DOI: 10.13009/EUCASS2022-7710
9TH EUROPEAN CONFERENCE FOR AERONAUTICS AND SPACE SCIENCES (EUCASS)

Investigation of Boundary Layer Ingestion for Civil Transport Aircraft Engine Installation

GODARD Jean-Luc, BARRIER Raphaël**, MOLTON Pascal***, DAVID Jean-Michel*****
**ONERA*
8 rue des Vertugadins, 92190, Meudon, France, jean-luc.godard@onera.fr
***ONERA*
8 rue des Vertugadins, 92190, Meudon, France, raphael.barrier@onera.fr
****ONERA*
8 rue des Vertugadins, 92190, Meudon, France, pascal.molton@onera.fr
*****ONERA*
29 avenue de la Division Leclerc, 92322, Chatillon, France, jean-michel.david@onera.fr

Abstract
This paper provides an overview of scientific activities performed at ONERA on Boundary Layer Ingestion (BLI) for civil transport aircraft. Four scientific challenges are treated: quantification of aircraft aerodynamic performance with BLI engine installation, prediction of engine fan surge with BLI flow distortion, prediction of flow distortion in an engine intake, prediction of aeroelastic behaviour of engine fan submitted to flow distortion. These activities include numerical and experimental work, with RANS, URANS CFD and aeroelastic computations as well as tests in research and industrial facilities. Some experimental databases are already delivered and compared to computations and preliminary conclusions are presented.

1. Introduction

The increase of civil aircraft engine dimensions, associated to the increase of their by-pass ratio to improve their efficiency, leads to closely-coupled engine integration. For such configuration, it could be interesting to introduce the Boundary Layer Ingestion (BLI) technology. This technology consists in the ingestion of the viscous layers developing on the aircraft by the engine intake. In aerodynamics, two beneficial effects are produced: on one side the ingestion of a flow with lower velocities increases the propulsive efficiency of the engine, and on the other side the aircraft viscous drag is reduced because the flow in the viscous layers is accelerated at the exit of the engine. Different studies have shown that a reduction of aircraft fuel consumption, by 5% up to 10%, could be obtained [1] [2] [3] [4]. But the ingestion of a non-uniform flow by the engine presents also important drawbacks: the fan blades are submitted to a flow presenting distortion, with variation of velocity and energy during the rotation, generating cyclic efforts on the blades. As a consequence, the aerodynamic performance of the blades is reduced and their structural strength can be compromised. This technology could be used for innovative aircraft configurations with engines installed on the rear fuselage, for instance with an annular engine, or with distributed propulsion configurations for tube and wing as well as blended wing body aircraft [5] [6] [7] [8] (Figure *1*).

ONERA NOVA configuration

NASA N3-X configuration

Figure 1: Aircraft innovative configurations with BLI engine installations.

DOI: 10.13009/EUCASS2022-7710

In order to improve its knowledge and competencies on the topic of BLI for transport aircraft, and try to quantify the possible benefits that could deliver this technology, ONERA has launched the SUBLIME project. This project addresses aerodynamics and aeroelasticity purposes only. Four main scientific challenges have been identified and are tackled in the project:

- The quantification of aircraft aerodynamic performance with BLI engine installation in transonic conditions;
- The prediction of surge margin for engine fans in flows presenting distortion due to BLI;
- The prediction of the flow characteristics, in particular distortion, in engine intakes in case of BLI;
- The prediction of risks of aeroelastic instability for engine fans submitted to distortion due to BLI.

Both challenges are treated through theoretical, numerical and experimental activities. For numerical activities, high-fidelity ONERA software are used to solve the RANS, URANS and coupled aero-elastic RANS equations. The experimental activities are performed in research test benches but also in industrial wind-tunnels. The activities are not limited to the validation of existing numerical and experimental tools but also include developments of software and of experimental techniques to better investigate this topic.
This paper presents all activities performed up to now, at mid duration of the project, the main results obtained today and some preliminary conclusions. The remaining activities are also described as well as the objectives to be reached at the end of the SUBLIME project.

2. Aircraft Aerodynamic Performance with BLI

The work performed on the quantification of the aerodynamic performance improvement with BLI includes 3 types of activities:

- Theoretical and numerical developments of post-processing software, analyzing CFD solutions and determining the aerodynamic performance, these software using far-field analysis techniques;
- Experiments in industrial wind-tunnels with a first test on a generic configuration to assess the maximum benefit procured by BLI and a second test on a more advanced configuration to assess the benefit expected for a more realistic shape;
- CFD computations to prepare wind-tunnel tests, in particular to design the model shape and the model support, followed by a numerical restitution of these tests.

ONERA has developed for several years different post-processing software performing the analysis of CFD solutions to determine the aerodynamic performance of a configuration, to deliver a drag breakdown into different physical sources and to localize in the field the sources of drag. These software are based on the so-called far-field drag analysis, using drag balance in different volumes in the flow. The ONERA software can handle Euler or RANS solutions, calculated on multi-block structured, structured with chimera technique or unstructured grids.
The first series of ffd software was based on momentum balance [9] [10]. In addition to the total drag it can evaluate the viscous drag due to viscous layers, the induced drag due to the lift and the wave drag due to shock waves. The initial version of the software could only analyze external flows. Within the SUBLIME project, theoretical and software developments have been done so that this software can analyze internal flows, for instance for turbomachines or for engine nozzles.
In addition to innovative approaches developed for the analysis of BLI configurations (e.g. the power balance method [12]), the more recent ffx software is based on exergy balance i.e. the useful part of the energy [11]. Such decomposition presents the advantage of being able to separate different sources of energy for complex configurations without defining a subjective drag/thrust bookkeeping, for instance if the engine and the airframe are closely coupled, and take into account thermal effects. Different theoretical and software developments have been done within the SUBLIME project to improve the quality of the analysis and to extend the capabilities of the software [13] [14], in particular to be able to treat flows with BLI. The main developments have concerned:

- The treatment at the interfaces between the different computational domains;
- The volumes selection based on physical sensors to calculate the different exergy components;
- The capability to perform an exergy balance in a rotating referential to treat propellers [15].

To illustrate these developments, Figure 2 shows the integration volumes of viscous and wave anergy components for the NASA CRM (Common Research Model) configuration at cruise conditions. Figure 3 shows the capability of the software to treat flows in a rotating referential, illustrated here by a computation of the HAD-1 propeller (design at ONERA for hybrid CTOL/VTOL concept) with a dedicated post-processing showing the exergy provided and consumed by the system or the local thermocompressible exergy in the field.

