Methods and Uncertainties for the Flow Investigation in a High-speed Multi-stage Compressor

Xavier Ottavy\textsuperscript{1} and Nicolas Courtiade\textsuperscript{2}

\textit{Laboratoire de Mécanique des Fluides et d’Acoustique, Ecole Centrale de Lyon, 69130 - Ecully, France}

and

Nicolas Gourdain\textsuperscript{3}

\textit{CERFACS, 31057 - Toulouse, France}

This study takes place in the frame of a research project to better understand the flow that develops in a multistage high-speed compressor. First, the paper presents the high-speed 3.5-stage compressor CREATE under investigation and the methods which are considered to increase the data reliability and the investigation capability with such realistic compressors. Second, the numerical model is presented and flow simulations, achieved with 3D unsteady RANS computations over the whole compressor spatial and temporal periodicities, are analyzed through a study of their sensitivity to some parameters such as the inlet conditions, the space and time discretization and some technological effects. Finally the paper focuses on the methodology used to compare the experimental and numerical results over the whole compressor spatial and temporal periodicities. The mean flow features are well predicted. The local spatial and temporal flow structures are also well estimated. The key is the advection of these structures which interact with each other and produce a significant part of the flow fluctuations in the downstream stages.

Nomenclature

\begin{align*}
H & = \text{span height [m]} \\
h/H & = \text{location of the investigated point in the spanwise direction [%]} \\
Q & = \text{mass flow rate [kg.s}^{-1}] \\
\end{align*}

\textsuperscript{1} Dr., Researcher at Centre National de la Recherche Scientifique, Turbomachinery team, xavier.ottavy@ec-lyon.fr.
\textsuperscript{2} Ph.D Student, Turbomachinery team, nicolas.courtiade@ec-lyon.fr.
\textsuperscript{3} Dr., Senior researcher, CFD Team, nicolas.gourdain@cerfacs.fr.
$R_p$ = total pressure ratio

$R_T$ = total temperature ratio

$u_x, u_\theta, u_r$ = axial, tangential and radial velocity components [m.s$^{-1}$]

$U_{\text{ref}}$ = blade rotating speed at the inlet mid-span (section 26A) [m.s$^{-1}$]

$t$ = time [s]

$T$ = compressor time period = $2\pi/16/\Omega$ [s]

Greek symbols:

$\alpha$ = angle between the tangential and the axial component of the velocity [degree]

$\phi$ = normalized mass flow rate: $\phi = Q/Q_{\text{nom,exp}}$

$\psi$ = normalized total pressure ratio: $\psi = (R_{p,\text{out}}/R_{p,\text{out,nom}})$

$\eta$ = normalized isentropic efficiency: $\eta = \eta/\eta_{\text{nom,exp}}$

$\Omega$ = rotation speed of the rotors [rad/s]

Indices:

$\text{atm}$ = for the atmospheric conditions

$\text{chocked}$ = for the chocked operating point

$\text{exp}$ = from the experiment

$i,j$ = indice for the time and space respectively

$\text{nom}$ = for the nominal operating point

$\text{stall}$ = for the operating point at the onset of stall

$t$ = for stagnation conditions

$_{\text{in}}$ = for the inlet conditions

$_{\text{out}}$ = for the outlet conditions

$_{d}$ = for the differential measurements compared to the atmospheric pressure

Averages:

$\bar{.}$ = time-averaged value

$\bar{\theta}$ = space-averaged value (circumferential direction)
I Introduction

THE design of efficient “green” gas turbines requires a better prediction of the components performance and understanding of unsteady flows. Among all components, the compressor remains a critical part, especially regarding its efficiency and stability. Therefore, some experimental test-rigs have been developed all around the world to study complex flows in these systems. Most rigs are low-speed and allow to study relatively “easily” unsteady flow phenomena and instabilities (such as rotating stall and surge) occurring in compressors. For example, the low-speed 4-stage compressor of Dresden already provides very useful data for the study of rotor-stator interactions and rotating stall [1-2]; However, only a few configurations consider flow conditions close to real engine operating conditions (high-speed flows, multistage machine), mainly due to their cost of development and exploitation. Some of these (rare) high-speed rigs dedicated to multistage compressors are presented in Table 1.

Complementary to experimental investigation, the numerical simulation of flows, commonly refereed as the Computational Fluid Dynamics, is a very promising way to investigate flows close to real operating conditions. While this technique is effective to reproduce flows in simple geometries, it remains a challenge for CFD to predict such complex flows as observed in multistage compressors with existing technological effects.

The present paper proposes thus to investigate the flow in a 3.5-stage high-speed compressor, representative of a high-pressure compressor of modern turbojet engines. Since the test rig is fully instrumented for unsteady pressure and laser measurements, it is an excellent opportunity to learn more about complex (unsteady) flows in multistage compressors. This compressor is also designed to become a reference test case for code validation that is the second topic of this paper. Unsteady flow simulations have been carried out on the whole compressor periodicities with a particular interest for the estimation of their sensitivity to several parameters. Finally, a detailed comparison between experimental and numerical database is performed to set the state of the art regarding the capability to predict and understand the flow in this compressor.

II Presentation of the compressor CREATE

The test case considered for this study is a research multistage compressor dedicated to aerothermal and aerodynamic studies. This 3½-stage axial compressor, named CREATE (Compresseur de Recherche pour l’Etude des effets Aérodynamiques et TÉchnologiques) has been designed and built by Snecma. Its geometry and its rotation
speed are representative of High Pressure Compressor median-rear blocks of modern turbojet engine. The number of stages was chosen in order to have a magnitude of the secondary flow effects similar to a real compressor, and to be within the rig torque power limitation. Snecma and the research team of the LMFA have taken into account technological constraints coming from the experimental part of the project, very early in the compressor design. In order to have traversal probes between blade and vane rows, the axial gap was slightly increased compared to current compressors and an outer-case moving-rings technology was implemented to perform probe measurements in the circumferential direction at constant radius location. The circumferential periodicity of the whole machine (obviously $2\pi$ in general case with primary blade numbers) has been reduced to $2\pi/16$ on the compressor CREATE, choosing the number of blades of each rotor and stator (Inlet Guide Vane -IGV- included) as a multiple of 16 (see Table 2). Consequently, measurements carried out over a sector of only $2\pi/16$ (namely 22.5 degrees) should contain all the spatial information (in the case of stabilized operating points) and are very useful when devoted to detailed studies, such as rotor-stator interaction analysis. The meridian view of the compressor and the inter-row measurement sections are presented in Fig. 1.

