NUMERICAL INVESTIGATION OF A TRANSONIC CENTRIFUGAL COMPRESSOR AT HIGH ROTATIONAL SPEED

A.L. Fiquet 1 - J. Dombard 2 - N. Poujol 1 - P. Duquesne 1

1 Univ. Lyon, École Centrale de Lyon, CNRS, Univ. Claude Bernard Lyon 1, INSA Lyon, LMFA, UMR5509, 69130, Écully, France

2 Centre Européen de Recherche et de Formation Avancée en Calcul Scientifique — CERFACS, 42 Avenue Gaspard Coriolis, 31100 Toulouse, France

ABSTRACT
Even though significant flow separation zones may occur in centrifugal compressors, they are still able to provide high efficiency before reaching the stability limit. However, numerical prediction of these separated zones is challenging in complex geometries such as compressors, where turbulence modeling requires special attention. In the European project FLORA, a research transonic centrifugal compressor stage designed and manufactured by Safran Helicopter Engines is experimentally tested and analyzed using numerical simulations. Three different numerical approaches are used to conduct the numerical investigation, including steady and unsteady Reynolds-Averaged Navier-Stokes calculations with the code elsA, while high-fidelity simulations (LES) are performed with AVBP. A brief description of the experimental methods and numerical approaches is given, and the predictivity of the performance at high rotation speed is investigated for the three numerical approaches. Performance prediction reveals discrepancies regarding pressure ratio and isentropic efficiency compared to experiments. At low mass flow rates, unsteady simulations develop an alternate flow separation pattern in the splittered radial diffuser. This phenomenon was also observed experimentally at different operating conditions. A comprehensive numerical analysis reveals that the alternate flow separation pattern originates from the interaction between several separation zones.

KEYWORDS
CENTRIFUGAL COMPRESSOR, AERODYNAMICS, STALL, CFD SIMULATION

NOMENCLATURE
\[ p \] Static pressure
\[ \Delta p \] Pressure fluctuation
\[ \Pi^{t-t} \] Total pressure ratio
\[ \eta_{is} \] Isentropic efficiency
\[ W_m \] Meridional velocity
\[ RANS \] Reynolds-Average Navier-Stokes
\[ LES \] Large-Eddy Simulation
\[ IGV/OGV \] Inlet/Outlet Guide Vane
\[ RD \] Radial diffuser
\[ P_t \] Total pressure
\[ T_t \] Total temperature
\[ \Pi^{t-s} \] Total-to-static pressure ratio
\[ C_p \] Pressure recovery coefficient
\[ \alpha_m \] Meridional flow angle
\[ URANS \] Unsteady RANS
\[ m_{std} \] Standard massflow rate
\[ IMP \] Impeller
\[ A \] Alternate rate

OPEN ACCESS
Downloaded from www.euroturbo.eu
Copyright © by the Authors
INTRODUCTION

Modern lightweight propulsion engines utilize compact high pressure ratio compressors such as centrifugal compressors to improve operability. Within this context, the centrifugal compressor design has to reach high efficiency, high pressure ratio and a large operating range. These requirements lead to highly-loaded centrifugal compressors with the use of high speed impellers with backswept blades and vaned diffusers where complex flow structures occur. Numerical prediction of sensitive flows such as flow separations, shock waves or small-scale disturbances is challenging for state-of-the-art methods in complex geometries and requires special attention for turbulence modeling. Since CFD simulations are commonly used for compressor design, accurate performance prediction is crucial.

Within the European project FLORA, a research transonic centrifugal compressor stage, designed and manufactured by Safran Helicopter Engines, is tested experimentally and numerically. The compressor module is mounted on a 1MW test facility at the Laboratoire de Mécanique des Fluides et d’Acoustique (LMFA), École Centrale de Lyon, France. Over the last decade, intense research has been conducted on this compressor to enhance the comprehension of aerodynamic phenomena and the interdependency between aerodynamics and aeroacoustics. A numerical investigation based on steady and unsteady RANS simulations has been conducted at different rotational speeds (Benichou and Trébinjac, 2016). The authors show that a large shroud-suction side corner separation occurs in the splitted vaned radial diffuser in every second passage near the stability limit at part-speed conditions. This leads to an alternate pattern in which one passage is totally stalled. Experimental observations confirmed these numerical results (Moënne-Loccoz et al., 2020). High discretization of the aerodynamic flow field at the diffuser leading edge is achieved through numerous unsteady pressure transducers distributed over the entire circumference. Experimental observations of the alternate flow pattern are based on an alternate rate derived by projecting all sensors in a single or two passages. Spectral analysis of pressure fluctuations reveals the lock-in phenomenon which may occur between the alternate pattern and the Helmholtz frequency of the test facility (Poujol et al., 2022). When the compressor is operated in mild surge, this pattern phase-locks with the associated planar pressure wave.

