

Article

Some RANS Modeling Results of the UHBR Fan: The Case of ECL5/CATANA[†]

Lorenzo Pinelli * , Maria Malcaus, Giovanni Giannini and Michele Marconcini 

Department of Industrial Engineering, University of Florence, Via S. Marta 3, 50139 Florence, Italy;
giovanni.giannini@unifi.it (G.G.); michele.marconcini@unifi.it (M.M.)

* Correspondence: lorenzo.pinelli@unifi.it

[†] This manuscript is an extended version of the ETC2025-162 meeting paper published in the Proceedings of the 16th European Turbomachinery Conference, Hannover, Germany, 24–28 March 2025.

Abstract

With the advancement of modern fan architectures, dedicated experimental benchmarks are becoming fundamental to improving the knowledge of flow physics, validating novel CFD methods, and fine-tuning existing methods. In this context the open test case ECL5/CATANA, representative of a modern Ultra High Bypass Ratio (UHBR) architecture, has been designed and experimentally investigated at École Centrale de Lyon (ECL) in a novel test facility with multi-physical instrumentation, providing a large database of high-quality aerodynamic and aeromechanic measurements. In this paper, a thorough numerical study of the fan stage aerodynamics was performed using the CFD TRAF code developed at the University of Florence. Fan stage performance was studied at design speed over the entire operating range. The results were discussed and compared with datasets provided by ECL. Detailed sensitivity on numerical schemes and state-of-the-art turbulence/transition models allowed for the selection of the best numerical setup to perform UHBR fan simulations. Moreover, to have a deeper understanding of the fan stall margin, unsteady simulations were also carried out. The results showed the appearance of blade tip instability, precursor of a rotating stall condition, which may generate non-synchronous blade vibrations.

Keywords: CFD; fan; UHBR



Academic Editor: Antoine Dazin

Received: 11 April 2025

Revised: 11 June 2025

Accepted: 26 June 2025

Published: 23 July 2025

Citation: Pinelli, L.; Malcaus, M.; Giannini, G.; Marconcini, M. Some RANS Modeling Results of the UHBR Fan: The Case of ECL5/CATANA. *Int. J. Turbomach. Propuls. Power* **2025**, *10*, 17. <https://doi.org/10.3390/ijtp10030017>

Copyright: © 2025 by the authors. Published by MDPI on behalf of the EUROTURBO. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY-NC-ND) license (<https://creativecommons.org/licenses/by-nc-nd/4.0/>).

1. Introduction

To cope with the growing demand for air traffic, engine manufacturers have sought to achieve higher efficiency while simultaneously reducing noise emissions.

Increasing the bypass ratio while reducing the fan pressure ratio is essential to increasing efficiency, making the Ultra High Bypass Ratio (UHBR) fan architecture a promising near-term solution. This trend involves increasing the fan diameter and reducing the rotational speed to avoid excessive relative tip Mach number. This can be achieved by coupling the fan and the low-pressure turbine shaft by means of a gearbox. The weight of the gearbox is partly counterbalanced by the use of slender fan blades usually made of composite material.

By employing lightweight 3D-shaped rotor blades featuring a reduced tip Mach number, the main phenomena that restrict the stable operating range differ from those observed in traditional metallic high-speed designs (like NASA rotors 37 and 67 or TU Darmstadt Rotor 1). These differences arise due to various factors:

- Low-speed fans predominantly operate on the flat part of the compression characteristic, making them more susceptible to stall-driven instability [1].
- The flutter frequencies are lower compared with high-speed design. Acoustic liners in the intake, which are designed to attenuate high-frequency noise, do not affect the modes relevant to aeroelastic instability.
- The intake length is shorter for low-speed fans, leading to stronger inflow asymmetry and altered acoustic interactions [2]. This gives rise to stronger broadband excitation and shifted resonance frequencies.
- The relative Mach number and shock strength are lower, and the tip clearance relative to the blade chord and solidity (blade chord length-to-pitch ratio) is smaller than that of the conventional direct-drive architecture and more sensitive to geometric variability [3].
- A strongly non-linear fluid–structure interaction has been observed at low frequencies for fans with low solidity, related to the pressure untwist of the blades. Under transonic inflow conditions, slight deviations of the local stagger angle at the blade tip can cause a fundamentally different shock structure between adjacent blades that affects the stability of different rotor sections [4]. This circumstance affects the applicability of promising methods such as intentional blade mistuning [5] for suppressing the development of circumferential propagating modes.

Furthermore, significant 3D blade features (in particular the forward sweep) are known to be beneficial for aerodynamic performance but emphasize aeroelastic sensitivity for torsional or chord-wise bending modes and particularly non-synchronous vibrations (NSVs). On the other hand, recent research has shown that the use of the anisotropic properties of composites in fans could help to improve both the mechanics and aerodynamics [6,7]. Composites also present the potential to control flutter by modification of the eigenmodes [8].

In order to enable further technological advancements in this direction, extensive research is necessary to identify and characterize the relevant instability mechanisms for the novel type of low-speed fan. In particular, the complex flow structure at part load and part speed is challenging for state-of-the-art numerical approaches and requires experimental benchmark data on representative geometries.

Test cases of representative geometries without industrial restrictions are a key element of an open scientific culture. In order to provide a multi-physical validation benchmark representative of near-future UHBR fan concepts, the open-test-case fan stage ECL5 was developed at École Centrale de Lyon (ECL) and was investigated within the European Clean Sky 2 project CATANA (Composite AeroelasTics ANd Aeroacoustics), with a focus on non-synchronous coupling mechanisms among aerodynamics, acoustics, and structure dynamics.

The aim of this work is to numerically investigate the ECL5 fan stage from an aerodynamic point of view. In particular, the fan stage is studied at design speed. All the numerical simulations are performed using the CFD TRAF code [9], an in-house solver that has been developed for more than 30 years at the University of Florence. Moreover, this aerodynamic study represents a fundamental step for further aeromechanical analyses planned in the near future. The aerodynamic analysis compares the influence of a transition model to the assumption of fully turbulent flow, and evaluates the application of a turbulence model correction based on the helicity [10]. The results are compared with the experimental and numerical data provided by ECL. Finally, URANS simulations are carried out to examine the flow behavior near the stability limit in order to gain an initial insight into the possible flow instability that may trigger unwanted non-synchronous vibrations.

2. Test Case

The fan stage has been designed at École Centrale de Lyon (ECL) with the intention to be representative of near-future composite low-speed fans in the following terms:

- General aerodynamic design parameters (Mach number, blade loading, solidity, aspect ratio, hub-to-tip ratio, mass flow rate, etc.).
- Aerodynamic flow structures due to their influence on instability mechanisms (shock patterns, radial flow migration, secondary flows, separations, etc.).

The main design parameters are summarized in Table 1.

Table 1. Fan stage design parameters [11].

Fan Geometric and Operating Parameters	
Rotor diameter	508 mm
Number of rotor blades	16
Number of stator vanes	31
Rotational speed	11,000 rpm
Blade tip speed	288 m/s
Tip Mach number	1.02
Mass flow rate	36.0 kg/s
Total pressure ratio	1.35
Isentropic efficiency	92.6%

The stage consists of 16 fan blades and 31 outlet guide vanes (OGVs). The fan blades are made of unidirectional carbon fibers and epoxy composite plies [12]. The blade external surface consists of only one ply, resulting in very low surface roughness without any sharp step. The OGVs consist of conventionally manufactured aluminum vanes. No specific design criteria were set for the OGVs, except for a minimization of the numerical corner separation under highly loaded conditions and minimum losses at the design point. They are located far downstream of the fan to minimize interactions, as the focus of the research project was on the fan itself.

As the facility comprises only a single flow channel, engine-representative OGV aerodynamics are not considered; only axial stage outflow and homogeneous radial conditions are required to ensure detailed performance analysis.

Test Facility, Instrumentation, and Experimental Procedures

The experiments were performed at the test facility ECL-B3 at École Centrale de Lyon. A schematic of the test cell is presented in Figure 1.

