Assessment of scale adaptive simulation of a rotor of high pressure compressor

Julien Marty, Hélène Gaible, Hervé Bézard

► To cite this version:

Julien Marty, Hélène Gaible, Hervé Bézard. Assessment of scale adaptive simulation of a rotor of high pressure compressor. ASME Turbo Expo, Jun 2018, OSLO, Norway. 10.1115/GT2018-76537 . hal-02339505

HAL Id: hal-02339505

<https://hal.science/hal-02339505>

Submitted on 30 Oct 2019

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

ASSESSMENT OF SCALE ADAPTIVE SIMULATION OF A ROTOR OF HIGH PRESSURE COMPRESSOR

J. Marty

ONERA - The French Aerospace Lab
Aerodynamic, Aeroelasticity
and Acoustics Department
8 rue des Vertugadins
92190 Meudon, France
Email: Julien.Marty@onera.fr

H. Gaible

ONERA - The French Aerospace Lab
Multi-Physics
and Energetics Department
2, avenue Édouard Belin
31055 Toulouse, France
Email: helene.gaible@onera.fr

H. Bézard

ONERA - The French Aerospace Lab
Multi-Physics
and Energetics Department
2, avenue Édouard Belin
31055 Toulouse, France
Email: herve.bezard@onera.fr

ABSTRACT

The main design objectives of a high pressure compressor are the aerodynamic efficiency and the operating range (e.g. the surge margin). Those quantities are impacted by secondary and leakage flows occurring in the blade passage such as corner separation or stall and tip leakage flows. The turbulence modeling influences strongly the prediction of the overall performances. The aims of the present study are (i) the validation of the combination of the SAS approach with the DRSM turbulence model by comparison to experimental data, especially to laser measurements in the tip of a rotor of a high pressure compressor and (ii) the discussion of the flow prediction improvements with respect to turbulence approaches classically used in CFD and industry: URANS simulations and standard SAS simulation i.e. combined with SST turbulence model. The SAS results are compared to experimental data and to URANS results (SST and DRSM). Only the simulations with IGV wakes predict the velocity fluctuations near tip gap, from the leading edge. Concerning the time-averaged performances, the stagnation pressure losses are slightly overestimated by SAS, especially with DRSM model. This is due to an amplification of the hub corner separation. Moreover, the isentropic efficiency is very sensitive to the SAS approach and to the turbulence model. The spectral analysis shows that the prediction of the amplitude and frequencies of the power spectral density of static pressure is improved using the SAS approach instead of URANS one. The SAS approach leads to PSD similar to ZDES, especially with the DRSM model.

Thus, the SAS-DRSM is able to well predict the tip leakage flow with the fine mesh. Nevertheless, this approach amplifies the hub corner separation leading to a strong underestimation of overall performances.

NOMENCLATURE

<i>BPF</i>	Blade Passing Frequency of the IGV for rotor 1
<i>DRSM</i>	Differential Reynolds Stress Model
<i>IGV</i>	Inlet Guide Vane
<i>LDA</i>	Laser-Doppler-Anemometry
<i>PSD</i>	Power Spectral Density
<i>SAS</i>	Scale Adaptive Simulation
<i>SST</i>	Shear Stress Transport
<i>(U)RANS</i>	(Unsteady) Reynolds-averaged Navier Stokes
$\Delta x^+, \Delta y^+, \Delta z^+$	Normalized wall cell dimensions
<i>ZDES</i>	Zonal Detached Eddy Simulation

INTRODUCTION

The high pressure compressor design is mainly driven by the search of the highest efficiency and the largest stable operating range for a given stagnation pressure ratio. These design objectives are directly and strongly impacted by the secondary flows such as the corner separation and the tip leakage vortex. As shown by Herzig *et al.* [1], the interaction of the tip flow with the

main flow, the boundary layer developing at the shroud and the incoming wakes are responsible for penalizing losses. According to Bindon [2], the two main sources of losses relative to tip clearance flow are the mixing and the gap shear, not the interaction with endwall. He also conceptualized the flow within the gap by hypothesizing that there are different kinds of channels with and without separation bubbles within the gap. The axial position of these different channels depends on the pressure distribution along the chord, and thus the loading. With a stationary end-wall Kang and Hirsch [3] highlights a tip flow pattern involving a tip leakage vortex, a secondary vortex and a tip separation vortex. Mailach *et al.* [4] studied the interaction between incoming wake and tip leakage flow. They highlighted the periodic variation of orientation and extension of tip clearance vortices. Suder and Celestina [5] investigated the tip clearance flow at different rotational speed highlighting the interaction with shock wave at nominal speed and the appearance of a second vortex only at partial speed.

The designers use more and more computational fluid dynamics (CFD) to design compressor as CFD offers a flexibility to evaluate numerous design parameters with low cost. Nonetheless, CFD results depend significantly on the Navier-Stokes closure i.e. on the used turbulence model and the modeling turbulence approach (RANS, DES, LES, DNS). Classically, the RANS turbulence model is based on the Boussinesq hypothesis by using a model with one or two transport equations such as the Spalart-Allmaras [6] or the Menter SST [7] models. Borello *et al.* [8] show that the turbulence is clearly anisotropic in the vicinity of the tip gap and second moment closure (Differential Reynolds Stress Model) is able to capture this characteristic. Several authors studied the tip leakage flow with high fidelity methods. Using IDDES, Shi *et al.* [9] investigate the tip flow interacting with shock wave resulting in a curved shock surface and shock oscillations. Zonal Detached Eddy Simulation was also performed by Riéra *et al.* [10, 11] to study the tip leakage vortex and its interaction with incoming wakes and vortices. You *et al.* [12] and Boudet *et al.* [13] performed Large Eddy Simulation (LES) of a clearance flow. Nonetheless, the cost of LES is still expensive.

The Scale Adaptive Simulation (SAS) approach is a way to enhance traditional RANS model with low implementation effort. It can improve dramatically URANS computations especially when massive flow separation occurs, even for non refined grids (obstacle in cross flow, airfoil beyond stall). A well-known example is the jet in crossflow [14]. The classical turbulence models (even Reynolds stress models) fail reproducing the diffusion of the jet downstream, whereas the same models coupled with the SAS approach reproduce perfectly the experiments, using classical RANS grids. The SAS approach was also used in previous studies in turbomachinery framework. Wang and Yuan [15] investigated the unsteady behavior of corner separation. Lucius and Brenner [16] performed SAS computations on a pump

in partial load conditions. Both URANS and SAS approaches are able to capture the rotating stall. Ravelli and Barigozzi [17] showed that the SAS approach improves the prediction of vortex shedding at the trailing edge and the film cooling effectiveness in the framework of high pressure turbine nozzle guide vane with trailing edge cutback film cooling.

The literature review shows that the combination of SAS approach with DRSM turbulence model is promising and has never been assessed in the turbomachinery framework, especially for the tip leakage flow prediction. The present study aims at validating this combination by comparison to experimental data, especially to laser measurements in the tip. The second objective of this paper is to highlight and discuss the improvement obtained with this combination SAS-DRSM by comparison to turbulence approaches classically used in CFD and industry: URANS simulations and standard SAS simulation i.e. combined with SST turbulence model. After a short description of the experimental facility CREATE (high pressure compressor), the numerical test bench is detailed and the turbulence models and approaches are presented. Then the numerical results are validated at several levels (from integral performances to spectral analysis according to Sagaut and Deck [18]) by comparison to experimental data.

CREATE COMPRESSOR

Experimental facility

The studied configuration is the experimental compressor CREATE [19] which is located at the Laboratory of Fluid Mechanics and Acoustics (LMFA) in Lyon, France. This axial compressor from SAFRAN AIRCRAFT ENGINES comprises three and a half stages presented in figure 1 and is representative of the median stages of modern high pressure compressors. The number of stages was chosen to have a magnitude of the secondary flow effects similar to a real compressor, and to be within the rig torque power limitation. SAFRAN AIRCRAFT ENGINES and the research team of LMFA have taken into account technological constraints coming from the experimental part of the project, very early in the compressor design. 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 2π in the general case with primary blade numbers to avoid generating resonant waves) has been reduced to $\frac{2\pi}{16}$ on the compressor CREATE, choosing the number of blades of each rotor and stator [inlet guide vane (IGV) included] as a multiple of 16. The blade number for each row is detailed in table 1. Thus the analysis of rotor-stator interaction does not require measurements over a full annulus sector as a sector of only 22.5° ($\frac{2\pi}{16}$) should contain all the spatial information useful to rotor-stator interaction analysis, at least for stabilized operating point.

