



**HAL**  
open science

# The High Level of Maturity of the elsA CFD Software for Turbomachinery Aerodynamics Applications

Sylvie Plot

► **To cite this version:**

Sylvie Plot. The High Level of Maturity of the elsA CFD Software for Turbomachinery Aerodynamics Applications. ISABE 2019, Sep 2019, CANBERRA, Australia. hal-02502342

**HAL Id: hal-02502342**

**<https://hal.science/hal-02502342v1>**

Submitted on 9 Mar 2020

**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.

# The High Level of Maturity of the elsA CFD Software for Turbomachinery Aerodynamics Applications

**Sylvie Plot**

**sylvie.plot@onera.fr**

ONERA – The French Aerospace Lab

Aeronautics Directorate

Châtillon

France

## **ABSTRACT**

The elsA CFD software developed at ONERA is both a software package capitalizing the innovative results of research over time and a multi-purpose tool for applied CFD and for multi-physics. elsA is intensively used by the Safran and Airbus groups and ONERA, as well as other industrial groups and research partners. This paper presents the main capabilities of elsA and recent numerical simulations using elsA for a wide range of challenging turbomachinery applications. The simulations address aerodynamics computations as well as aero-structure and aeroacoustics ones. Each simulation enables to underline differentiating modelling and numerical methods implemented within elsA and shows that this software has reached a very high level of maturity and reliability.

**Keywords:** Navier-Stokes; Aerodynamics; Acoustics; Turbulence; Transition; Meshing; turbomachinery

## NOMENCLATURE

CFD	Computational Fluid Dynamics
elsA	ensemble Logiciel de Simulation en Aérodynamique
UHBR	Ultra-High Bypass Ratio
CROR	contra rotating open rotor
(U)RANS	(unsteady) Reynolds Averaged Navier-Stokes
EARSM/DRSM	Explicit Algebraic/Differential Reynolds Stress Model
SAS	Scale Adaptative Simulation
(Z)DES/LES	(Zonal) Detached/Large Eddy Simulation
DNS	Direct Numerical Simulation
OO	Object-Oriented
MFA	Multiple Frequency phase-lagged Approach
NGV / OGV	Nozzle Guide Vane / Outlet Guide Vane
BD / AD	Body-Forces / Actuator Disk approaches
FWH	Ffowcs-Williams & Hawkings
WMLES	Wall Modeled LES

## 1.0 INTRODUCTION

Advanced CFD is one key differentiating factor for designing efficient transportation systems in particular turbomachinery. Accuracy and efficiency are crucial for engine manufacturers to optimize the design of their products so that the limit of the aircraft flight envelopes is extended. Accurate and robust numerical simulation capabilities are essential to take into account complex phenomena involved in flow physics and to simulate real complex geometries. Numerical methods have to meet the constantly rising requirements in terms of efficiency in order to reduce the response time for complex simulations.

To this end, the Airbus and Safran groups use the elsA [1] software (ONERA-Airbus-Safran property), a multi-purpose tool for applied CFD and multi-physics. Indeed, while it is rather unusual for a CFD software package to deal both with external flows around aircrafts or helicopters and with internal flows in turbomachinery, elsA is today used as a reliable tool by Airbus for transport aircraft configurations [2][3] and helicopter applications [4][5], by Safran for turbomachinery flow simulations [6][7]. Among other industrial users, let us mention MBDA for missile configurations and Électricité de France for steam turbine applications [8][9].

The high levels of accuracy and reliability required today in aeronautic design are obtained through long term expertise and innovative research present at ONERA in various areas: physical modelling, algorithms, numerical methods, software efficiency on rapidly evolving hardware and validation and verification of aerodynamics results by comparison with detailed experimental data. The elsA software benefits directly from the above, as well as from multiple skills of many of its research partners. Among them the followings establishments have participated or participate to its development and validation mainly in the following indicated areas: Cerfacs for studies dealing in particular with numerical methods [10][11][12][13] and CPU efficiency [14] the LMFA and LTDS labs of École Centrale de Lyon [15][16][17] for turbomachinery applications and special interests in acoustics and vibrations, Cenaero (Belgium) [18][19], the Von Karman Institute [20] and ISAE institute [21] for turbomachinery flow simulation, the Dynfluid lab (Arts et Métiers ParisTech) for high accuracy numerical schemes [22], the Institute of Technology Antoine de Saint-Exupéry [23] and the Delft University of Technology [24] for design optimizations.

Among topics of prime importance in turbomachine design, one can mention the complexity of the geometries, the unsteadiness of the flows, the need to properly model the turbulence and accurately compute the onset of laminar turbulence transition, the impact of the engine-airframes interactions on the aircraft performances, the acoustic issues linked to close coupled engine integration, the taking into account of blade

deformations, likewise ground effect on fan forced response. Hereafter an illustration of these different themes is provided.

Many experimental and numerical studies have shown that taking into account geometrical components such as casing treatments, grooves, blade slots, cooling holes or gaps separating fixed and rotating walls can have a crucial impact on the performance of gas turbines. Therefore it is important for aeroengine designers to simulate more and more accurately the very complex geometries encountered on industrial turbomachines by integrating in the design process these geometrical components, at a reasonable cost. Besides, the relative motion between adjacent rotor and stator blade rows present in turbomachinery configurations gives rise to a wide range of unsteady flow mechanisms such as wake interactions, potential effects, hot streak migrations, shock wave propagations, or unsteady transitional flows. All these phenomena can have a crucial impact on the performance of gas turbines and cannot be captured accurately with a steady mixing-plane approach since the averaging treatment at the rotor/stator interface filters all unsteady effects. So unsteadiness must be addressed in the design process. Furthermore, the search of the highest efficiency and largest stable operating range for a given stagnation pressure ratio are design objectives for high pressure compressors. Those quantities are directly and strongly impacted by secondary and leakage flows occurring in the blade passage such as corner separation or stall and tip leakage flows. The turbulence modelling influences strongly the prediction of the overall performances. As for turbines, an accurate prediction of boundary layer laminar-turbulent transition is key to obtain a good design. Indeed, the boundary layer around turbine blades can be laminar for a significant portion of the chord, even in a high turbulence level environment. This is especially true for low-pressure turbines. Moreover, surface roughness can have a significant effect on boundary layer transition. In today's context of an increased focus on fuel efficiency and environmental impact, UHBR engines become an interesting option but the resulting size of the nacelles poses challenges in terms of installation on the airframe and impact on overall performance. Another related issue for UHBR is the jet noise, and the need to rely on reliable unsteady approaches to simulate the turbulent mechanisms which generate noise. Furthermore, the large fan diameter and the small ground clearance of these configurations can lead to aerodynamic interaction with the ground during take-off phases and lead to possible failure due to high cycle fatigue. So the ability to investigate the ground effect on the forced response of a large civil fan is necessary. Taking into account of the real manufactured blade shape in the investigation of the blade deformations due to centrifugal forces and aerodynamic loads during the flight is key for an accurate prediction of CROR performances for instance.

This paper intends to demonstrate that the elsA software contains accurate and robust numerical and modelling features to address all these topics. The main capabilities of elsA [1] are recalled in section 2.0 . Section 3.0 highlights recent numerical studies [25][26][27][28][29][30][7][31] performed at ONERA for a wide range of challenging turbomachinery applications. Finally perspectives and current work of prime importance is quickly presented in section 4.0 .

## **2.0 GENERAL DESCRIPTION OF THE ELSA CFD SOFTWARE**

For about 20 years, the elsA software is simultaneously a basis for CFD research, a software package capitalizing on the innovative results of research over time, a tool allowing investigation and understanding of flow physics, and a multipurpose tool for applied CFD and multi-physics [1][32]. The range of aerospace applications covered by elsA is very wide: aircraft [33][30], helicopters [34], tilt-rotors [35], turbomachinery [6][36], wind turbines [37], missiles, unmanned aerial vehicles, launchers... For turbomachinery, all types of multi-stage axial, radial or mixed-flow configurations are addressed: compressors [25][26][27], turbines [28], fans [29][7], propellers or contra-rotating rotors [31].