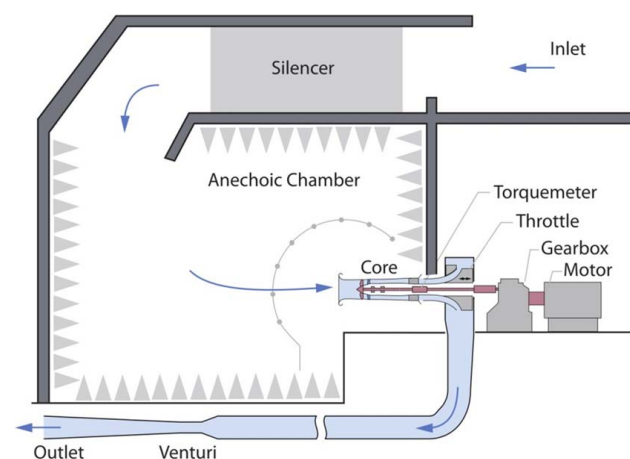


Figure 1. Test cell schematic [13].

A gearbox, placed between the motor and the rotor shaft, enables rotational speeds of up to 16,000 rpm. A torque meter, located between the fan and the gearbox, allows for the measurement of the applied power.

The machine is operated in an open cycle (see Figure 1). Air is sucked in from the roof and flows through a set of silencers and into the anechoic chamber containing the machine. After passing the core, where the fan stage is situated, the compressed air passes an axisymmetric cone throttle that enables accurate control of the mass flow. In order to measure the mass flow, a Venturi nozzle is installed after a 25 m long circular tube section. Finally, the air passes through a diffuser and another set of silencers and then is extracted to the outside environment.

Inside the anechoic test chamber, a traversable microphone antenna allows for the localization and quantification of noise sources in the machine. A turbulence control screen is installed in front of the machine core to ensure homogeneous inflow conditions and reduce large-scale turbulence [14].

The inner core design is depicted in Figure 2. To reduce complexity, the setup does not separate bypass and core flow. Instead, the OGV has been designed to provide representative conditions at the rotor exit with just a single flow channel. Instrumented rakes are placed both upstream and downstream of the stage to conduct performance measurements.

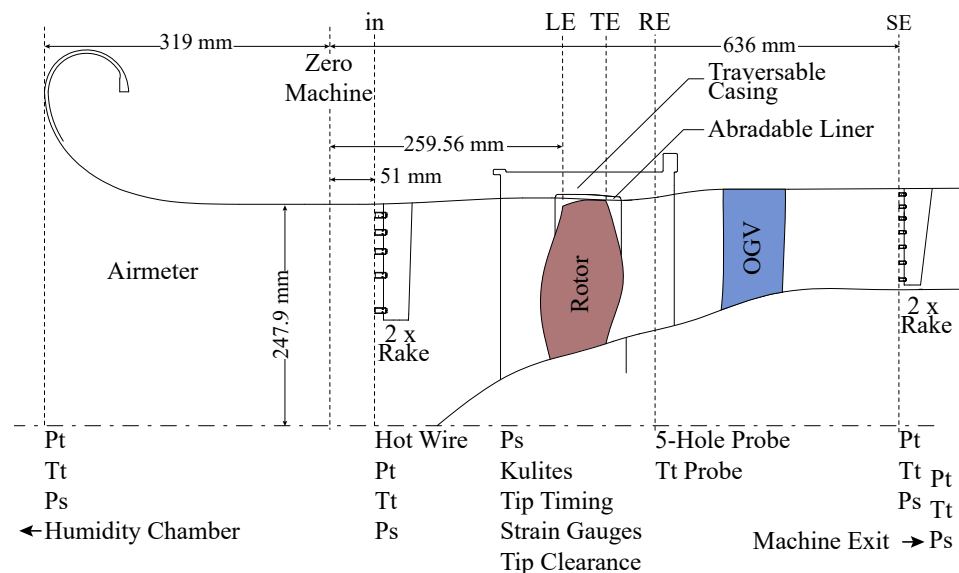


Figure 2. Meridional view of machine core and probe positions [11].

In the machine intake (in), two sets of rakes are integrated to measure total pressure $p_{t,in}$ and total temperature $T_{t,in}$. These parameters are obtained by mass-averaging values from different radial measurement positions, denoted by $\overline{(\cdot)}$. The mass flow rate $\dot{m}_{std}$ through the fan stage is determined using static pressure $p_{s,in}$, total pressure, and total temperature in the stage intake. Humidity and atmospheric conditions are measured in the chamber, allowing for the humidity correction of gas constants and the adjustments of the mass flow rate to International Standard Atmosphere (ISA) conditions. When the stage is operated under stable conditions, downstream of the stage (“SE” in Figure 2), two rakes separated by 180° are traversed around the machine axis to obtain total pressure maps.

To calculate the fan stage characteristics, the static pressure is measured at both the hub and the casing and is assumed to evolve linearly across the channel height to allow for

the mass-weighted averaging of the total pressure ratio $\Pi_{t,stage}$ and stage efficiency η_{stage} based on shaft power P_{mec} :

$$\Pi_{t,stage} = \frac{\overline{\overline{p}}_{t,SE}}{\overline{\overline{p}}_{t,in}} \quad (1)$$

$$\eta_{stage} = c_p T_t \dot{m} \frac{(\Pi_{t,stage})^{\frac{\gamma-1}{\gamma}} - 1}{P_{mec}} \quad (2)$$

At selected operating points, 5-hole probe and total temperature measurements are used to determine the radial profiles of the total pressure ratio, flow angle, and total temperature between rotor and OGV (“RE” in Figure 2). Unsteady instrumentation in several axial and circumferential positions is used to resolve machine and rotor aerodynamics and aeroelastic behavior. Fast-response wall pressure transducers (WPT, Kulite XCS-062) in multiple axial and circumferential positions allow for the detailed analysis of rotor tip aerodynamics, shock structure under transonic operating conditions, circumferential flow inhomogeneities and convective aerodynamic disturbances, and stall onset.

3. Numerical Framework

3.1. TRAF Code

All the calculations were carried out using the TRAF code, a 3D steady/unsteady, viscous, multi-row, multi-block RANS solver developed at the University of Florence [9].

The code was designed for turbomachinery flow predictions and includes several techniques to achieve computational efficiency and accuracy. Different turbulence closures have been implemented in the code, ranging from algebraic to more complex one- and two-equation models. The viscous terms are discretized using second-order accurate central differences, whereas for the inviscid fluxes, two different options are available: a second-order cell-centered scheme and Roe’s upwind scheme. For the upwind scheme [15], a higher order of spatial accuracy is achieved through a MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) extrapolation scheme. To avoid numerical instabilities, a TVD (Total Variation Diminishing) scheme is applied [16]. A generalized minmod limiter is available, which, through the setting of a coefficient, can present various levels of dissipation.

3.2. Domain Discretization

The computational grids were generated by using in-house tools. The grids adopted in this study were H-type structured meshes, selected for their versatility to adapt to the significant twisting of the fan blades. The grid wall spacing was selected to provide a y^+ value lower than 2.0 for the first grid point above the wall and a growing rate of around 1.2.

A single passage was modeled with four blocks, one upstream of the stage, one for the fan, one for the OGVs, and one downstream of the stage, as shown in Figure 3.

The first block consists of 2.5×10^5 cells, the fan block of 5.5×10^6 cells, the OGV block of 2.8×10^6 cells, and the last block of 3.8×10^5 cells. Each block is discretized with 160 cells in the radial direction. The mesh size was selected on the basis of grid sensitivity analyses carried out in previous studies (e.g., [17,18]).

The strategy used to model the fan tip clearance region is commonly referred to as “open tip” [19]. It involves extending the grid from the blade tip to the casing while maintaining the tangential blade thickness and passing the fluxes between pressure-side and suction-side cells in this region (12 radial cells were included in the clearance area). Periodicity boundary conditions are enforced between the two airfoil sides within the clearance region: the values of all the conservative variables for each phantom cell on one side of the grid are interpolated, at the same axial position, from the corresponding cells on

the other side. Considering the very low thickness of the blades at the tip, the use of the open tip model was deemed sufficiently accurate.