FIGURE 1. CREATE COMPRESSOR MERIDIAN VIEW

TABLE 1. BLADE NUMBER FOR THE ROWS

Row	IGV	R1	S1	R2	S2	R3	S3
Blade number	32	64	96	80	112	80	128

TABLE 2. CHARACTERISTICS AT DESIGN OPERATING POINT

Outer casing diameter	0.52 m
Rotating speed	11543 rpm
Mass flow	12.7 kg.s ⁻¹
Inlet Mach number at tip of R1	0.92

Some characteristics are summarized in table 2. Measurements are carried out with both pneumatic and unsteady pressure probes [19], which are located between the axial sections 25A and 26A, visible in figure 1. The measurement of the overall compressor performance was performed at several speeds, including the stall line position. At several operating points (near maximum efficiency, near choke and near surge), conventional pressure, temperature, and angle probe traverses were performed downstream of each blade and vane. The four-holes probes had dual pressure and temperature sensors, together with two extra static pressure sensors (mounted on each side of the probe) providing the flow angle when balanced. Thus the Mach number and also the three velocity components can be computed. The quantities measured with pneumatic probes are then averaged circumferentially to obtain the radial profile and averaged circumferentially and radially to obtain the performances (pressure ratio, efficiency). The same method is used in CFD.

Moreover, two-dimensional velocity measurements are available through Laser-Doppler-Anemometry. Two pairs of beams ($\lambda = 488$ nm and $\lambda = 514.5$ nm) are used for simultaneously measuring two velocity components which lead to the de-

termination of the axial and tangential velocity components. The focal length of the front lens is 250 mm. A diameter of 76 mm and a length of around 0.9 mm characterize the measurement volume.

The pneumatic probe and LDA measurements are performed downstream of each row, at mid-distance between rotor and stator. For the pneumatic probe, a system with rotating ring allows the scanning at different azimuthal position over a 22.5° sector and the measurements can be performed from 5% to 95% relative height. For LDA, windows are not only placed downstream each row with a size allowing measurements over the same azimuthal sector but also in front of the rotor tip. The measurements were triggered with the rotation frequency of the machine, in such a way that the flowfield is described either inside a single-blade passage, or within several blade passages covering the circumferential periodicity of the whole machine. More details about LDA measurements are given in [20, 21, 19].

CFD setup

Computational domain In the present study, only the first rotor of the compressor (R1) is investigated at the nominal rotational speed and the nominal operating point. The inlet boundary of the test bench is set to section 25A of the experimental test rig, located between the IGV and the R1. The reason is that measurements are available on the experimental compressor for different operating points at this section. Besides, the possible comparisons with experimental data at section 25A make the numerical setup more reliable for the evaluation of the method. The location of the outlet boundary is based on the hypothesis that the outflow is axisymmetric, therefore, it must be far from the trailing edge of the R1. Furthermore experimental measurements are available at section 26A, so this section has to be inside the computational domain for validation purposes. Nonetheless, the mesh size is a key element in the domain definition. This is why the outlet boundary of the computational domain is defined at two axial chords downstream the R1.

Mesh and boundary conditions In this study, two meshes are used. All grids are obtained with a multi block approach using an O4H topology. The rotor tip clearance is meshed with an additional O-H block. The normalized wall cell dimension normal to the wall fulfils Δy^+ of the order of 1 in every zone. The first grid is composed of one blade passage (1/64 of the compressor) and can be considered as a fine mesh for URANS simulation. This mesh is called "M1" and consists in 4.3 million points. The IGV wakes are not taken into account in the simulation: the inlet boundary conditions are based on the radial profiles of stagnation pressure, stagnation enthalpy, velocity direction and turbulent primitive variables obtained by azimuthal average of the two-dimensional cartography used with the second grid.

The second grid (called "M2") is the same used for Zonal

FIGURE 2. Computational domain

Detached Eddy Simulation by Riéra *et al.* [10, 11]. The normalized wall cell dimension Δx^+ and Δz^+ are detailed in [11]. Two passages are simulated in order to fit the spatial periodicity of the IGV. The whole grid comprises 88 million points in order to model 1/32 of the compressor. The computational domain of mesh M2 is shown in figure 2. For the first mesh, the boundaries are located at the same location except the periodic boundaries by rotation as only one blade passage is simulated. The mesh at hub and blades walls is depicted in figure 3 with one point over two in all mesh directions. The mesh density is clearly high near leading and trailing edge but also in the blade passage in order to capture the tip leakage vortex. In the real configuration, there are two leakage flows at the hub. The first one is upstream of the rotor and the second one is linked to the labyrinth seal under the first stator. These two leakage flows and blade fillets are not included in the computational domain as the massflow are very small and the impact on tip leakage vortex is several order of magnitude smaller than the IGV wakes and vortices.

In this configuration, the IGV effects are reproduced with specific inlet boundary conditions based on 2D cartography resulting from a previous IGV-R1 computation [11]. It consists in a rotating distortion cartography, whose method was used for centrifugal compressor by [22], based on a Fourier decomposition with 60 harmonics of the two-dimensional map of the flow. The IGV wakes and vortices are observed at the 25A section as shown in figure 4 which depicts the axial velocity and the entropy variation fields. The convection of this secondary flows originating from the IGV across the rotor passage is shown in figure 5. Only

FIGURE 5. Instantaneous entropy variation field at several axial sections, from 25A (foreground) to 26A sections (background) for the simulation M2-SAS-DRSM-wlin

the entropy variation is plotted as the absolute value of entropy is always arbitrary. The entropy variation Δs [J/kg/K] between the current location and a reference state $(\cdot)_{\text{ref}}$ is defined by:

$$\Delta s = Cp \ln \left(\frac{T_s}{T_{s\text{ref}}} \right) - r \ln \left(\frac{P_s}{P_{s\text{ref}}} \right) \quad (1)$$

where the reference state is taken at the computational domain inlet. The physical values for stagnation pressure, stagnation enthalpy, primitive turbulence quantities of the turbulence model, and direction of the flow are given at each cell of the map. Therefore, with this rotating boundary condition, the unsteady distortion effects induced by the IGV are modeled in the simulation of R1 flow.

For the outlet boundary condition, the static pressure is specified at the hub and then a simplified radial equilibrium law. All simulations are based on the same static pressure value as in [11]. Finally, a classic rotation periodicity condition is set at the azimuthal boundaries and a no-slip adiabatic wall condition is applied at all wall surfaces.

Numerical methods The numerical simulations have been performed using the *elsa* software [23], developed at ONERA and co-owned by AIRBUS, SAFRAN and ONERA. This software is based on a cell centered finite volume technique and

FIGURE 3. Computational domain and mesh over hub and blade walls (one point over two in all mesh directions)

structured multiblock meshes. The viscous fluxes are computed with a second-order centered scheme. For efficiency, an implicit time integration is employed to deal with the very small grid size encountered near the wall.