### Table 1 Overview of some research high-speed multistage compressor test rigs

<table>
<thead>
<tr>
<th>Reference</th>
<th>Localization</th>
<th>Number of stages</th>
<th>Tip Mach number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sanders et al. [3]</td>
<td>Purdue University (USA)</td>
<td>1.5</td>
<td>0.92</td>
</tr>
<tr>
<td>Nakakita et al. [5]</td>
<td>Ishikawajima Harima Heavy Industries, Tokyo (Japan)</td>
<td>3</td>
<td>1.35</td>
</tr>
<tr>
<td>Bennington et al. [6]</td>
<td>University of Notre Dame (USA)</td>
<td>1.5</td>
<td>1.10</td>
</tr>
<tr>
<td>Biela et al. [7]</td>
<td>Technische Universität Darmstadt (Germany)</td>
<td>1.5</td>
<td>1.35</td>
</tr>
<tr>
<td>Ernst et al. [8]</td>
<td>Aachen University (Germany)</td>
<td>2</td>
<td>0.89</td>
</tr>
<tr>
<td>Present test case - CREATE</td>
<td>Ecole Centrale de Lyon (France)</td>
<td>3.5</td>
<td>0.92</td>
</tr>
</tbody>
</table>

### Table 2 Blades number of the compressor rows

<table>
<thead>
<tr>
<th>Row</th>
<th>IGV</th>
<th>R1</th>
<th>S1</th>
<th>R2</th>
<th>S2</th>
<th>R3</th>
<th>S3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades per row (for $2\pi$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>64</td>
<td>96</td>
<td>80</td>
<td>112</td>
<td>80</td>
<td>128</td>
<td></td>
</tr>
<tr>
<td>Number of blades for $2\pi/16$</td>
<td>2</td>
<td>4</td>
<td>6</td>
<td>5</td>
<td>7</td>
<td>5</td>
<td>8</td>
</tr>
</tbody>
</table>
### Table 3 Characteristic of the compressor at design rotating speed

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylindrical outer casing diameter</td>
<td>0.52 m</td>
</tr>
<tr>
<td>Core Speed</td>
<td>11,543 rpm</td>
</tr>
<tr>
<td>Rotor1 tip speed</td>
<td>313 m/s</td>
</tr>
</tbody>
</table>

Fig. 1 Meridional view of the 3.5-stage axial compressor CREATE.

### III Global aerodynamic performance

The studied compressor is a very complex test case, both from the experimental and numerical point of views. The complexity comes from the flow itself (high Reynolds number) but it also comes from the geometry (3½ stages) and technological effects (gaps between fix and moving parts). Indeed, the flow analysis is very tricky and there are many sources of errors and/or uncertainties. In that context, it is meaningless to compare the numerical characteristic with the experimental one without quantifying the measurements and the numerical uncertainties. For example, for complex test cases, the experimental facility and the numerical domain do not reproduce the same geometry (technological devices, inflow conditions, etc.). One of the objectives of this paper is to provide magnitude orders for some of these uncertainties. Fig. 2 proposes a comparison of the experimental and numerical compressor test case performance following this idea. Numerical data are computed with a classical RANS simulation with the fine grid described in section V.1. The detail of the method to estimate the uncertainties is described in section IV (experimental data) and section V (numerical data). The shape of the numerical performance curve is identical to the experimental one. The stable operating ranges (i.e. the range from the choked mass flow $\phi_{\text{chocked}}$ to the last stable operating point $\phi_{\text{stall}}$) measured and predicted with the numerical simulation are also similar (difference is less than 1%). However, the numerical performance curves are shifted towards higher mass flow: the numerical values for $\phi_{\text{chocked}}$ and $\phi_{\text{stall}}$ are both increased by 2% with respect to experimental values. Another important difference is that
Experimental uncertainties are quite small ($\Delta \phi = \pm 0.35\%$, $\Delta \psi = \pm 0.07\%$ and $\Delta \eta = \pm 0.20\%$) compared to the numerical ones ($\Delta \phi = \pm 2.05\%$, $\Delta \psi = \pm 2.06\%$ and $\Delta \eta = \pm 1.21\%$). This observation is correlated with the difficulty for the numerical simulation to account for the unknown “details” of the compressor geometry (such as technological effects). Inflow conditions also influence the compressor performance. Finally, when uncertainties are taken into account, the numerical characteristic compares reasonably well with the experimental data. However, the high levels of uncertainties relative to the test case performance indicate that the numerical simulation and measurements should be considered as two different means with their own bias to investigate the flow: as a consequence there is no sense to rigorously seek for the same results with both methods.

Fig. 2 Compressor aerodynamic performance – comparison of numerical and experimental data, including uncertainties (see section IV and section V for details)

IV Experimental facilities and related uncertainties

The compressor CREATE is tested at Ecole Centrale de Lyon in LMFA. The test stand is designed as an open loop. The ambient air is led into the compressor (Fig. 3-1) through a settling chamber (Fig. 3-2) with air filter and a throttle that drops the inlet total pressure to 0.74 of the atmospheric pressure and enables to decrease the needed electric power. The rotors shaft is driven at the design speed of 11,543 rpm by a 2 MW three-phase AC drive (Fig. 3-3) coupled with a gearbox (Fig. 3-4). At this rotational speed, the Mach number at the tip of the first stage is 0.92. Indeed the flow is slightly transonic in the first stage and fully subsonic in the two last ones. A throttling butterfly-
type valve (Fig. 3-5) is used downstream the collector (Fig. 3-6) to control the mass flow rate, which is measured using a Venturi nozzle (not presented in Fig. 3) located just before the exhaust of the test-rig. The anti-surge valve (Fig. 3-7) is located just upstream the throttling butterfly-type valve. In case of surge, this valve is very quickly operated to discharge the system.

![Image of the compressor CREATE test rig in LMFA at Ecole Centrale Lyon.](image)

**Fig. 3** The compressor CREATE test rig in LMFA at Ecole Centrale Lyon.

### A Rig instrumentation

1. **Global performance measurements**

   Ten sensors measure the inlet total temperature in the settling chamber. The inlet total pressure is obtained with a Kiel probe in front of the compressor. This choice has been fixed after the use of several ways to obtain the inlet total pressure. Total pressure and total temperature probe rakes, located at six circumferential and five radial positions in the outlet section upstream the discharge collector, give the exit flow conditions. As mentioned before, the operating point of the compressor is adjusted by throttling a butterfly-type valve downstream the compressor. The characteristic curves of the compressor are obtained from the pressure ratio, the isentropic efficiency and the mass flow rate, with the relations hereafter.

