HPT with the addition of inlet turbulent fluctuation. The results, particularly if the high-resolution mesh is used, can allow to establish some conclusions on the impact of free stream turbulence on the predictions in conjunction with wall-modeled LES.

- A wall-resolved LES of the MT1 HPT stage can be performed to improve the validation of the method on this realistic configuration. Such a simulation can also provide more information on the general capacity of LES to predict aerodynamic losses, transition, tip clearance flows and heat transfer, areas where considerable gains are expected from the LES formalism. Additionally, it can serve as a benchmark and provide a solid database for future comparisons, for example with lower cost simulations experimenting with novel wall models. However, the cost for such a simulation is expected to be be upwards of 5M CPUhours due to the explicit time integration of the solver.
Concerning indirect combustion noise, a simulation with multifrequency forcing can be performed to examine the levels of the generated noise as a function of the frequency. The realistic fully-coupled combustor/turbine simulation can also be continued and analyzed from this perspective to evaluate the combustion noise generated across the turbine from real entropy fluctuations instead of model entropy waves. Studies of non-linear effects should also be performed, as the simulations revealed non-linear interactions that could impact the generated noise.

Finally, computing the fully coupled case using a more accurate representation of the industrial geometry (the rotor was strongly modified) will allow to establish more reliable conclusions on the aerothermal flow field across the turbine. Such predictions from a real configuration with an accurate turbine inflow will also help to validate observations and conclusions drawn from cheaper RANS/URANS simulations and correlations or other low-order models.

The biggest challenge that is envisioned after this Ph.D is a fully coupled LES investigation of the entire high-pressure spool of a gas turbine: high-pressure compressor, combustion chamber and high-pressure turbine. The proposed tools are not only applicable to single-stage turbine configurations but in multi-stage compressor/turbines as well. Such a simulation can be within reach only if the necessary computational resources become available in the future, which first requires an accurate evaluation of the CPU cost of such simulations. The biggest difficulty in this evaluation is to determine the minimum number of blades/combustor sectors that have to be used in order to obtain a good ratio between accuracy loss due to scaling and computational cost gains. An additional parameter for this evaluation is the engine type, as large gas turbines typically have multi-stage compressors and turbines while smaller helicopter engines have one or more centrifugal compressors.

The information that could be extracted is considerable and of interest for all three components. For example, the off-design predictions in compressors could be addressed by such tools and they can also contribute to the understanding of thermoacoustic instabilities in the combustors as well as the combustion noise transmission and hot-streak migration across the turbine.
General conclusions and perspectives
Lire la troisième partie de la thèse