DOI: 10.13009/EUCASS2022-7710

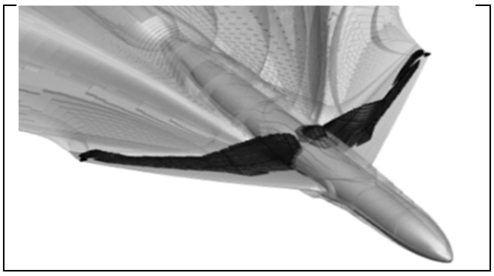

Figure 2: Volumes of integration for the viscous (grey) and wave (black) components of exergy for the NASA-CRM configuration at cruise [13]

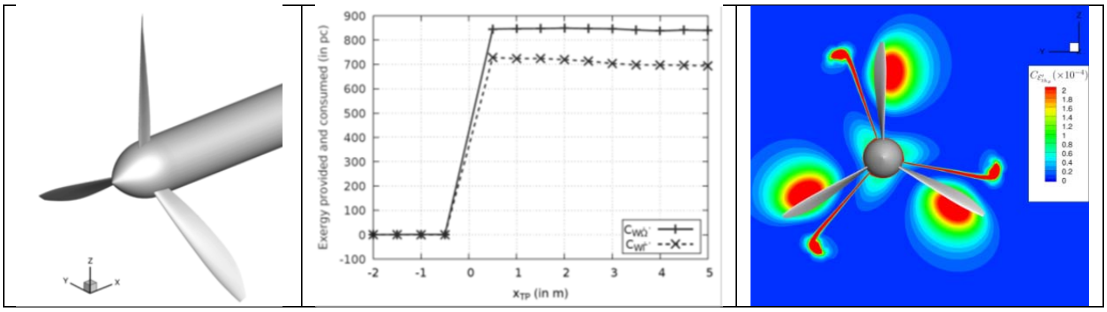

Figure 3: HAD-1 propeller (left), exergy provided and consumed by the system (middle), contours of thermocompressible exergy at propeller level (right) [15]

Two different wind-tunnel tests are planned to be performed in the ONERA S1MA transonic wind-tunnel to evaluate the potential of BLI technology on aircraft aerodynamic performance. S1MA is an atmospheric industrial wind-tunnel, with a test section of 8 m diameter roughly, for subsonic and transonic tests up to Mach number 1 [16]. Previous similar tests have already been performed at ONERA in low speed conditions, and have shown a benefit on aerodynamic efficiency with BLI close to 20% when 100% of the boundary layer is ingested by the engine. The tests at S1MA wind tunnel are aimed at confirming these benefits in transonic conditions.
The first test is aimed at evaluating the maximum benefit that could be obtained on a generic configuration. This configuration is composed of an axis-symmetrical body and an engine simulator placed downstream of the body to swallow its wake. The configuration is installed at the wind-tunnel floor. The efforts on both bodies, on the aircraft simulator and on the simulator fan are measured with different balances to perform a detailed drag analysis. When the simulator is aligned with the body, the engine ingests the maximum portion of the wake, leading to the maximum benefit due to BLI. The lateral position of the body can be modified to change the portion of the wake ingested by the engine. The body shape, the relative position between the body and the engine, and the model support shape have been selected with RANS computations followed by drag balance. The body is 1.50 m long and has a relative thickness of 30%. The engine simulator is placed 0.50 m downstream of the body. The engine simulator was developed outside the SUBLIME project and is representative of a UHBR engine. It is powered by pressurized air and has a fan diameter of 0.27 m. Figure 4 shows a schematic view of the model and of the test set-up in the S1MA wind-tunnel. The test is planned to be performed during year 2023. The test program will include variations of the Mach number from subsonic conditions to Mach number 0.82 roughly, variations of the engine regime rpm and different lateral positions of the engine versus the body.

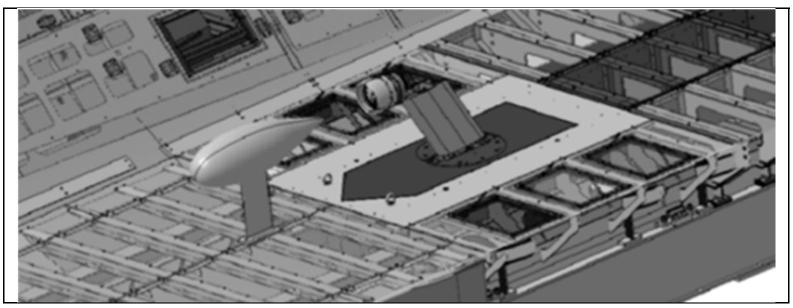

Figure 4: Test set-up for the generic configuration performance test at S1MA.

DOI: 10.13009/EUCASS2022-7710

The second test is done on a more realistic configuration, composed of a body with an engine installation partly buried in it so that the body viscous layer is partly ingested by the engine simulator. For this test, the model is supported by a downstream strut. The same engine simulator as for the previous test will be used. The efforts on the model and on the engine fan are measured with different balances. The engine installation has been designed with RANS computations. This test will deliver an assessment of BLI benefits that could be obtained on a more realistic configuration, closer to a civil aircraft one. Figure 5 shows a schematic view of the model and the set-up in the S1MA wind-tunnel. This test is planned to be performed during year 2024. The test program will include variations of the Mach number from subsonic conditions up to Mach number 0.82 roughly, variations of the model angle of attack and variations of the engine regime rpm.

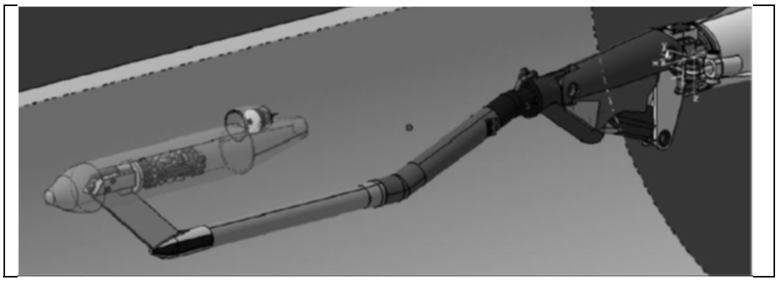

Figure 5: Test set-up for the second configuration performance test at S1MA.

Different RANS simulations of the configurations selected for the tests have been done prior to the tests, using the ONERA-Airbus-SAFRAN CFD solver elsA [17] and the previously described ONERA ffx post-processing software. For the generic configuration test, first computations have been used to design the configuration shape as well as the test set-up. Then, different computations have been done for different variants of the configuration, corresponding to different lateral positions of the engine versus the body, and to different portions of the body wake ingested by the engine simulator. A power saving coefficient PSC can be defined as the difference between the power delivered by the engine without and with BLI, for the thrust/drag equilibrium conditions, divided by the power delivered without BLI. This coefficient is maximum, equal to 7% roughly, when the wake is completely ingested by the engine. It decreases quickly when the engine is moved along the lateral direction because the portion of the wake ingested by the engine is quickly decreasing (Figure 6).
For the second test, different RANS computations have been done to design the engine installation of the wind-tunnel model. The benefit produced by BLI on the aerodynamic performance was also evaluated.
These preliminary performance assessments will have to be confirmed by the tests measurements. After the tests, a more complete numerical campaign will be realized for different model configurations and aerodynamic conditions studied during the tests. Then, a detailed comparison between computations and experiments will be possible in order to draw conclusions on the benefits on aerodynamic performance that can be expected from the BLI technology for civil transport aircraft.

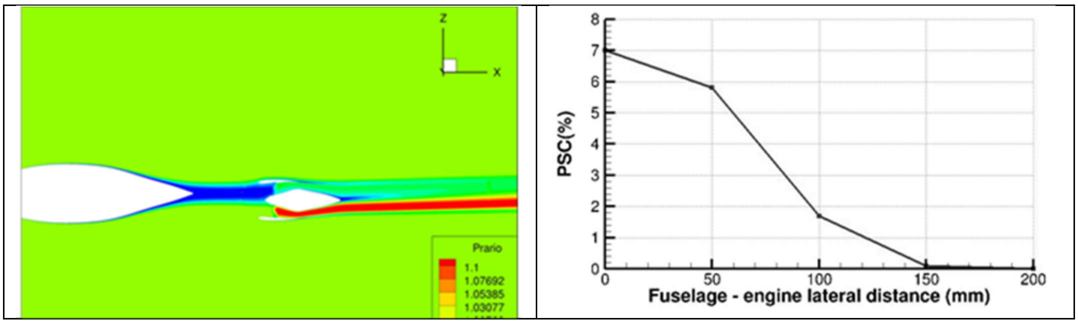

Figure 6: Total pressure around generic configuration at cruise Mach number 0.82 (left) and benefit on engine power according to the lateral position of the engine (right).