Previous work attests highly three-dimensional phenomena such as tip-leakage flow, shock-wave and boundary layer interaction and jet-wake which interact with each other and tend to be very sensitive to disturbances with a large range of frequencies and wavelengths. In this paper, the sensitivity of the performance prediction depending on the numerical approach is investigated near peak efficiency at high rotational speed. To obtain comparable data, experimental methods are applied to numerical results. Reducing the massflow rate, high-fidelity simulations reveal the establishment of an alternate pattern in the radial diffuser as observed experimentally. A comprehensive analysis of numerical results is addressed to highlight the main mechanism behind the alternate pattern.

METHODOLOGY

Experimental setup

The research centrifugal compressor rig is composed of struts, inlet guide vanes (IGV), a backswept splitted unshrouded impeller (IMP), a splitted vaned radial diffuser (RD) and axial outlet guide vanes (OGV), as shown in Figure 1. The compressor stage is mounted on a 1MW test facility at the LMFA. Pressure and temperature are measured with 130 steady sensors to derive performance. Unsteady pressure measurements are also carried out using 52 case-mounted Kulite transducers to investigate aerodynamic phenomena, see (Moënne-Loccoz et al., 2020; Poujol et al., 2022).
2022) for details. Regarding some specifics, the overall stage total-to-static pressure ratio is around 4, and the Mach number at the exit of the impeller is about 1.2 at the rotational speed of interest derived from the peripheral velocity.

**Numerical setup**

The computational domain considers the IGVs, the impeller, the radial diffuser and the OGVs. Since an alternate pattern in two adjacent passages of the radial diffuser has been observed experimentally, the mesh for steady calculations is created with a dual-passage for the stationary domains and a single-passage for the rotating domain. Thanks to the natural circumferential periodicity of the whole compressor stage, only one natural periodic sector is considered for unsteady simulations without any simplification of the geometry, leading to the identical number of simulated passages per component between URANS and LES simulations. Within the FLORA project, the effect of suction flow control in the radial diffuser is investigated. The suction actuators are defined on the hub surface using small slots. These slots are modelled in LES simulations, but not activated. The time-averaged results over ten rotations are used for unsteady simulations.

**Reynolds-Averaged Navier-Stokes Simulation**

Steady and unsteady RANS calculations are performed with the flow solver elsA developed at Office National d’Études et de Recherche Aéronautiques (ONERA) (Cambier et al., 2013). Numeca AutoGrid5 is used to mesh the domain with structured blocks, with 55 million cells for the unsteady domain. Figure 2a) shows the meridional view of the impeller’s main blade surface, with about 30 layers in the radial direction to mesh the tip gap.

The Roe scheme with Harten’s entropic correction is used with second order accuracy in space, as well as the $k − l$ Smith to calculate turbulence. Unsteady simulations are time-accurate with a first-order backward Euler scheme with around 10,000 physical time steps per rotation. This temporal discretization ensures more than 1,000 physical time steps per blade passing period. Comparison with a second-order Gear scheme at one representative operating point reveals that the first-order Euler scheme was more appropriate for the presented case. Since steady and unsteady simulations are wall resolved, $y^+$ lies between 1 and 5 in the whole domain. As the computational domain includes stators and a rotor, each interface between two adjacent rows is modelled with a sliding-mesh for the unsteady simulation, instead of a mixing-plane in steady calculations. Unsteady simulations are initialized with steady calculations. Boundary condition which specify total pressure, total temperature, velocity angles and turbulent variables is applied at the inlet.
At the outlet, static pressure at mid-span, calculated with a quadratic throttle law with a radial equilibrium, is applied. Each calculation is distributed on 168 processors, and certain simulations require at least 40 rotations to reach periodic convergence. The computational cost amounts to approximately 200 000 CPU hours for one operating point.