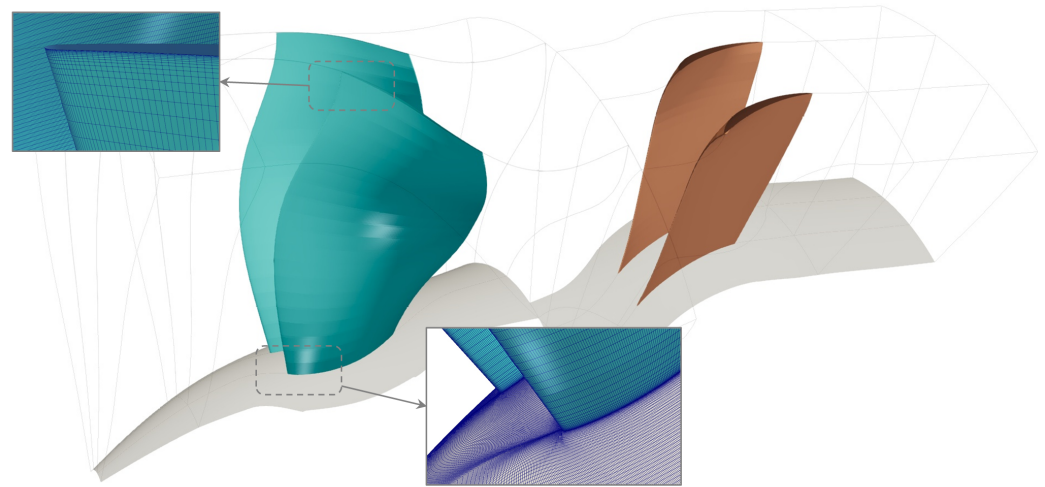


Figure 3. Computational domain and mesh details.

The inlet section of the numerical domain was placed 105 mm upstream of the fan LE at the hub. The interface between the fan block and the OGV block was located at the rotor exit (“RE” in Figure 2). The domain outlet was placed at the stator exit (“SE” in Figure 2). The inlet section could not be placed at the experimental inlet (“in” in Figure 2), because the structured grid would have led to collapsed cells at the edge of the nose cone. It was assumed that the flow between the experimental inlet and the numerical inlet was subjected to minimal variations. The boundary layer thickness growth, from the experimental inlet to the numerical inlet, was taken into account for the inlet boundary condition.

3.3. Boundary Conditions

The boundary conditions prescribed at the inlet section coming from the experimental measurements were the following:

- A yaw angle equal to zero.
- A pitch angle equal to zero.
- A constant total temperature equal to 288.15 K.
- Span-wise distribution of total pressure to take into account the endwalls boundary layer: the total pressure value at mid-span was 101,320 Pa.
- A free-stream turbulence level equal to 1%.
- A turbulence length scale equal to 0.8% of the fan chord at mid-span.

At the outlet section, a static pressure value was imposed at the hub, and the span-wise distribution was established by radial equilibrium. The static pressure value was varied across calculations to simulate the throttling along the speedline from choke to stall conditions. All the simulations were run at fan design speed, i.e., 11,000 rpm.

3.4. Numerical Setup

Steady-state simulations were performed on a single blade passage by using a mixing plane model with non-reflecting treatment to couple the fan and the OGV domains. For the discretization of the inviscid fluxes, the second-order Roe’s upwind scheme with MUSCL extrapolation in conjunction with a TVD scheme with dissipation comparable to the van Albada slope limiter was applied.

The impact of different turbulence closures was assessed. The baseline turbulence model was the high-Reynolds version of Wilcox’s $k - \omega$ model [20]. A helicity-corrected ver-

sion of the baseline model, recently used to predict the performance of transonic centrifugal impellers [10,21], was also used for comparison.

The possible presence of laminar flow was investigated by coupling the $k - \omega$ model, with and without the helicity correction, and the Langtry–Menter $\gamma - Re_\theta$ transition model [22]. The mesh spacing near the wall allowed for wall integration and avoided the use of wall functions. For all the simulations the perfect gas hypothesis was assumed.

Time-accurate calculations were performed using a dual-time-stepping approach. A single simulation period, corresponding to a complete fan rotation, was divided into 1550 time steps in order to capture up to the third harmonic of the blade passing frequency, leading to $\Delta t = 0.0035$ ms between physical time steps. Between each time step, the solution is advanced in a non-physical time domain, employing convergence acceleration strategies such as local time stepping, implicit residual smoothing, and multi-grid techniques to speed up the residual reduction.

The circumferential periodicity conditions were imposed using the full annulus approach, so that the whole wheel was simulated. Non-reflecting boundary conditions together with sponge zone treatments were imposed at the inlet/outlet to avoid spurious reflections. The criterion for considering the periodic convergence of the unsteady simulation was based on the evolution of the blade lift coefficient spectrum between consecutive periods.

4. Steady Results

4.1. Fully Turbulent, Standard $k - \omega$ Model

Figure 4 compares the TRAF speedline obtained with the baseline fully turbulent model with experimental and numerical results from ECL and numerical results from Imperial College and Turbostream.

ECL numerical simulations employed a structured mesh of a single passage, created using AutoGrid5, with 4.4×10^6 points. The wall resolution of the mesh was below $y^+ = 1$ for design conditions. The $k - \omega$ Kok model was used, and no transition model was applied. Imperial College used the URANS code AU3D with an unstructured 1.7×10^6 -cell mesh and wall modeling, while Turbostream simulations employed wall functions on a mesh with 3.1×10^6 cells in the rotor and 0.94×10^6 in the stator [11].

The total pressure ratio was calculated as in Equation (1). The experimental value of $\Pi_{t,stage}$ was based on the values measured by the rakes located at stage inlet and exit, which do not capture the region near the endwalls. To improve comparability, the ECL and TRAF numerical values of $\Pi_{t,stage}$ do not take into account the region close to the hub and the casing.

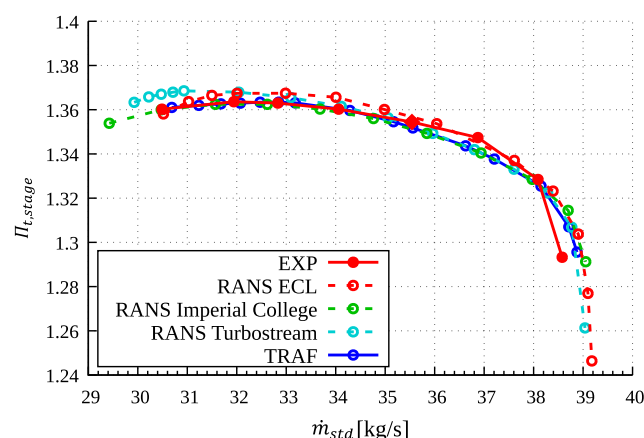


Figure 4. Fan stage speedline: comparison with results from the research community [11].

The TRAF speedline shows good agreement with the experimental speedline, especially toward lower mass flow rates. In the knee region, a lower total pressure ratio is reasonably predicted, whereas in the near-choke region, the mass flow rate is overestimated. The ECL numerical speedline overestimates the total pressure ratio near stall, is in good accordance in the knee region, and overestimates the mass flow rate near choke. The Imperial College speedline is very similar to the TRAF speedline. The Turbostream simulations overestimate the total pressure ratio near stall; otherwise, they are extremely close to the TRAF calculations.

All the simulations overestimate the mass flow rate in the near-choke region. The mass flow rate overestimation near choke can be explained considering that the numerical simulations all employ the average stagger angle of 60° calculated by FEM for peak efficiency conditions, while the experimental measurements show that under the near-choke condition, the stagger angle increases to almost 63° . A lower stagger angle corresponds to decreased incidence, resulting in an increased mass flow rate [11].

4.2. Impact of Boundary Layer Transition

As the turbulence level at the inlet is relatively low, since the air is sucked in from an anechoic chamber, it is reasonable to assume that the boundary layer transition may play an important role in the flow characterization.

The boundary layer transition can have a significant effect on blade performance, both on operating range and efficiency. A laminar boundary layer results in lower profile losses compared with a turbulent boundary layer. On the other hand, boundary layer separation is more common in laminar flows due to the limited momentum transport across the boundary layer, and depending on the extent of the separated flow, the loss increase can be very high. Under the “fully turbulent” hypothesis, the boundary layer is considered to be turbulent for the entire length of the blade.