First, let us recall the main features of the elsA multi-application CFD simulation software. This software deals with internal and external aerodynamics from the low subsonic to the high supersonic flow regime and relies on the solving of the compressible 3-D Navier-Stokes equations.

elsA allows for the simulation of the flow around moving deformable bodies in absolute or relative frames. A large variety of turbulence models from eddy viscosity to full DRSM are implemented for the RANS equations [38][39]. We can mention numerous one- or two-equation turbulence models (Spalart-Allmaras,  $k-\omega$  and  $k-\epsilon$  families,  $k-l$  Smith models...), more advanced ones, such as the non-eddy viscosity EARSM [39] and several versions for the DRSM, as well as the SAS approach which is available in elsA for  $k-\omega$  and DRSM models [40][27]. Various laminar-turbulent transition modelling exist in elsA to address natural or by-pass transition [38]. The approaches that have initially been implemented rely either on local or non-local criteria, or on solving additional transport equations as the Menter-Langtry [41][42]. Then developments around criteria have enabled the increase of the Mach domain application [43] and also a huge simplification for the user (criteria transition model written under transport equations formalism). As for the Menter-Langtry model, several correlation functions are available included some developed at ONERA [44] and taking into account roughness is now possible [28]. A database approach for natural transition prediction [45] also called Parabola method, as well as a new laminar kinetic energy model [46] for bypass transition have also been implemented and tested in elsA. Various approaches for DES [47][48][30] and LES are also available.

Complex geometrical configurations may be handled using highly flexible techniques involving multi-block structured body-fitted meshes: these techniques include patched grid and overset capabilities (Chimera technique) [49][50][51][33][36]. From this initial multi-block structured meshing paradigm, elsA has evolved toward a quite complete multiple gridding paradigm including the local use of unstructured grids [52][25] in some blocks of a multi-block configuration, as well as adaptive Cartesian grids [49][50][53].

The system of equations is solved using a cell centered finite-volume method. Space discretization schemes include classical second order centered or upwind schemes and higher order schemes. The mostly used integration of the semi-discrete equations relies on a backward Euler technique with implicit schemes solved by robust LU relaxation methods. The convergence is accelerated by the use of multigrid techniques for steady flows. The implicit Dual Time Stepping method or the Gear scheme is employed for time accurate computations. elsA also includes an aeroelastic subsystem [54][31] which gives access in a unified formulation to various types of aeroelastic simulations. The simulation types range from non-linear and linearized harmonic forced motion computations, to static coupling and consistent dynamic coupling simulations in the time-domain, with different levels of structural modelling ("reduced flexibility matrix" for static coupling, modal approach, or full finite element structural model). The elsA software offers also a framework for optimization design thanks to its optimization module for the calculation of sensitivities by linearized equation solution or by adjoint solver techniques [55][56][3][23][24].

The elsA platform refers to the elsA solver and several pre and post-processing Python modules, such as Cassiopée [51], used in particular for transforming input data, for Chimera pre-processing and post-processing. elsA is based on an OO design method and is coded in three programming languages: C++ as the main language for implementing the OO design, Fortran for implementing time-costly scientific computation methods, and the Python interpreted OO language for the user interface. The CGNS/Python mapping of the CGNS/SIDS is used for interoperability and coupling with elsA. The software has been ported to most HPC platforms, achieving good CPU efficiency on both scalar multi-core computers and vector computers [14].

A list of more than 130 references based on elsA in peer-reviewed journals is available on <http://elsa.onera.fr/publications.html>.

## 3.0 RECENT CHALLENGING STUDIES

### 3.1 Preamble

The reader must refer to the main references mentioned in each following subsection in order to obtain more details on the computed configurations, the meshing and numerical steps, the original sources for considered methods and models, as well as a deeper interpretation of the results and their comparison to experiments when appropriate. In the current paper only main conclusions are provided.

### 3.2 A hybrid structured/non structured grid strategy for the modelling of technological effects on complex turbomachinery applications

As already mentioned technological effects present in turbomachinery configurations must be taken into account in numerical computations. Such geometrical details, characterized by their small dimension, can be addressed with elsA with a multi-block structured approach by using the Chimera technique which offers flexibility in the grid generation. Yet the main drawback of the Chimera method is that it induces local conservation losses, leading to uncertainties on the evaluation of the massflow which is key parameter in turbomachinery applications. Paper [25] deals with an alternative grid approach, based on the use of hybrid structured/non structured grids. The main principle of the grid approach is to mix within the computational domain structured and non structured zones which are connected with conformal matching frontiers in order to be conservative. Both structured and unstructured grids are generated separately (precision and efficiency for the structured part, flexibility for the meshing of complex geometries with unstructured elements). Two complex configurations are studied. The first case is a multi-row film-cooled NGV, and the second is a multistage compressor configuration including labyrinth seals.

#### 3.2.1 Multi-row film-cooled turbine blade

The first application is a film-cooled turbine nozzle guide (Figure 1 -left). This highly film cooled NGV, composed of 113 holes distributed along 10 rows, is representative of an advanced industrial turbine blade. The Chimera approach has already been used on this configuration [36]. In the current work one investigates the hybrid grid strategy illustrated on Figure 1 (right). It allows for the meshing of the cooling holes with unstructured elements, keeping structured elements in the main channel flow.

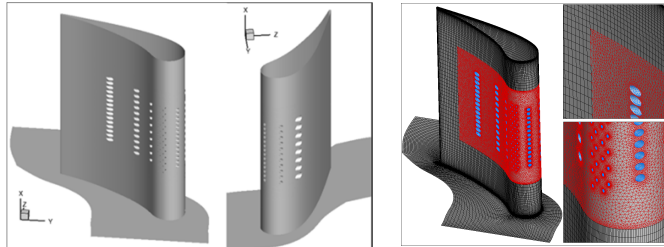


Figure 1: 3D view of the EPFL configuration (left) – Hybrid grid (right)

Such grid strategy enables to simulate the film-cooling interaction between the main channel flow and the cooling flow as illustrated hereafter. Figure 2 (left) represents a mid-span view of the Mach number distribution: as expected the main flow is deviated and accelerates within the NGV passage. The total temperature field on the blade and on a mid-span slice represented in Figure 2 (center) exhibits qualitative satisfactory behavior: the convection of the coolant flow leaking on the blade surface and then in the blade channel. The wall heat transfer coefficient values can be computed and compared to available experimental data. Figure 2 (right) represents the heat transfer distribution obtained using three turbulence models: Spalart-Allmaras, (k-l) of Smith, and (k- $\omega$ ) of Wilcox for the experimental point corresponding to an injected coolant mass flow of 10.33 g/s. One can notice that qualitatively the computations reproduce a satisfactory agreement with the experimental results.

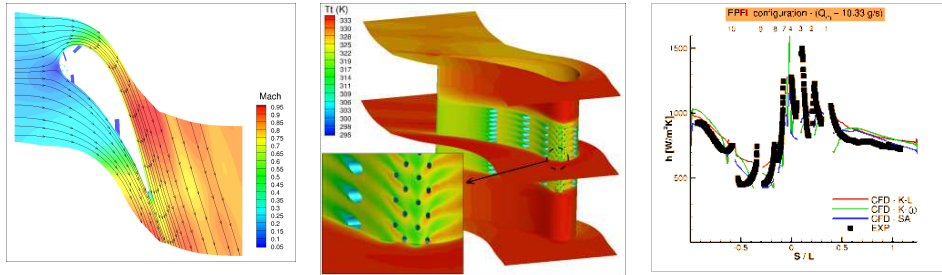


Figure 2: Mach number distribution at mid-span (left) – Total temperature field on the blade and on a mid-span slice (center) - Heat transfer distribution (right)

### 3.2.2 Multistage compressor stage with labyrinth seals

The second application presented in [25] is the 3.5 stage axial experimental compressor CREATE tested in LMFA laboratory [57]. A meridian view of the compressor with the investigated experimental planes is presented in Figure 3 (left). In this study the modelling of the labyrinth seals beneath the three stator blades is discussed. The main flow channel grid has a classical structured O-4H type topology, the labyrinth is meshed with non-structured elements (Figure 3 –right).