   - Pressure and temperature ratio:

   
   
   \[
   R_p = \frac{P_{t\text{ out}}}{P_{t\text{ in}}} \quad \text{and} \quad R_T = \frac{T_{t\text{ out}}}{T_{t\text{ in}}} \tag{1}
   \]


where

$$P_{i\_out} = \frac{1}{n_{P\_out}} \sum_{j=1}^{n_{P\_out}} P_{i\_d\_out,j} + P_{\text{atm}} = P_{i\_d\_out} + P_{\text{atm}} \quad \text{and} \quad P_{i\_in} = P_{i\_d\_in} + P_{\text{atm}}$$  \hspace{1cm} (2)

$$T_{i\_out} = \frac{1}{n_{T\_out}} \sum_{j=1}^{n_{T\_out}} T_{i\_out,j} \quad \text{and} \quad T_{i\_in} = \frac{1}{n_{T\_in}} \sum_{j=1}^{n_{T\_in}} T_{i\_in,j}$$  \hspace{1cm} (3)

with the subscripts \( P \) for pressure, \( T \) for temperature, \(_{\text{in}}\) for inlet, \(_{\text{out}}\) for outlet, and \(_d\) for differential pressure compare to the atmospheric one.

- Isentropic efficiency:
  \[
  \eta_s = \frac{(R_p)^{\frac{\gamma-1}{\gamma}} - 1}{R - 1}
  \]  \hspace{1cm} (4)

- Mass flow rate

The classical expression used to measure the mass flow rate \( \phi \) with a Venturi Nozzle is (see the norm AFNOR [9])

$$\phi = C E \varepsilon \frac{\pi d^2}{4} \sqrt{B}$$  \hspace{1cm} (5)

In this expression, \( C \) is a discharge coefficient, \( E \) is a geometric coefficient, \( \varepsilon \) is a compressible effects coefficient, \( d \) is the throat diameter of the Nozzle and \( B \) is obtained with the measured quantities at the inlet and at the throat of the Venturi Nozzle.

In order to have a global picture of the aerodynamical and mechanical state of the rig, a set of 210 measurements is performed in less than 2 seconds. It contains temperature and pressure measurements for the flow and the oil, as well as accelerometers and strain gauges.

All acquisition devices and computers dedicated to the operating functions and the surveillance of the test rig are connected together with a server that saves all the measurements and information in a MySQL data base. The server is also the time reference to date all the measurements and events. The campaigns are carried out in the frame of a quality plan based on the ISO-9001 version 2000 model.
2 Quantification of uncertainties

All the relative uncertainty levels of the measurements have been calculated using the Guide to the Expression of Uncertainty in Measurements [10] published by the International Organization for Standardization (ISO) and revised by Coleman and Steele [11], assuming a Gaussian distribution error. For the nominal operating point, they are summarized in Table 4 with a 95% coverage estimate of the uncertainty in the results. This method has been preferred to the sum of the absolute value of all the uncertainty contributions which is not really representative of the quality and the reliability of the measurements achieved with this kind of experimental setup. The confrontation of the measurements acquired during different campaigns reinforces the authors in their choice. The uncertainty bars plotted in Fig. 2 have been calculated with these considerations for each measurement points.

| Table 4 Relative experimental uncertainty levels at the nominal operating point |
|--------------------------------------------------|-------------------------------|
| Pressure ratio \( \psi \)                       | ±0.07%                        |
| Isentropic efficiency \( \eta \)                 | ±0.20%                        |
| Mass flow \( \phi \)                            | ±0.35%                        |

Remarks about Table 4 Relative experimental uncertainty levels at the nominal operating point:

- In order to have stabilized temperatures and performance during a test session, the first measurements to characterize an operating point of the compressor is achieved 30 minutes after the first revolutions of the machine. Then a time gap of 10 minutes between each new operating point is needed.
- Concerning the calculation of the relative uncertainty level of the pressure ratio, and thus of the efficiency, the calibration process uncertainty of the probe rakes in the outlet section of the machine has not been taken into account.
- In relation (5), the coefficient \( C \) depends of a losses coefficient \( \xi \) that is unknown. In the literature, only the range [0.975 ; 1.00] of this coefficient \( C \) is given, depending on the geometric characteristic of the Venturi nozzle. A calibration of our Venturi Nozzle has been performed using the results of the previous campaign (same machine with a second Venturi Nozzle). The value of \( C \) has then been fixed to 0.985. An error on this value would introduce a bias in the mass flow measurements, but would not change the repeatability of the measurements. Taking all the possible value in the range [0.975 ; 0.995] would increase the relative uncertainty of the mass flow rate to ±1.32%.
- The uncertainty on the mass flow rate takes into account the uncertainty induced by the standardization of the results by the inlet pressure and temperature.
- The influence of the inlet temperature on the heat capacities \( C_p \) and \( C_v \) is taken into account.
- The influence of the hygrometry has been checked and is negligible here.

### B Specific Instrumentation

A backscatter Laser Doppler Anemometer (LDA), designed and built by Dantec, was used to perform all the unsteady measurements presented in this paper. Two pairs of beams \((l=488 \text{ nm and } l=514.5 \text{ nm})\) are used for measuring simultaneously two velocity components which lead to the determination of the axial and tangential velocity components. The focal length of the front lens is 250 mm. A diameter of 76 µm and a length of around 0.9 mm characterize the measurement volume. The signals were treated by two Dantec real-time signal analyzers.

The measurements were triggered with the rotation frequency of the machine, in such a way that the flow field is described either inside a single blade passage, or within several blade passages covering the circumferential periodicity of the whole machine. The data reduction process, which consists in a phase-lag average, filters the random time scales of the turbulent flow. Thus, the unsteadiness captured only relates to phenomena clocked with the rotor passing frequency. Concerning the turbulence, parts of this information can then be found in the standard deviation of the velocities histograms.

The anemometer was carried on a six-axis robot allowing the displacement of the measurement point. This system prevents the optical assembly of the anemometer being sensitive to machine vibrations. Due to the axial thrust and thermal dilatation of the machine, a procedure was defined for positioning the LDA control volume during the compressor operation. The uncertainty of the location of any measurement point is then estimated to ±0.15 mm. The spatial and temporal discretizations for these measurements were chosen in agreement with the methodology described in [12] to minimize the interpolation errors in rotor/stator interaction analysis.

