IOP Conference Series: Materials Science and Engineering

Aerodynamic Analysis of a scaled UHBR Fan

To cite this article: N. Natale *et al* 2022 *IOP Conf. Ser.: Mater. Sci. Eng.* **1226** 012007

Aerodynamic Analysis of a scaled UHBR Fan

Natale N.[1], Egger T.[2], Friedrichs J.[2], Russo S.[1]
[1]Dream Innovation, via F. Parri 1 81030 Caserta - Italy
[2]Institute of Jet Propulsion and Turbomachinery, Technische Universität Braunschweig 38108 Braunschweig – Germany

nunzio.natale@dreaminnovation.it
t.eggers@ifas.tu-braunschweig.de

Abstract. In the frame of the CA3ViAR Clean Sky 2 (Composite fan Aerodynamic, Aeroelastic, and Aeroacoustic Validation Rig), the main objective is to design a low-speed (low-transonic) fan typical of a future large aircraft UHBR engine, in terms of aerodynamic shaping as well as structural design and analysis to make sure the test article experiences aerodynamic and aeroelastic instabilities in an expected way during wind tunnel (WT) operations. Eventually, open access to all the produced models will be provided, with the objective to establish an "open test-case" for the whole European scientific community, unique in the engine fans landscape. A preliminary fan stage design is presented in this paper, details about the aerodynamic design process and the results of the CFD analysis of the stage are shown. The present UHBR fan design fulfils the initial aerodynamic requirements and represents the starting point for the structural and aeroelastic analysis within the multidisciplinary design process employed to design the final CA3ViAR fan stage.

1. Introduction

In order to achieve a higher propulsive efficiency, it is necessary to decrease the nozzle exit velocity along with increasing the total engine mass flow with a lower pressure ratio.
The consequence of this leads to the design of fans with larger diameters and thus higher bypass ratios. The design of such new engines poses inevitably new challenges to designers, as a matter of fact, at constant nozzle design the take-off and approach operating points move towards lower mass flows reducing the stall margin and increasing the risk of flutter [1], [2].
Flutter is a self-excited aeroelastic instability, where the interaction of blade vibration and unsteady aerodynamic forces lead to an energy transfer from the flow into the blade motion. Flutter can occur during part-load operation when the machine is operating close to the stall boundary. Therefore, it is of particular interest for designers of ultra-high bypass ratio (UHBR) fans. Mechanisms of stall flutter in a modern turbofan have been investigated in several studies, e.g. [3], [4]. Besides the aerodynamic requirements, other challenges have to be faced by designers. Increasing the fan diameter will inevitably lead to a higher engine weight. This effect has to be balanced by using lighter and stiffer materials, such as carbon fiber reinforced polymers (CFPR) instead of more conventional titanium alloys. CA3ViAR (grant agreement No 864256, [5]) is contributing to the achievement of Clean Sky 2 goals by designing and testing a highly instrumented composite low-transonic fan (LTF). The fan object of this work has been designed to experience forced response and instabilities, such as flutter, inside a wind tunnel (Propulsion Test Facility of the Institute of Jet Propulsion and Turbomachinery - Technische Universität

Braunschweig). The present work will focus on the evaluation of aerodynamic performance of the preliminary design developed by the consortium.

	DP@10.7 Km	Rig@0 Km	
Operating Point	Cruise	Cruise	Take-off
Bypass ratio	17	17	17
Polytropic Efficiency, η_{poly}	0.89	0.89	0.87
Total pressure ratio, π_t	1.37	1.37	1.32
Mass flow, $\dot{m}$ $\left(\frac{kg}{s}\right)$	272.24	63.39	57.15
Rotational speed, n (RPM)	2375	8667	8095
Fan tip speed (m/s)	272	295	275
Fan tip radius (m)	1.093	0.325	0.325
Hub- to -tip ratio	0.26	0.26	0.26