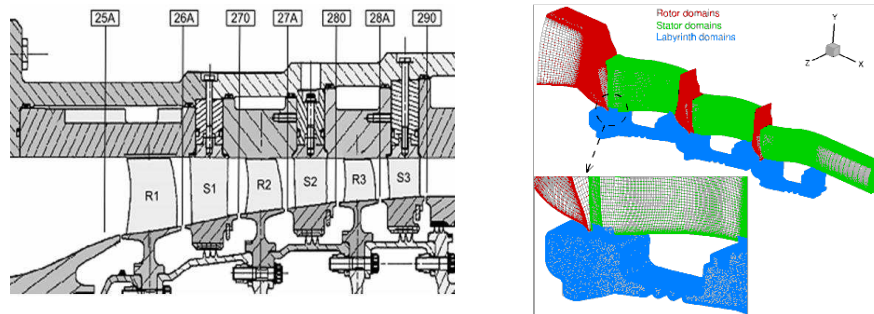


Figure 3: Meridian view of CREATE with measurement positions. (left) – Global View of the mesh. (right).

Different steady mixing plane computations have been performed on this configuration. A view of the relative Mach number at mid-span and inside the labyrinth seals for a computation at nominal operating point is plotted in Figure 4 (left).

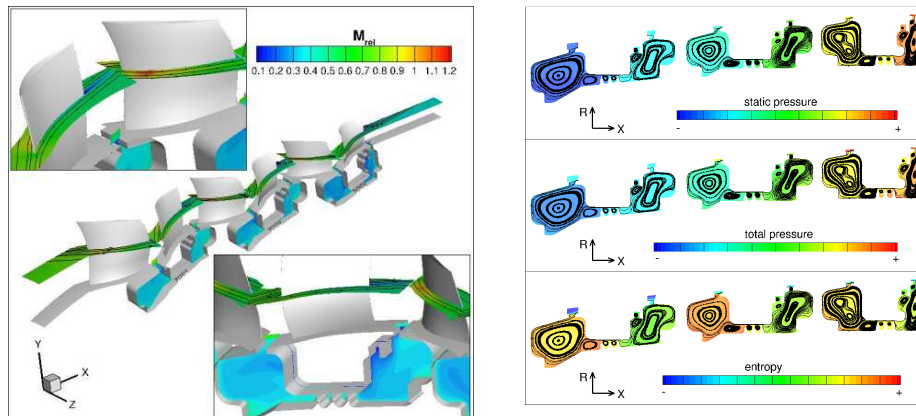


Figure 4: Mach number distribution at mid-span and in the cavity (left) – Cavity flow within the three labyrinths (right).

The result is acceptable from a qualitative point of view. The streamlines and the main flow pattern within the flow channel are well captured, while at the same time the flow within the labyrinth is simulated. Such types of computations give indeed access to the complex cavity flow highlighted in Figure 4 (right).

### 3.3 Multiple-frequency phase-Lagged approach

The computational cost of a time-accurate full-annulus computation remains very high, despite the increase of HPC resources. It is therefore important to have access to

numerical methods that reduce the computational domain and are efficient enough to simulate accurately the main unsteady effects. To alleviate the problem, the phase-lagged method is commonly used for unsteady rotor/stator simulations to reduce computing resources. This method, limited to flow configurations including a single periodic perturbation (as one single-stage or one row under vibrations), enables the computation of the periodic flow around one blade per row, using appropriate phase-shifted boundary conditions at the pitchwise boundaries. A generalization of the phase-lagged approach, called MFA allows unsteady computations through several rows, whilst still limiting the computational domain to one single blade-to-blade passage in each row is available in elsA. An evaluation of the MFA approach is presented in [26]. The first part of the paper presents the method and discusses the associated assumptions and limitations. The method is evaluated on the 3.5 stage axial compressor CREATE [57] (in Figure 3). The CFD results are analyzed and compared both with experimental data and with a elsA reference multipassage computation (RC) based on a sliding mesh approach [57]. An example of flowfield reconstruction is illustrated in Figure 5. They are compared to snapshots obtained with the RC and the mixing-plane steady computations (MPC). One can observe that the main flow structures seen in the RC computation are quite well captured by the MFA simulation.

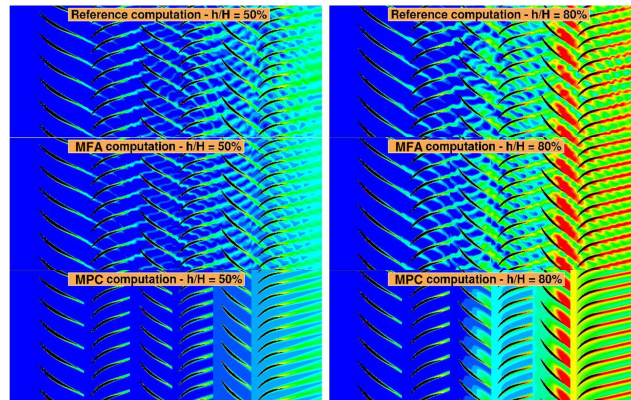


Figure 5: Entropy snapshots at  $h/H = 50\%$  and  $h/H = 80\%$ .

The MFA approach enables the simulation of unsteady effects on a multistage turbomachinery and access to unsteady information that would not be available with a MPC. A detailed analysis underlines the limits of the MFA. If the method is capable of capturing unsteady effects linked to the adjacent upstream and downstream blade rows passing frequency, it fails modelling clocking effects, i.e., the relative influence between rows  $N$  and  $N+2$ .

### 3.4 Assessment of scale adaptive simulation

CFD results depend significantly on the Navier-Stokes closure i.e. on the used turbulence model and 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 or the Menter SST models. The turbulence is clearly anisotropic in the vicinity of the tip gap and second moment closure DRSM is able to capture this characteristic. Zonal DES was also performed at ONERA [58][59] to study the tip leakage vortex and its interaction with incoming wakes and vortices. The SAS approach is a way to enhance traditional RANS model with low implementation effort. It can improve dramatically unsteady RANS computations especially when massive flow separation occurs, even for non-refined grids. A well-known example is the jet in crossflow [40]. The classical turbulence models (even DRSM) fail at reproducing the diffusion of the jet downstream, whereas the same models coupled with the SAS approach perfectly reproduce the experiments, using classical RANS grids. The standard SAS approach of Menter associated to the SST  $k-\omega$  model [60] is present in elsA for years [40]. The SAS approach can be generalized to any RANS turbulence model. It has been implemented in elsA [27] for the DRSM model (only the term, responsible for the decrease of the eddy viscosity in case of occurrence of flow instabilities, has to be added to right side of the  $\omega$ -scale transport equation). The study presented in [27] aims at validating this combination.



The first rotor (R1) of the axial compressor CREATE [57] (in Figure 3) is investigated. CFD setup computational domain, mesh and boundary conditions and numerical methods are detailed in [27]. Two meshes are used. This first one (M1) can be considered as a fine mesh for URANS simulation, the second grid is the same used for ZDES simulation of [58][59]. SAS-DRSM computation is compared to URANS using  $k-\omega$  SST or DRSM models, as well to SAS-SST and ZDES computations, and experimental data.