As one of the aims of this study is to compare SAS approach coupled with turbulence models to ZDES-SA method, the same numerical parameters as Riéra *et al.* [11] are used. The computation is performed in the rotor frame of reference. The casing wall is then moving regarding this relative frame. The spatial discretization scheme for the inviscid fluxes [24] is based on the third order accurate Advection Upstream Splitting Method for low Mach numbers (AUSM+P), initially developed by Edwards and Liou [25]. The numerical dissipation is proportional to the local fluid velocity. A classic second order centred formulation is used for the viscous fluxes. The time discretization scheme is the second order accurate Gear scheme. At each time step, an approximate Newton method based on the LU factorization solves the non-linear problem. The time step is set to $1.6 \cdot 10^{-7}$ s which leads to a Courant-Friedrich-Levy number lower than 1 except for the boundary layers. It corresponds to 1000 time steps per IGV passing period. The use of 8 sub-iterations per time-step is required to reach a decay superior to 1.5 orders of magnitude for the residuals. This criterion is a compromise between precision and cost of the computation and takes into account the small time step involved [26]. Moreover, as the physical time-step is very small ($CFL < 1$ outside the boundary layer, $CFL \approx 15$ near the wall). Daude *et al.* [26] use a ratio of CFL/N equal to

2 where N is the sub-iteration number, to perform LES in order to reduce CPU cost in comparison to an explicit time integration scheme while the result quality is identical. In the present case, the maximum CFL is equal to 15. Thus 8 sub-iterations ensure the fully-converged solutions.

TURBULENCE MODELING

SST Menter $k - \omega$ model

The transport equations for the turbulent kinetic energy k and for the turbulent specific dissipation rate ω are the following:

$$\frac{\partial \rho k}{\partial t} + u_j \frac{\partial \rho k}{\partial x_j} = P_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] \quad (2)$$

$$\begin{aligned} \frac{\partial \rho \omega}{\partial t} + u_j \frac{\partial \rho \omega}{\partial x_j} = & \gamma \mu_t P_k - \beta \rho \omega^2 \\ & + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] \\ & + 2(1 - F_1) \sigma_{\omega 2} \frac{\rho}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (3)$$

FIGURE 4. IGV wakes and vortices at section 25A (simulation M2-SAS-DRSM-wlin)

with the eddy viscosity $\mu_t = \frac{\rho k}{\omega}$. The F1 function allows to switch from the Wilcox $k - \omega$ model in the inner part of the boundary layer (F1=1) to a pseudo $k - \varepsilon$ model in the outer part (F1=0). It reads:

$$\begin{aligned}
 F_1 &= \tanh \zeta^4 \\
 \zeta &= \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega d}; \frac{500\nu}{\omega d^2} \right); \frac{4\sigma_{\omega 2}k}{CD_{k\omega}d^2} \right] \\
 CD_{k\omega} &= \max \left(2\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}; 10^{-20} \right)
 \end{aligned} \quad (4)$$

The constants of the model are a combination of the inner [$k - \omega$ model (index 1)] and outer [$k - \varepsilon$ model (index 2)] models:

$$C = F_1 C_1 + (1 - F_1) C_2 \quad (5)$$

$$\sigma_{k1} = 0.5 \quad \sigma_{\omega 1} = 0.5 \quad \beta_1 = 0.075 \quad (6)$$

$$\sigma_{k2} = 0.5 \quad \sigma_{\omega 2} = 0.856 \quad \beta_2 = 0.0828 \quad (7)$$

The γ constant is defined by equation (8) which gives $\gamma_1 = 0.555$ and $\gamma_2 = 0.664$.

$$\gamma = \frac{\beta}{\beta^*} - \frac{\sigma_{\omega} \kappa^2}{\sqrt{\beta^*}} \quad \kappa = 0.41 \quad \beta^* = 0.09 \quad (8)$$

Wall condition on ω

The ω -scale behaves theoretically as $\frac{1}{d^2}$ when approaching a wall, which is hard to deal with from the numerical point of view. Different wall conditions on ω can be specified at a wall cell interface. Menter suggests to prescribe the ω wall value as $\frac{60\nu}{d_1^2}$ where d_1 is the wall distance to the first cell point. It is a kind of quadratic extrapolation and can lead to great ω wall values. A more robust way implemented in *elsA* code is to extrapolate linearly the first cell ω value. Computations performed on different cases showed no influence of these conditions on the solution (friction coefficient, velocity profiles). Wilcox [27] introduces the notion of slightly-rough surface and prescribes the ω wall value as $\frac{40,000\nu}{ks^2}$ where the sand-grain roughness height ks should be taken to ensure a reduced wall value ks^+ below than 5. However this condition produces a small increase of the flat plate friction coefficient of about 5%. Both conditions (linear extrapolation and pseudo-roughness) were tested in the present simulations.

DRSM model

The Differential Reynolds Stress Models transport seven equations: six equations for the turbulent stresses and one equation for a turbulent scale (ε , ω , ...). On the contrary to standard two-equation models, the DRSM features many physical behaviors of the turbulence: anisotropy, rotation, curvature, history ef-

fects, strong shear. The model formally reads:

$$\frac{DR_{ij}}{Dt} = P_{ij} - \varepsilon_{ij} + \Phi_{ij} + D_{ij} + C_{ij} \quad (9)$$

where P_{ij} is the production (no modelization), ε_{ij} the dissipation, Φ_{ij} the pressure strain, D_{ij} the diffusion and C_{ij} the rotation term in case of a rotating relative frame (no modelization). Many different expressions exist for ε_{ij} , Φ_{ij} and D_{ij} and are not detailed here. In this paper the LRR/SSG- ω DRSM of Einfeld *et al.* is used (see [28] for more details). It is based on a blending of the LRR and SSG redistribution terms, a classical isotropic dissipation term, a simple or generalized gradient diffusion term and the BSL Menter ω -scale transport equation (without SST correction on the eddy viscosity).

SAS model

Standard model The Scale Adaptive Simulation (SAS) approach of Menter [29,30,31] can be considered of an improved URANS model which mimics the behavior of hybrid RANS/LES methods: when the flow is attached, the standard RANS solution is obtained; when the flow is separated, an extra term in the scale equation becomes active and reduces the eddy viscosity, which allows capturing large turbulent eddies and resolve part of the turbulence spectrum.

The standard SAS approach of Menter associated to the SST $k - \omega$ model is here presented. The k transport equation is the same as before. The ω transport equation is slightly modified with the addition of a Q_{SAS} term on the right side (source term). This term is responsible for the decrease of the eddy viscosity in case of occurrence of flow instabilities. It has the following expression:

$$\begin{aligned} \frac{\partial \rho \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} &= \gamma \mu_t P_k + Q_{SAS} - \beta \rho \omega^2 \\ &+ \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] \\ &+ 2(1 - F_1) \sigma_{\omega 2} \frac{\rho}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (10)$$

The Q_{SAS} term, which is responsible for the decrease of eddy viscosity in case of flow instabilities, has the following expression:

$$\begin{aligned} Q_{SAS} &= \max \left[\rho \zeta_2 \kappa S_2 \left(\frac{L}{L_{vK}} \right)^2 \right. \\ &\quad \left. - C \frac{2\rho k}{\sigma_\Phi} \max \left(\frac{1}{k^2} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j}, \frac{1}{\omega^2} \frac{\partial \omega}{\partial x_j} \frac{\partial \omega}{\partial x_j} \right), 0 \right] \end{aligned} \quad (11)$$

with the following constants: $\zeta_2 = 3.51$, $\sigma_\Phi = \frac{2}{3}$ and $C = 2$. In the Q_{SAS} expression, L is the length scale deduced from the turbulent scales transported by the model $L = \frac{k^{\frac{3}{2}}}{\varepsilon} = \frac{\sqrt{k}}{C_\mu^{\frac{1}{4}} \omega}$.

L_{vK} is a 3D generalization of the von Kármán length scale expression coming from the boundary layer and defined by $L_{vK} = \frac{\kappa S}{\|\Delta U\|}$ where S is the second invariant of the strain rate tensor: $S = \sqrt{2S_{ij}S_{ij}}$ and the denominator is the norm of the Laplacian of the velocity vector: $\|\Delta U\| = \sqrt{\frac{\partial^2 U_i}{\partial x_j^2} \frac{\partial^2 U_j}{\partial x_i^2}}$.