DOI: 10.13009/EUCASS2022-7710

3. Engine Fan Surge in case of BLI

The activities on the prediction of surge margin for engine fans in case of BLI are of two types:

- Execution of tests at the LMFL CME2 compressor bench with upstream flow distortion. The objective is to create an experimental database on surge, to better understand the physical phenomena and to evaluate the impact of the upstream flow distortion for different types of topology or intensity of the distortion;
- CFD computations of the CME2 compressor configuration and for the operating conditions of the tests. The first objective of this activity is to validate numerical software by comparison to the test measurements, but the computations can also be useful to better characterize the aerodynamic phenomena.

The CME2 compressor bench from LMFL Lille is a research bench dedicated to the study of fan instability regimes, in particular surge [18]. Its fan diameter is 0.55 m roughly. The fan is equipped with 30 blades and the stator with 40 blades. Its main nominal characteristics are approximately a massflow of 10.5 kg/s, a pressure ratio of 1.15 and a rotation speed of 6 000 rpm (Figure 7).
The activities performed within the SUBLIME project consisted in the test preparation and its execution. The test preparation included bench adaptations, to be able to perform the desired measurements (torque meter installation for power measurement, installation of total pressure probes upstream of the fan for distortion characterization, optical access for endoscopic PIV), and the design and manufacturing of different wireframes that are installed upstream of the fan to generate the flow distortion. The wireframes have angular sector shapes and the geometrical parameters used for their definition are the sector angle and the porosity of the wireframes. These parameters were selected to have a range of flow distortion characteristics representative of what is usually studied in literature. The distortion characteristics can be deduced from the total pressure loss through the wireframe and this was evaluated from parametric laws [19], with verification by comparison to CFD computations for few cases. At last, the selected wireframes have sector angles ranging from 15° to 150° and porosities from 0.1 to 0.5 (Figure 7).

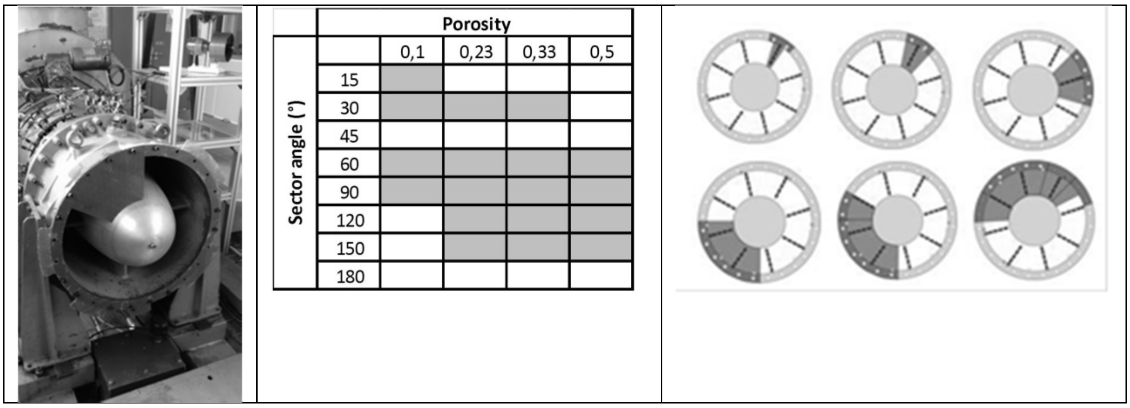

Figure 7: CME2 bench with wireframe (left), wireframe sector angles and porosities (middle and right).

The test has been fully executed and was completed at the beginning of year 2022. The test program included test points without and with flow distortion, using the previously described wireframes. The procedure for the test execution was to fix the velocity to 3 200 rpm and then to decrease the massflow to reach surge conditions. The following measurements have been done during the test: the operating conditions (regime, massflow, …), the torque on the bench shaft, local measurement of total pressure profiles with Pitot rakes and local velocity measurements with the PIV technique.
From the measurements, it is possible to determine the average loss of total pressure generated by the wireframe upstream of the fan and to deduce distortion coefficients. The measurements give also the pressure ratio generated by the fan and the efficiency of the fan for all operating conditions. As an example, Figure 8 presents the average total pressure loss generated by the 90° angle wireframes of different porosities, showing a maximum value close to 2 500 Pa. The same figure shows the total pressure drop due to the fan with the same wireframes. As expected, the pressure drop is reduced when distortion is increased.
From each fan pressure drop curve, and by comparison to the same curve without distortion, it is possible to determine a stall margin value, for one wireframe. Using the measurements from the different wireframes it was possible to generate a global database of surge margin for the different sector angles and porosities of the wireframes used in the test (Figure 9). The evolution of surge margin with the sector angle shows a decrease up to an angle between 60° and 90°, followed by a region with a relatively low variation of surge margin. This behavior has been

usually observed in the past. The critical angle between the two regions depends only on the pressure loss due to the wireframe.
At last, the unsteady pressure measurements around the fan allowed the identification of the type of instability, of spike or modal type. From these measurements it has been observed that the instability is of spike type if the porosity is low and of modal type if the porosity is high.

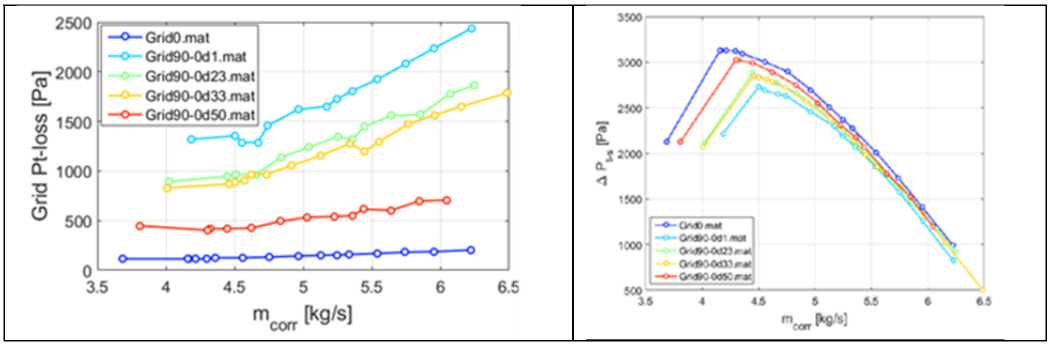

Figure 8: Total pressure loss due to 90° wireframe for porosities 0, 0.1, 0.23, 0.33, 0.50 (left) and fan pressure drop versus corrected massflow (right).

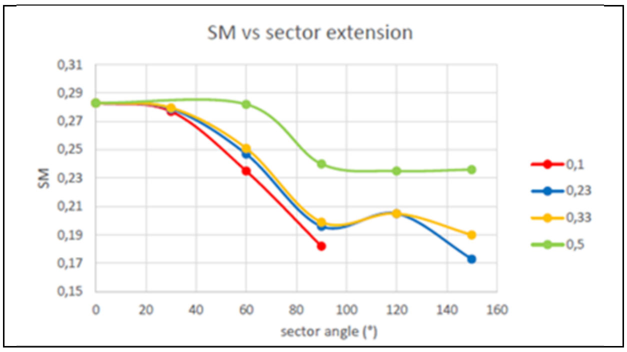

Figure 9: Surge margin versus the wireframe sector angle for the different porosities.