Figure 5 shows that the speedline obtained by using the Langtry–Menter $\gamma - Re_\theta$ transition model is in better agreement with the experimental speedline in the knee region, while in the region to the left of the DP, the predicted total pressure ratio is lower than the measured one. The transitional boundary layer simulations show numerical instabilities in the lower-mass-flow-rate region.

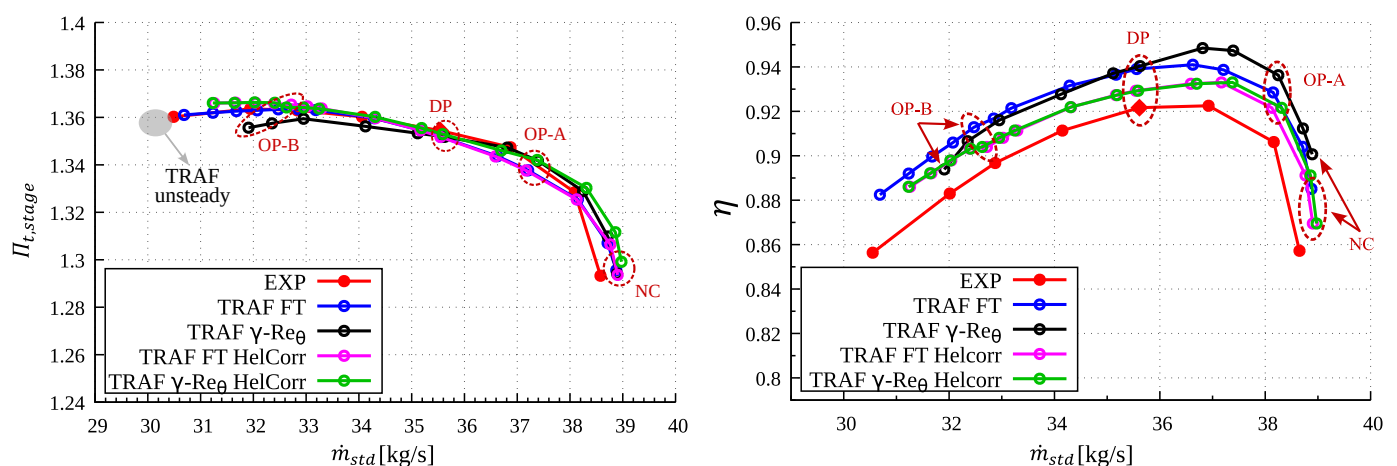


Figure 5. Performance characteristics: fully turbulent and transition baseline models comparison with helicity-corrected versions.

Both the experimental and numerical efficiency are calculated with Equation (2). Again, $\Pi_{t,stage}$ does not take into account the region near the endwall to be coherent with the experimental values. The mechanical power P_{mec} coming from the TRAF simulations is the power transferred from the fan blades to the flow and does not take into account

the efficiency of the mechanical transmission. On the other hand, the experimental P_{mec} is measured with a torquemeter; this is why the numerical P_{mec} is expected to be lower than the experimental one, so the η_{stage} is overestimated by the numerical simulations.

The efficiency predicted with the transition model is higher for the operating points in the knee region and lower for those between near stall and the design point compared with the efficiency obtained with fully turbulent simulations.

The distribution of the skin friction on the blade surface can serve as a detector of boundary layer transition, as the laminar boundary layer has a lower value of skin friction than the turbulent boundary layer. Furthermore, as per its definition, a negative value of skin friction indicates reverse flow. The typical structure of a separation bubble presents the separation point, where the velocity gradient at the wall becomes zero, and downstream of the separation point, reverse flow develops between the surface and the separated shear layer. In contrast, three-dimensional separation can occur without reverse flow and with non-zero friction [23].

Shown in Figure 6 are the Mach number and skin friction distributions on the fan blade surface at 95% of the span for the four operating points marked in Figure 5: NC, OP-A, DP, and OP-B. OP-A is located in the knee region, where the use of the transition model gives better agreement with the experimental results. OP-B is located in the low-mass-flow-rate area, where the application of the transition model predicts poorer agreement with the experimental speedline compared with the fully turbulent speedline.

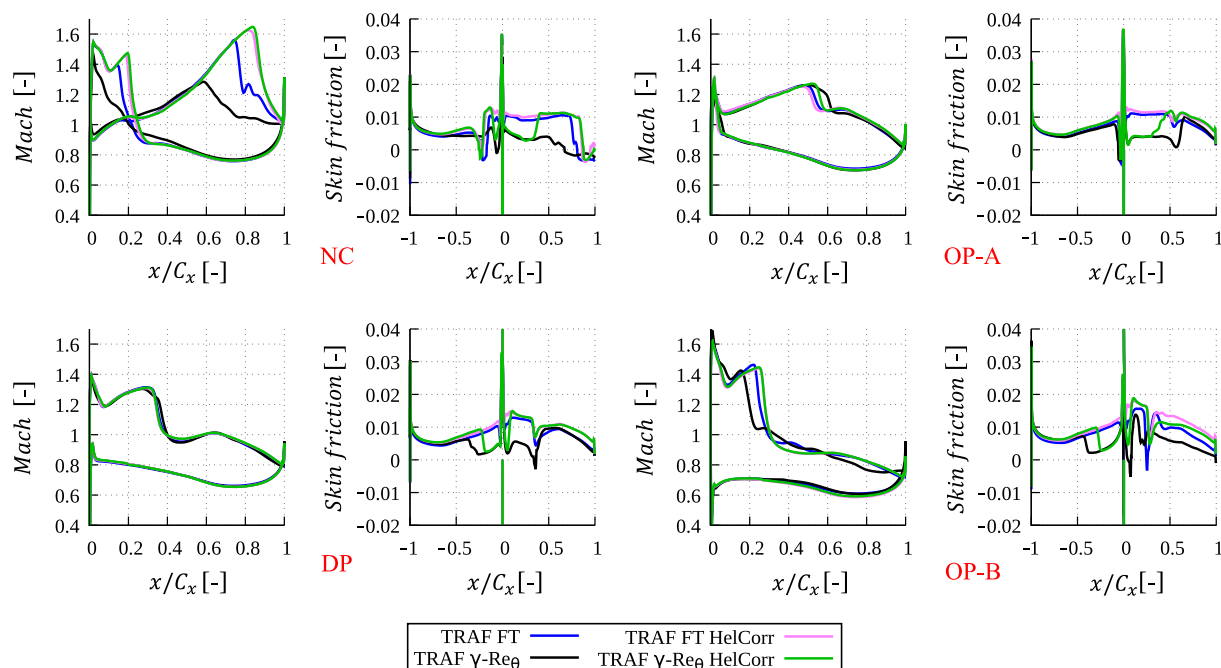


Figure 6. Mach and skin friction distributions at 95% of fan blade span.

Both the fully turbulent and the transitional simulations show that the velocity peak and the region of impact of the shock on the SS move closer to the LE for more throttled operating points. The fully turbulent skin friction distribution displays negative values for NC and OP-A on the PS and for NC and OP-B on the SS. There are significant differences between the two Mach and skin friction distributions. The skin friction distribution obtained with the transitional model shows that a portion of laminar boundary layer is present on the PS for DP and OP-B and negative values of skin friction are visible on the SS at DP.

Again, Figure 6 shows that the velocity peak at DP and OP-B is lower for the simulation with the transition model. At OP-B, the Mach number at the TE is higher than that predicted by the fully turbulent model. At OP-A, the region near the velocity peak has higher Mach values, and the blade load is higher for the transitional simulation. At OP-A and DP, both simulations predict the same skin friction value at the TE on the SS, while at OP-B, the skin friction is lower for the simulation with the transition model. At OP-A, the portion of laminar boundary layer increases efficiency and the total pressure at the stage exit, whereas at OP-B, the laminar boundary layer leads to a significant separation on the SS; thus, the efficiency and the total pressure at the stage exit are lower.

Figure 7 presents a comparison of the total pressure maps at SE normalized with mass-averaged stage total pressure ratio at DP. The experimental stage exit plane was measured by ECL with two rakes separated by 180° . The two rakes were traversed around the machine axis and recorded 15 circumferential points per OGV passage (represented by the black dots in Figure 7). A stator wake is clearly visible from about 25% to 82% of the channel height. Close to the hub and the casing, regions of reduced pressure ratio compared with mid-channel values indicate corner separation.