Small scales limiter To limit the classical phenomenon of energy accumulation at small turbulent scales, Menter *et al.* introduce lower bounds on the von Kármán length scale, which are function of the cell size Δ :

$$L_{vK \text{ new}} = \max(L_{vK}, L_{vK}^{\min}) \quad (12)$$

There can be of two types:

➤ Smagorinsky limiter with $C_S = 0.11$ and $\Delta = \sqrt[3]{\Omega_{CV}}$ where Ω_{CV} is the cell volume.

$$L_{vK}^{\min} = C_S \sqrt{\frac{\zeta_2 \kappa}{\left(\frac{\beta}{C_\mu}\right) - \gamma}} \Delta \quad (13)$$

➤ WALE-LES limiter of Nicoud and Ducros [32], with

$$\begin{aligned} C_W &= 0.4, S_W = \frac{1}{\sqrt{2}} \frac{(4S_{ij}^d S_{ij}^d)^{\frac{3}{2}}}{S^5 + (4S_{ij}^d S_{ij}^d)^{\frac{5}{4}}} \text{ and } S_{ij}^d = S_{ik} S_{kj} + \Omega_{ik} \Omega_{kj} - \\ &\frac{1}{3} \delta_{ij} (S_{mn} S_{mn} - \Omega_{mn} \Omega_{mn}). \end{aligned}$$

$$L_{vK}^{\min} = C_W \sqrt{\frac{\zeta_2 \kappa}{\left(\frac{\beta}{C_\mu}\right) - \gamma} - \frac{S_W}{S}} \Delta \quad (14)$$

αL correction It can be shown that in the original SAS expression of Menter, the von Kármán length scale goes to infinity in the center of a 2D mixing layer. This implies that the Q_{SAS} term is zero in a region close to the mixing layer center. This unwanted feature prevents or causes delay to the development of mixing layer instabilities (Kelvin-Helmholtz). A correction was introduced by Benyoucef *et al.* [33] to limit the upper bound of the von Kármán length scale:

$$\widetilde{L}_{vK} = \min(L_{vK}, F_1 L_{vK} + [1 - F_1] \alpha L) \quad (15)$$

where L is the large turbulent length scale, α is equal to 2.5, F_1 is the Menter's $k - \omega$ model blending function. The blending

is present here to prevent this new expression to be active in the inner boundary layer and ensure a RANS behavior in the attached boundary layer.

Extension to DRSM models

As mentioned before, the SAS approach can be generalized to any RANS turbulence model. It was thus extended to the DRSM- ω models already implemented in *elsA* code. The implementation is thus straightforward as only the Q_{SAS} term is added to the ω -scale transport equation. Using DRSM-SAS approach ensures that both the modeled and the resolved part of the turbulence are well captured, on the contrary to SAS-SST approach where the modeled part of turbulence is isotropic. Moreover the DRSM are more sensible to flow instabilities than standard eddy viscosity models, which ensures a better development of the scale resolving mode when coupled to the SAS approach. The DRSM-SAS was successfully used on a heated square cylinder in a crossflow as shown by Didorally *et al.* [34]. It exhibits very close behavior to ZDES simulations in terms of turbulence spectrum, or mean and rms velocity and temperature profiles.

Turbulence model summary

The present study aims at evaluating the SAS approach coupled with two different RANS turbulence models (SST and DRSM). For the SST model, two meshes are used and for the finest grid, the IGV wakes can be taken into account through the inlet boundary condition (the rotating cartography). Thus, the number of simulations is significant. The name of simulation is normalized according to the different options used. The table 3 summarizes the simulations performed in this study.

RESULTS

Unsteady flow in blade passage

The instantaneous entropy variation field at four axial locations (22.3% x/c, 31% x/c, 46% x/c and 96% x/c) for all simulations is depicted in figure 6. In figure 6(n), there is not only the tip leakage vortex but also the IGV tip vortex which can be observed at 22.3%, 31% and 46% x/c planes. The analysis of the entropy variation field shows that the switch from URANS to SAS approach with both SST and DRSM turbulence model, is responsible for a decrease of losses near the casing. The vortices seem to be more numerous but also more coherent, especially with the DRSM model. The effect of SAS is not visible at the leading edge as flow topology is very similar between URANS and SAS. However, significant discrepancies are observed from 46% x/c. This is partly due the tip leakage vortex dissipation across the weak shock as shown by Riéra *et al.* [11]. It should be noticed that the rebound effect linked to the induced tip vortex is better captured with SAS approach, especially with DRSM turbulence model. Moreover, secondary vortices can be observed

with these approaches, in particular in the vicinity of the trailing edge while all URANS-SST simulations are unable to capture them. The only discrepancy between M1-URANS-SST-wlin and M1-URANS-DRSM-wlin on one hand, between M1-SAS-SST-wlin and M1-SAS-DRSM-wlin on the other hand, is the use of the DRSM turbulence model instead of the SST one. It should be noticed that the suffix "-wlin" refers to the linear extrapolation for the ω value at the wall. Without this suffix, the pseudo roughness condition is used. Since the generation of the tip leakage vortex due to pressure difference between pressure and suction sides leads to a shear layer at the blade tip, there is a discrepancy between DRSM and SST models. The roll-up is quite different and less stable with DRSM. At 31% x/c, the tip leakage vortex is less spread. After the weak shock located just downstream the last mentioned plane, the dissipation is very different between the two RANS turbulence models. The tip leakage vortex is still coherent with DRSM. With SAS approach, the differences are more significant as the shear layer is more unstable leading to a more fluctuating vortex. The dissipation reduction and the fluctuating tip vortex lead to decreased losses (both area and levels). The comparison between M1-URANS-SST and M1-URANS-SST-wlin shows that the ω value at the wall based on linear extrapolation leads to a small reduction of entropy i.e. of losses in the tip region. The high loss area linked to the tip leakage vortex has both smaller level and extent. With SAS approach, this is also observed, except in the last plane where the losses are higher with linear extrapolation. This is due to a combination of boundary condition and turbulent viscosity destruction with SAS. The comparison of mesh M1 and M2 shows that, for URANS, the mesh density has quite no significant effect on entropy generation. For SAS, some discrepancies appear from the 46% x/c plane, especially in the tip shear layer. The amplitudes of the ω value effect and of mesh size are at least one order of magnitude smaller than the other modeling parameters. The comparison between all DRSM results obtained on both M1 and M2 meshes shows that the mesh has quite no effect on URANS predictions while the entropy field predicted by SAS approach is reduced with the finest mesh, especially in the tip leakage vortex core and in the vicinity of the casing. The combination of the DRSM model with the SAS approach allows the generation of a consequent content of fluctuations near the tip without any strong dissipation of the tip leakage vortex. Far from the IGV wakes, the tip leakage vortex is very similar for URANS and SAS approaches at plane 22.3% x/c and 31% x/c even if the rebound effect is slightly differently predicted. At plane 91% x/c, the SAS simulation highlights a second vortex at mid-passage which is not observed as well without IGV wakes and vortices. Moreover, in this plane, the entropy level is smaller meaning that the losses are smaller with incoming wakes and vortices. Figures 6(i) and 6(j) depict the instant where there is no significant interaction between the tip leakage vortex and the IGV wakes and vortices. An another instant is depicted in figures 6(m) and

FIGURE 6. INSTANTANEOUS ENTROPY VARIATION FIELD AT 22.3% X/C, 31% X/C, 46% X/C AND 96% X/C PLANES (FROM TOP TO BOTTOM).

TABLE 3. SUMMARY OF SIMULATIONS

Name	Mesh	SST vs. DRSM	URANS vs. SAS	ω_{wall}	IGV wakes
M1-URANS-SST	M1	SST	URANS	pseudo roughness	✗
M1-SAS-SST	M1	SST	SAS	pseudo roughness	✗
M1-URANS-SST-wlin	M1	SST	URANS	linear extrapolation	✗
M1-SAS-SST-wlin	M1	SST	SAS	linear extrapolation	✗
M1-URANS-DRSM-wlin	M1	DRSM	URANS	linear extrapolation	✗
M1-SAS-DRSM-wlin	M1	DRSM	SAS	linear extrapolation	✗
M2-URANS-SST	M2	SST	URANS	pseudo roughness	✗
M2-SAS-SST	M2	SST	SAS	pseudo roughness	✗
M2-URANS-SST-wake	M2	SST	URANS	pseudo roughness	✓
M2-SAS-SST-wake	M2	SST	SAS	pseudo roughness	✓
M2-URANS-DRSM-wlin	M2	DRSM	URANS	linear extrapolation	✗
M2-SAS-DRSM-wlin	M2	DRSM	SAS	linear extrapolation	✗

6(n) in order to highlight this interaction, especially at the 46% x/c plane. This significantly impacts the tip leakage vortex topology, especially with SAS approach. As previously observed by Riéra *et al.* [11], this interaction is responsible for the tip leakage vortex flutter leading to the breakdown of the tip leakage vortex into small coherent structures. This interaction modifies also the loss area (radial and azimuthal extents, loss level). Losses are reduced thanks to the interaction with IGV wakes and vortices.