The compressor was seeded with a polydisperse aerosol of paraffin oil. The size of the particles at the outlet of the seeding generator was measured and its mean value is smaller than 1 µm. Seeding was performed upstream of the settling chamber. In such a flow configuration (low centrifugal forces and moderated decelerations), Ottavy et al. [13] proved that this technique was reliable. However, the seeding is identified to be the most important source of bias in this configuration: the oil, needed to operate the compressor, is also present in the compressor vein, with
some particles whose size is not controlled and may be larger than 1 µm. The concentration of these particles is higher in the wakes of the rotors, because centrifuged and spread by the blades at the trailing edge.

Once all the quantifiable uncertainties have been taken into account, the measurement relative error of the velocity components does not exceed ±1.0%. The velocity angles are calculated from the velocity components, and the uncertainties on the angles are deduced from the ones of the velocity components.

V Numerical Method

A. Flow solver and boundary conditions

The governing equations are the compressible Navier-Stokes equations that describe the conservation of mass, momentum and energy. In conservative form, it can be expressed in three-dimensional coordinates as:

$$\frac{dW}{dt} + \text{div} F = 0$$  \hspace{1cm} (6)

where $W$ is the vector of primary variables, $F = (f, f_v, g, g_v, h, h_v)$ is the flux tensor; $f, g, h$ are the inviscid fluxes and $f_v, g_v, h_v$ are the viscous fluxes (including the potential contribution of turbulence models through the value of the turbulent viscosity $\nu_t$). The fluid follows the ideal gas law $P = \rho r T$ where $r$ is the mixture gas constant. The viscous stress tensor and the heat diffusion vector use classical gradient approaches. The fluid viscosity follows Sutherland’s law and the heat flux follows Fourier’s law.

The elsA software uses a cell centered approach on structured multi block meshes. More information about this flow solver can be found in [14]-[15]. For this application, convective fluxes are computed with a third-order scheme, based on the flux splitting method [16]. Diffusive fluxes are calculated with a classical second order centered scheme. For (steady) RANS simulations, the pseudo time-marching is performed by using an implicit time integration scheme, based on the backward Euler scheme and a scalar Lower-Upper (LU) Symmetric Successive Over-Relaxation (SSOR) method [17]. To obtain a time consistent solution (for unsteady RANS simulations), the time marching method is based on the second order Dual Time Stepping technique proposed by Jameson [18].
turbulent viscosity $\nu_t$ is computed with the two equations model of Wilcox [19] based on a $k-\omega$ formulation. For this application, the flow is assumed to be fully turbulent since the Reynolds number based on the chord is around $10^6$.

Boundary conditions are detailed in [20]. The inlet of the calculation domain is located at position 25A (see Fig. 1) where experimental pressure probe measurements are available to define the inlet conditions. The outlet duct behavior is modeled using a throttle condition, coupled with a simplified radial equilibrium law. The outlet static pressure $P_{s_{out}}$ is set at each time step by means of the relation:

$$P_{s_{out}}(t) = P_{t_{in}} + \lambda Q(t)^2$$

where $P_{t_{in}}$ is the upstream infinite total pressure, $Q(t)$ is the instantaneous mass flow through the exit section and $\lambda$ is a parameter. At the rotor-stator interfaces, a mixing plane method is used for RANS simulations and a sliding mesh condition is used for unsteady RANS simulations [21]. Only one blade passage of each row is considered for steady flow simulations. For unsteady flow simulations, the whole natural spatial periodicity of the compressor (a $2\pi/16$ sector) is used with classical condition of spatial periodicity for lateral boundaries.

![Fig. 4 Evolution of the normalized mass flow $\phi$ at the compressor inlet during an unsteady RANS simulation](image)
B. Computational cost

All the simulations are estimated with the fine grid described in section V.C.1 and calculations are performed on a SGI Altix scalar computer with 32 computing cores (Intel Harpertown). At the nominal operating point, about 5000 iterations are needed to converge a RANS simulation. It requires 250 CPU hours. As shown in Fig. 4 for unsteady RANS simulations, the physical time needed to reach a periodic state is one rotation at the choked operating point and up to two rotations for the last stable operating point (close to the surge line). A rotation of the compressor is completed in 7800 CPU hours (with the small time step defined in section V.C.1).

C. Quantification of some numerical uncertainties

It is well known that numerical simulations of flows are still far to be predictive, especially when considering complex industrial configurations such as multistage compressors. As recently discussed by Denton [22], four kinds of errors are responsible for this problem: numerical errors (induced by mesh grids, numerical schemes, etc.), modeling (RANS, URANS, LES, etc.), boundary conditions (that are often not well known) and geometry (that is never identical in the experiment and the simulation). This paragraph mainly focuses on the quantification of some of these “uncertainties” on the compressor performance and details about the flow physics are indicated only if necessary.

1. Mesh grid and time step refinements

The numerical domain is discretized with a multi block approach, using an O-H meshing strategy for each passage of the compressor (Fig. 5). To minimize the computational cost, the wall cell size is set to ensure a normalized wall distance $y^+$ below 20 everywhere in the domain. Wall functions are then applied to improve the quality of numerical results [23]. This approach gives very satisfying results for non-separated boundary layers but results for massively separated flows, which are not the case in this work, should be considered with caution.

Two mesh grid qualities are considered. For the coarse grid, a blade passage is meshed with about 250,000 points (145 points around the blade and 57 in the radial direction, including 13 points in the tip clearance). The total number of nodes to represent the three compressor stages is 1.51 millions nodes for steady RANS simulations (only one passage per blade row is meshed) and 8.39 million for unsteady RANS simulations (a $2\pi/16$ sector is meshed).

The fine grid corresponds to the “standard” industrial grid quality with 1.08 million nodes to mesh one passage (289 points around the blade and 109 in the radial direction, including 25 points in the tip clearance). The total
number of nodes to represent the three compressor stages with the fine grid is 6.46 million for steady RANS simulations and 35.98 million for unsteady RANS simulations.