Table 1 – Fan design parameters

2. Aerodynamic Design

The design presented in this work is the result of an iterative design process, briefly, considering only the aerodynamic design, the first step is an engine cycle design which leads to the full-size engine dimensions. This first step is then followed by a scaling of the dimensions to the final rig dimensions. Thereby, geometrical and Mach similarity will be ensured. The resulting fan is in the scale of approximately 1:3.3 based on the full-scale design, and it has 18 rotor blades and 40 stator blades. The running tip clearance is chosen to be 0.5 mm. The scaled engine dimensions are then the input for the turbomachinery design code [6] which led to the geometry described in Figure 1. It is not the objective of this work to show in detail the design process which led to the preliminary design under investigation. All details can be found in [7]. Some design parameters are shown in Table 1. At design point (DP) in cruise conditions, the fan stage is operated with a flow coefficient ϕ=0.69 at a polytropic efficiency of 0.89 and a total pressure ratio equal to 1.37. This performance is consistent with the design of an advanced fan stage reported in [8].

3. Numerical Set-up

3.1 Computational Grid

To achieve high-quality results a high-resolution mesh with low Reynolds configuration was used. A volume mesh was created with NUMECA AutoGrid5 for the fan stage. The dimensionless wall distance y^+ has been set to 1 at the blade, hub and shroud. The suitable element density has been found using the grid convergence index (GCI) approach.

With a maximum GCI of about 0.01% for the mass flow the results can be considered as mesh independent [9]. The mesh size for the fan stage is about $4 * 10^6$ cells with 137 cells in radial direction in a single pitch periodic domain. A section of the computational grid employed for the calculations is presented in Figure 2. All the simulations for the hot shape have been carried out at steady-state conditions. The two-equation k-ω SST turbulence model with modifications of the turbulence production near stagnation points and in areas with high streamline curvature is chosen for evaluating the fan performance. It is known that this turbulence model can provide a good compromise between accuracy of the results and computational costs [10].

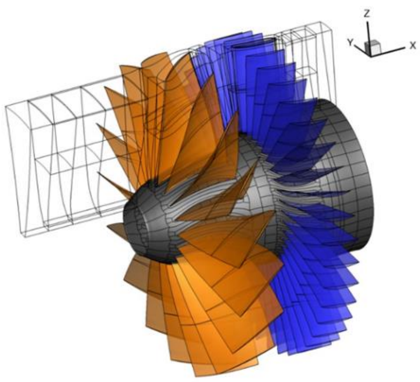

Figure 1. *UHBR preliminary design*

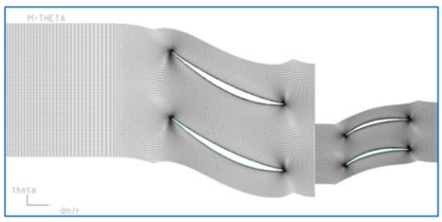

Figure 2. *Computational mesh*

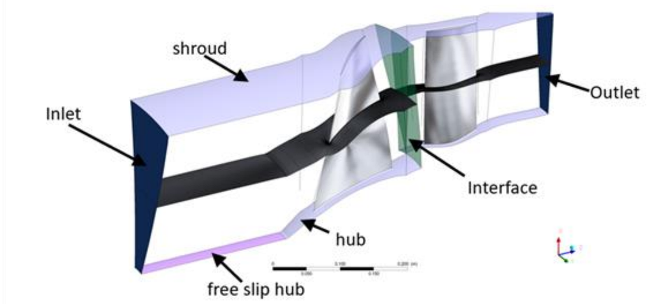

Figure 3. *Boundary conditions*

3.2 CFD Solver Set-up

The interface between the rotating fan domain and the stationary OGV domain is modelled with a mixing plane approach. Figure 3 shows the boundary conditions applied, the hub is defined as a non-viscous wall in the inlet area, because the hub contour is not modelled down to the machine axis. The inlet boundary condition is defined by homogeneous ambient conditions at sea level considering a loss of 1% due to the presence of the intake. Two types of boundary conditions have been applied to the outlet depending on the needs. Typically, the points on the extreme of each speed line have been simulated by applying static pressure boundary condition. Mass flow outlet boundary condition has been used as well for all the points which do not present particular convergence difficulties.