Time-averaged flow in blade passage

Figure 7 and 9 compare the experimental data to numerical results in terms of axial and circumferential velocities and standard deviations of these quantities in two axial locations. The first one is located at 21.7% chord and 98% of relative height (h/H) while the second one matches the 26A plane where compressor performances are measured. The radial location of the second plane is chosen in order to highlight the presence of tip leakage vortex. In each figure, the different numerical results highlight the influence of grid density (meshes "M1" vs. "M2"), of ω value at the wall (ω_{wall} , 'roughness' vs. 'linear extrapolation'), of RANS turbulence model ('SST' vs. 'DRSM'), of SAS model ('URANS' vs. 'SAS') and of IGV wake and tip vortices.

In the first plane (21.7% chord), the mesh size, the value of ω_{wall} and RANS turbulence model have no significant effect as no velocity fluctuations are observed. These fluctuations can be seen as root mean square value of velocity components. It should be noticed that the SAS modeling generates small fluctuations. This is still better than URANS modeling. Nonetheless, this

FIGURE 8. TIME-AVERAGED AXIAL MOMENTUM UPSTREAM OF THE COMPRESSOR ROTOR.

is far from being satisfactory. The comparison to experimental data shows that the IGV wakes and vortices must be included in CFD setup in order to sufficiently generate fluctuations. These fluctuations are consequently due to the periodic perturbations

(a) Axial velocity

(b) Circumferential velocity

(c) Standard deviation of axial velocity

(d) Standard deviation of circumferential velocity

FIGURE 7. TIME-AVERAGED VELOCITY FIELD NEAR THE CASING AT 21.7% CHORD AND 98% h/H.

of the tip leakage vortices by the IGV tip vortex. The SAS model increases the velocity fluctuations by comparison to URANS results. It should be noticed that the combination of SAS approach and DRSM model on mesh M2 produces some fluctuations near the suction side. Nevertheless, these fluctuations are clearly insufficient to recover the experimental data. Despite the fact that circumferential velocity fluctuations are well captured, the axial velocity fluctuations are underpredicted by simulations. Thus, this seems that some axial perturbations are not taken into account in the simulations. A part of the underprediction can be due to the absence of the downstream stator row.

In terms of time-average of velocity components, the agreement with experimental data is quite good, especially for the circumferential velocity. A deficit of axial velocity is not well captured. This discrepancy can be explained by the average used to compute the time-averaged data. Numerically, as the simulation

is performed in the rotor frame, a simple time average is performed. Experimentally, in order to be in the same relative position to the rotor while the LDA measurements are performed in absolute frame, the phase average is also used. This phase average implies that the IGV wakes and vortices are not azimuthally averaged. On the contrary, as the time-average of numerical results is performed on 20 IGV periods, the IGV wakes and vortices can not be directly observed. Only the influence on the velocity field around the rotor blade can be captured.

Time-averaged flow downstream of the rotor

The axial and circumferential velocities and standard deviations of these quantities in plane 26A (performance measurements plane) are plotted in figure 9.

The profile of the circumferential velocity is well predicted by all simulations in terms of location and width of the rotor wake. A small discrepancy is observed on the amplitude of the

(a) Axial velocity

(b) Circumferential velocity

(c) Standard deviation of axial velocity

(d) Standard deviation of circumferential velocity

FIGURE 9. TIME-AVERAGED VELOCITY FIELD NEAR THE CASING AT 26A PLANE.

peak within the wake which is over-predicted by CFD. There are two exceptions. The first one concerns the URANS-DRSM, for both M1 and M2 meshes, which does not predict the good shape of the velocity profile as the peak is not well located and gradients are not similar to the experimental profile. Indeed, experimentally, the gradient is stronger at the smallest azimuthal edge of the peak (around 5° - 5.5°) than at the other edge (around 5.5° - 7°). The opposite behavior is observed for URANS-DRSM simulations, despite the discrepancy is reduced with the finest mesh M2. The SAS-DRSM with mesh M1 is the second exception. Despite the fact that the shape of the velocity profile is well captured, the peak level is under-predicted by this simulation. On the mesh M2, the circumferential velocity profile is quite well captured by SAS-DRSM as the wake is only a little bit thinner than the experimental profile.

The experimental profile of axial velocity is similar to the circumferential velocity except that there is a deficit of axial ve-

locity within the wake instead of an increase of azimuthal velocity. As for the other velocity component, the shape of the axial velocity profile is well predicted outside and within the wake. The velocity deficit is well located by simulations. Nonetheless, the width of the peak relative to the velocity deficit is under-predicted by CFD. As for the the circumferential velocity, for both meshes, the URANS-DRSM fails correctly predicting the shape of the axial velocity profile (location of the peak and gradients) while SAS-DRSM underestimates the velocity deficit. It can be noticed that the finest mesh M2 leads to a thinner wake, especially with the SAS approach. The comparison between the resultats obtained with both meshes (URANS-SST and SAS-SST) shows that there is a very small impact on the velocity profile. On the contrary, the velocity shape is significantly sensitive to the presence of IGV wake and vortices in the simulation as the width is reduced with these rotating secondary flows at the inlet.

The rms value of circumferential velocity represented by the

standard deviation of this quantity is not well predicted in terms of level by any simulation. With the first mesh "M1", some fluctuations are captured only if the SAS model is used, especially with the DRSM turbulence model. For these simulations, the location of the peak is well reproduced. A similar behavior is obtained with SAS-SST or SAS-DRSM and the second mesh "M2". Nonetheless, the width of the peak is too small. The inclusion of IGV wakes and vortices allows the prediction of a wider peak. All URANS simulations fail predicting the shape of standard deviation profile. With the second mesh "M2" and IGV vortices, the URANS-SST-wake is able to capture some fluctuations. However, the shape of the profile is far from being well reproduced.

The standard deviation of axial velocity is shown in figure 9(c). The experimental profile is characterized by a double peak in the rotor wake at 5.25° and 6.25° . This double peak shape is well captured by all SAS simulations (SST or DRSM models). It should be noticed that the simulation M2-SAS-DRSM predicts a second peak too close to the first one. This is linked to the thinner wake as observed on axial velocity. Despite the good agreement in double peak locations, the fluctuation level and the double peak width are not well predicted. This can be due to the absence of the interaction of the first rotor and the first stator located downstream as the computational domain includes only the rotor and IGV wakes are taken into account through the inlet boundary condition. Except with the mesh "M2" and the IGV vortices, all URANS simulations fail correctly predicting the axial velocity fluctuations.

Time-averaged performances

The overall performances of the first rotor, computed between the experimental planes 25A (upstream of the rotor as shown in figure 1) and 26A (downstream) are plotted in figure 10. These figures depict the evolution of stagnation pressure ratio (figure 10(a)) and isentropic efficiency (figure 10(b)) against the massflow. In the present study, only the nominal point is computed. Three nominal isospeeds predicted by RANS simulations are also plotted in order to show the trend of the evolution of stagnation pressure ratio and isentropic efficiency with respect to massflow. The main discrepancy of stagnation pressure ratio between experimental data and numerical results is due to the choice of the operating point. The static pressure specified at the hub was set in order to have the best agreement with the experimental axial momentum, especially in the tip region as shown in figure 8. This outlet boundary condition allows a consistent comparison to experimental data near the tip, especially for LDA measurements. Considering the evolution of the stagnation pressure ratio against massflow, the definition of the operating point seems to be responsible for an error about 0.75% for the stagnation pressure ratio and 1% for the massflow.