**Effect on the compressor performance map**

Fig. 6 shows the compressor performance map (pressure rise and isentropic efficiency) predicted with a RANS simulation at nominal rotating speed with both coarse and fine grids. This comparison indicates that the coarse grid gives roughly the same results than the fine grid, including the value of the mass flow at chocked conditions ($\phi = 1.07$) and peak efficiency ($\phi = 1.00$). Most important differences are found at near stall conditions (for $\phi < 0.966$). The coarse grid predicts the stability limit at $\phi = 0.932$ while the fine grid found it at $\phi = 0.941$ (difference is less than 1%). At $\phi = 0.941$, the difference for efficiency is 0.6%. The use of a coarse grid (with a 3rd order numerical scheme) is thus sufficient to predict the compressor performance map with a reasonable accuracy. However, this conclusion is valid only because a “high” order scheme is used in this case for convective fluxes (simulations with second order schemes show differences close to 5% for pressure ratio and mass flow between coarse and fine grids).

**Effect on unsteady flows**

Unsteady flows are sensitive to both mesh and time step refinements. Unsteady RANS simulations are performed with two different time steps. The “large” time step is equal to the rotor-passing period $T$ divided by 3200 (50 time steps are used to describe the blade passing frequency of the first blade row). The “small” time step is equal to the “large” time step divided by 2 (the blade passing frequency of the first blade row is discretized by 100 time steps). Fig. 7 shows the normalized axial velocity fluctuations $u_x'(t)$ at the interface between the rotor 3 and the stator 3 (section 28A), close to the casing wall ($h/H = 83.7\%$). For clarity reasons, only five rotor blade passages (i.e. the compressor periodicity) are represented in the figures. At each blade passage, two “negative” peaks are observed: the most important one (for example at $t=0.3$) is related to the rotor wake and the second one (at $t=0.25$) corresponds to the tip leakage flow. On the one hand, simulations with the “large” time step show that coarse and fine grids predict the same phenomena with the same magnitude order (Fig. 7a). On the other hand, the simulation with a “small” time step predicts a different behavior than the simulation with the “large” time step. More precisely, the trajectory and the strength of the tip leakage flow are not identical in both cases. While the contributions of wake and tip leakage flow are well separated in Fig. 7a (two different peaks at each blade passage), the velocity deficits due to the wake and the tip leakage tend to merge together with the “small” time step in Fig. 7b (only one peak is observed at each blade passage). Since the flow stability in this compressor is related to the wall tip flow, this
behavior is expected to have an influence on the onset of the instability. In all cases, the amplitude of the fluctuations is close to 15% of the mean axial velocity.

Moreover, experimental data shows a similar behavior where only one peak is observed (see Fig. 15 in section VI). This observation is interesting since it illustrates the necessity to refine both the mesh and the time step to obtain a good accuracy in numerical simulations. For turbomachinery applications, a method to ensure coherence between the mesh size and the time step has been proposed in [24].

Fig. 5 View of the compressor grid at h/H=50% (1 point over 2) - 2π/16 sector.

Fig. 6 Effect of mesh grid refinement on total pressure rise and isentropic efficiency (RANS simulations).
(a) Effect of mesh refinement

(b) Effect of time step refinement

Fig. 7 Effect of mesh (a) and time step (b) refinements on the flow unsteadiness - temporal fluctuation of the axial velocity $u_x$ normalized with the mean axial velocity at $h/H = 83.7\%$ (section 28A, nominal operating conditions)

2 Influence of the time-integration algorithm

The simulation of the turbulent spectrum in a multistage compressor is still out of range with current means. Based on the study of Gomar et al. [25], a Large Eddy Simulation (LES) in the CREATE compressor will need a 3,500M points grid (i.e. a 100 bigger grid size than the current grid for unsteady RANS simulations). Indeed, it is expected that a LES should not be feasible in this compressor before 2020 (if computing resources still follow the Moore’s law). Since turbulence needs to be fully modeled, the only unsteady flow that can be investigated with an unsteady RANS approach is the deterministic flow (i.e. the unsteadiness induced by the relative motions between blade and vane rows such as wakes, potential effects, secondary flows and their interactions). Fig. 8 shows the compressor performance map estimated with RANS and unsteady RANS simulations at the nominal rotation speed with the fine grid. The comparison of results indicates that both numerical methods predict identical pressure rise coefficient $\psi$ and efficiency $\eta$ from $\phi=1.00$ (design operating point) to $\phi=1.07$ (chocked conditions).
At near stall conditions ($\phi=0.945$), the RANS simulation over-estimates the isentropic efficiency by 1% with respect to unsteady RANS predictions. Another difference lies in the estimation of the stability limit: the RANS simulation predicts it at $\phi=0.941$ while the unsteady RANS simulation found it at $\phi=0.927$ (difference is 1.5%). To give an order of magnitude, such a difference on the stability limit represents 10% of the stable operating range.

For this application, the RANS approach appears to be sufficient to predict the compressor performance with a quite good accuracy, except in the “close to surge” region where an accurate estimation of the compressor aerodynamic stability requires unsteady flow simulations. However, local differences between RANS and unsteady RANS simulations are pointed out in Fig. 9a (nominal operating point, $\phi=1.00$) and Fig. 9b (near stall operating point, $\phi=0.945$) that show the flow field at the last stage interface (section 28A) colored with the local mass flow. The difference between both solutions $\zeta$ is defined as

$$\zeta = (\phi_{\text{RANS}}/\phi_{\text{URANS}}) - 1 \quad (8)$$
Fig. 9 Time-averaged unsteady flow solution, steady flow solution and difference between the two solutions. Local normalized mass flow $\phi$ at section 28A at (a) nominal operating point ($\phi=1.00$) and (b) near stall operating conditions ($\phi=0.945$)
The comparison between RANS and unsteady RANS solutions shows that differences are mainly observed in two regions. Close to the corner between the suction side and the casing walls, the RANS approach underestimates the local mass flow by 25% (with respect to the unsteady RANS solution). Behind the blade trailing edge, RANS shows an overestimation higher than 25% of the local mass flow in the wake region (the deficit of axial velocity in the wake is thus more important with the unsteady RANS simulation). This difference is more important at lower mass flow (Fig. 9b) since the local mass flow predicted with RANS in the wake region is increased by 50% when compared to the unsteady RANS solution.

No assumption is done in this paragraph about which is the “correct” solution, however one of the feature of RANS is to filter all the unsteady flows. It also means that the differences observed in Fig. 9a and Fig. 9b are only related to the time-averaged effect of unsteady flows. The last part of the paper shows clearly the importance of the interactions between the flow structures generated by the upstream rows to predict correctly the flow fluctuations (in space and time). Moreover, as already mentioned in section V.C.1, the capability for numerical simulations to help in the understanding of compressor stall inception is largely related to its ability to accurately describe the flow in the tip region. Based on this analysis, an unsteady RANS simulation is the most suited method to investigate the flow in this multistage compressor.