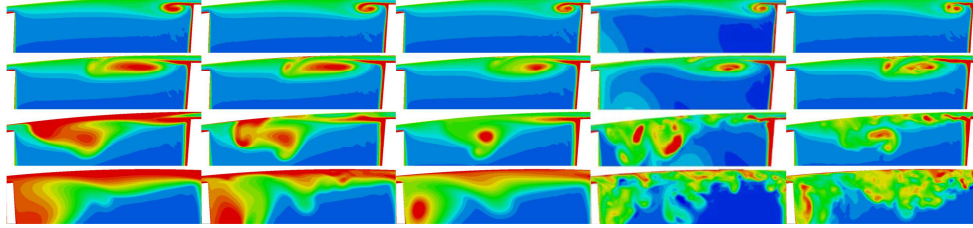


Figure 6: Instantaneous entropy fields at 22.3%, 31%, 46% and 96%  $x/c$  for (from left to right): M2-URANS-SST, M2-SAS-SST, M2-URANS-DRSM, M2-ZDES, M2-SAS-DRSM

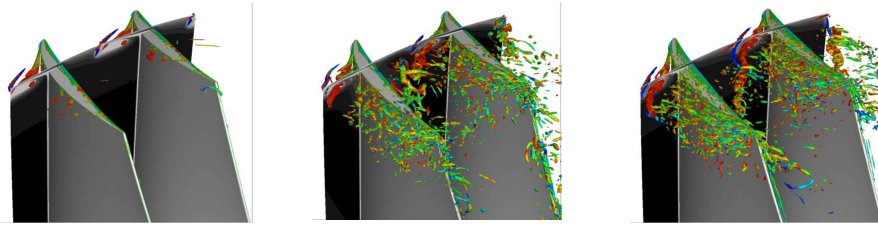


Figure 7: Instantaneous view of the tip leakage flow for (from left to right): M2-SAS-SST, M2-ZDES and M2-SAS-DRSM

The instantaneous entropy variation field at four axial locations for all simulations is depicted in Figure 6, and an instantaneous view of the tip leakage flow is shown for SAS and ZDES computations in Figure 7. The SAS-DRSM approach much better captures turbulent structures than URANS/SST, URANS/DRSM and SAS-SST computations. The global representation of turbulent structures with SAS-DRSM is close to ZDES.

Reference [27] provides a full analysis of the computations by exploring the time-averaged flow in blade passage and downstream of the rotor, as well as the time-averaged radial profiles, and time-averaged overall performances and comparison to experiments. Besides, all power spectral densities (PSD) are depicted (in Figure 8).

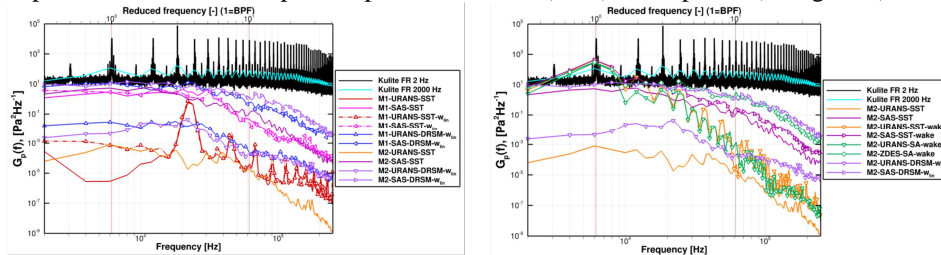


Figure 8: PSD of static pressure for all meshes (left) and only M2 mesh (right)

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 20<sup>th</sup> harmonics) while the first mesh is ten times smaller.

### 3.5 Implementation of a surface roughness-based transition onset correction in the Menter-Langtry transition model

The  $\gamma$ - $Re_{\theta t}$  transition model of Menter [61] gives rather accurate results on turbine blade calculations for smooth cases [42] [20] [62] [63]. However, surface roughness can have a significant effect on boundary layer transition. Indeed, given a certain minimum height, transition will move upstream as the roughness height increases. The reference [28] presents an extension of the  $\gamma$ - $Re_{\theta t}$  model that aims to take into account the effect

of a specific type of surface roughness on the transition location. Stripf roughness correlation [64] correction has been selected for multiple reasons. First, the fact that it takes the form of a correction of the smooth value of  $R_{\theta t}$  allows it to be used in many transition methods in a relatively automatic manner. Second, it was established through experiments on turbine blades, making it well suited to turbomachinery cases. In this study, the  $\gamma$ - $Re_{\theta t}$  model is coupled with the  $k$ - $\omega$  turbulence model of Wilcox [65]. As the transition Reynolds number takes multiple forms in the  $\gamma$ - $Re_{\theta t}$  model, different implementations of the Stripf correction were tested and are discussed in [28]. The best results are obtained by correcting the value of the critical Reynolds number  $R_{\theta C}$ . Computation results on a rough low-pressure turbine vane are presented and compared to experimental results. There is a good agreement on the transition location between the computations and experiments. In particular, the model displays accurate sensitivity to the roughness height, though the influence of the roughness spacing, which is of second order in these experiments, is not as well captured.

### 3.6 Method comparison for fan performance in short intake nacelle

The aim of this numerical study [29] is to demonstrate that elsA is able to assess the strong aerodynamic interactions between the secondary fan/OGV stage with the airframe engine integration system and to determine if intake/fan interactions prevent the use of simplified methods which are available in the software. This study has been conducted in the frame of the European ASPIRE project (in collaboration with Airbus, DLR and NLR).

Several numerical methods have been tested. They range from RANS computations where the engine is modelled using simplified boundary conditions to full  $360^\circ$  URANS computations including the rotating fan blades. Intermediate methods such as Body-Forces and Actuator Disk approaches have also been assessed. The configuration studied is a double-flux isolated nacelle including a UHBR engine (fan/OGV stage). In order to obtain comparable and relevant results, a special mesh procedure is set-up to study the different fan modelling approaches (multi-block structured meshes): (i) the nacelle is meshed independently; (ii) the fan stage zone is left empty to be filled with a specific mesh for each case. Figure 9 shows the nacelle with the 2 AD (the first AD models the fan and a second AD models the OGV), the nacelle with the two Body-Force volumes at the fan (front red volume) and OGV (rear red volume) blades locations and the complete  $360^\circ$  case which includes all engine components (fan, OGV, core, nacelle and nozzle).

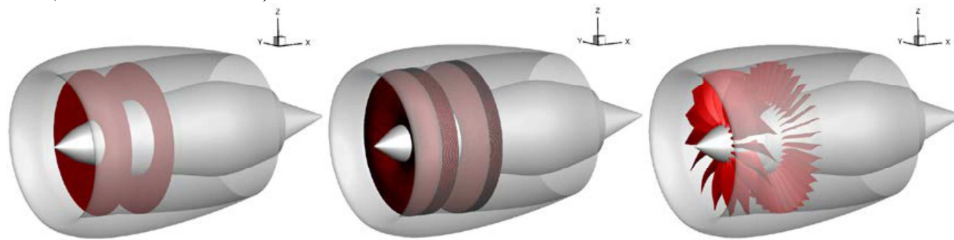


Figure 9: Visualization of the three tested approaches: (a) nacelle with Actuator Disk; (b) nacelle with Body-Forces; (c) nacelle with all fan and OGV blades.

The purpose of using the AD approach is to reduce computational time as required in a pre-design stage, while having a realistic modelling of the fan stage. The BF method consists in applying source terms to the RANS equations to model the effect of blades. In this study, the BF method is used to model the fan and the OGV blades. For RANS  $360^\circ$  computations, a mixing plane boundary condition is applied at the three interfaces (between nacelle and fan domains in the air intake, between fan and OGV domains; between OGV and nacelle domains) and for the URANS computations, an interpolation on the sliding mesh is performed in order to take into the account the unsteady effect. To draw more relevant conclusions, a grid convergence study is performed for all methods with three grid levels. Two operating conditions are considered (transonic cruise conditions and take-off).

In term of global values, these results have shown that in cruise conditions, all methods are able to predict the primary and secondary mass flows with a deviation lower than 0.3% with the medium and fine grids compared to the specifications provided by

Airbus. Regarding the Fan Pressure Ratio (FPR), the accuracy with respect to the specifications is about 0.1% in cruise and 0.3% at low-speed. Finally regarding fan and OGV efficiencies, the fan modelling methods and the grid size have a strong impact. Indeed, with the AD and BF approaches, the deviations are between 2 and 14% compared to the reference URANS computations on the medium grid. The differences between RANS and URANS are rather small.