Large-Eddy Simulation

An unstructured mesh is used for LES simulations, with a total number of tetrahedral elements of 323 million for both stationary and rotating domains. Figure 2b) and c) present visualizations of a perpendicular crinkly cut near the leading edge of the impeller. The element size is relatively homogeneous in the computational domain with a refinement near the walls, providing an averaged $y^+$ lower than 10 in the impeller and approximately 100-250 in the radial diffuser. The code TurboAVBP is used to perform compressible simulations on the natural circumferential periodicity of the compressor stage. TurboAVBP, developed at CERFACS (Schönfeld and Rudgyard, 1999), is a single program–multiple data paradigm that encapsulates two instances of the unstructured massively parallel LES solver AVBP. Both rotor and stator domains are run simultaneously within the code, and primitive variables are interpolated and exchanged using overset grids with the CWIPI library of Onera (Duchaine et al., 2015). The convective operator is discretized by the Lax–Wendroff scheme (2nd-order accurate) and an explicit time advancement (Lax and Wendroff, 2005). The setup is detailed in a previous paper (Dombard et al., 2018). The Sigma (Franck et al., 2011) Sub-Grid Scale model (SGS) is used and standard log-law is applied on all solid boundaries. The physical time step used in the LES simulations results in approximately 20 000 iterations per rotation. Each simulation is distributed over 2 800 processors, and the computational cost for one rotation is approximately 31 000 CPU hours. To achieve convergence, the required number of simulated rotations is between 15 and 40, depending on the operating conditions.

![Figure 2: Mesh visualization of a) impeller surface in URANS simulation, b) perpendicular crinkly cut near the leading edge of the impeller in LES simulation and c) zoom around the tip clearance in the LES mesh.](image)

NUMERICAL RESULTS

Compressor stage performance

Figure 3 presents the total-to-static pressure ratio of the entire compressor stage (IGV - IMP - RD) as function of the normalized massflow rate for high rotational speed, as well as the isentropic efficiency, given by Eq. 1.
\[ \Pi^{t-s} = \frac{p_t}{P_{t,0}} \quad \text{and} \quad \eta_{is} = \frac{T_{t,0}(\Pi^{t-s} \frac{\gamma - 1}{\gamma} - 1)}{T_{t,3} - T_{t,0}} \tag{1} \]

Experimental data based on steady measurements are depicted as black dots. The measurement uncertainties at the design point for the massflow rate, the total-to-static pressure ratio and the isentropic efficiency are, respectively \( \pm 0.11\% \), \( \pm 0.13\% \) and \( \pm 0.21\% \). Steady performance and average performance from converged unsteady simulations are also shown in Fig. 3. Performance of the compressor stage is computed between planes (0) and (3), identified in Figure 1. The total pressure at plane 0 is derived experimentally with an in-house pressure drop correlation based on the total pressure measured in the settling chamber, far upstream the IGVs (Moënée-Loccoz et al., 2020). Since this part is not included in the numerical domain, the total pressure at plane 0 is obtained with the massflow averaged value at plane 0 for numerical results. Apart from this, the same method to compute the performance is applied to both experimental and numerical data. The characteristics show a general trend which is representative of centrifugal compressors. The choke massflow rate predicted by steady simulations presents an overestimation of 2\% compared to experiments. Two operating points denoted as OP-1 and OP-2 will be investigated in this paper, corresponding respectively to a normalized massflow rate of \( \dot{m}_{\text{OP-1}} = 1.04 \) and \( \dot{m}_{\text{OP-2}} = 0.96 \). At OP-1, steady simulation (RANS) and unsteady simulations (URANS and LES) are available, while only LES simulation has been converged at OP-2. The compressor stage operates near peak efficiency for OP-1, in contrast to OP-2 where the compressor is highly loaded. Performance prediction at OP-1 shows slight discrepancies depending on the numerical approach in terms of total-to-static pressure ratio and isentropic efficiency. Steady RANS simulation and time-averaged performance derived from high-fidelity LES result predict an overestimation of more than 5\% for total-to-static pressure ratio, while time-averaged performance from URANS simulation gives an underestimation of less than 2\%. The same trend is also observed for the isentropic efficiency.