The numerical activity consisted in unsteady RANS computations of the CME2 bench. The configuration considered includes the fan, the stator and the struts installed upstream of the fan. The mesh has been generated with the NUMECA Autogrid5 software [20] and presents 122 million nodes for the 360° full wheel. The mesh around the fan must be axis-symmetrical at its boundaries to be able to perform computations with a rotating fan (Figure 10).

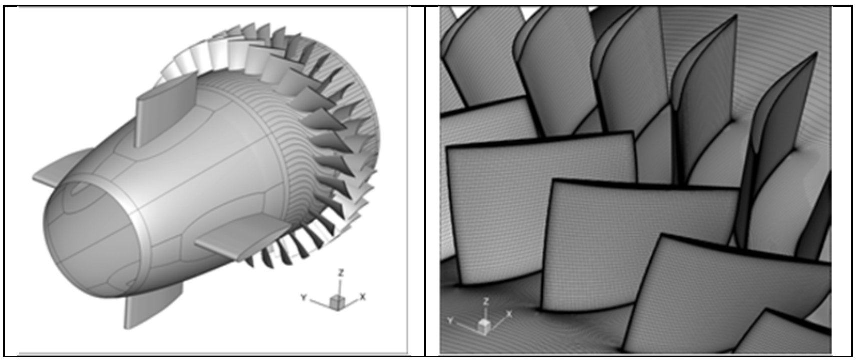

Figure 10: CME2 configuration and detailed view of the mesh (the shroud is blanked in these views).

The URANS computations have been done with the ONERA-Airbus-SAFRAN CFD solver elsA [17], for the fan in rotation. The k-l Smith turbulence model was used in these calculations. An upstream condition on total pressure and temperature and on the turbulent variables k and l was applied, with a turbulence level equal to 2%. A downstream condition on static pressure was applied and a sliding mesh approach was used to take into account the relative

DOI: 10.13009/EUCASS2022-7710

motion of the grids at the rotor/stator interface. First calculations have been performed for a massflow of 4.85 kg/s with or without upstream flow distortion. The convergence was obtained after calculating 6 complete rotations of the fan, with 3600 iterations for each rotation. To illustrate the analysis of the result, the flow solution has been extracted at several points around the fan and close to its leading edge, corresponding to the positions of unsteady pressure probes in the CME2 tests. The numerical unsteady pressure signal is compared to the experimental signal for one probe in Figure 11: this comparison shows a good coherency between tests and computations.

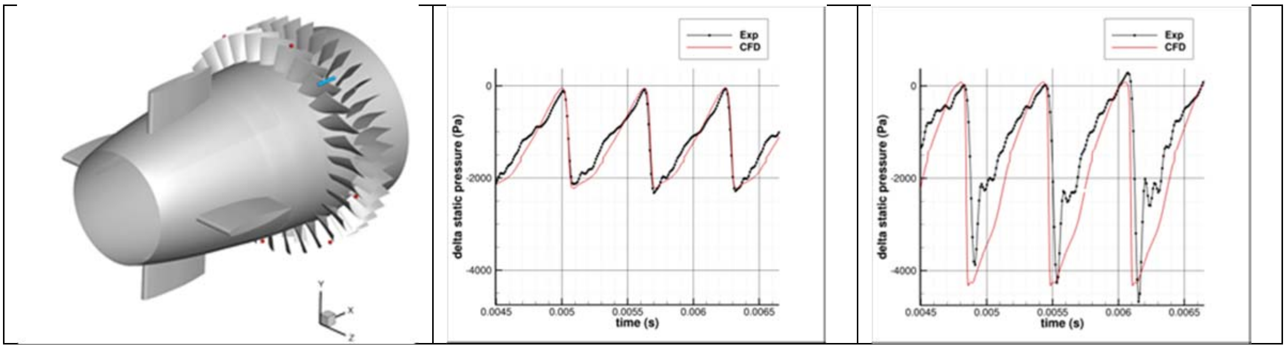

Figure 11: Probes location (left) and comparison of experimental and numerical pressure signals without distortion grid (middle) and with a 60° and 33% porosity grid (right) (pressure probe at shroud near rotor leading edge).

4. Evaluation of Fan Distortion with BLI

The activities on the evaluation of fan distortion with BLI are the following ones:

- Execution of tests at the ONERA S3Ch wind-tunnel on semi-buried engine inlets. The objective is to create an experimental database on flow distortion in engine intakes in case of BLI, for different topologies and intensities of distortion, and to better understand the physical phenomena;
- CFD computations to prepare the previous wind-tunnel test, in particular to design the model shapes, followed by computations after the tests to validate numerical software and to have additional information on the phenomena than the ones delivered by the experiments.

The ONERA S3Ch facility is an atmospheric continuous-flow closed-circuit wind-tunnel that can be operated from Mach number 0.30 to 1.20. Its test section is 0.8 x 0.8 m roughly. It is equipped with a suction circuit, generating flow suction up to 4 kg/s, to simulate for instance an engine massflow. This research wind-tunnel can be easily modified to accept different models set-up such as models supported by a strut or installed at the wind-tunnel walls. Different measurement techniques can be used in this wind tunnel for a detailed analysis of the phenomena, such as pressure measurements on models or velocity measurements in the flow with a PIV technique for instance (Figure 12).

For the SUBLIME project test it was requested to test 2 semi-buried engine intakes in BLI conditions. The intakes must have different geometries (level of burying inside the floor, width) so that they ingest different “quantities” of viscous layers and generate different types of flow distortion. Such simulation can be done at S3Ch by installing the intakes on the lateral wall of the test section so that they ingest the boundary layer developing at the wall. The ingestion is produced by the suction circuit of the wind-tunnel. Due to the size of the test section and the limitation on the suction massflow, models with a fan plane diameter of 164 mm have been selected (Figure 12).

The test program includes Mach number variations from 0.3 to 0.8 roughly. The test set-up has been studied and manufactured so that the model slideslip angle can be adjusted to 0°, 4°, 8° and 12°. The test procedure will be the following one: for a given Mach number and sideslip angle, the massflow will be decreased from the maximum value up to a low value for which an extended separation generating distortion is present inside the intake, by discrete values of the massflow.

The measurement techniques used for the test are steady an unsteady pressure measurements on the model surface, in particular inside the inlet, unsteady total pressure measurements in the intake fan plane with a rake equipped with 40 Kulite sensors, and also velocity measurements in the fan plane with the PIV technique. Two variants of the models will be used for the measurements with the rake and for the PIV measurements, with glasses on certain parts of the models for the PIV measurements (Figure 13). In addition, boundary layer profile measurements are performed upstream of the inlets with a specific motorized total pressure probe.

DOI: 10.13009/EUCASS2022-7710

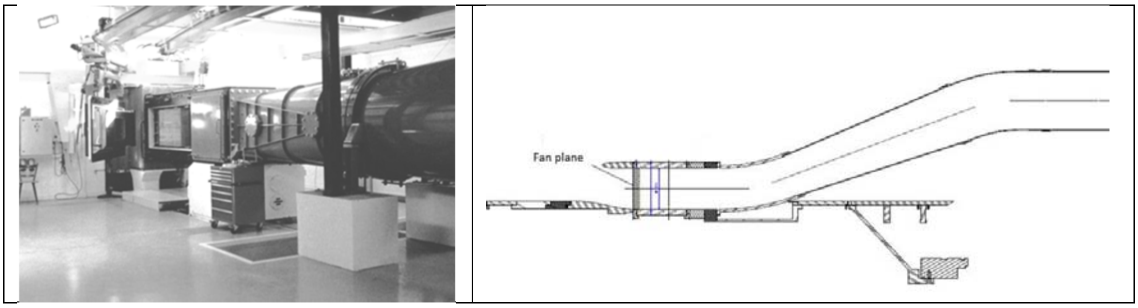

Figure 12: S3Ch wind-tunnel (left) and model set-up with suction circuit (right).