Radial and surface distributions of static and stagnation pressure have also been compared. Results show that among the simplified approaches only the BF one is able to capture the main tendencies regarding the radial distributions but also the distortion maps downstream of the OGV blades (Figure 10). The 360° RANS computations provide relatively accurate results for the radial distributions but cannot correctly predict the distortion maps downstream of the OGV due to the azimuthal averaging of the flow using the mixing plane boundary condition.

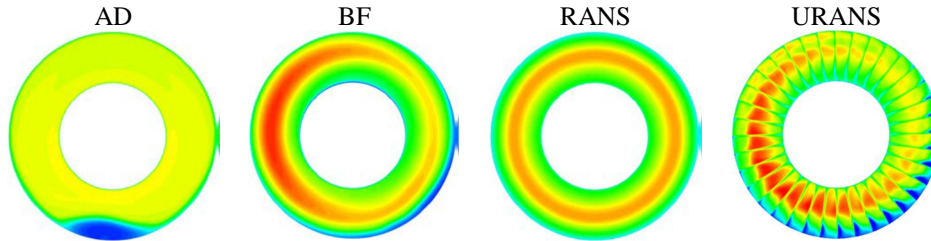


Figure 10: Stagnation Pressure maps downstream of the OGV for the different fan modelling approaches (Medium grid computations) at a low speed condition.

In conclusion, to correctly model the fan in terms of mass flow or FPR, simplified methods like the BF can provide realistic results but 360° computations are required to have a good estimation of the fan and OGV efficiencies. With high distortion levels in the air inlet, the steady RANS computations are not able to correctly transfer these distortions through the fan and OGV stages, which could be critical in off-design conditions (at low-speed or in cross-wind conditions).

### 3.7 Jet noise of a UHBR nozzle using ZDES: external boundary layer thickness and installation effects

In order to compute accurately jet noise, reliable unsteady approaches are needed to simulate the turbulent mechanisms which generate noise. LES has proven to be efficient to do so but this approach is challenging to implement in an industrial framework when dealing with complex geometries for which it does not seem to be reasonable nor relevant to resolve the turbulence production mechanisms in all attached boundary layers (BL). Hybrid RANS/LES methods have been developed to model the attached BL (or at least the internal part) and resolve turbulence in free shear layers.

The ZDES [48] approach available in elsA for years has been used with success for different academic and technical configurations. In [30], ZDES in its automatic mode (namely mode 2, [66]) is used to simulate industrial configurations of jets and evaluate the effect of the external BL thickness and of the wing/flap installation. Very small differences have been observed on the aerodynamic properties of the jet when increasing the nacelle external BL thickness. Figure 11 shows that the thicker external BL leads to a slightly shorter jet.

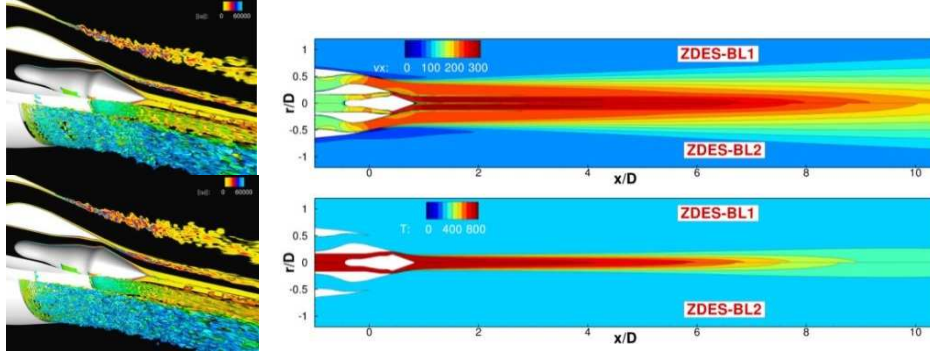


Figure 11: Q criterion isosurfaces and contours of vorticity (left) for the thin (BL1 - above) and thicker (BL2 - below) boundary layer - Streamwise velocity and static temperature fields (right)

An acoustic analysis is performed with the FWH approach and recent ONERA developments allowing the use of closed surfaces. At medium frequencies the results display high noise levels attributed to vortex roll-ups at the nozzle exit. The noise variation between the baseline BL case and the thick BL case varies within a range of  $\pm 0.5$  dB. This low sensitivity of the radiated noise may be attributed to the fact that the baseline BL is already rather thick for this type of flow.

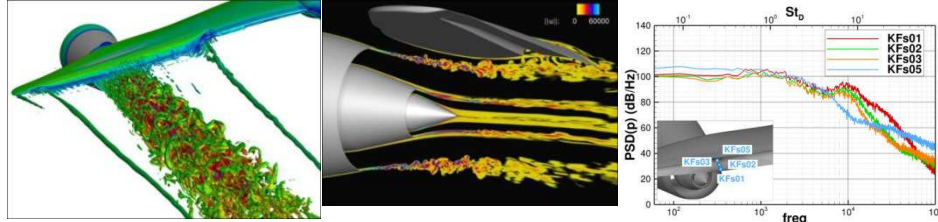


Figure 12: Q criterion isosurface colored by streamwise velocity - Norm of the vorticity (center) - Wall pressure spectra on the upper side of the flap (right)

Q criterion isosurface and norm of the vorticity are plotted in Figure 12 for the installed configuration. Regarding the installation effect it was shown that the jet length is reduced by 30% due to the presence of the wing and flap and that the mean jet direction is deviated due to the jet impinging the trailing edge of the flap (which participates to the jet length reduction). This interaction leads to some increased temperature and pressure fluctuations on the flap which creates noise sources at  $f = 10$  kHz (Figure 12 right). The presence of the wing entails a shielding effect of the acoustic waves and the near-field fluctuating pressure levels are decreased in the direction above the wing. Work related to jet noise is currently devoted to the use of ZDES mode 3 to enable WMLES inside the nozzle and alleviate the RANS-to-LES transition related noise.

### 3.8 Ground effect on fan forced response

The ability to predict the level of response at the design stage allows for implementing design solutions preventing possible failure due to high cycle fatigue and reduces design margins. In [7] the ground effect on the forced response of a large civil fan operating near the ground under crosswind conditions is investigated. Under these conditions a vortex can form between the ground and the intake and can lead to possible failure due to high cycle fatigue. An initial validation of the 3D RANS approach has been carried out against experimental data on an academic intake. The numerical approach has been used to compute the flow field around the intake of the full-scale turbofan demonstrator MASCOT 2 with (Figure 13 –left- shows the flow field topology in the presence of the ground performed with elsA) and without the ground. The distortion induced by the ground vortex has been characterized by comparing total pressure and swirl angle maps obtained for configurations with and without ground. The ground vortex is responsible for a total pressure loss confined at its core and strong positive and negative swirl angles impacting a large radial area.

Both decoupled and coupled approaches have been used to estimate fan forced response on the MASCOT2 at the crossing of the 2<sup>nd</sup> bending mode (2F) and the 4<sup>th</sup> engine order. These approaches are based on aerodynamic URANS calculations. Arbitrary Lagrangian–Eulerian formulation is used to take into account the deformation of the mesh at each time step [54] and linear structural behaviour. The decoupled approach, based on superposition principle and linear aerodynamic behaviour, was used with two twin elsA calculations with different prescribed harmonic motion. The aerodynamic excitation forces due to distortion and the aerodynamic response due to blade motion are then obtained through the analysis of generalized aerodynamic forces from the two twin calculations. The coupled approach is based on temporal integration for which the fluid and the structure are solved iteratively until convergence for each time step. Four steps (mesh deformation, fluid resolution, computations of generalized aerodynamics forces, structure resolution giving displacements) are repeated several times until convergence for the same time step (only 3 loops are necessary) before switching to the next time step. This approach is much more time consuming than the decoupled one. Both approaches have led to very close results which validates the assumptions of superposition and linear aerodynamic behaviour used in the decoupled approach.