![Figure 3: Compressor characteristics at high rotational speed from experiments, steady and unsteady results: (a) total-to-static pressure ratio, (b) isentropic efficiency](image)

The compressor stage total pressure ratio and the total temperature rise downstream of the radial diffuser are plotted against the channel height in Figure 4 for OP-1. The trend of the total pressure ratio profile is well captured regardless of the numerical method. The pressure ratio is uniformly distributed over the channel height, with a reduction towards the casing. A smooth distribution of the total temperature rise in the radial direction is also observed from the hub to
80% channel height. According to the total temperature rise profile, the maximum work is reached between 80% of channel height and the casing for all simulations. Both RANS and URANS simulations predict an increase towards the casing. In comparison, LES result shows a reduction of work, most probably due to the prediction of secondary flows, which differs between numerical methods.

The trend of the radial evolution of both the total pressure ratio and the total temperature rise is well captured regardless of the numerical method. As expected, discrepancies are more prominent near the hub and the casing. No significant difference is observed, which suggests that the averaged aerodynamic field is similar between the three approaches.

**Impeller performance**

The meridional velocity and flow angle profiles at the impeller inlet from steady and time-averaged unsteady results at OP-1 are illustrated in Figures 5(a, b) respectively. Meridional velocity profiles are clearly comparable at the impeller inlet for all numerical approaches, as the flow angle presents a variation less than $\Delta \alpha_m = 1^\circ$. Figure 6 shows the axial plane 1 countered by the Mach number for both time-averaged unsteady results. IGV wakes and channel flow field present identical topology and amplitudes. This indicates that the impeller inflow is similar in all simulations at OP-1.

Since the total pressure at plane 2 (see Figure 1) is not measured in experiments, the Euler’s equations are used to get an estimation (Cumpsty, 1989). Even if the flow is strongly non-uniform at the impeller outlet it is possible to treat the flow in inviscid terms to get an idea of the behavior. This methodology is applied to both numerical and experimental data to obtain the impeller total pressure ratio, presented in Figure 7. This figure shows that RANS simulations are comparable to experiments for massflow rates higher than 0.95. At OP-1, a significant pressure drop of 4.2% is observed for the time-averaged URANS prediction, in contrast to the time-averaged LES result which shows an increase of 6.4%.

The work provided by the impeller differs depending on the numerical method. It is illustrated by the total pressure ratio and the total temperature rise profiles downstream of the impeller at OP-1 plotted in Figure 8(a, b). Total pressure ratio is quite uniformly distributed in the spanwise direction up to 80% channel height. A strong variation is noticed in the tip region and rapid decay
Figure 5: Impeller inlet: (a) meridional velocity and (b) meridional flow angle profiles from steady and time-averaged unsteady results.

Figure 6: Mach number visualization at the impeller inlet from (a) time-averaged URANS and (b) time-averaged LES results for OP-1.

Figure 7: Estimation of the impeller total pressure ratio between planes 0 and 2.

is predicted for all results due to the involved tip leakage flow. It is noticed that LES results are similar to URANS simulations from the hub to 40% channel height, and an increase in total pressure ratio is observed towards the casing. The work illustrated by the total temperature rise presents strong differences near the hub and in the tip region, which is driven by secondary flows,
such as boundary layer and tip leakage flow.

Flow structures at the impeller outlet are strongly non-uniform in centrifugal compressors due to complex geometry with meridional and blade-to-blade curvatures. Figure 9 shows the relative Mach number at the impeller outlet from time-averaged unsteady results. The wakes induced by the main blade (MB) and the splitter blade (SB) are comparable between both results. The predicted topology is similar, with a clear distinction between channel A and channel B. A large tip leakage flow observed in A is induced by the main blade, whereas a smaller one is triggered by the splitter in B. The jet-wake structure is clearly visible in both visualizations. A deeper analysis is provided by the visualization of the normal component of the vorticity at the impeller outlet in Figure 10 from time-averaged URANS (a) and time-averaged LES (b) results. Both visualizations are significantly different in terms of topology and amplitude of vortex structures. URANS simulation shows large vortex structures near walls with high amplitudes, related to the blade-to-blade curvature as detailed in (Bulot et al., 2009). In contrast, LES simulations develops structures due to impeller wakes with very low amplitudes. No traces of vortex are observed near walls. These observations are coherent with the prediction of total pressure ratio illustrated in Figure 7.

Figure 8: Impeller characteristics: (a) total pressure ratio and (b) total temperature rise profiles downstream of the impeller at OP-1 from steady and time-averaged unsteady results.

Figure 9: Relative Mach number visualization at the impeller outlet from (a) time-averaged URANS and (b) time-average LES results for OP-1.