3 Inlet and outlet boundary conditions

Inlet and outlet boundary conditions are another source of uncertainty for flow solvers. If a study considers only the stable operating range (it excludes surge and rotating stall), the outlet boundary condition can be described following the strategy proposed in paragraph V.A. In this case it is sufficient to fix the problem at the outlet. The problem is much more complex for the inlet. A correct characterization of the flow at the compressor entrance requires to take into account the whole compressor geometry, including inlet ducts, struts and the intake (for a real engine). Since such an approach largely increases the computational cost, the authors chose another strategy for this application. To reduce the cost, the numerical domain does not account for the struts and the inlet guide vane. The definition of the inlet boundary condition only relies on experimental measurements at section 25A with pressure probes. The result of the experimental data interpolation on the numerical grid is shown in Fig. 10a (the dark strip corresponds to the inlet guide vane wakes, with two black spots induced by the IGV bottom and tip leakage flows close to the hub and the casing). Theoretically, it ensures similar inflow conditions both for experiments and
numerical simulations. However, experimental values are also affected by uncertainties. Indeed, it is necessary to quantify the sensibility of the numerical solution to the variation of the inflow conditions.

![Diagram](image_url)

**Fig. 10**  (a) View of the total pressure ratio at the inlet of the calculation domain and (b) influence of the inlet flow angle variation (RANS simulation). The dashed line with light symbols corresponds to the experimental value for the inlet flow angle \( \alpha_{\text{in}} \)

Fig. 10b shows a performance map (pressure rise and efficiency) obtained with three different flow angles at the inlet: the experimental value \( \alpha_{\text{in}} \), the experimental value \( \alpha_{\text{in}} + 2^\circ \) and the experimental value \( \alpha_{\text{in}} - 2^\circ \). Most of the time the experimental uncertainty on the angle does not exceed \( \pm 0.5^\circ \), but in the regions where it exists a strong pressure gradient, as in the wakes or in the tip leakage flows, this uncertainty may reach 2\(^\circ\). All simulations have been performed with a RANS approach considering the fine grid. The results underline the dependency of the choked mass flow to the inlet flow angle \( \alpha \): with respect to the reference case, the choked mass flow \( \Phi_{\text{choked}} \) is either increased by 1.8\% (\( \alpha_{\text{in}} + 2^\circ \)) or decreased by 0.7\% (\( \alpha_{\text{in}} - 2^\circ \)). The shape of the compressor performance is not modified when the flow angle is modified (efficiency and pressure ratio curves are simply “shifted” with respect to the reference). The stable operating range is similar for the case “(\( \alpha_{\text{in}} \))” and “(\( \alpha_{\text{in}} - 2^\circ \))”, and it is reduced by 5\% for the case “(\( \alpha_{\text{in}} + 2^\circ \))”.
4 Influence of some technological effects

Technological devices are related to the design constraints such as gaps between fix and moving parts. The influence of two effects is quantified in this study: the tip gap (the distance between the top of the rotor blade and the casing) and the hub gap (the distance between rotating and fix parts of the hub)\(^4\). Both technological effects induce leakage flows that modify the flow topology and the compressor performance. Numerical simulations commonly consider the tip gap. However an accurate value of the rotor tip gap is not easy to obtain, mainly due to thermal effects and measurements uncertainties. For example, the uncertainty related to the current measurement methodology is 3%. The compressor geometry is modified to take into account for an increase by 3% in the size of the tip gap. The compressor performance is plot in Fig. 11a (all simulations are performed with a RANS approach considering the fine grid). No modification of the performance map is observed until the nominal operating point (\(\phi=1.00\)). At the last stable operating point (\(\phi=0.941\)), the efficiency \(\eta\) is reduced by about 0.5% with the larger tip gap and the pressure rise coefficient \(\psi\) is decreased by 1% (these values are similar to the difference observed when comparing fine and coarse grids).

The influence of the gap at the hub between fix and moving parts is quantified using the method described in [26]-[27]. The technique relies on the Chimera method to add the technological devices responsible for the leakage flows. Results are shown in Fig. 11b. For the configuration with hub leakage flows, the chocked mass flow \(\phi_{\text{choked}}\) is decreased by 1% with respect to the reference and both the pressure rise coefficient \(\psi\) and the efficiency \(\eta\) are reduced by 3% and 2% respectively. However, the stable operating range remains similar to the reference configuration. The main influence of the hub leakage flow is thus to shift the compressor characteristic towards lower mass flow with a 2% penalty on the efficiency (which is more important than the difference observed between steady and unsteady RANS solutions).

\(^4\) The technological effect that led to the flow recirculation under the stator vanes is visible in Fig. 1 (for example, the flow is aspirated at section 270, then it goes below the stator before to be injected in front of the stator leading edge at section 26A).
Fig. 11  Influence of technological effects on the compressor performance (RANS simulation)

(a) increase of the tip gap size by 3% and (b) simulation with hub leakage flows.

5  Estimation of the numerical test rig uncertainties

The fine grid provides grid independent results (for mean performance) and the unsteady RANS simulation is chosen as the “reference” to take account for the mean unsteady flow effects. It has been shown in the previous paragraphs that the numerical uncertainty $\Delta x$ of a variable $x$ depends on the uncertainties for the geometry (tip and hub gaps), the inlet boundary conditions, the modeling errors (RANS/URANS) and the grid density. In the present case, the two main sources of errors are the geometry and the inlet conditions, so $\Delta x$ is approximated with:

$$\Delta x = \Delta x_{\text{inlet condition}} + \Delta x_{\text{geometry}}$$  \hspace{1cm} (9)

The calculations of the relative uncertainty levels for the main global parameters are summarized in Table 5.

<table>
<thead>
<tr>
<th>Table 5 Mean relative numerical uncertainty levels</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure ratio $\psi$</td>
</tr>
<tr>
<td>Isentropic efficiency $\eta$</td>
</tr>
<tr>
<td>Mass flow $\phi$</td>
</tr>
</tbody>
</table>
VI  Comparisons between experimental and numerical data

1  Post-processing methodology

The unsteady RANS simulations with the small time step and the fine grid, without the hub leakage flows, provided all the numerical data which are compared with the experiment hereafter.

For the global performance of the compressor presented in Fig. 4, the total pressure rise coefficient \( \psi \) and the isentropic efficiency \( \eta \) are not computed by considering the whole outlet and inlet sections: only the information extracted at the experimental probe positions (mentioned in section IV.A.1) is used. While not rigorous from the mathematical and physical points of view, this method ensures that the same sets of data are compared. Finally, all data are post-processed with the same software, without distinction of their origin, numerical simulations or measurements.