4. Results

Several speed lines have been calculated to fully define the aerodynamic behaviour of the machine. In Figure 4 one can see the performance in terms of total pressure ratio and polytropic efficiency for several rotational speeds. Particularly, cruise speed line (red) and take-off speed line (green) refer to the two design conditions highlighted in Table 1, the light-blue mark refers to the design point of the corresponding speed line. Comparing the expected performance parameters in Table 1 and the actual CFD output in terms of pressure ratio and polytropic efficiency, one can see the preliminary design matches the required performance. To fully define the aerodynamic behaviour along the blade span, the contour Mach number for three different sections spanwise (10%, 50% and 90% span) are presented in Figure 5, Figure 6, and Figure 7 for the cruise condition and design point (which corresponds to an outlet mass flow of 63.4 kg/s at 8667 RPM). Looking at Figure 7 and Figure 8, it is clearly visible the presence of a shock wave located between 45 and 50 % of the normalized chord. During the design phase particular attention has been placed to the shock pattern on the fan suction side. The camber of the transonic flow sections was progressively reduced to change the shock pattern by controlling the throat area. Figure 8 also shows how the loading progressively increases spanwise on both rotor and stator blade. Moreover, a pressure plateau is visible at 10% span on the suction surface for the rotor blade (red curve) starting about 85 per cent of the chord length. This pressure plateau is due to a separated zone present close the hub towards the trailing edge of the blade which is clearly visible in Figure 9 where entropy and skin friction lines (Figure 9a) and the streamlines in the proximity of the hub for the rotor suction surface (Figure 9b) are presented.

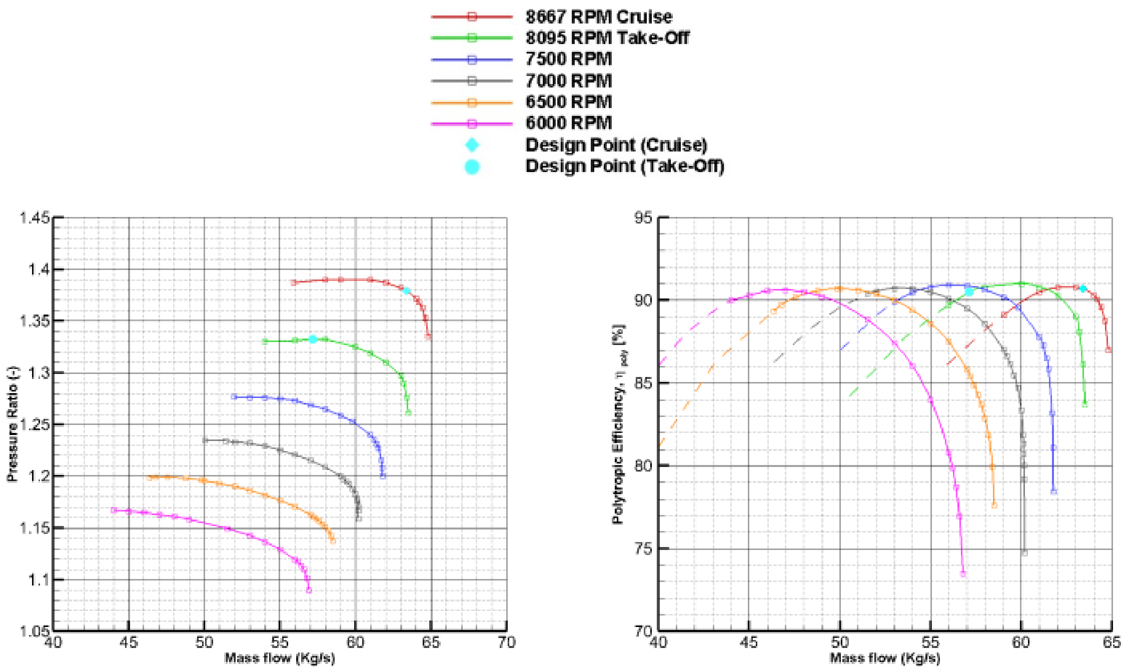

Figure 4. *Compressor map*

Figure 10 shows the spanwise distribution for the efficiency and axial velocity, it is clear looking specially at Figure 10a how the present design can ensure a quite uniform efficiency distribution except for the first 15 % of the blade span and the very last part of the blade, where stronger shocks and tip clearance vortex reduce the efficiency. It should be noted the precise control of the shock pattern allows to attain a larger amount of efficiency in the transonic part of the rotor blade, where a not negligible decreasing in terms of efficiency can be noted only starting from 92 % of the rotor blade span.