Figure 10: Visualization of the normal component of the vorticity at the impeller outlet from time-averaged URANS (a) and time-averaged LES (b) results.
Radial diffuser performance

The pressure recovery coefficient for the radial diffuser given by Equation 2 is plotted in Figure 11.

\[
C_{p, RD} = \frac{p_3 - p_2}{p_{t,3} - p_3}
\]  

(2)

For OP-1, steady simulation shows a higher value compared to experiments, whereas LES prediction is comparable. The overestimation predicted by steady simulations can be attributed to the mixing-plane treatment between rotating and stationary domains, leading to an incorrect inflow at the diffuser inlet. The URANS result lies between both. It is important to notice that the radial diffuser inflow is slightly different between all approaches, since the impeller provides different work input depending on the numerical method (see Figure 8). This point is highly relevant since the overall total-to-static pressure ratio presented in Figure 3 may lead to an incorrect analysis and conclusions. Based on the overall performance, it is observed that averaged URANS performance is more coherent to experiments than averaged LES performance. According to Figure 7, both unsteady methods do not predict impeller performance with accuracy, affecting the inflow of the diffuser (see Figure 11). However, both time-averaged unsteady results show that the flow field in the radial diffuser is similar in every channel for OP-1, as illustrated in Figure 12. The blade-to-blade view at 50% channel height countered by the Mach number reveals a supersonic zone at the
diffuser inlet from averaged URANS (a), which is slightly reduced in the averaged LES result (b). No significant differences between the channels A and B are observed.

Performance prediction of the present centrifugal compressor is highly sensitive to the numerical method and requires further experimental investigations to clearly identify zones which are inaccurately predicted. The followings bullets summarize the section:

- As expected, steady RANS simulations overpredict static-to-total pressure ratio and isentropic efficiency, mainly due to the use of mixing planes.

- Even if the overall URANS performance is close to the experiments, the performance of each component differs from experiments when considers separately, which reveals the difficulty to predict the correct flow features.

- The main difference is observed in the impeller, while the pressure recovery coefficient of the radial diffuser is comparable between URANS, LES and experiments – meaning that the flow prediction in the diffuser may not be representative of the reality.

![Figure 12: Contours at 50% channel height of Mach number in the radial diffuser from time-averaged (a) URANS and (b) LES results with black isocontour at sonic speed.](image)

**Throttling towards OP-2**

LES simulation at lower massflow rate has been converged, whereas URANS simulation could not be throttled further due to the development of acoustic reflexions in the domain. For OP-2 (see Figure 3), the overall total-to-static pressure ratio is comparable to experiments. The total pressure ratio of the impeller is also presented in Figure 7 leading to similar observations as OP-1. Even if the total pressure losses in the impeller are underestimated, the radial diffuser achieves a comparable pressure recovery coefficient compared to experiments, as shown in Figure 11.

In papers (Moënne-Loccoz et al., 2020; Poujol et al., 2022), experimental analysis at part-speed conditions based on pressure transducers reveals the establishment of an alternate flow pattern in the radial diffuser at low massflow rates. To identify this phenomenon, the pressure sensors distributed over the circumference at the diffuser leading edge are used to derive the experimental alternate rate $A$ detailed given by equation 3 with $p_{1p}(p_{2p})$ the pressure projected in 1 channel (2 channels), $\theta$ the circumferential coordinate and $\theta_{RD}$ the pitch of the radial diffuser.

$$A = \frac{1}{2\theta_{RD}} \int_0^{2\theta_{RD}} \frac{|p_{1p}(\theta) - p_{2p}(\theta)|}{\sqrt{p_{1p}/p_{2p}}} d\theta$$  

(3)
For numerical results, the circumferential distribution of pressure fluctuations at the same radial and axial positions as pressure measurements is considered. The alternate rate is derived using the continuous circumferential distribution over two adjacent channels. The alternate rate based on the time-averaged LES result at OP-2 indicates the occurrence of an alternate pattern with \( \mathcal{A} = 0.049 \), while only traces of an alternate pattern have been observed in experiments. A similar alternate pattern has been experimentally observed at lower massflow rates. Figure 13 shows the flow field in the radial diffuser with (a) a blade-to-blade view at 50% channel height countered by the Mach number with black iso-contour at sonic velocity to identify the shock structure and (b) the considered circumferential distribution over two channels near the LE of the pressure fluctuation. Each channel of the radial diffuser presents a supersonic zone at its inlet. The shock topology in channel A differs from B which has a reduced supersonic zone (see Figure 13(a)). Figure 13(b) illustrates the circumferential static pressure distribution at the diffuser inlet over two channels (blue dots) compared to the duplication of the pressure distribution over channel A (red dots). A significant difference is observed between both channels, with lower pressure fluctuations in channel B.