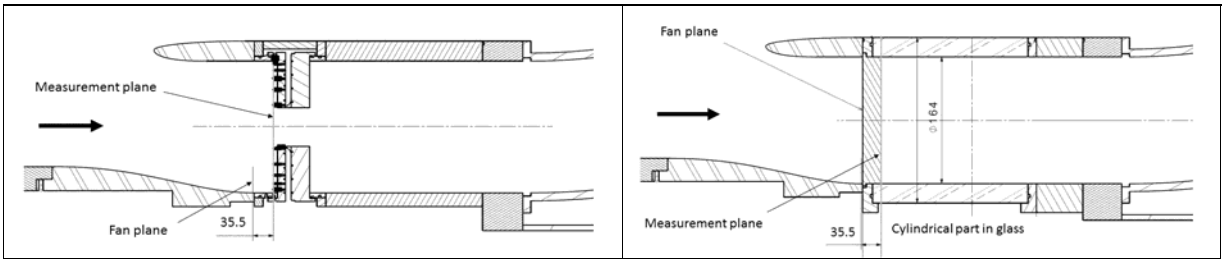

Figure 13: Model for total pressure (left) and PIV (right) measurements.

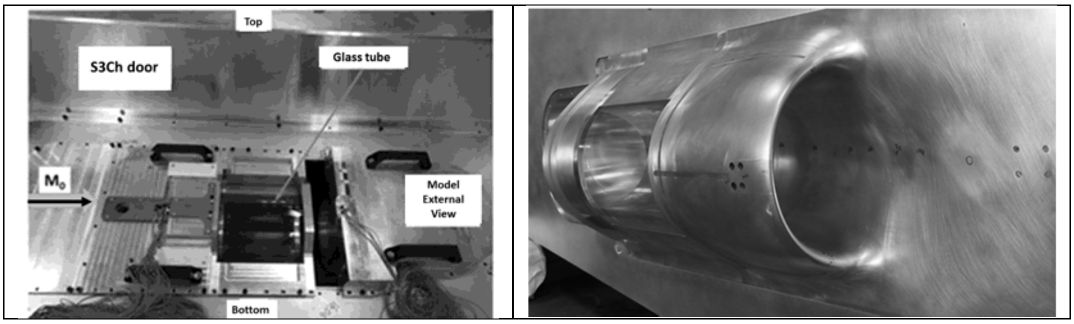

Figure 14: Views of the models installed in the test section.

The numerical activities consist in RANS computations of the semi-buried intakes investigated during the S3Ch wind-tunnel tests. The calculations are performed with the ONERA-Airbus-SAFRAN CFD solver elsA [17].
In a first step, the 2 engine intake models have been designed numerically. The objective was to have a first intake higher and less buried inside the wall and a second inlet lower and more buried, with a bigger width, so that different proportions of the wall boundary layer were ingested by the 2 models, in order to have different aerodynamic phenomena, in particular distortion, inside the intakes. The selected inlets have the following geometrical characteristics (Figure 15):

- Inlet 1: burying level equal to 0.20 (height of the fan under the wall divided by fan diameter), height over fan diameter ratio 0.88, width over fan diameter ratio 1.17, total inlet length over fan diameter ratio 1.50;
- Inlet 2: burying level equal to 0.40, height over fan diameter ration 0.68, width over fan diameter ratio 1.34, total inlet length over fan diameter ratio 2.75.

In a second step and before the tests, different computations have been done on the 2 models for different aerodynamic conditions to be tested. The objective was to determine roughly for which conditions a flow separation was present in the intakes and to compare the aerodynamic phenomena observed in the inlets. The computations clearly show the separated regions for the low inlet massflows. The extent of these regions and the levels of total pressure inside these regions are quite different according to the inlet considered (Figure 16, Figure 17).
At last, for some conditions, RANS computations have been done with different turbulence models (Spalart-Allmaras, Spalart-Allmaras with QCR correction, k-w SST). They have shown different extent of the separation regions and total pressure loss according to the model considered. The distortion coefficients calculated with these flow solutions are consequently different (Figure 18). These results have to be confirmed by the experimental measurements.

DOI: 10.13009/EUCASS2022-7710

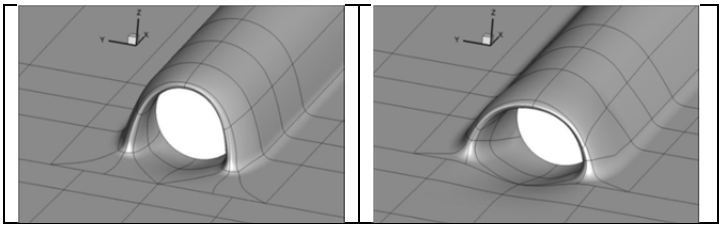

Figure 15: Semi-buried inlets 1 (left) and 2 (right).

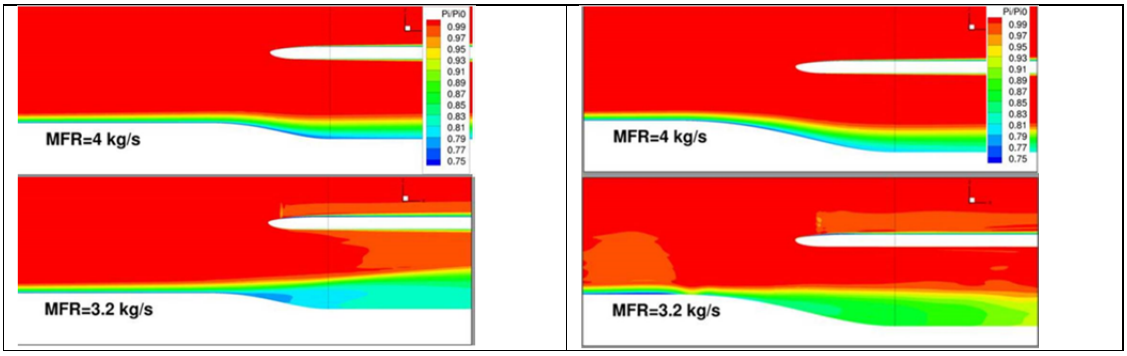

Figure 16: Total pressure in symmetry plane for different inlet massflows for inlet 1 (left) and 2 (right) - Mach 0.82 Sideslip angle 0°.

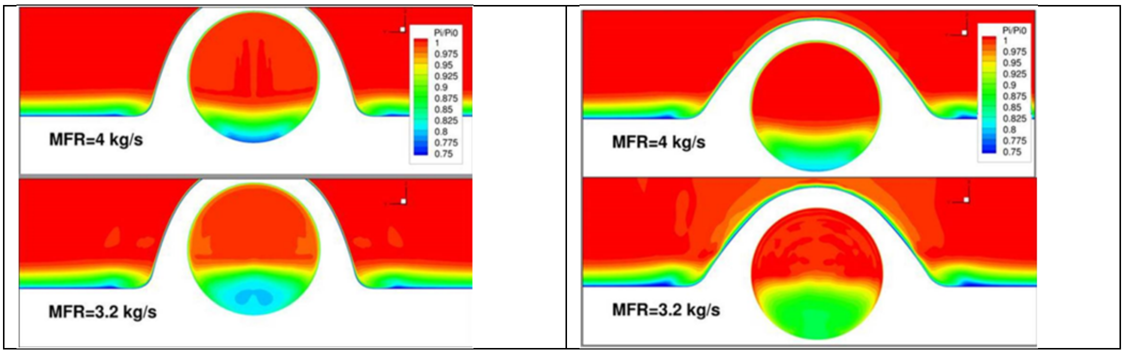

Figure 17: Total pressure in fan plane for different inlet massflows for inlet 1 (left) and 2 (right) - Mach 0.82 Sideslip angle 0°.