The Mach number contours along the span already presented for the cruise are reproposed for the design point of take-off condition (Figure 11, Figure 12, and Figure 13) which corresponds to 57.15 Kg/s mass flow outlet at 8095 RPM. The preliminary design under investigation has also been verified in terms of stall margin.

$$SM = 1 - \frac{\pi_{t,DP}}{\pi_{t,SL}} \frac{\dot{m}_{SL}}{\dot{m}_{DP}} \quad \textbf{(1)}$$

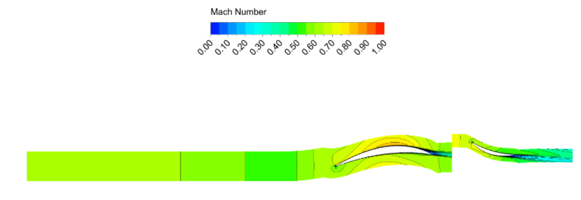

Figure 5*. Mach number contour : cruise condition, design point, 10% span*

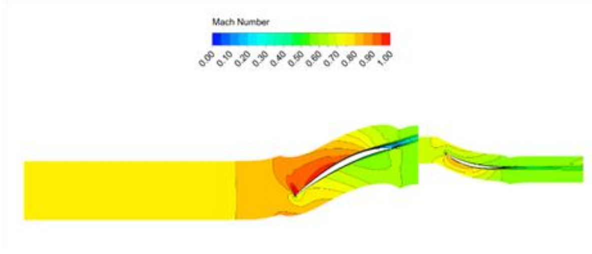

Figure 6. Mach number contour : cruise condition, design point, 50% span

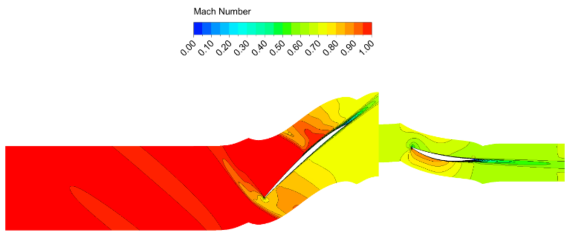

Figure 7. Mach number contour : cruise condition, design point, 90% span

Considering this parameter, the performance can be considered satisfactory as well leading to a value at least higher than 11 per cent, which was the initial design target. Figure 14 shows the total pressure ratio distribution and flow turning along the rotor blade span, it can be seen looking at Figure 14a how the present design results slightly more loaded towards the blade tip. The blade tip is going to get more and more loaded at part-load and this behaviour will eventually end up with a fan-driven stall due to separation near the tip.

5. Conclusions and Outlooks

Following the activity within the CA3ViAR consortium, the aerodynamic performance analysis of the preliminary design (Case 132) developed for a scaled ultra-high bypass ratio (UHBR) fan has been carried out. The preliminary design presented in this work fulfil all the aerodynamic design specifications. Although this preliminary design (Case 132) fulfils the target performance from the aerodynamic point of view, inputs from the structural/aeroelastic side have led to some modifications of the initial design, in particular, to a thickness reduction to 75 %.

Nonetheless, the presented design can be treated as the baseline or starting point for the final CA3ViAR design, considering the inputs from the structural side are not expected to change the design radically. The complete aerodynamic analysis of the designs derived from Case 132 is not part of the present work, but a complete overview of the aerodynamic design performed for CA3ViAR project will be part of future works.