The numerical predictions of fan forced response were then compared to experimental results and showed a good agreement for both with and without ground cases. The increase of the vibration level induced by the ground vortex is well captured by the numerical approach. However, the maximum vibration levels are slightly overestimated by the numerical approach (in Figure 13-right).

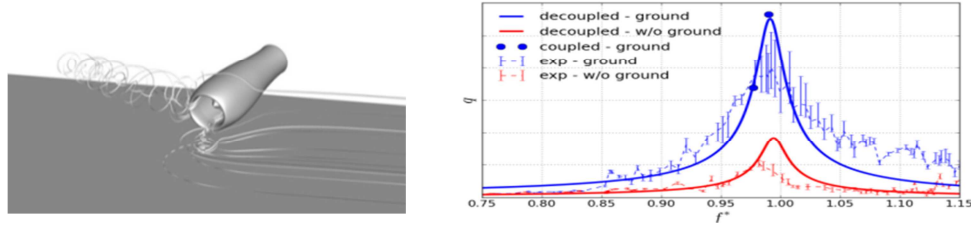


Figure 13: Flow field topology in MASCOT2 case (left) - Forced response for the 2F-4/rev crossing with and without ground (right).

A current work consists in taking into account the non-linear behaviour of the structure in the case of another configuration.

### 3.9 Comparison between coupled CFD/CSM hot shape prediction and AIPX-7 CROR experimental data

In [31] CROR wind tunnel tests results are compared to numerical hot shape (i.e. the deformed shape due to centrifugal forces and aerodynamic loads during the flight) computations. In a previous paper numerical computations were performed using the design shape in both CFD and CSM computations. The current study is a step forward. The blade manufacturing failures (Figure 14 –center) are introduced and the finite element model is tuned considering these new shapes. The flowchart for the CFD/CSM static coupling is shown in Figure 14 (right). Once the flow has converged toward a steady solution, the aerodynamic forces and moments are extracted from the CFD grid to be interpolated on the CSM grid. This is done inside the aeroelastic interface. These structural loads are then injected in the CSM solver (©Nastran) which computes the displacements of the structure submitted to these loads. The aeroelastic interface then interpolates these displacements from CSM grid to CFD grid aeroelastic interfaces and propagates the displacement in the aerodynamic volume mesh using a mesh deformation tool. This process is continued until convergence to equilibrium between the loads and the structural displacements, using a relaxed fixed point procedure.

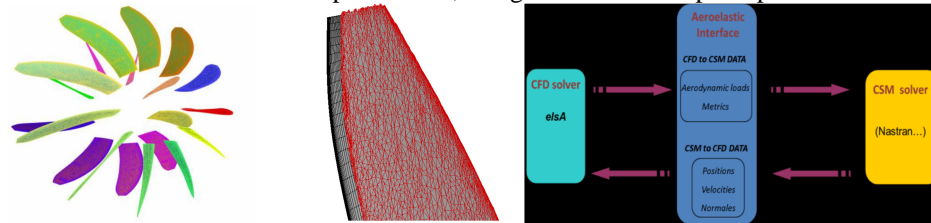


Figure 14: Rotor blades (left) - CAD blade mesh surface in black and manufactured blade mesh surface in red (center) - Static coupling strategy elsA/Nastran (right)

Then, blade displacements and blade pressure distributions are compared. If some differences are still noticeable concerning the blade displacements, this methodology improved a lot the blade pressure distributions correlations with the experimental data. These encouraging results show that manufacturing failures can have a non-negligible impact on the numerical / experimental comparison and must be evaluated and, if necessary, be taken into account.

## 4.0 CONCLUSION AND PERSPECTIVES

The previous section has presented recent challenging turbomachinery studies. That demonstrates that the elsA software has become a very mature and reliable tool for CFD studies, which can be used for industrial design and for advanced research in turbomachinery field.

An important work which revolutionizes the architecture of the software is in progress. The first results in terms of CPU provide very important gains (ensuring exactly the same numerical solution as with the former architecture). One of the test cases is the CREATE axial compressor presented above. For a RANS computation with Spalart-Allmaras model the CPU gains range from a factor of 3 with a structured mesh to a factor bigger than 6 with an unstructured mesh.

The elsA V4.0 release produced in 2018 already contains these achievements for a limited scope of the numerical and modelling parameters present in elsA. Details will be published shortly by the elsA team.

## ACKNOWLEDGMENTS

The studies presented in this paper are making use of the elsA-ONERA software, whose the co-owners are Airbus, Safran, and ONERA. Since 2015, the development and validation of the elsA software are driven by a close research and financial cooperation, as well as a common governance, between ONERA, Airbus and Safran groups.

The author wants to acknowledge all the people which have carried out the simulations whose results are presented in this paper: R. Barrier, T. Berthelon, H. Bézard, A. Burlot, L. Castillon, A. Dugeai, H. Gaible, F. Gand, A. Geeraert, T. J. Marty, Y. Mauffrey, B. Maugars, M. Méheut, B. Michel, A. Minot, J. Perraud, F. Sartor, M. Soismier, M. Vergez, S. Verley.

The author would also like to thank all who have made the elsA software project so successful for many years.

## REFERENCES

- [1] L. CAMBIER, S. HEIB, AND S. PLOT. "The Onera elsA CFD software: input from research and feedback from industry". *Mechanics & Industry*, 14(3):159–174, 2013.
- [2] W. THOLLET, G. DUFOUR, X. CARBONNEAU, F. BLANC, "Body-force modeling for aerodynamic analysis of air intake – fan interactions", September 2016, *International Journal of Numerical Methods for Heat and Fluid Flow* 26(7):2048-2065.
- [3] G. CARRIER, D. DESTARAC, A. DUMONT, M. MÉHEUT, I. SALAH EL DIN, J. PETER AND S. BEN KHELIL, J. BREZILLON, M. PESTANA, "Gradient-Based Aerodynamic Optimization with the elsA Software", *52nd Aerospace Sciences Meeting - AIAA SciTech 2014*, National Harbor, Maryland. January 2014.
- [4] VAN DER WALL, B.; KESSLER, C.; DELRIEUX, Y.; BEAUMIER, P.; GERVAIS, M.; HIRSCH, J-F.; PENGEL, K.; CROZIER, P., "From Aeroacoustics Basic Research to a Modern Low-Noise Rotor Blade", *Journal of the American Helicopter Society*, Volume 62, Number 4, October 2017, pp. 1-16(16)
- [5] LÉON, E. R.; LE PAPE, A.; COSTES, M.; DÉSIDÉRI, J.-A.; ALFANO, D., "Concurrent Aerodynamic Optimization of Rotor Blades Using a Nash Game Method", *Journal of the American Helicopter Society*, Volume 61, Number 2, April 2016, pp. 1-13(13)
- [6] D. GUEGAN, M. SCHVALLINGER, F. JULIENNE, N. GOURDAIN, AND M. GAZAIX. "Three-dimensional Full Annulus Unsteady RANS Simulation of an Integrated Propulsion System", *51st AIAA/SAE/ASEE Joint Propulsion Conference, AIAA Propulsion and Energy Forum*, AIAA 2015-4021.
- [7] T. BERTHELON, A. DUGEAI, J. LANGRIDGE, F. THOUVEREZ, "Ground Effect on Fan Forced Response", *Proc. of the 15th International Symposium on Unsteady Aerodynamics, Aeroacoustics & Aeroelasticity of Turbomachines*, 2018, ISUAAAT15-094
- [8] F. BLONDEL, M. STANCIU, F. LEOEUF, M. LANCE, " Modeling Unsteadiness and Polydispersion in wet steam flows using the quadrature method of moments and a two-equation model", *10<sup>th</sup> European Conference on Turbomachinery Fluid dynamics & Thermodynamics*, April 15-19, 2013, Lappeenranta, Finland.
- [9] STANCIU, M., MARCELET, M., DOREY, J.M., "Numerical Investigation of Condenser Pressure Effect on Last Stage Operation of Low Pressure Wet Steam Turbines", *Conference: ASME Turbo Expo 2013: Turbine Technical Conference and Exposition*, June 2013.
- [10] G. DUFOUR, F. SICOT, G. PUIGT, C. LIAUZUN, A. DUGEAI, "Oscillating-Flap Simulations with the Time-Spectral and Linearized Methods", *AIAA Journal* 48(4):788- 797, April 2010.