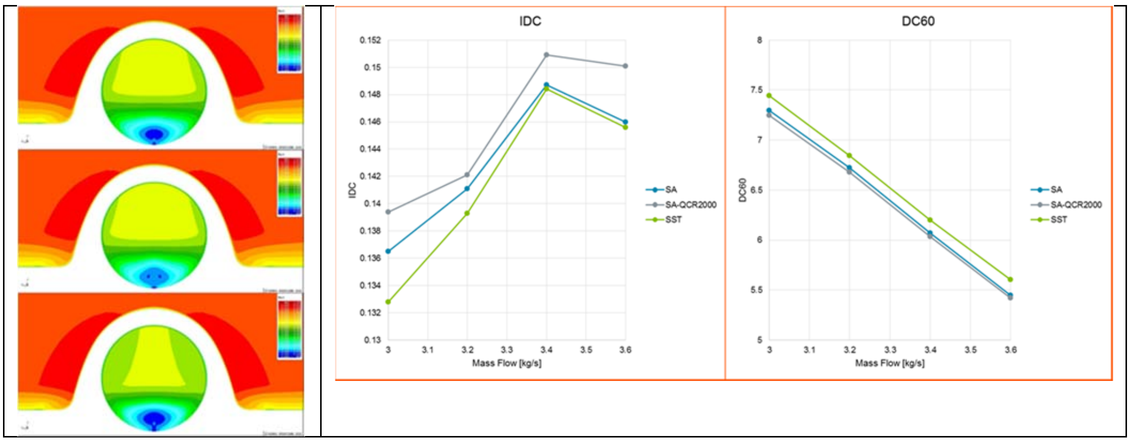

Figure 18: Mach number in fan plane for SA (top), SA with QCR (middle) and k-w SST (bottom) turbulence models (Mach 0.82, massflow 3.2 kg/s) and IDC and DC60 distortion coefficients.

DOI: 10.13009/EUCASS2022-7710

5. Engine Fan Aeroelastic Behaviour for BLI conditions

The different activities to be performed to investigate the aeroelastic behavior of engine fans, when submitted to a flow with distortion due to BLI, are the following ones:

- A numerical study of the aeroelastic behavior of an existing engine simulator to evaluate the risk of instability when it is operating under an upstream flow with distortion due to BLI;
- An aeroelastic wind tunnel test on a big size fan at LMFA ECL-B3 bench, with and without simulations of flow distortion. The objective is to generate an experimental database on aeroelastic instability for fans;
- Aeroelastic computations of the configuration tested at LMFA for a better understanding of the phenomena and for validation of ONERA numerical aeroelastic approach ;
- In parallel, a so-called tip-timing technique has been developed to measure the fan blade deformations during wind-tunnel tests.

An engine simulator has been developed for previous studies on engine installation with UHBR engines. Its nominal characteristics at the Mach number 0.82 are a nominal regime of 24 300 rpm, a power of 340 kW, a pressure ratio of 1.4, a fan massflow of 9.4 kg/s. It represents a turbofan engine of by-pass ratio 16 roughly. It is planned to be used in the S1MA wind-tunnel tests on aerodynamic performance assessment although it was not initially developed for BLI investigations. So, it was decided to conduct a numerical study to evaluate the risk of aeroelastic instability for the fan in case of flow with and without distortion.
The aeroelastic investigation consisted, in a first step, in a study of the forced response of the blades without and with flow distortion. This work was performed numerically with high-fidelity CFD and FEM computations and included:

- Computations of the static behavior of the rotating fan blades for 70% and 100% of the nominal regime, under the rotation and aerodynamic forces. This study has shown that there was no risk of breaking blades due to centrifugal and aerodynamic forces acting together ;
- An analysis of the dynamic behavior of the blades. A Campbell diagram has been established for the different regimes rpm of the fan to check if coincidences could occur between the structural frequencies of the blades in rotation and the frequencies (fundamental and harmonics) of rotation of the fan (Figure 19-left). Possible frequency coincidences could occur for the structural mode 2 (2F) at 70% nominal rpm with the 3rd harmonic (4N) and with the 2nd harmonic (3N) at 100% nominal rpm and also for the structural mode 1 (1F) with the fundamental (1N) and/or 1st harmonic (2N) at the 70% nominal rpm. But 70% nominal rpm regime corresponds to a short duration of the flight and then this situation is not critical for the fan ;
- The determination of the aerodynamic damping of the 3 main structural modes of the blades (Figure 19), at different nodal diameters (n) of the blades for both 70% and 100% of the nominal regimes. The damping coefficients being positive (Figure 19-middle and right), no blade damage is expected due to this phenomenon.

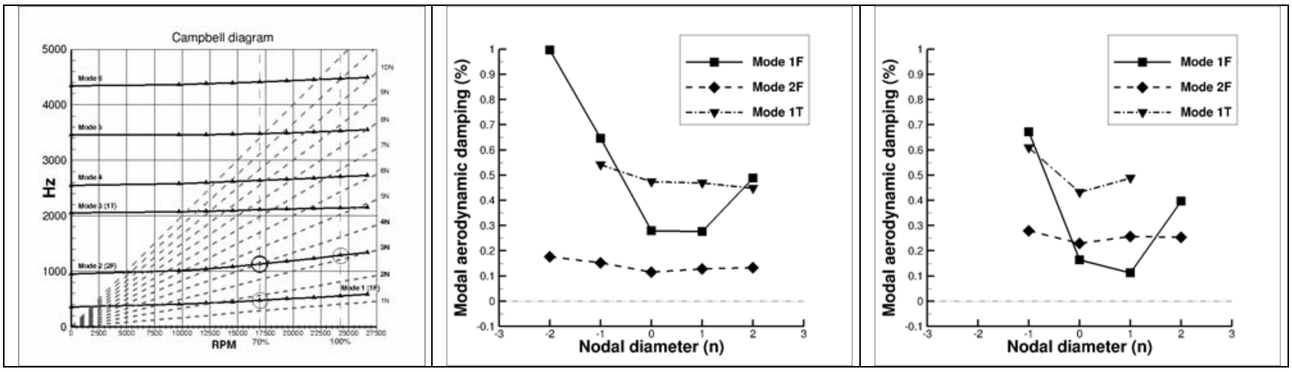

*Figure 19: Campbell diagram for the first six modes of the blade (left) and aerodynamic damping for the first three modes: **without distortion** (middle), **with distortion** (right).*

The aeroelastic investigation consisted, in a second step, in a study of the impact of distortion on the aeroelastic vibratory response stability of the fan. Hifi CFD computations have been performed with the elsA software to determine the distortion in the flow and the unsteady aerodynamic forces on the blades (Figure 20-left). Then, computations of the vibratory response of the fan excited by the unsteady aerodynamic excitations have been done, for the 3 main structural modes of the blades (1F, 2F, 1T) and for the interesting conditions identified from the Campbell diagram analysis. The vibratory response of a blade is first of all calculated mode by mode, for all rotation frequencies (fundamental and its harmonics). The modal blade response corresponds to the crossing between the FRF