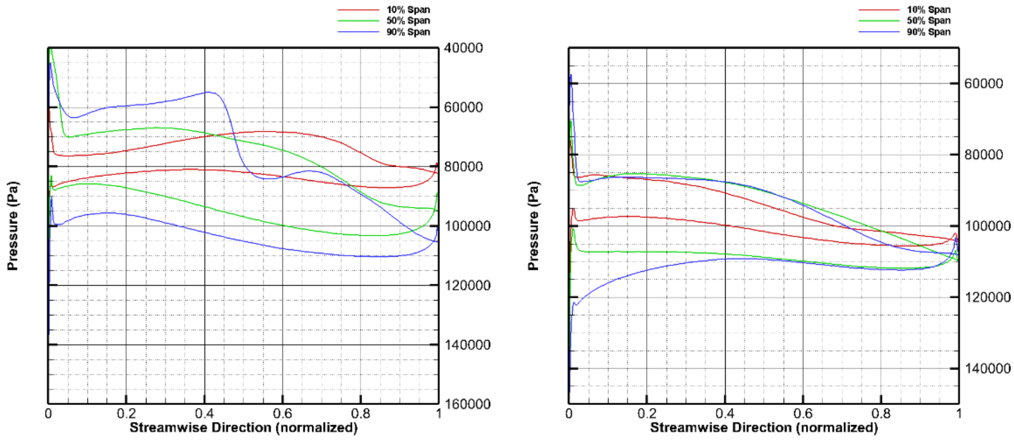

Figure 8. *Pressure distribution for rotor (a) and stator (b)*

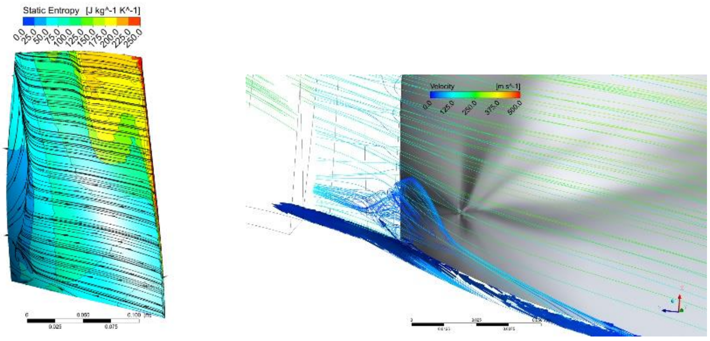

Figure 9. *Entropy and skin-friction lines on suction surface of rotor (a) and streamlines at rotor hub close to the trailing edge (b)*

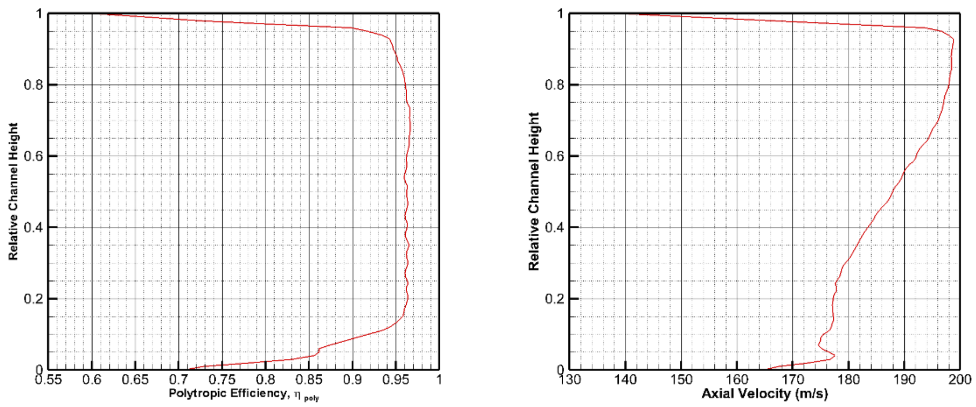

Figure 10. *Spanwise efficiency distribution (a) Spanwise axial velocity distribution (b)*

IOP Conf. Series: Materials Science and Engineering 1226 (2022) 012007 doi:10.1088/1757-899X/1226/1/012007