In the following sections, the comparisons in the inter-row sections are performed using experimental data deduced from the LDA-2C measurements (velocities will be compared). To reduce the source of errors during the comparison, the experimental procedure is used to average numerical data. It means that numerical data are interpolated on the experimental mesh (in space and time) and are post-processed with the same methods and software.

At a specific geometrical position in the compressor, the time evolution of a measured quantity is obtained by a phase-lag average based on the compressor periodicity \( T=2\pi/16\Omega \). As already mentioned, this time evolution is a deterministic evolution and corresponds to the spatial evolution of that quantity in the rotating frame in the calculation. The time discretization being constant in the experiment, the time average of a quantity \( x_{j,i} \) measured for \( n \) time steps over a time period is calculated using a simple arithmetic average

\[
\overline{x_j} = \frac{\sum_{i=1}^{n} x_{j,i}}{n}
\]

(10)

where \( i \) and \( j \) are the indices for time and space respectively.
The space-average of time-averaged quantities, such as the velocities, should then be weighted by the momentum. The density being not accessible in the experiment with the LDA, the time-and-space-average, which leads to the axisymmetric value $x_{axi}$ is thus weighted with a coefficient $\alpha$ that depends only on the axial velocity $u_x$.

$$x_{axi} = \bar{x}_j = \frac{\sum_{j=1}^{p} \alpha_i \Delta \theta_j \bar{x}_j}{\sum_{j=1}^{p} \alpha_i \Delta \theta_j} \quad \text{with} \quad \alpha_j = \sqrt{u_{x,j}^2} . \quad (11)$$

In the next section the radial distribution of the axial velocity are compared using this time-and-space average (i.e. time-and-azimuthal average).

Moreover, the size of the LDA measurement volume can be neglected in the azimuthal direction but not in the optical axis direction which corresponds to the radial direction during the measurements. This is due to the fact that the size of this integration volume in the radial direction reaches 0.9 mm and corresponds to several percent of the span height in the last stages. The light intensity distribution in the focus volume has been measured and its distribution has been applied to a control volume used to integrate the velocity distribution in the calculation. The differences induced by this treatment reaches 1% in some specific zones, which is comparable to the LDA measurement uncertainty.

2 **Radial evolution at the inter-row positions**

A comparison of the radial evolution of the time-and-azimuthal-averaged axial velocity $u_x$ is shown in Fig. 12 (nominal operating point) and Fig. 13 (near stall operating point) at four locations: section 26A, 27A, 28A and 280 (see Fig. 1 for definition). The simulation is able to reproduce the correct shape of the experimental curves at all locations, at nominal and near stall operating points. Both set of data agree quite well for the value of the axial velocity $u_x$ close to the casing. At the nominal operating point (Fig. 14), discrepancies tend to increase when moving closer to the hub, especially at sections 26A and 27A (i.e. section at the rotor exit).
The comparison shows that the numerical simulation predicts a lower values of the axial velocity than the experiments below $h/H=30\%$ (difference at section 26A at $h/H=10\%$ reaches 9\%). The hub leakage flows are not considered in the unsteady RANS simulations, meaning this discrepancy is mainly due to the influence of these secondary flows on the radial profiles. At section 280, the difference between experimental and numerical data is close to 5\%. Contrary to sections at the rotor exits, the difference is constant with the radius at the stator exit. At near stall conditions (Fig. 13), both set of data agree well at section 27A (except close to the hub due to the hub leakage flows). At section 28A, results agree very well, especially close to the hub while the unsteady flow
simulation does not account for the hub leakage flows. One can note that the mean effect of the momentum deficit induced by the shroud boundary layer and the tip leakage flow is a bit overestimated by the calculation.

(a) Section 27A (rotor 1 exit)  
(b) Section 28A (rotor 2 exit)

![Graphs showing radial distribution of time-averaged axial velocity comparison of numerical results with experimental data at the near stall operating point (\(\phi=0.945\)).](image)

3 **Spatial fluctuations at the inter-row positions**

In order to point out the non-uniformity of the flow along the azimuth, data are time-averaged and the mean value is suppressed, following the relation:

\[
x'(\theta) = \frac{1}{n \Delta t} \sum x(t, \theta) - x_{axi} \quad \text{or} \quad x'_j = \overline{x_j} - x_{axi}
\]

The result is a tangential distribution that highlights the influence of the stator vanes. The distribution of the axial velocity is shown in Fig. 14 over a \(2\pi/16\) periodic sector close to the casing (\(h/H=83\%\)), at sections 26A, 27A, 28A and 280. This is a challenging location for the numerical flow solver since strong interactions occur at this place between upstream wakes, potential effects and the tip leakage flow. Experimental and numerical data predict a similar order of magnitude of these spatial fluctuations, except at section 28A, where the simulation predicts only half of the magnitude of the experimental fluctuations. What is interesting is that both the numerical and the experimental data agree at all sections about the number of observed “periods”. For example at section 26A (rotor 1 exit), potential effects of stator 1 drive the mean axial velocity signal (6 periods over a \(2\pi/16\) sector, i.e. 96 periods over a \(2\pi\) sector - cf. table 2) and the influence of the 2 wakes of the IGV modulates the signal (2 periods).
At section 27A, both sets of data indicate that the dominant influence still comes from the 6 wakes of stator 1. This analysis indicates that stator 1 has an influence far downstream (2 rows) of its position and the potential effects coming from the 7 blades of the stator 2 only modulate the signal (1 period = 7-6). At section 28A, seven periods are...
observed over a $2\pi/16$ sector, meaning the second stator generates most of the spatial distortions in the last stage. In this compressor the effects of the wakes coming from upstream are stronger than the one of the potential effects induced by the downstream row.

4 Quantification of the flow unsteadiness

The following section focuses on the unsteadiness that develops in the compressor. The comparison between unsteady numerical and experimental data is achieved with the temporal fluctuations $x''$ defined for a quantity $x$ as:

$$x''(t, \theta) = x(t, \theta) - \overline{x'(\theta)}$$

or

$$x'_{j,i} = x_{j,i} - \overline{x'_{j}}$$

(13)

Fig 15 presents the temporal fluctuation of the normalized axial velocity at the same locations in the compressor than in Fig 14 (inter-row sections at $h/H$=83%). The temporal distributions are plotted over the whole compressor temporal periodicity, normalized by $2\pi/16\Omega$, for three specific azimuthal positions (-0.25°, 0.55° and 2.15°) drawn in Fig 14 with dashed lines. Because the CFD and the experiment have no common origin for the time, it has been chosen to use the wakes shed by the first rotors to calibrate this time origin. The same time origin is then used for all the comparisons.