Except the SAS-DRSM simulation, all computations pre-

dict the stagnation pressure with an error less than 1%. With another operating point and the trend given by RANS-SST isospeed, this discrepancy could be reduced up to 0.5% for both stagnation pressure ratio and massflow. The mesh density has a small impact on performances as stagnation pressure ratio and massflow obtained by M1-URANS-SST and M2-URANS-SST are very similar. The mesh effect is quite identical for SAS-SST model. For DRSM computations, the mesh has a small influence with URANS approach, mainly on massflow (reduction with the finest mesh). The combination with SAS is much more sensitive to the mesh density. With the finest mesh, the stagnation pressure ratio is significantly reduced and a discrepancy close to 2.5% is observed on this ratio (2% for the first mesh M1). The value of ω at the wall has a significant impact on performances as stagnation pressure ratio is higher with the linear extrapolation, for both URANS and SAS approaches. On the finest mesh "M2", the influence of the IGV wakes and vortices is not similar between URANS and SAS. For URANS, the stagnation pressure is decreased while there is an increase for SAS model. It should be noticed that the trend observed with SAS is similar to those of Zonal Detached Eddy Simulation as shown by Riéra *et al.* [10]. The main discrepancy between experimental data and numerical results is obtained with SAS-DRSM approach for the mesh "M2". The stagnation pressure ratio is significantly underpredicted for both meshes with SAS-DRSM. This can be due to an overestimation of separation and blockage, especially near the hub corner.

The isentropic efficiency is depicted in figure 10(b). As only the first rotor of the 3.5 stage compressor is simulated, there is no experimental data. Thus this is not possible to highlight the wrong prediction of an approach. Nevertheless, the comparison between all numerical results shows that there is a significant sensitivity of the isentropic efficiency to the following parameters:

- turbulence modeling as SAS approaches predict smaller isentropic efficiency than URANS up to 0.75 point for SST model (M2-URANS-SST-wake vs. M2-SAS-SST-wake) and 1.5 points for DRSM model (M2-URANS-DRSM-wlin vs. M2-SAS-DRSM-wlin). Moreover the RANS turbulence model (DRSM vs. SST) influences the prediction of isentropic efficiency with a difference about 0.5 point for both URANS and SAS approaches on mesh M1.
- ω value at wall as the linear extrapolation from the first cell center leads to an isentropic efficiency increased by 0.3 point. This effect is compensated by the effect of RANS model between results on mesh M2 with URANS approach (M2-URANS-SST vs M2-URANS-DRSM-wlin). On contrary, with SAS approach, the compensation is not sufficient and the combination with DRSM model leads to a decrease of 1 point of efficiency (M2-SAS-SST vs M2-SAS-DRSM-wlin).
- the IGV wakes and vortices, especially for SAS approach

(a) Stagnation pressure ratio

(b) Isentropic efficiency

FIGURE 10. TIME-AVERAGED OVERALL PERFORMANCES OF THE COMPRESSOR ROTOR.

coupled with SST model, with a difference of 0.5 point (M2-SAS-SST vs. M2-SAS-SST-wake). The URANS approach is less sensitive to the incoming wakes and vortices as the difference is only of 0.2 point.

- ➡ mesh density at least for URANS simulations with a discrepancy of 0.2 point. The mesh effect is amplified with DRSM model and SAS approach (0.75 point).

In the experimental plane 26A, the time-averaged numerical data are azimuthally averaged in order to obtain the radial profile of some quantities. In figure 11 are depicted the radial profiles of absolute deviation angle, of absolute stagnation pressure and of absolute stagnation temperature.

The comparison between experimental and numerical data shows that there is a good agreement for the absolute deviation angle, except near the hub and near the casing. The discrepancy between all numerical results is about 1° which is the order of magnitude of experimental uncertainty. Near the hub, similar levels are obtained by all simulations. The discrepancy with experimental data can be due to both (i) a misprediction of the hub corner flow as a small hub corner separation is predicted leading to a decrease of the absolute deviation angle and (ii) the leakage flow generated by the gap between fixed and rotating wall at the hub [35]. At the casing, the discrepancy of prediction of tip leakage flow, especially the vortices, leads to a significant difference in terms of level and radial gradient. The highest dis-

crepancy concerns the M1-SAS-DRSM-wlin simulation with an underestimation about 4° . The analysis of the different modeling parameters (mesh density, ω value at the wall, RANS modeling, SAS modeling and IGV wakes) shows that:

- ➡ the value of ω at the wall has no influence on absolute deviation angle.
- ➡ the mesh refinement does not modify significantly the radial profile of this angle.
- ➡ the IGV wakes and vortices impact significantly the prediction of the tip leakage flow. It should be noticed that the gradient near the casing is well captured with URANS approach.
- ➡ the use of DRSM model instead of SST modifies strongly the radial profile, especially when combined with the SAS approach especially in the vicinity of the tip leakage flow.

The radial profile of absolute stagnation pressure in plane 26A is shown in figure 11(b). As the absolute deviation angle, the agreement with experimental data is globally good for all computations except the M1-SAS-DRSM-wlin and M2-SAS-DRSM-wlin ones. With this turbulence modeling, the absolute stagnation pressure is significantly underpredicted along the whole span, with the highest discrepancy between 15% and 60% of the vane height (h/H). Thus the underestimation of stagnation pressure ratio (figure 10(a)) is not only due to an amplified hub corner separation but also to an amplified stagnation pressure loss outside the influence area of endwall. This underprediction is

(a) Absolute deviation angle

(b) Absolute stagnation pressure

(c) Absolute stagnation temperature

FIGURE 11. TIME-AVERAGED RADIAL PROFILES IN 26A PLANE.

increased with the finest mesh M2. The comparison between all other simulations shows that the main discrepancy is due to the choice between SAS and URANS approaches. The effects of mesh density and ω value at the wall are smaller than the switch from URANS to SAS approach i.e. two families of nu-

merical results are obtained: SAS on one hand, URANS on the other hand. For each family, the discrepancy between the predictions is smaller than the difference between the two experimental data sets (pneumatic probes vs. time-averaged unsteady probes). With the DRSM model, the stagnation pressure within the tip

leakage flow is significantly overpredicted (close to the 85% relative height), especially with the mesh M2. It should be noticed that the underprediction of the stagnation pressure near the hub is due to the absence of the leakage flow itself due to the axial gap between rotating and fixed endwall. The analysis of the radial profile between 80% and 90% h/H shows that the IGV wakes impact significantly the prediction of stagnation pressure as the gradient prediction is better when these wakes are taken into account through the inlet boundary condition.

Figure 11(c) depicts the radial profile of absolute stagnation temperature. The agreement with experimental data is good as the maximum discrepancy is about 0.9%. The comparison between all numerical results shows that the SAS approach modifies strongly the profile near the casing, with a discrepancy of 0.75%. The modification of the radial profile shape is one of the explanation of the difference in terms of isentropic efficiency. The other variants of flow modeling (mesh density, turbulence modeling) have a very small impact on stagnation temperature prediction. For these secondary influence parameters, the highest effect is obtained with DRSM model which predicts a smaller stagnation temperature.

Spectral analysis

In the following, all power spectral densities (PSD) are computed with the Welch method [36] using an overlap of 50% and at least 10 Hann windows with a linear mean for each. The PSD of the static pressure signal recorded at the casing in plane 26A is plotted in figure 12. All simulations are depicted and the two signals obtained by Riéra *et al.* [11] are also added as references ("M2-URANS-SA-wake" and "M2-ZDES-SA-wake", the last one being obtained with Zonal Detached Eddy Simulation based on Spalart-Allmaras turbulence model (noted SA) [6]). First of all, the frequency resolution is 2000 Hz for the numerical data set arising from a temporal range of 20 IGV periods (even if the IGV is not taken into account in the simulation). The experimental data (frequency of 2 Hz) is resampled in order to have the same resolution frequency as numerical results and to perform consistent comparison as explained in [11].