DOI: 10.13009/EUCASS2022-7710

of the mode and the rotation frequencies, as illustrated on Figure 20-middle, where blades are excited by six frequencies. The overall blade displacement is then calculated by modal superposition. The maximum blade displacement observed for the two 70% and 100% rpm regimes was close to the blade tip at leading edge and was always lower than 0.5 mm; so this structural blade behavior is not critical for the fan. As an immediate consequence of the vibratory response analysis of the fan under unsteady forces, polycyclic fatigue of the blades was investigated. Polycyclic fatigue of a blade is commonly characterized by the constitution of Haigh diagram (Figure 20-right). This diagram represents the evolution during the rotation cycles of the fan of the dynamic blade stress σ_a (due to the blade vibration) versus the static blade stress σ_m (due to the fan static permanent loading). Domain (σ_m, σ_a) is separated in two areas (whose one is to avoid for the blades) limited by the Goodman curve (this curve is perfectly defined for isotropic materials as Titane, Steel, … and a number of cycles of 10^9).
(σ_m, σ_a) scatter plot observed on (Figure 20-right) for the fan at 100% nominal rpm is finally not critical for the blades after more than 700 operating hours of the fan.

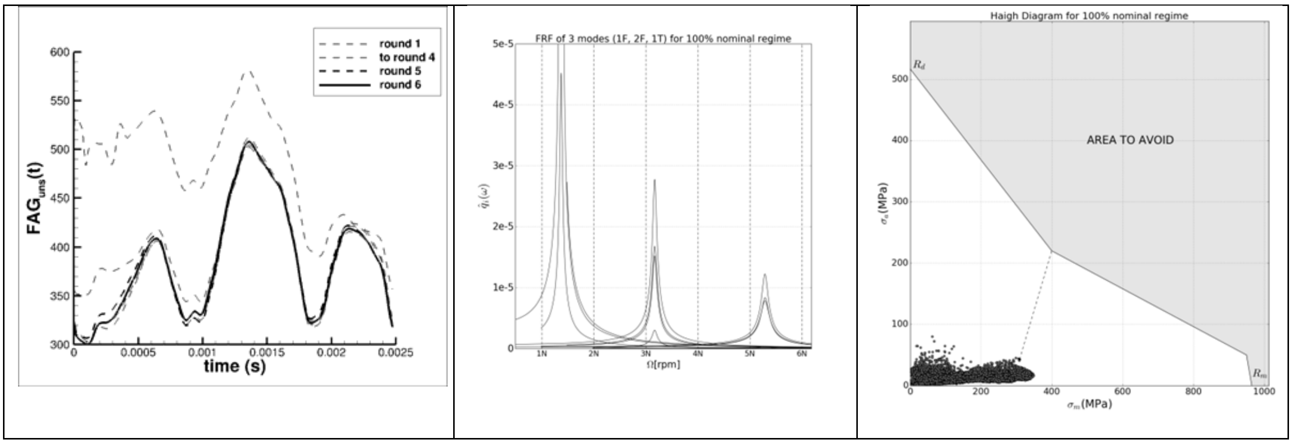

Figure 20: Unsteady force after six fan rotations for the mode 1F, FRF of 3 modes 1F, 2F and 1T and Haigh diagram of blade fatigue, for 100% nominal regime.

An aeroelastic test is planned to be executed in the PHARE bench of the LMFA center [21]. This bench is aimed at investigating multi-physics problems for turbomachines, in particular the phenomena of aerodynamic and aeroelastic instability or aeroacoustic phenomena. Its characteristics are: a power of 3 MW, a maximum pressure ratio of 1.8, a maximum velocity of 16 000 rpm and a maximum massflow of 45 kg/s (Figure 21).
It is planned to tests the ECL5 generic fan, designed by LMFA, this fan having a scale 1/3 roughly compared to modern civil engine fans. The test program includes various conditions with some of them close to surge, and tests with and without upstream flow distortion, this distortion being generated by wireframes installed upstream of the fan. The measurement techniques will include wall pressure and temperature measurements, loads measurements with strain gauges, fan blades deformations measurements and also flow measurements with optical methods. The test is planned to be executed during year 2024. This test will generate a valuable database for a better understanding of the aeroelastic instability close to surge and for the validation of CFD and aeroelastic software.

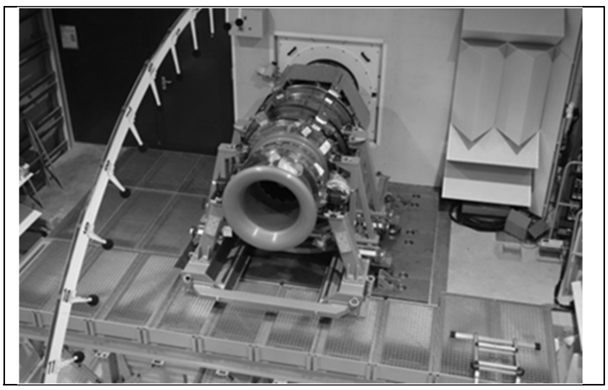

Figure 21: Multi-physics ECL-B3 test bench.

DOI: 10.13009/EUCASS2022-7710

A numerical activity will be conducted for the restitution of the previously described test using the elsA software with the aeroelastic module Ael. The software can simulate steady and unsteady flows around a rotating fan. So, it is possible to determine the steady and unsteady aerodynamic forces acting on the fan blades. It can take into account flow distortion so that the computations can be done in the same conditions than the experiments. The aeroelastic module can determine the local strengths and the vibratory levels of deformations of the fan blades. With all this information it will be possible to perform a precise analysis of the forced response of the fan under distortion. The comparison to the experimental data will be useful to judge the capabilities of prediction and the limitations of the ONERA software for the study of the aeroelastic unstability due to a flow distortion due to BLI.

To complete the activities on fan aeroelastic behavior in case of BLI, an experimental technique to measure the fan blade deformations during wind-tunnel tests has been developed in the SUBLIME project. This so-called “tip-timing” technique is based on the measurements of the blades positions at its tip with optical sensors installed on the fixed part of the inlet around the fan. The analysis of the different signals of the sensors and the data treatment is done by specific software which can finally calculate the deformations of the blades. The software and sensors have not been developed at ONERA but were delivered by suppliers. ONERA have performed the system installation study for its specific needs. Then, different tests have been performed on specific benches to validate this technique and to adapt it the model. The technique being now validated, it will be used during the S1MA wind tunnel tests dedicated to the analysis of the aircraft aerodynamic performance in case of BLI.

6. Conclusion

The SUBLIME project has been set-up to improve the knowledge and the competencies of ONERA on the topic of BLI for transport aircraft. The project is aimed at investigating four main scientific challenges:

- The quantification of aircraft aerodynamic performance with BLI engine installation in transonic conditions;
- The prediction of surge margin for engine fans in flows presenting distortion due to BLI;
- The prediction of the flow characteristics, in particular distortion, in engine intakes in case of BLI;
- The prediction of risks of aeroelastic instability for engine fans submitted to distortion due to BLI.

The challenges are addressed through theoretical, numerical and experimental activities.

Concerning the quantification of the aerodynamic performance with BLI, different developments have been done in the ONERA software *ffd* and *ffx* for the post-processing of CFD solutions and the determination of the far-field drag and exergy components. The numerical activities performed on simplified configurations have shown that a gain of 7% on the engine power can be obtained with BLI in transonic conditions. Different tests in the industrial ONERA S1MA wind-tunnel have been prepared and will be executed in the coming years. They are aimed at confirming the expected gain due to BLI in transonic conditions.