The first 3 plots (a) (b) and (c) show a very good agreement between the experiment and the calculation in section 26A. The four wakes downstream the rotor 1 are well described in terms of width and deficit. The influence of the IGV and the potential effects generated by the stator 1 are well captured and are the reason for the flow heterogeneities observed between the three azimuthal positions. These rotor/stator interactions especially impact the influence of the rotor 1 tip leakage flow at this span height.

In section 27A, downstream the rotor 2 - plots (d), (e) and (f) -, the flow is mostly driven by the five wakes and tip leakage flows of the rotor 2 (the tip leakage flows impact the pressure side close to the trailing edge of the rotor blades and produce some extra velocity deficit at the left side of the wakes in the figure). The second major influence in the time domain is the clocking effect, generated by the four wakes of the rotor 1 and the 5 wakes of the rotor 2, which modulate the signal with a time period equal to the compressor one. This means that the wakes of the rotor 1 are quite well transported through the stator 2 and the rotor 2. In this section the differences between the three azimuthal positions are mostly driven by the wakes of stator 1, as observed in Fig 14, but the potential effects induced by the stator 2, and what remains from the IGV wakes, influence the flow field. The complexity of the rotor/stator interactions leads to the small differences between the experiment and the calculation (around 5%).
Fig. 15  Axial velocity fluctuation $u_x''(t, \theta)$ normalized with the mean velocity $u_{x, \text{mean}}$ at given compressor azimuths ($h/H=83.7\%$) – Nominal operating point ($\phi=1.00$).
In section 28A, downstream the rotor 3, the energy associated to the unsteadiness obviously increases. Here the most important differences between the experiment and the calculation are located outside the wakes generated by the rotor 3. The trajectory of the tip leakage flow is modified and tends to be more directed towards the leading edge of the adjacent blade. In the experiment the tip leakage flows are mixed with the wakes, whereas in the calculation this is not always the case, especially in the plot (i), where they are detached from the pressure side of the wakes. This means that the influence of the stator 2 combined with the rotor 2 is weaker in the calculation.

The last plots (j) (k) and (l) are for the section 280, downstream the stator 2. Here the time fluctuations are obviously much lower, because this location is downstream a steady row. The fluctuations are based on 5 periods, induced by the 5 wakes of the rotor 2, modulated by the potential effect induced by the 5 blades of the rotor 3. They reach their maximum intensity in the wake region of the stator 2 (e.g. the azimuth θ=±0.55°) and correspond to an oscillation of the wake induced by the interaction with the rotors 2 and 3.

To summarize, Fig. 14 and Fig. 15 show that the main flow features and the local structures are correctly simulated, which means that the numerical model is able to reproduce the local physic. However, the advection of these structures in the compressor is too dissipative and a part of the flow information is lost before it interacts with the other blade rows. The consequence of this dissipation is an underestimation of the rotor/stator interaction, in the last stages of the compressor.

VII Summary and Conclusion

The work presented in this paper is done following the philosophy that the coupling between experimental and numerical approaches is the most promising way to understand the flow physics that develops in a realistic multistage compressor. An application of this philosophy is achieved here with the 3.5 stage compressor CREATE which remains a rare test case with detailed unsteady measurements and numerical simulations.

The confrontation between numerical simulations and experimental data require a perfect knowledge of the hypotheses and the compromises imposed by:

- the complexity of the machine itself (geometry and technological effects),
- the high cost of numerical simulation of complex unsteady flows (space and time discretizations, numerical schemes, modeling of the turbulence,\ldots)
- the high cost of experimental campaigns that have to obtain accurate unsteady measurements.

The comparisons of the results are finally imposed by the experiment. The numerical simulation becomes a numerical test rig where probes are located at the same positions as in the experimental case, in order to measure the same quantities. As the measurements, the post-processing of the data must be performed exactly with the same methodology in both cases. However, once the confidence in the numerical simulation is established, the numerical results can be used to enhance the database and to conduct more detailed analysis.

Judging from the uncertainty analysis presented here, there is no sense to compare experimental and numerical results with the aim of obtaining small discrepancies if the inlet conditions and the technological effects are not accurately taken into account. As demonstrated in the paper, the numerical simulations are very sensitive to the inlet flow angle and the recirculation at the hub below the stators, with huge consequences (several percent) on the mass flow rate and the pressure ratio (which is a good point since it means the numerical simulation reacts to these uncertainties).

The unsteady feature of the flow is mostly induced by the direct contributions of the rows, but it becomes much more dependent on the rotor/stator interactions when moving towards the last stages of the machine. These rotor/stator interactions are very sensitive to the transport of the secondary flows, and are thus affected by the numerical dissipation related to numerical simulation. For example in a rotor blade passage, the interactions between the wakes spread by the row upstream and the tip leakage flow belong to the unsteady structures of the flow involved in the blockage and the losses in the upper part of the vein. The transport of these secondary flows and their interactions is obviously involved in the onset of instabilities. Indeed the operating range of the compressor predicted by the simulations depends on the quality of this transport.

Finally, despite the difficulty related to uncertainties on boundary conditions and geometry, the comparison of unsteady RANS results with measurements match pretty well, both for time-averaged quantities and unsteady fluctuations. The numerical simulation correctly reproduces the local flow in the three stages of the compressor, including the interactions of the tip leakage flow with the main flow. Such results are very promising since the computational cost of the unsteady RANS method proposed in this paper is moderate and it should be applied for industrial design with current computing means.
Acknowledgments

The authors are grateful to SNECMA for permission to publish results for the CREATE compressor. The experimental part of this work is supported by SNECMA Moteurs and SPAé. Special thanks to G. Halter, P. Krikorian, H. Navière, B. Paoletti and A. Willier for their technical works in the experimental part at LMFA. The authors are also grateful to CNRS, Région Rhône-Alpes and French Research Ministry in providing grants for metrology purchase. The computational results were obtained using the elsA software, developed by ONERA and CERFACS. In this regard, the whole software development team is greatly acknowledged with special thanks to Michel Gazaix (ONERA) and Marc Montagnac (CERFACS). The authors also grateful to GENCI-CINES for providing computing resources under the project fac6074.

References