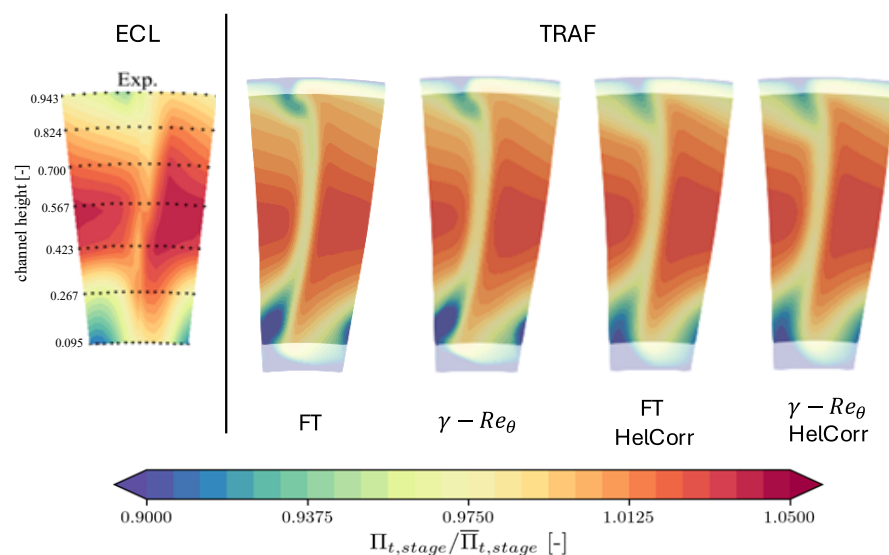


Figure 7. DP normalized stage total pressure ratio.

The corner separation at the hub and the casing predicted by numerical simulations run with a conventional turbulence model is more extended and intense than the experimental one.

4.3. Impact of Helicity-Based Correction

As previously mentioned, the corner separation at the stage exit has been overestimated by the simulations presented so far. In order to improve the prediction of the corner separation, a helicity-based correction was applied to the $k - \omega$ turbulence model for both the fully turbulent and the transitional simulations.

The speedline predicted by the simulation using the helicity-based correction and the fully turbulent model is very close to that using the fully turbulent model and no helicity-based correction, except for a slight overestimation of the stage total pressure ratio at a lower mass flow rate. The speedline calculated using the transition model and the helicity-based correction has a similar behavior to the one using the standard turbulence model coupled with the transition model speedline in the knee region, so it improves the agreement with the experimental curve in that area compared with the fully turbulent

speedlines. Furthermore, the speedline maintains good agreement with the experimental curve even towards lower mass flow rates, as displayed in Figure 5.

The operating points calculated using the helicity-based correction present higher mass flow rates than those calculated without the correction for the same outlet static pressure. This is to be expected, since a smaller corner separation results in a smaller blockage and thus a higher mass flow rate.

The efficiency is lower for the operating points calculated with the helicity-based correction. The efficiency is calculated with Equation (2), and as shown by the speedline comparison, both the mass flow rate and the stage total pressure ratio increase with the use of the helicity correction, which would result in higher efficiency. On the other hand, the mechanical power also increases, which leads to lower efficiency values. The two curves obtained with the turbulence model correction are identical over the entire operating range, except for the higher mass flow rate region, where the simulations using the transition model predict slightly higher efficiency.

Figure 7 shows that the use of the helicity-based correction of the turbulence model improves the prediction of the corner separation at the hub, both in extension and intensity. Moreover, the region outside the corner separation near the tip displays a lower total pressure value compared with that obtained without the model correction and thus a more similar distribution to the experimental map. The radial profiles of five-hole probe measurements downstream of the fan (rotor exit plane, RE) are presented in Figure 8 for two operating points at design speed: near choke and design point.

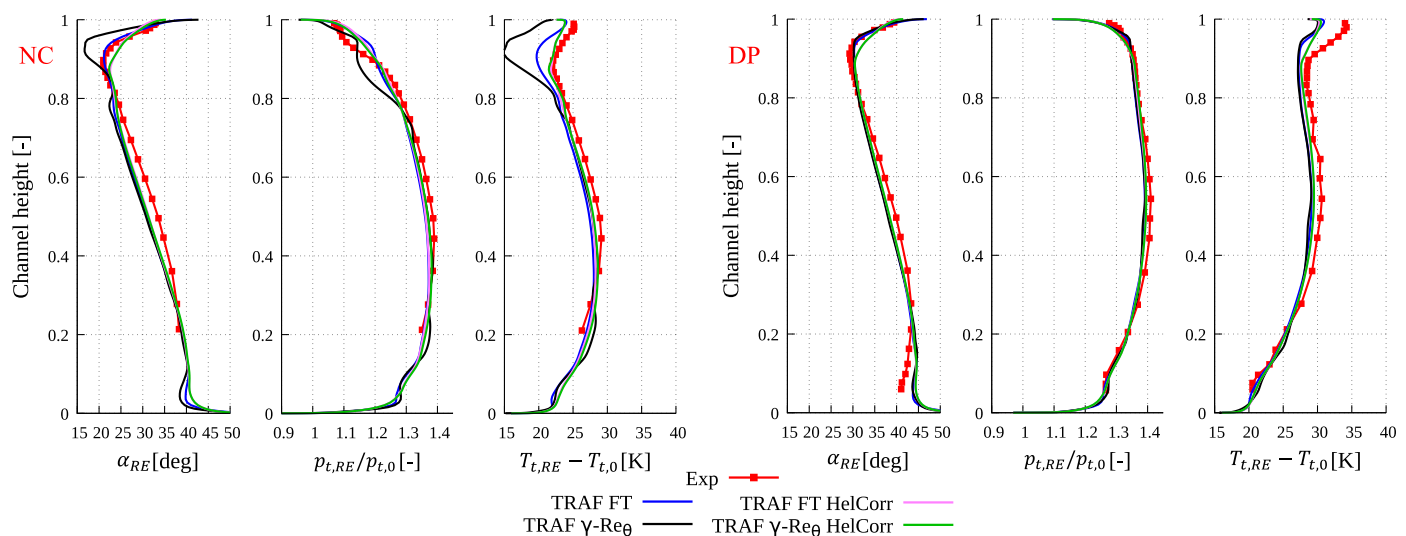


Figure 8. NC (left) and DP (right) span-wise distributions of flow angle, total pressure, and total temperature at RE.

Overall, there is good agreement between the experimental and numerical radial profiles. The absolute swirl angle, at constant rotational speed, is indicative of the mass flow rate. The lower the swirl angle, the higher the mass flow rate. The span-wise distributions of flow angle, total pressure, and total temperature obtained using the transition model at NC are greatly improved in the tip region with the use of the turbulence model correction. At DP the total temperature distribution predicted with the helicity-based correction is in better agreement with the experimental curve data in the tip region: near the casing, the higher total temperature radial extension is better predicted (Figure 8).

The numerical–experimental differences in the swirl angle, particularly visible for the NC condition, may be attributed to the blade stagger variation caused by the blade load not considered in the current CFD simulations. Differences in the blade stagger angle may cause a different work input in the fluid, thus leading to a different total temperature radial distribution. Moreover, since RANS simulations do not resolve the turbulent mixing, flow features like wakes, secondary flows, and corner vortex can be overestimated, leading to different radial distributions.

Using the helicity-based correction, the static pressure maps at the casing present lower pressure values on the pressure side near the LE at NC. The transition model map is improved with the use of the turbulence model correction, as shown in Figure 9. At DP the tip leakage flow extension is overestimated by the standard turbulence model and underestimated by the corrected turbulence model.