- 
- [11] F. SICOT, G. DUFOUR, N. GOURDAIN, "A time-domain harmonic balance method for rotor/stator interactions", *J. of Turbomachinery*, 134(1):011001 (13 pp.), 2012.
- [12] Y. COLIN, H. DENIAU, J.-F. BOUSSUGE, "A robust low speed preconditioning formulation for viscous flow computations", *Computers and Fluids*, 47(1):1-15, 2011
- [13] S. LE BRAS, H. DENIAU, C. BOGEY, G. DAVILLER, "Development of Compressible Large-Eddy Simulations Combining High-Order Schemes and Wall Modeling", *AIAA Journal* Vol. 55, No. 4, pp. 1152-1163, Avril 2017
- [14] N. GOURDAIN, M. MONTAGNAC, F. WLASSOW AND M. GAZAIX, "High-performance Computing to Simulate Large-scale Industrial Flows in Multistage Compressors". *Int. Journal of High Performance Computing Applications*, vol. 24, 2010.
- [15] G. DESPRÉS, G. NGO BOUM, F. LEBOEUF, D. CHALET, P. CHESSE, A. LEFEBVRE, "Simulation of near surge instabilities onset in a turbocharger compressor" - *Proc. of the Inst. Mech. Eng., Part A, J. of Power Energy* 227, 665–673, 2013.
- [16] P. GOUGEON, G. NGO BOUM, "Aerodynamic interactions between a high-pressure turbine and the first low-pressure stator", *J. of Turbomach.* 136, 071010 (11 pages), 2014.
- [17] CADEL A, NGO BOUM G, THOUVEREZ F, DUGEAI A, PARENT M. "Computing Fluid Structure Interaction Coupling Time Spectral Method and Harmonic Balance Method", *ASME Turbo Expo 2017: Turbo. Technical Conference and Exposition*, Volume 7B: Structures and Dynamics, Charlotte, NC, USA, June, 2017
- [18] I LEPLLOT, M LEBORGNE, R SCHNELL, J YIN, G. DELATTRE, F. FALISSARD, J TALBOTEC, "Aero-mechanical optimization of a contra-rotating open rotor and assessment of its aerodynamic and acoustic characteristics", *9th ETC*, Istanbul , mars 2011
- [19] M. MESBAH, J.-F. THOMAS, F. THIRIFAY, R. VIGUIÉ, F. DURIEU, V. ILIOPOULOU, O. DE VRIENDT, "Investigation of compressor blade vibrations due to subharmonic aerodynamic excitations", *13th Int. Symp. on Unsteady Aerodynamics, Aeroacoustics and Aeroelasticity of Turbomachines*, Tokyo, 2012.
- [20] J. BABAJEE, T. ARTS, "Investigation of the laminar separation-induced transition with the  $\gamma$ - $Re_{\theta}$  transition model on one very high-lift low-pressure turbine (T2) and one engine-like scale low-pressure turbine (TX) rotor blades at steady conditions and freestream turbulence", *47th 3AF Int. Symp. of Applied Aerodynamics*, Paris, 2012.
- [21] DUFOUR, G., CARBONNEAU, X., THOLLET, W., BLANC, F, "Body-force modeling for aerodynamic analysis of air intake – fan interactions", *International Journal of Numerical Methods for Heat & Fluid Flow* (2016), 2048, 26
- [22] A. LERAT, P. CINNELLA, B. MICHEL, F. FALISSARD, "High order residual-based compact schemes for aerodynamics and aeroacoustics", *Computers & Fluids*, Vol. 61, pp. 31-38, 2012
- [23] GUÉNOT, D. ; GALLARD, F. ; BREZILLON, J. ; MÉRILLAC, Y., "Aerodynamic optimization of a parametrized engine pylon on a mission path using the adjoint method", June 2018, *ECCM-ECFD Conference 2018*, Glasgow, UK
- [24] K. JOVANOV, R. DE BREUKER, M. ABDALLA, AND C. BLONDEAU. "A linear aerodynamics-based preconditioner for high-fidelity aeroelastic analysis and sensitivity analysis", *17th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference*, AIAA AVIATION Forum, AIAA 2016-4123
- [25] L. CASTILLON, M. SOISMIER, M.C. LE PAPE, B. MAUGARS, B. MICHEL, "A hybrid structured/unstructured grid strategy for the CFD modeling of technological effects on complex turbomachinery applications", *ISABE-2019-24009 Submitted*
- [26] CASTILLON, L., GOURDAIN, N., AND OTTAVY, X., "Multiple-Frequency Phase-Lagged Unsteady Simulations of Experimental Axial Compressor", *J. of Propulsion and Power*, (2015), accessed January 06, 2015, volume 31, issue 1 on pages 444-455.
- [27] MARTY, J., GAIBLE, H., BÉZARD, H., "Assessment of Scale Adaptative Simulation of a Rotor of High Pressure Compressor", *Proceedings of ASME Turbo Expo 2018*, GT 2018, June 5-11, 2018, Oslo, Norway
- [28] MINOT, A., MARTY, J., PERRAUD J. CASALIS G., "Implementation of a surface roughness-based transition onset correction in the Menter-Langtry transition model". *ASME Turbo Expo 2017*, GT2017-63237, June, 2017, Charlotte, NC
- [29] A. BURLLOT, F. SARTOR, M. VERGEZ, M. MÉHEUT, R. BARRIER, "Method comparison for fan performance in short intake nacelle", *Applied Aerodynamics Conference*, AIAA AVIATION Forum, AIAA 2018-4204
- [30] F. GAND, M. HUET, T. LE GARREC AND F. CLÉRO, « Jet noise of a UHBR nozzle using ZDES: external boundary layer thickness and installation effects, in *AIAA AVIATION*