Figure 13: Flow in radial diffuser from time-averaged LES simulation (a) Blade-to-blade view of the Mach number at 50% channel height with black iso-contour at sonic speed and (b) Circumferential distribution of pressure fluctuation near the blade leading edge over two passages.

Low Mach number zones are observed in the radial diffuser, as shown in Figure 13. To detect a separation zone, analysis of critical points is necessary (Duquesne et al., 2022). Figure 14 presents the average topological structure of the velocity field near the hub in two adjacent channels in the radial diffuser. A blade-to-blade view at first cells near the hub contoured by the velocity magnitude is illustrated, with the visualizations of the related critical points emphasized by friction lines. Focus and saddle points are denoted (F) and (S). As the suction flow control is not activated, flow recirculation near the suction side around 30% of chord of the main blade occurs through each slot, as observed in this figure.

In Figure 14a), a large flow separation zone is observed on the pressure side of MB1 near the trailing edge. The flow separation structure is mainly composed by a large focus at the blade surface, which attracts the flow from the hub and shroud. The pressure side of MB2 shows smooth streamlines without traces of a flow separation zone, as shown in Figure 14b). At the MB1 suction side in Figure 14c), a large corner separation zone is noticed, including numerous focus and saddle points. This flow separation zone induces a flow over deviation in channel B. In contrast, the corner separation at the MB2 suction side, as shown in Figure 14c), is smaller and restricted between the two saddle points. The flow deviation in channel A is less prominent. A high flow incidence is
predicted at the leading edge of SB1, producing a separation zone at the suction side with two saddle points and one focus point, as observed in Figure 14d). The streamlines with black arrows in channel B indicate a flow deviation towards the pressure side of the blade MB2, which prevents large flow separation on this side. In the channel A, the flow near the hub surface is aligned with SB2, directing the flow in the meridional direction. The adverse pressure gradient grows up, leading to a separation at the blade MB1 surface. The occurring alternate pattern arises from the interaction between several separation zones.

Figure 14: Topological structure of two adjacent channels in the radial diffuser at OP-2 at first cells from time-average LES results.

CONCLUSIONS

Steady (RANS) and unsteady (URANS and LES) simulations were carried out in a transonic centrifugal compressor at high rotational speed. Near peak efficiency, a comprehensive analysis was addressed to highlight the sensitivity of performance prediction depending on the numerical approach. This study shows that major discrepancies were located in the tip region of the impeller, where secondary flows are prominent. Since secondary structures depend directly on the turbulence modelling, it requires special attention and has a significant impact on the overall performance. Wall-resolved URANS calculation and wall-modelled LES simulation near peak efficiency present different behaviour of the impeller, which directly impacts the radial diffuser inflow. The necessity to investigate carefully each component has been illustrated through this investigation to avoid any misleading conclusions.

At higher loaded conditions, time-averaged LES results reveal the emergence of an alternate pattern in the radial diffuser, whereas only traces of an alternate pattern between two adjacent channels are experimentally observed. Such behaviour develops at lower massflow rate in experiments. Numerical results show that a large flow separation occurs in every second passage on the main blade pressure side in the radial diffuser. The mechanism of the occurring alternate pattern is detailed based on the topological analysis of critical points. Further investigations at high rotational
speed will be addressed in the future to overcome the lack of experimental data. The flow will be characterized in detail through LDA measurements, which are planned in future experiments.

ACKNOWLEDGEMENTS

The authors are truly grateful for the technical advice and contributions of Pierre Laucher, Sébastien Goguet, Gilbert Halter and Benoit Paoletti. The authors thank Safran Helicopter Engines for permission to publish these results, and we are grateful for the comments of Jacques Demolis, Nicolas Buffaz, Fabien Artus and Jérôme Porodo during the preparation of this paper. This project has received funding from the Clean Sky 2 Joint Undertaking (JU) under grand agreement No.820099. The JU receives support from the European Union’s Horizon 2020 research and innovation program and the Clean Sky 2 JU members other than the Union. This publication reflects only the author’s view, and the JU is not responsible for any use that may be made of the information it contains.

References