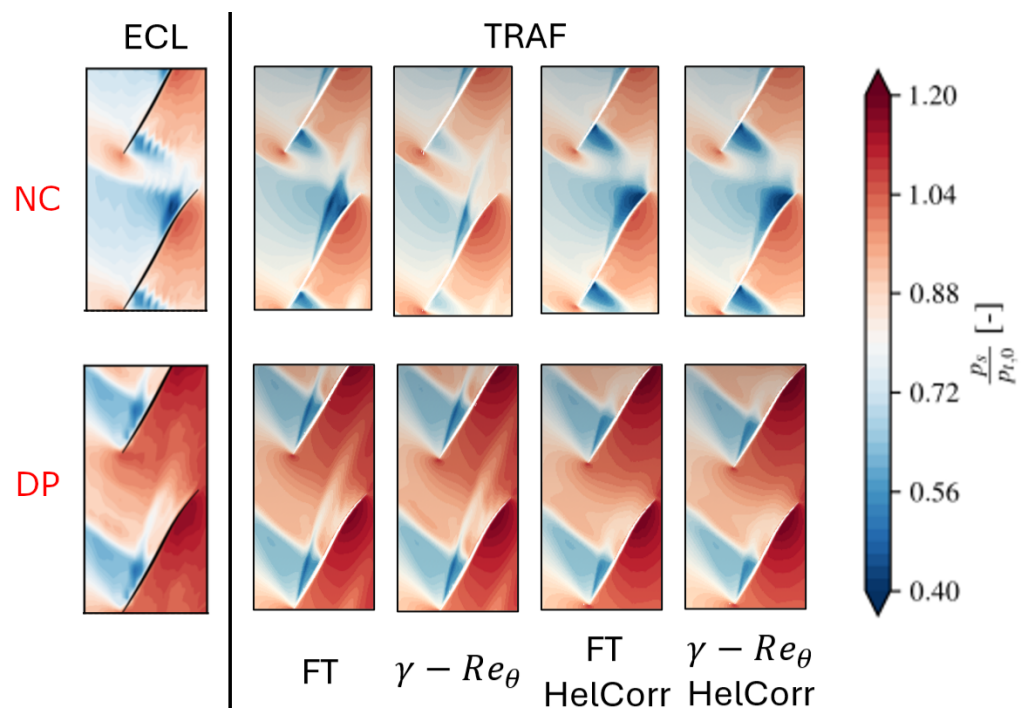


Figure 9. NC and DP normalized static pressure maps at rotor casing.

The skin friction distributions reported in Figure 6 show that the boundary layer transition occurs closer to the LE for the modified turbulence model simulations, thus when the shock wave interacts with the boundary layer, the latter is turbulent. This results in smaller separations, as shown in Figure 10a,b, and explains the higher values of stage total pressure ratio and efficiency previously mentioned. Overall, the skin friction values obtained with the helicity correction are higher than those obtained with the unmodified turbulent model. The blade load predicted is also higher. In contrast with the simulations using the transition model and the standard turbulence model, the calculated Mach number at the TE is the same as that calculated with the fully turbulent model at DP and OP-B. Furthermore, at OP-B, the velocity peak moves downstream.

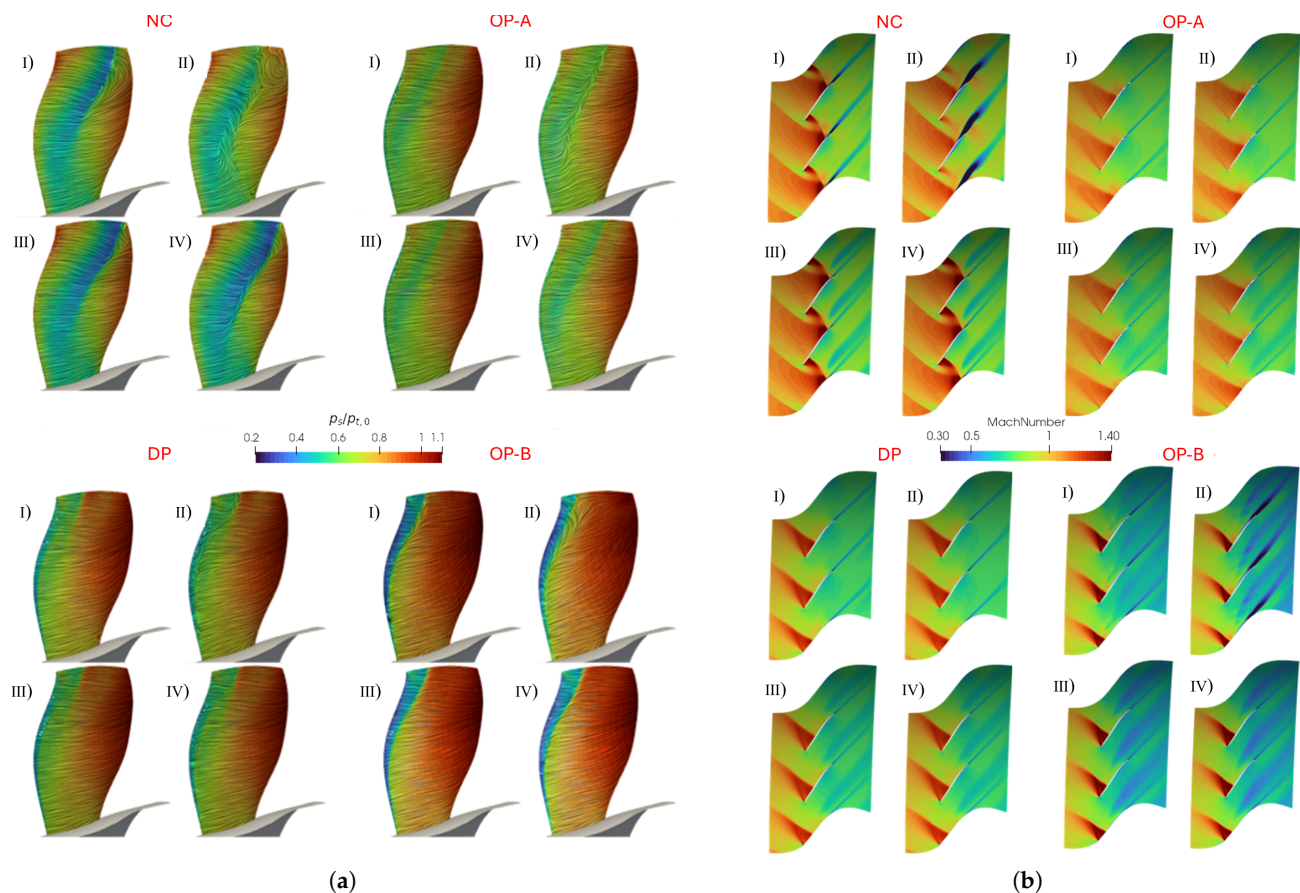


Figure 10. (a) Streamlines and static pressure maps on fan SS and (b) Mach number maps at 95% of fan span: (I) Fully turbulent and no helicity correction. (II) Transition model and no helicity correction. (III) Fully turbulent and helicity correction. (IV) Transition model and helicity correction.

5. Unsteady Results

The steady-state simulations become numerically unstable when run close to the stall line due to the increased influence of unsteady phenomena at low mass flow rates. For this reason a full annulus unsteady simulation was set up to explore the left part of the curve, possibly highlighting the tip instability phenomena. Such phenomena are the precursors of fan stall and may lead to non-synchronous vibrations (NSVs) depending on their frequency and tangential wavelength [24]. Under the design conditions, no NSVs are experimentally detected [11], and this is in line with the numerical results.

To do so, a full annulus unsteady simulation was started from the last stable steady-state solution by further increasing the stage pressure ratio. In total, 15 rotor revolution were needed to reach a pseudo-stable conditions with stage performance extracted after each period lying in the gray dot in Figure 5. A successive increase in pressure ratio led to a drop in the total pressure ratio and mass flow rate, suggesting the begin of the stall condition.

To monitor flow unsteadiness, a lift coefficient based on the blade surface pressure was utilized. The lift coefficient DFT of the last unsteady period for a single fan blade is plotted in Figure 11. The harmonics corresponding to the number of OGVs and to double the number of vanes, i.e., 31 and 62, are easily noticeable in the fan blade lift spectrum. Moreover, the spectrum also presents low-frequency content in the BPF subsynchronous region (around engine order 6 and highlighted by an arrow in Figure 11) related to flow tip instability due to the generation of a tornado-like vortex. High-frequency content is related to numerical noise.