For the first mesh, as there is no rotor-stator interaction (no IGV wake and absence of first stator in computational domain), no specific frequency is observed. The only exceptions are M1-URANS-SST and M1-URANS-SST-wlin for which there is quite no fluctuations as the PSD is very close to zero and a specific frequency and its harmonics arises from the PSD. This frequency is clearly artificial and has no physical meaning. Switching to DRSM turbulence model, the fluctuations of static pressure increases leading to a higher PSD on the whole frequency range. For both SST and DRSM turbulence models, the ω value at the wall has no significant impact on PSD, especially with SAS approach. Switching from URANS to SAS approach, for both SST and DRSM turbulence models, the fluctuations

amplitude are increased up to 2.5 orders of magnitude (for URANS approach). Nonetheless, the PSD is still far from the experimental data.

Considering all PSD obtained with the second mesh "M2", three main effects are highlighted:

- ➡ the IGV wakes. Without any IGV wakes and vortices, the blade passing frequency of IGV (BPF) and its harmonics cannot be captured. There is only a broadband spectral content with a discrepancy of 5 orders of magnitude. With the IGV wakes and vortices specified at the inlet boundary condition, the fluctuations are higher leading to a higher PSD, especially at the lowest frequencies. Moreover, the BPF and harmonics can be observed. Despite the good agreement for the first three harmonics of the BPF, the cutoff frequency arises too early. It should be noticed that SST and SA turbulence models lead to similar PSDs (URANS approach). In terms of physical mechanisms, the IGV tip vortex is responsible for the flutter of the rotor tip leakage vortex. Due to the fact that the rotor blade count is twice the IGV one, the flutter appears at a specific frequency linked to BPF. In compressor with prime numbers, this frequency is not directly linked to the compressor rotational speed but to rotor-stator interaction (Tyler-Sofrin mode).
- ➡ the URANS vs. SAS approach. Switching from URANS to SAS approach, the PSD level without IGV is increased and the broadband spectral content is closer to experimental data. With IGV wakes, the broadband spectral content is also amplified and the BPF and some of its harmonics are captured. It should be noticed that the switch from URANS to SAS approach delays the frequency where the numerical results become far from experimental data and the PSD gradient is not captured.
- ➡ the switch from SST to DRSM turbulence model increases the broadband spectral content on the whole frequency range, by three orders of magnitude. With SAS approach, the PSD is close to the result of the reference PSD obtained by Riéra *et al.* [11]. Nevertheless the BPF and its harmonics are not captured by M2-SAS-DRSM-wlin simulation as the IGV wakes are not taken into account in the simulation.

The comparison to the reference PSD obtained by Riéra *et al.* [11] shows that the SAS approach improves certainly the prediction of static pressure PSD by comparison to URANS method, especially at low and medium frequencies where the SAS is very close to ZDES. With SST turbulence model and with the IGV wakes, a good agreement is obtained up to the fifth harmonics of IGV BPF (third harmonics for URANS approach). This can be explained by the cutoff effect of modeled eddy viscosity on high frequencies. With the DRSM turbulence model, the agreement is even better at high frequency range. A better agreement should be expected with the IGV wakes at the compu-

(a) All meshes, without IGV wakes

(b) Only mesh M2, wake effect and comparison to ZDES-SA [11]

FIGURE 12. POWER SPECTRAL DENSITY OF STATIC PRESSURE IN PLANE 26A AND AT CASING.

tational domain inlet. The comparison of the two meshes shows that with the first mesh, the SAS-DRSM technique is able to capture the same PSD as the second mesh up to the ninth harmonics (on the same mesh, the discrepancy between ZDES and SAS-DRSM arises at the 20th harmonics) while the first mesh is ten times smaller (4.3 million points vs 44 million points per blade passage). This means also that the PSD prediction could be im-

proved by using a more elaborated RANS model (DRSM) in the ZDES framework [37, 38].

CONCLUSION

The prediction of secondary flows (corner separation and tip leakage flow) and of the compressor performances (compression

ratio, efficiency, operating range) is influenced by the turbulence modeling. The aims of the present study are (i) the validation of the combination of the SAS approach with the DRSM turbulence model by comparison to experimental data, especially to laser measurements in the tip of a rotor of a high pressure compressor and (ii) the discussion of the flow prediction improvements with respect to turbulence approaches classically used in CFD and industry: URANS simulations and standard SAS simulation i.e. combined with SST turbulence model. On one side, the SAS approach can be viewed as an improved URANS one as the turbulent viscosity is reduced allowing the appearance of instability, especially in shear layer. For this reason, one mesh adapted to RANS simulation is used. On another side, this approach acts with the same idea of Detached Eddy Simulation: the reduction of eddy viscosity in detached flow region. Thus a second mesh adapted to Zonal Detached Eddy Simulation was used, allowing to highlight the mesh effect on URANS and SAS approaches. The RANS turbulence model plays also a significant role in URANS prediction. Thus two turbulence models are assessed both with URANS and SAS simulations: $k - \omega$ SST model and a DRSM-SSG/LRR- ω model.

The analysis of entropy field and the comparison to experimental data (LDA measurements, performances) show that the tip leakage vortex predicted by SAS and/or DRSM is significantly different from URANS-SST simulation. The vortex dissipation through the weak shock is strongly reduced. The tip leakage vortex is still observed in the vicinity of the trailing edge. Only the simulations with IGV wakes predict the velocity fluctuations near tip gap, from the leading edge. Except the combination SAS with DRSM model and with the first mesh, all simulations are able to capture the time-average velocity in the vicinity of the tip. The prediction of SAS-DRSM is clearly improved by using a fine mesh. Concerning the time-averaged performances, the stagnation pressure losses are slightly overestimated by SAS. This is due to an amplification of the hub corner separation. The combination with DRSM model increases the pressure stagnation losses. Moreover, the isentropic efficiency is very sensitive to the SAS approach and to the turbulence model. The spectral analysis shows that the prediction of the amplitude and frequencies of the power spectral density of static pressure is improved using the SAS approach instead of URANS one, especially with the DRSM turbulence model. The SAS approach leads to PSD similar to ZDES, especially with the DRSM model. Thus, the SAS-DRSM is able to well predict the tip leakage flow with the fine mesh. Nevertheless, this approach amplifies the hub corner separation leading to a strong underestimation of overall performances.

Future work will concern the simulation of SAS-DRSM with incoming wakes and further analysis at design point, especially in the flow field and at near surge point in order to better understand the effect of turbulence modeling. Moreover,

the comparison between SAS and ZDES will be pursued. The present configuration is quite simple with one rotor subjected to incoming wakes and vortices. The next step is to extend the present study to real rotor-stator interactions by including the downstream rows in the computational domain. Then, technological effects such as axial gap (and seal teeth), fillets will be added to the computational domain.

ACKNOWLEDGMENT

All simulations presented in this article have been funded in the framework of the *elsA* three-party agreement between AIRBUS, SAFRAN and ONERA which are co-owners of this software. The support of the different partners for the research compressor CREATE program is fully acknowledged. The authors thank gratefully X. Ottavy (LMFA) for providing the experimental data.