Concerning the prediction of the surge margin for engine fans with flow distortion, an important test has been done in the LMFL CME2 compressor bench, the fan being operated in surge conditions and with upstream flow distortion. A large number of devices, generating various flow distortions, have been used and a wide database on the surge phenomenon has been generated. Numerical activities, with unsteady RANS computations considering a rotating fan, have begun. The first results have shown a correct comparison between computations and experiments. The activities have to be continued before drawing complete conclusions.

Concerning the prediction of flow distortion in the engine fan plane in case of upstream flow with BLI, a numerical design of semi-buried inlets generating different flow distortions has been done. Two wind-tunnel models have been designed and manufactured with these shapes. A test in the ONERA S3Ch research wind-tunnel will be performed at the end of year 2022 to generate an experimental database on flow distortion in the fan plane, with unsteady measurements of total pressure in this plane and velocity measurements. CFD RANS simulations on the same shapes have shown the influence of the turbulence model on the separation extent in the inlet and on the total pressure distribution in the fan plane. The future tests will help to conclude on the most appropriate model to represent this phenomenon.

Concerning the risk of aeroelastic instability for an engine fan submitted to an upstream flow with distortion, a numerical analysis of the behavior of an existing engine simulator has been done. It has been concluded that the simulator can be used during several wind-tunnel tests planned in the SUBLIME project. An aeroelastic test at the LMFA PHARE bench is under preparation. It will generate a database on the aeroelastic instability of a fan under flow distortion. Further high-fidelity aeroelastic computations will be performed to assess the capabilities of ONERA software to predict this phenomenon.

DOI: 10.13009/EUCASS2022-7710

7. Acknowledgments

The author would like to acknowledge French DGAC Directorate for funding these activities. The author would like to thank his ONERA colleagues (JC Abart, O Atinault, D Bailly, R Barrier, I Berhouni, G Billonnet, I Cafarelli, M Carini, JM David, C Illoul, G Losfeld, Y Mauffrey, P Molton, G Outtier, I Petropoulos, JF Séchaud, JP Tobeli, JP Vieira, C Wervaecke), A Dazin from LMFL laboratory and X Ottavy from LMFA laboratory for their contributions to the technical activities within this project.

8. References

[1] Smith L. H. Jr., Wake Ingestion Propulsion Benefit, *Journal of Propulsion and Power*, Vol. 9, No. 1, 1993, pp. 74-82.

[2] Plas, A.P., Sargeant, M.A., Madani, V., Chrichton, D., Greitzer, E.M., Hynes, T.P., Hall, C.A. Performance of a Boundary Layer Ingesting (BLI) Propulsion System, AIAA 2007-450, *45th AIAA Aerospace Sciences Meeting and Exhibit*, 8 - 11 January 2007, Reno, Nevada.

[3] Arend, D.J., Tillmann, G., O'Brien, W.F. Generation After Next Propulsor Research: Robust Design for Embedded Engine Systems, AIAA 2012-4041, *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, 30 July - 01 August 2012, Atlanta, Georgia.

[4] Pandya S. A., Huang A., Espitia A., Uranga A., Computational Assessment of the Boundary Layer Ingesting Nacelle Design of the D8 Aircraft, AIAA 2014-0907, *AIAA SciTech 52nd Aerospace Sciences Meeting*, January 13-17, 2014, National Harbor, Maryland.

[5] Uranga A., and al., Preliminary Experimental Assessment of the Boundary Layer Ingestion Benefit for the D8 Aircraft, *AIAA SciTech 52nd Aerospace Sciences Meeting*, January 13-17, 2014, National Harbor, Maryland.

[6] Wiart L., and al., Aeropropulsive Performance Analysis of the NOVA Configurations, *30th Congress of the International Council of the Aeronautical Sciences*, September 25-30, 2018, DCC, Daejeon, Korea.

[7] Stuckl S., van Toor J., Lobentanzer H., VoltAir – The All Electric Propulsion Concept Platform – A Vision for Atmospheric Friendly Flight, *28th International Congress of the Aeronautical Sciences*, September 23-28, 2012, Brisbane, Australia.

[8] Felder J.L., Brown G.V., Kim H.D., Chu J., Turboelectric Distributed Propulsion in a Hybrid Wing Body Aircraft, *20th International Society for Airbreathing Engines (ISABE 2011)*, September 12-16, 2011, Gothenburg, Sweden.

[9] Destarac, D., Far-Field / Near-Field Drag Balance Applications of Drag Extraction in CFD, *CFD-Based Aircraft Drag Prediction and Reduction*, VKI Lecture Series 2003-02, von Karman Institute for Fluid Dynamics, Rhode-Saint-Genèse, Belgium, 2003.

[10] Van der Vooren, J., and Destarac, D., Drag / Thrust Analysis of Jet-propelled Transonic Transport Aircraft; Definition of Physical Drag Components, *Aerospace Science and Technology*, vol.8(6), pp. 545-556, 2004.

[11] Arntz, A., Civil Aircraft Aero-thermo-propulsive Performance Assessment by an Exergy Analysis of High-fidelity CFD-RANS Flow Solutions, PhD Thesis, Lille 1 University, 2014.

[12] Drela, M., Power Balance in Aerodynamic Flows, AIAA Paper 2009-3762, 2011.

[13] Petropoulos I., Wervaecke C., Bailly D., Derweduwen T. (2019), Numerical investigations of the exergy balance method for aerodynamic performance evaluation, *AIAA Aviation Forum*, AIAA Paper 2019-2926, Dallas, TX, USA.

[14] Wervaecke C., Petropoulos I., Bailly D. (2021), Assessment of exergy analysis of CFD simulations for the evaluation of aero-thermo-propulsive performance of aerial vehicles, *In: Advances in Evolutionary and Deterministic Methods for Design, Optimization and Control in Engineering and Sciences*. Computational Methods in Applied Sciences (pp. 261-275), vol 55. Springer, 2021.

[15] Berhouni, I., Bailly, D., Petropoulos, I., Extension of the Exergy Balance to the Rotating Frames of Reference: Application to a Propeller Configuration, *AIAA SciTech 2022 Forum*, January 3-7, 2022, San Diego, CA, USA.

[16] https://www.onera.fr/fr/windtunnel/s1ma-continuous-flow-wind-tunnel-atmospheric-mach-005-mach-1

[17] Cambier, L., Heib, S., Plot, S., The Onera elsA CFD software: input from research and feedback from industry, *Mechanics & Industry*, Vol. 14, No. 3, 2013, pp. 159-174. https://doi.org/10.1051/meca/2013056, URL https://doi.org/10.1051/meca/2013056.

[18] https://lmfl.cnrs.fr/en/plateforms/rotating-machines/

[19] I.E. Idel'Cik, s.d. Memento des Pertes de Charge. S.I. :s.n.

[20] NUMECA, 2008. *User Manual Autogrid5 Release 8.4: A full automatic hexahedral grid generator for all types of rotating macManual*, s.l.:s.n.

[21] Brandstetter, C., Pagès, V., Duquenne, P., Paoletti, B., Aubert, S., Ottavy, X., Project PHARE-2 – a High-Speed UHBR Fan Test Facility for a New Open-Test Case, *15th International Symposium on Unsteady Aerodynamics, Aeroacoustics & Aeroelasticity of Tubomachines*, September 24-27, 2018, Oxford University, UK.