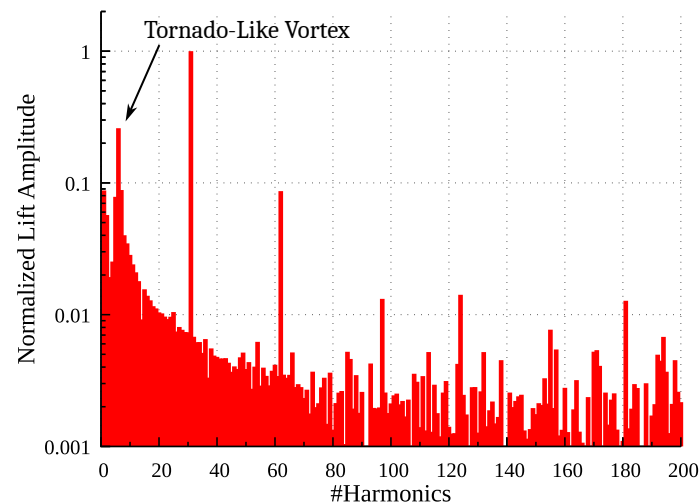


Figure 11. Unsteady lift spectrum for fan blade. Low-frequency content pointed out by the black arrow.

During the unsteady evolution of the flow, a leading-edge separation occurs near the tip and grows to form the tornado-like vortex connecting the blade suction surface and the casing wall. The casing-side leg of the tornado-like vortex, which generates a low-pressure region on the casing, moves circumferentially as the vortex grows. When the vortex casing-side leg approaches the adjacent blade, it causes a leading-edge separation near the tip of the adjacent blade due to its blockage effect. As a result, the leading-edge separation propagates through the rotor, forming the tornado-like separation vortex and eventually developing into the multiple stall cells under the mild-stall condition [24–27].

The time evolution of radial vorticity on a blade-to-blade surface at 95% of the fan blade span is shown in Figure 12. The maps are captured with a time interval of $1/20$ of the fan rotation period. In Figure 12a the vortex, presenting positive radial vorticity, i.e., counterclockwise vorticity, is about to impact on the pressure side of the adjacent blade; then it interacts with the adjacent blade in Figure 12b and subsequently propagates in the adjacent blade passage in Figure 12c. In Figure 12d the vortex is once again about to impact on the adjacent blade. This time the vortex radial intensity is higher compared with the first image, and there is a negative radial vorticity region in the middle of the blade passage close to the TE. This second vortex starts developing at the TE in Figure 12b, and it grows in Figure 12c and detaches from the TE in the last frame. The differences between Figure 12a,d show that the phenomenon is not perfectly periodic.

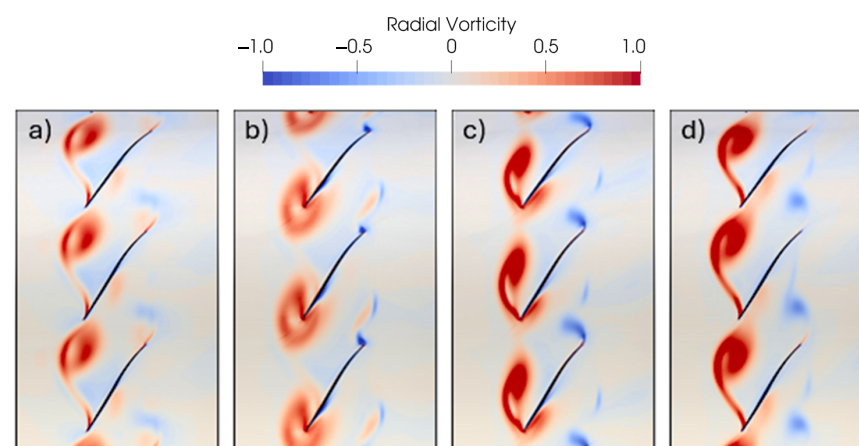


Figure 12. Time evolution of blade-to-blade radial vorticity at 95% of fan blade span over four instants (a–d) with a time interval of $1/20$ of the fan rotation period.

These results confirm the capability of URANS simulations to capture the occurrence of the blade tip instability that may be responsible for NSVs when the frequency of the instability locks on to the natural frequency of the blade. In the present case, engine order 6 has a frequency of 1100 Hz, which is quite different from the first blade natural frequency leading to stall conditions without passing through NSVs. This has been confirmed by the experimental campaign [24] where NSV phenomena were only observed at different rotational speeds (80% and 55% of the design speed).

6. Conclusions

In this paper the ECL5/CATANA fan stage was numerically investigated from an aerodynamic point of view using a RANS approach. In order to assess the prediction capability of the numerical framework, the speedline at design rotational speed was simulated over the entire operating range.

With the use of a standard $k - \omega$ turbulence model, the stage total pressure ratio and the mass flow rate are underestimated in the knee region and are overestimated in the near-choke region. This fact is in agreement with the numerical results obtained by other researchers and is suspected to be due to the fan blade stagger angle variation with the mass flow rate.

The influence of laminar flow was investigated by coupling the turbulence model with the Langtry–Menter transition model. This model improved the prediction of the pressure ratio in the knee region, but in this case, a premature numerical stall was detected, compared with the fully turbulent approach.

The decreased prediction of the operating range was associated with an overestimation of the blade corner stall and an increased shock wave–boundary layer interaction due to the presence of laminar flow. The use of a helicity correction for the turbulence model coupled with the transition model was able to maintain good agreement in the knee region of the pressure ratio characteristic, preserving a wider operating range closer to the experimental data.

The prediction improvement was confirmed by the better agreement of total pressure maps obtained downstream the stage and the span-wise shapes of total pressure, total temperature, and absolute flow angle. A local analysis of the flow field confirmed that the use of the helicity correction was able to reduce the blade corner stall and the shock wave–boundary layer interaction in the tip region.

Finally, the speedline region near the stability limit was investigated with unsteady simulations, starting from the last converged operating point obtained with the steady-state calculations. By throttling the exit pressure, the slope of the characteristic turned to positive values, and a pseudo-periodic convergence was reached before a sudden drop in the pressure ratio. An analysis of the lift coefficient highlighted the amplification of sub-synchronous frequencies around the sixth harmonic order. The study of the instantaneous solution revealed the presence of a tornado-shaped vortex in each of the blade passages, with the open end attached to the casing and the other to the suction side of the blade. The propagation of this “mini-cell” occurs when the casing end of the vortex moves towards the pressure side of the neighboring blade, where it triggers a new vortex due to its blockage effect. This mechanism of vortex propagation through the rotor can be associated with non-synchronous vibrations (NSVs) of the fan blades when vortex frequency locks on to one of the blade natural frequencies and requires further investigations.

Author Contributions: Conceptualization, L.P.; validation, M.M. (Maria Malcaus) and G.G.; investigation, M.M. (Maria Malcaus) and G.G.; writing—original draft preparation, M.M. (Maria Malcaus); writing—review and editing, M.M. (Maria Malcaus), L.P., G.G., and M.M. (Michele Marconcini);

supervision, L.P. and M.M. (Michele Marconcini). All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Data are fully available for sharing upon direct request to the main author of this paper.

Acknowledgments: The authors would like to thank Christoph Brandstetter from École Centrale de Lyon for providing the data and supporting materials necessary to reproduce the numerical results. Moreover, the Authors would like to acknowledge ASME to grant the permission to use the Figures 1, 2 and 4.