REFERENCES

- [1] Herzig, H., Hansen, A., and Costello, G., 1954. A visualization study of secondary flows in cascades. Tech. rep., Report 1163-National Advisory Committee for Aeronautics.
- [2] Bindon, J., 1989. "The Measurement and Formation of Tip Clearance Loss". *Journal of Turbomachinery*, **111**, pp. 257–263.
- [3] Kang, S., and Hirsch, C., 1993. "Experimental Study of the Three-Dimensionnal Flow within a Compressor Cascade with Tip Clearance : Part II - The Tip Leakage Vortex". *Journal of Turbomachinery*, **115**, July, pp. 444–450.
- [4] Mailach, R., Lehmann, I., and Vogeler, K., 2008. "Periodical Unsteady Flow within a Rotor Blade Row of an Axial Compressor - Part II : Wake - Tip Clearance Vortex Interaction". *Journal of Turbomachinery*, **130**, October, p. 041005.
- [5] Suder, K., and Celestina, M., 1996. "Experimental and Computational Investigation of the Tip Clearance Flow in a Transonic Axial Compressor Rotor". *Journal of Turbomachinery*, **116**, April, pp. 218–229.
- [6] Spalart, P., and Allmaras, S., 1994. "A One-Equation Turbulence Model for Aerodynamic Flows". *La Recherche Aérospatiale*, **1**, pp. 5–21.
- [7] Menter, F., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". *AIAA Journal*, **32**(8), August, pp. 1598–1605.
- [8] Borello, D., Hanjalic, K., and Rispoli, F., 2007. "Computation of tip-leakage flow in a linear compressor cascade with a second-moment turbulence closure". *International Journal of Heat and Fluid Flow*, **28**(4), August, pp. 587–601.
- [9] Shi, K., Xiao, Z., Chen, H., and Fu, S., 2013. "Numerical

- study of tip leakage flow of a transonic compressor rotor with iddes method". In ISABE, 1116.
- [10] Riéra, W., Castillon, L., Marty, J., and Leboeuf, F., 2014. "Inlet condition effects on the tip clearance flow with Zonal Detached Eddy Simulation". *Journal of Turbomachinery*, **136**(4)(041018), April, pp. 1–10. Editor : R. Bunker.
 - [11] Riéra, W., Marty, J., Castillon, L., and Deck, S., 2016. "Zonal detached-eddy simulation applied to the tip-clearance flow in an axial compressor". *AIAA Journal*, **54**(8), August, pp. 2377–2391.
 - [12] You, D., Mittal, R., Wang, M., and Moin, P., 2004. "Computational methodology for large-eddy simulation of tip-clearance flows". *AIAA Journal*, **42**(2), February, pp. 271–279.
 - [13] Boudet, J., Caro, J., and Jacob, M., 2010. "Large-eddy simulation of a single airfoil tip-clearance flow". In 16th AIAA/CEAS Aeroacoustics Conference.
 - [14] Duda, B. M., Menter, F. R., Esteve, M.-J., Hansen, T., Deck, S., and Bézard, H., 2011. "Numerical investigations on a hot jet in cross flow using scale-resolving simulations". In 7th International Symposium on Turbulence and Shear Flow Phenomena (TSFP-7), Ottawa, Canada.
 - [15] Wang, Z.-N., and Yuan, X., 2013. "Unsteady mechanisms of compressor corner separation over a range of incidences based on hybrid les/rans". In ASME Turbo Expo, June 3-7, San Antonio, Texas, GT2013-95300.
 - [16] Lucius, A., and Brenner, G., 2010. "Unsteady cfd simulations of a pump in part load conditions using scale-adaptive simulation". *International Journal of Heat and Fluid Flow*, **31**, pp. 1113–1118.
 - [17] Ravelli, S., and Barigozzi, G., 2014. "Application of unsteady computational fluid dynamics methods to trailing edge cutback film cooling". *Journal of Turbomachinery*, **136**, December, p. 121006.
 - [18] Sagaut, P., and Deck, S., 2009. "Large eddy simulation for aerodynamics: status and perspectives". *Phil. Trans. R. Soc.*, **367**(1899), pp. 2849–2860.
 - [19] Ottavy, X., Courtiade, N., and Gourdain, N., 2012. "Experimental and Computational Methods for Flow Investigation in High-Speed Multistage Compressor". *Journal of Propulsion and Power*, **28**(6), pp. 1141–1155.
 - [20] Arnaud, D., Ottavy, X., and Vouillarmet, A., 2004. "Experimental Investigation of the Rotor-Stator Interactions, within a High Speed, Multistage, Axial Compressor. Part 1 - Experimental Facilities and Results". In ASME Turbo Expo 2004, June 14-17, Vienne, Austria, ASME Paper GT2004-53778.
 - [21] Arnaud, D., Ottavy, X., and Vouillarmet, A., 2004. "Experimental Investigation of the Rotor-Stator Interactions, within a High Speed, Multistage, Axial Compressor. Part 2 - Modal Analysis of the Interactions". In ASME Turbo Expo 2004, June 14-17, Vienne, Austria, ASME Paper GT2004-53778.
 - [22] Tartousi, H., Kulisa, P., Leboeuf, F., Ngo Boum, G., Lefebvre, A., and Yammine, A., 2011. "Numerical Investigation of a Turbocharger Centrifugal Compressor: Volute Influence on the Performance of The Compressor". In European Turbomachinery Conference, Istanbul, Turkey, M. Sen, G. Bois, M. Manna, and T. Arts, eds., pp. 925–934.
 - [23] Cambier, L., Heib, S., and Plot, S., 2013. "The Onera elsA CFD Software: Input from Research and Feedback from Industry". *Mechanics & Industry*, **14**, 1, pp. 159–174.
 - [24] Mary, I., Sagaut, P., and Deville, M., 2000. "An algorithm for low mach number unsteady flows". *Computers & Fluids*, **29**, pp. 119–147.
 - [25] Edwards, J., and Liou, M.-S., 1998. "Low diffusion flux-splitting methods for flows at all speeds". In AIAA, Vol. 36, pp. 1610–1617.
 - [26] Daude, F., 2007. "Méthode d'Intégration Temporelle Implicite pour la Simulation des Grandes Échelles Application à la Réduction du Bruit de Cavité". PhD thesis, Université de Poitiers.
 - [27] Wilcox, D., 2008. "Formulation of the $k - \omega$ turbulence model revisited". *AIAA Journal*, **46**(11), November, pp. 2823–2838.
 - [28] Eisfeld, B., Rumsey, C., and Togiti, V., 2016. "Verification and validation of a second-moment-closure model". *AIAA Journal*, **54**(5), pp. 1524–1541.
 - [29] Menter, F., and Egorov, Y., 2005. "A scale-adaptive simulation model using two-equation models". In 43rd AIAA Aerospace Sciences Meeting and Exhibit. AIAA 2005-1095. Reno, Nevada.
 - [30] Menter, F., and Egorov, Y., 2006. "Revisiting the turbulent scale equation". In Proceeding IUTAM Symposium "One Hundred Years of Boundary Layer Research". Gottingen, Springer.
 - [31] Menter, F., and Egorov, Y., 2010. "The scale-adaptive simulation method for unsteady turbulent flow predictions. part 1: Theory and model description". *Flow, Turbulence and Combustion*, **85**(1), pp. 113–138.
 - [32] Nicoud, F., and Ducros, F., 1999. "Subgrid-scale stress modelling based on the square of the velocity gradient tensor". *Flow, Turbulence and Combustion*, **62**(3), pp. 183–200.
 - [33] Benyoucef, F., Bézard, H., Aupoix, B., and Michel, B., 2012. "SAS-SST model assessment and improvement". In Conference on Modelling Fluid Flow (CMFF'12). Budapest, Hungary.
 - [34] Didorally, S., Bézard, H., and Laroche, E., 2012. "Numerical simulation of the unsteady flow past a heated square cylinder". In 7th International Symposium on Turbulence, Heat and Mass Transfer, Palermo, Sicily, Italy.
 - [35] Marty, J., and Aupoix, B., 2012. "Interaction of shrouded stator flow and main flow and its influence on performances

- of a three-stage high pressure compressor”. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, **226**(A4), pp. 489–500.
- [36] Welch, P., 1967. “The Use of Fast Fourier Transform for the Estimation of Power Spectra : A Method Based on Time Averaging Over Short, Modified Periodograms”. *IEEE Transactions on audio and electroacoustics*, **2**(AU-15), pp. 70–73.
 - [37] Uribe, C., Marty, J., and Gerolymos, G., 2017. “Zonal Detached Eddy Simulation extension to $k - \omega$ models”. In *Proceedings of the AIAA Aviation 2017*, Denver, United States, p. 4279.
 - [38] Uribe, C., Marty, J., and Gerolymos, G., 2017. “Evaluation of Zonal Detached Eddy Simulation $k - \omega$ and Scale Adaptive Simulation methods on a realistic rotor of a high-pressure compressor”. In *Proceedings of the CFM 2017*, Lille, France.