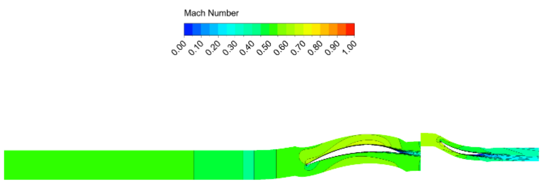

Figure 11. *Mach number contour: take -off condition, design point, 10% span*

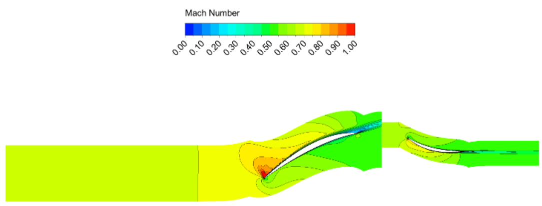

Figure 12. *Mach number contour: take -off condition, design point, 50% span*

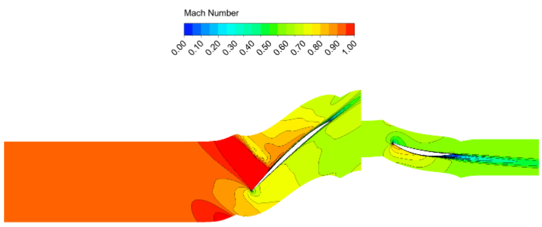

Figure 13. *Mach number contour: take -off condition, design point, 90% span*

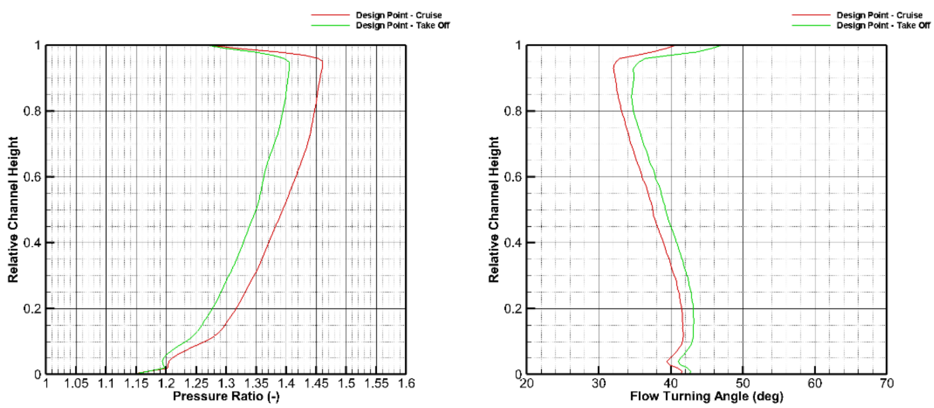

Figure 14. *Spanwise pressure ratio distribution (a), Flow turning angle (b)*

6. Acknowledgements

This project has received funding from the Clean Sky 2 Joint Undertaking (JU) under grant agreement No 864256. The JU receives support from the European Union's Horizon 2020 research and innovation programme and the Clean Sky 2 JU members other than the Union.

7. Disclaimer

The present work reflects only the authors' view and the European Commission and Clean Sky 2 JU are not responsible for any use that may be made of the information contained herein.

References

[1] M. Vahdati, A. I. Sayma, J. C. Marshall and M. Imregun, "Mechanisms and prediction methods for fan blade stall flutter," *Journal of Propulsion and Power,* p. 1100–1108, 2001.

[2] M. Vahdati, G. Simpson and M. Imregun, "Mechanisms for wide-chord fan blade flutter," *Journal of Turbomachinery,* p. 54, 2011.

[3] K. B. Lee, M. Wilson and M. Vahdati, "Numerical study on aeroelastic instability for a low speed fan," *Journal of Turbomachinery.,* 2017.

[4] I. J. Day, "Stall, surge, and 75 years of research," *Journal of Turbomachinery,* 2016.

[5] "https://cordis.europa.eu/project/id/864256," [Online].

[6] D. Giesecke, M. Lehmler, J. Friedrichs, J. Blinstrub, L. Bertsch and W. Heinze, "Evaluation of ultrahigh bypass ratio engines for an over-wing aircraft configuration," *Journal of the Global Power and Propulsion Society,* 2018.

[7] T. Eggers, J. Friedrichs, J. Goessling, J. R. Seume, N. Natale, J. P. Flüh and N. Paletta, "Composite UHBR fan for forced response and flutter," in *Proceedings of the ASME 2021 Turbomachinery Technical Conference & Exposition*, Pittsburgh, PA, USA, 2021.

[8] B. Kaplan, "Design of an Advanced Fan Stage With Ultra High Bypass Ratio and Comparison with Experimental Results," DLR, 2010.

[9] I. B. Celik, U. Ghia, P. J. Roache, C. J. Freitas, H. Coleman and P. E. Raad, "Procedure for estimation and reporting of uncertainty due to discretization in cfd applications," *Journal of Fluids Engineering,* 2008.

[10] D. C. Wilcox, Turbulence modeling for CFD, DCW Industries, Inc., 2006.