Conflicts of Interest: Author Maria Malcaus was employed by the company Baker Hughes. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

Nomenclature

The following nomenclature is used in this manuscript:

Acronyms

CFD	Multidisciplinary Digital Publishing Institute
DFT	Discrete Fourier Transform
DP	design point
ECL	École Centrale de Lyon
FT	fully turbulent
LE	leading edge
NC	near choke
NSV	non-synchronous vibration
OGV	Outlet Guide Vane
PS	pressure side
RE	rotor exit
SE	stage exit
SS	suction side
TE	trailing edge
UHBR	Ultra High Bypass Ratio
URANS	Unsteady Reynolds-Averaged Navier–Stokes
WPT	wall pressure transducer

Greek

η	efficiency
Π	pressure ratio
γ	specific heat ratio

Symbols

$\dot{m}$	mass flow rate
c_p	specific heat at constant pressure
C_x	axial chord
P	power
p	pressure
T	temperature
Ma	Mach

Subscripts

in	inlet
mec	mechanical
s	static
std	standard

Superscripts

$\overline{(\cdot)}$	mass-averaged value along the radial direction
----------------------	--

References

1. Lee, K.B.; Wilson, M.; Vahdati, M. Validation of a Numerical Model for Predicting Stalled Flows in a Low-Speed Fan—Part I: Modification of Spalart–Allmaras Turbulence Model. *J. Turbomach.* **2018**, *140*, 051008. [\[CrossRef\]](#)
2. Peters, A.; Spakovszky, Z.S.; Lord, W.K.; Rose, B. Ultrashort Nacelles for Low Fan Pressure Ratio Propulsors. *J. Turbomach.* **2014**, *137*, 021001. [\[CrossRef\]](#)
3. Wilson, M.J.; Imregun, M.; Sayma, A.I. The Effect of Stagger Variability in Gas Turbine Fan Assemblies. *J. Turbomach.* **2006**, *129*, 404–411. [\[CrossRef\]](#)
4. Lu, Y.; Lad, B.; Green, J.; Stapelfeldt, S.; Vahdati, M. Effect of geometry variability on transonic fan blade untwist. *Int. J. Turbomach. Propuls. Power* **2019**, *4*, 24. [\[CrossRef\]](#)
5. Zhai, Y.; Bladh, R.; Dyverfeldt, G. Aeroelastic Stability Assessment of an Industrial Compressor Blade Including Mistuning Effects. *J. Turbomach.* **2012**, *134*, 060903. [\[CrossRef\]](#)
6. Goerke, D.; Le Denmat, A.L.; Schmidt, T.; Kocian, F.; Nicke, E. Aerodynamic and Mechanical Optimization of CF/PEEK Blades of a Counter Rotating Fan. In Proceedings of the ASME Turbo Expo 2012, Copenhagen, Denmark, 11–15 June 2012; pp. 21–33. [\[CrossRef\]](#)
7. Schmid, T.E.; Lengyel-Kampmann, T.; Schmidt, T.; Nicke, E. Optimization of a carbon-fiber composite blade of a counter-rotating fan for aircraft engines. In Proceedings of the 13th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Lausanne, Switzerland, 8–12 April 2019. [\[CrossRef\]](#)
8. Reiber, C.; Blocher, M. Potential of aeroelastic tailoring to improve flutter stability of turbomachinery compressor blades. In Proceedings of the 12th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Stockholm, Sweden, 3–7 April 2017. [\[CrossRef\]](#)
9. Arnone, A. Viscous Analysis of Three-Dimensional Rotor Flow Using a Multigrid Method. *J. Turbomach.* **1994**, *116*, 435–445. [\[CrossRef\]](#)
10. Liu, Y.; Tang, Y.; Scillitoe, A.D.; Tucker, P.G. Modification of Shear Stress Transport Turbulence Model Using Helicity for Predicting Corner Separation Flow in a Linear Compressor Cascade. *J. Turbomach.* **2020**, *142*, 021004. [\[CrossRef\]](#)
11. Schneider, A.; Fiquet, A.L.; Paoletti, B.; Ottavy, X.; Brandstetter, C. Experiments on tuned UHBR open-test-case Fan ECL5/CATANA: Performance and aerodynamics. *J. Turbomach.* **2024**, *146*, 081001. [\[CrossRef\]](#)
12. Pagès, V.; Duquesne, P.; Aubert, S.; Blanc, L.; Ferrand, P.; Ottavy, X.; Brandstetter, C. UHBR Open-Test-Case Fan ECL5/CATANA, Part 2: Mechanical and Aeroelastic Stability Analysis. In Proceedings of the 14th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Virtual, 12–16 April 2021. [\[CrossRef\]](#)
13. Brandstetter, C.; Pages, V.; Duquesne, P.; Paoletti, B.; Aubert, S.; Ottavy, X. Project PHARE-2—A High-Speed UHBR Fan Test Facility for a New Open-Test Case. *J. Turbomach.* **2019**, *141*, 101004. [\[CrossRef\]](#)
14. Schneider, A.P.; Paoletti, B.; Ottavy, X.; Brandstetter, C. Influence of a Turbulence Control Screen on the Aerodynamic and Aeroelastic Behavior of a UHBR Fan. In Proceedings of the ASME Turbo Expo 2022, Rotterdam, The Netherlands, 13–17 June 2022; p. V08AT21A011. [\[CrossRef\]](#)
15. Roe, P.L.; Baines, M.J. Algorithms for advection and shock problems. In Proceedings of the Conference on Numerical Methods in Fluid Mechanics, Paris, France, 7–9 October 1981.
16. Arora, M.; Roe, P.L. A Well-Behaved TVD Limiter for High-Resolution Calculations of Unsteady Flow. *J. Comput. Phys.* **1997**, *132*, 3–11. [\[CrossRef\]](#)
17. Burberi, C.; Michelassi, V.; Scotti del Greco, A.; Lorusso, S.; Tapinassi, L.; Marconcini, M.; Pacciani, R. Validation of steady and unsteady CFD strategies in the design of axial compressors for gas turbine engines. *Aerosp. Sci. Technol.* **2020**, *107*, 106307. [\[CrossRef\]](#)
18. Pinelli, L.; Burberi, C.; Ignesti, M.; Michelassi, V.; Tapinassi, L.; Abati, A.; Pacciani, R. Forced response in a multi-stage aeroderivative axial compressor with acoustic excitations: Method validation with field data. *Aerosp. Sci. Technol.* **2023**, *138*, 108305. [\[CrossRef\]](#)
19. Chima, R.V. Calculation of Tip Clearance Effects in a Transonic Compressor Rotor. *J. Turbomach.* **1998**, *120*, 131–140. [\[CrossRef\]](#)
20. Wilcox, D.C. *Turbulence Modeling for CFD*, 2nd ed.; DCW Ind. Inc.: La Canada, CA, USA, 1998.

21. Pela, A.; Marconcini, M.; Arnone, A.; Toni, L.; Valente, R.; Agnolucci, A.; Grimaldi, A.; Pacciani, R. On the Feasibility of Helicity-Based Corrections of Turbulence Models for Improving RANS Predictions of Centrifugal Compressor Stages. In Proceedings of the ASME Turbo Expo 2023, Boston, MA, USA, 26–30 June 2023; p. V13DT35A026. [\[CrossRef\]](#)
22. Langtry, R.B.; Menter, F. Correlation-Based Transition Modeling for Unstructured Parallelized Computational Fluid Dynamics Codes. *AIAA J.* **2009**, *47*. [\[CrossRef\]](#)
23. Gbadebo, S.A.; Cumpsty, N.A.; Hynes, T.P. Three-Dimensional Separations in Axial Compressors. *J. Turbomach.* **2005**, *127*, 331–339. [\[CrossRef\]](#)
24. Tharreau, P.; Hardy-Falch, M.; Stapelfeldt, S.; Brandstetter, C. Non-Synchronous Vibration: Characterisation of the Aerodynamic Disturbance and its Dependency on Local Tip Clearance. In Proceedings of the ASME Turbo Expo 2024, London UK, 24–28 June 2024; p. V10AT21A021. [\[CrossRef\]](#)
25. Hewkin-Smith, M.; Pullan, G.; Grimshaw, S.D.; Greitzer, E.M.; Spakovszky, Z.S. The Role of Tip Leakage Flow in Spike-Type Rotating Stall Inception. *J. Turbomach.* **2019**, *141*, 061010. [\[CrossRef\]](#)
26. Inoue, M.; Kuromaru, M.; Tanino, T.; Yoshida, S.; Furukawa, M. Comparative Studies on Short and Long Length-Scale Stall Cell Propagating in an Axial Compressor Rotor. *J. Turbomach.* **2000**, *123*, 24–30. [\[CrossRef\]](#)
27. Brandstetter, C.; Schneider, A.P.; Fiquet, A.L.; Paoletti, B.; Ottavy, X. Experiments on Structurally Mistuned UHBR Open-Test-Case Fan ECL5/CATANA. In Proceedings of the ASME Turbo Expo 2024, London UK, 24–28 June 2024; p. V10AT21A020. [\[CrossRef\]](#)

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.