- Forum, 23rd AIAA/CEAS Aeroacoustics Conference, AIAA-2017-3526, 5-9 June 2017, Denver, Colorado.*
- [31] MAUFFREY, Y., GEERAERT, A., VERLEY, S., “Comparison between coupled CFD/CSM hot shape prediction and AIPX-7 CROR experimental data”, AIAA Paper No 2017-3915, *35th AIAA Applied Aerodynamics Conference*, Denver, Colorado, 2017.
- [32] L. CAMBIER, M. GAZAIX, S. HEIB, S. PLOT, M. POINOT, J.-P. VEUILLOT, J.-F. BOUSSUGE, M. MONTAGNAC, “An Overview of the Multi-Purpose elsA Flow Solver”, *Aerospace Lab*, Issue 2, 2011.
- [33] D HUE, S PÉRON, L WIART, O ATINAULT, E GOURNAY, P RAUD, C BENOIT, J. MAYEUR, “Validation of a near-body and off-body grid partitioning methodology for aircraft aerodynamic performance prediction”, *Computers & Fluids* vol. 117, pp. 196-211, 2015.
- [34] C. LIENARD, I. SALAH EL DIN, T. RENAUD, R. FUKARI “RACER high-speed demonstrator: Rotor and rotor-head wake interactions with tail unit”, *American Helicopter Society, 74th Annual Forum*, Phoenix, Arizona, May 14-17, 2018
- [35] J. DECOURS, P. BEAUMIER, W. KHIER, T. KNEISCH, M. VALENTINI, L. VIGEVANO, “Experimental Validation of Tilt-Rotor Aerodynamic Predictions”, *40<sup>th</sup> European Rotorcraft Forum*, September 2-5 2014, Southampton, U.K.
- [36] L. CASTILLON; G. BILLONNET; J. RIOU; S. PÉRON; C. BENOIT, "A Technological Effect Modeling on Complex Turbomachinery Applications With an Overset Grid Numerical Method", *Journal of Turbomachinery*, October 2014; 136(10):101005-101005-11.
- [37] C. LIENARD, R. BOISARD, AND C. DAUDIN, "Aerodynamic behavior of a floating offshore wind turbine", *AIAA Scitech 2019 Forum*, AIAA 2019-1575.
- [38] B. AUPOIX, D. ARNAL, H. BÉZARD, B. CHAOUAT, F. CHEDEVERGNE, S. DECK, V. GLEIZE, P. GRECARD, E. LAROCHE, “Turbulence and transition modeling”, *Aerospace Lab*, Issue 2, 2011.
- [39] H. BÉZARD, T. DARIS, “Calibrating the Length Scale Equation with an Explicit Algebraic Reynolds Stress Constitutive Relation”, *6<sup>th</sup> Int. Symp. on Engineering Turbulence Modelling and Measurements*, W. Rodi and M. Mulas editors, Elsevier, pp. 77-86, 2005.
- [40] DUDA, B. M., MENTER, F. R., ESTEVE, M.-J., HANSEN, T., DECK, S., AND BÉZARD, H., “Numerical investigations on a hot jet in cross flow using scale-resolving simulations”. *7<sup>th</sup> Int. Symposium on Turbulence and Shear Flow Phenomena*, Ottawa, Canada.
- [41] LANGTRY, R. B., AND MENTER, F. R., “Correlation-Based Transition Modeling for Unstructured Parallelized Computational Fluid Dynamics Codes”, *AIAA Journal*, 2009, 47(12), Dec., pp. 2894–2906.
- [42] BENYAHIA, A., CASTILLON, L., AND HOUEVILLE, R., “Prediction of separation-induced transition on high lift low pressure turbine blade”. In *ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition*, pp. 1835–1846.
- [43] J. PERRAUD, A. DURANT, “ Stability-Based Mach Zero to Four Longitudinal Transition Prediction Criterion”, *Journal of Spacecraft and Rockets*, Vol. 53, No 4, pp. 730-742, 2016.
- [44] CONTENT, C., AND HOUEVILLE, R., “Application of the  $\gamma$ - $Re_{\theta t}$  laminar-turbulent transition model in Navier-Stokes computations”. *AIAA, 40th Fluid Dynamics Conference and Exhibit*, 2010-4445, June 2010.
- [45] G. BÉGOU, H. DENIAU, O. VERMEERSCH, G. CASALIS, “Database Approach for Laminar-Turbulent Transition Prediction: Navier-Stokes Compatible Reformulation”, *AIAA Journal*, Vol. 55, No. 11 (2017), pp. 3648-3660.
- [46] L. JECKER, O. VERMEERSCH, H. DENIAU, G. CASALIS, E. CRONER, “A New Laminar Kinetic Energy Model for RANS Simulations of Bypass Transition”, *AIAA Aviation forum 2017*, Denver, U.S.A, 05-09 Juin 2017.
- [47] S. DECK, “Zonal-Detached Eddy Simulation of the Flow over a High-Lift Configuration”, *AIAA Journal*, Vol. 43, pp. 2372-2384, 2005.
- [48] S. DECK, “Recent improvements of the Zonal Detached Eddy Simulation formulation”, *Theoretical and Computational Fluid Dynamics*, Vol. 26, Issue 6, pp 523-550, 2012.
- [49] T. RENAUD, M. COSTES, S. PÉRON, “Computation of GOAHEAD configuration with Chimera assembly”, *Aerospace Science and Technology*, Vol. 19, Issue 1, pp. 50–57, 2012.
- [50] S. PÉRON, C. BENOIT, “Automatic off-body overset adaptive Cartesian mesh method based on an octree approach”, *Journal of Computational Physics*, Vol. 232, Issue 1, pp. 153–173, 2013.
- [51] BENOIT, B., PÉRON, S., AND LANDIER, S., “Cassiopee: A CFD pre- and post-processing tool,” *Aerospace Science and Technology*, Vol. 45, No. Supplement C, 2015, pp. 272 - 283.
- [52] M. SOISMIER, “Stratégie de résolution hybride structurée / non structurée pour la simulation d'effets technologiques en turbomachines”, Thèse de doctorat en Mécanique des fluides Energétique, France, *PhD Thesis*, 2016
- [53] B. ORTUN, “A coupled RANS/lifting-line analysis for modelling the aerodynamics of distributed propulsion”, *AHS Aeromechanics & Design for Transformative Vertical Flight*, San Francisco, CA, January 16-18, 2018



- 
- [54] A. DUGEAI, Y. MAUFFREY, F. SICOT, “Aeroelastic capabilities of the *elsA* solver for rotating machines applications”, *International Forum on Aeroelasticity and Structural Dynamics*, Paris, 2011.
- [55] J. PETER, G. CARRIER, D. BAILLY, P. KLOTZ, M. MARCELET, F. RENAC, “Local and global search methods for design in aeronautics”, *Aerospace Lab*, Issue 2, 2011.
- [56] J. PETER, R. DWIGHT, “Numerical sensitivity analysis for aerodynamic optimization: A survey of approaches”, *Computers & Fluids*, Vol. 39, Issue 3, pp. 373-391, 2010.
- [57] GOURDAIN, N., OTTAVY, X., AND VOULLARMET, A., “Experimental and Numerical Investigation of Unsteady Flows in a High-Speed Three Stages Compressor,” *8th European Turbomachinery Conference*, VKI, Belgium, 2009, pp. 247–266.
- [58] RIÉRA, W., CASTILLON, L., MARTY, J., AND LEBOEUF, F., “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, 2014.
- [59] RIÉRA, W., MARTY, J., CASTILLON, L., AND DECK, S., “Zonal detached-eddy simulation applied to the tipclearance flow in an axial compressor”. *AIAA Journal*, 54(8), August, pp. 2377–2391, 2016.
- [60] MENTER, F., AND EGOROV, Y., “A scale-adaptive simulation model using two-equation models”. In 43rd AIAA Aerospace Sciences Meeting and Exhibit. AIAA 2005-1095. Reno, Nevada, 2005.
- [61] LANGTRY, R. B., AND MENTER, F. R., “Correlation-Based Transition Modeling for Unstructured Parallelized Computational Fluid Dynamics Codes”. *AIAA Journal*, 47(12), Dec., pp. 2894–2906, 2009.
- [62] BENYAHIA, A., CASTILLON, L., AND HOUEVILLE, R., “Prediction of separation-induced transition on high lift low pressure turbine blade”. In ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition, American Society of Mechanical Engineers, pp. 1835–1846, 2011.
- [63] MINOT, A., EL-DIN, I. S., BARRIER, R., BONIFACE, J.-C., AND MARTY, J., “Improvement of laminar-turbulent transition modeling within a low-pressure turbine.”. In ASME Turbo Expo 2016, no. GT2016-57637, American Society of Mechanical Engineers.
- [64] STRIPF, M., SCHULZ, A., BAUER, H.-J., AND WITTIG, S., “Extended Models for Transitional Rough Wall Boundary Layers With Heat Transfer Part II: Model Validation and Benchmarking”. *J. of Turbomachinery*, 2009, 131(3), Apr., pp. 031017–031017–11.
- [65] WILCOX, D. C., “Reassessment of the scale determining equation for advanced turbulence models”. *AIAA Journal*, 1988, 26(11), Nov., pp. 1299–1310.
- [66] P.R. SPALART ET AL., "A new version of Detached-eddy Simulation, resistant to ambiguous grid densities," *Theoretical and Computational Fluid Dynamics*, vol. 20, pp. 181-195, 2006.