NUMERICAL INVESTIGATIONS OF TIP CLEARANCE EFFECTS ON THE ROTOR STAGES OF A CONTRA-ROTATING FAN ENGINE

N. Herter*, M. Schöpplein*, S. Hawner*, A. Hupfer*
* Institut für Aeronautical Engineering, Universität der Bundeswehr München (UniBwM), Deutschland

Abstract

A research focus at the Institute of Aeronautical Engineering at the University of the Bundeswehr Munich is the investigation of tip clearance effects in aircraft engines. The smallest possible gap between rotating and stationary parts are absolutely essential, especially for aerodynamically high loaded stages with small flow cross sections, in order to achieve the required efficiency, safety, reliability and operating dynamics Ref. [1]. Electrically powered aircraft engines represent a significant step towards the goal of emission-free and highly dynamic flight operations. The ELAPSED research project (Electrified Aircraft Propulsion - safe, efficient, digitally linked) within the cluster dtec.bw – digitalization and technology research center of the federal armed forces of Germany is investigating the use of a shrouded, contra-rotating and electrically powered fan engine. This approach makes it possible to extend the operating range of electric drives in terms of flight altitude, speed and facilitates integration compared to electrically powered propeller engines Ref. [2] and Ref. [3]. Tip clearance effects are mainly found between the blade tips and the casing. In this publication, different tip clearences (TC) over the rotors of a contra-rotating ductec fan engine (CRF) designed in ELAPSED are analysed numerically with regard to performance and flow stability. In consideration of these specific requirements, there are different effects and possible influences across the respective stages. The results confirm that for efficient operation the clearences must be minimized. The results also show the potential that can be used with the innovative tip clearance control system also developed in the project. Ref. [4].

Keywords
contra-rotating electric fan engine; electric propulsion; tip clearance

1. INTRODUCTION

Electric aircraft engines are a key step in achieving emission-free and climate-neutral aviation. The EU's Green Deal [5] and Flightpath 2050 [6] have highlighted the growing importance of hybrid-electric and fully electric propulsion. The general aviation sector is capable of replacing the combustion engine with an electric engine while maintaining an open propeller as the propulsion-generating component. For the demand of higher flight speeds, altitudes and longer range, it is recommended to employ a shrouded fan engine. This type of engine offers advantages in terms of integration options, noise reduction and safety in case of a blade off incident. However, the efficiency of a ducted fan is lower at low airspeeds compared to an open propeller. The efficiency advantages of open systems decrease as flight speed increases and the operating range is limited to low subsonic flight Mach numbers. Shrouded engines have larger operational potential in terms of flight speed. High subsonic Mach numbers can be achieved. This requires larger total pressure ratios across the propulsor, which can only be achieved efficiently with casing (shroud) and static pressure increase as well as a downstream nozzle. The challenge is to ensure high efficiency despite

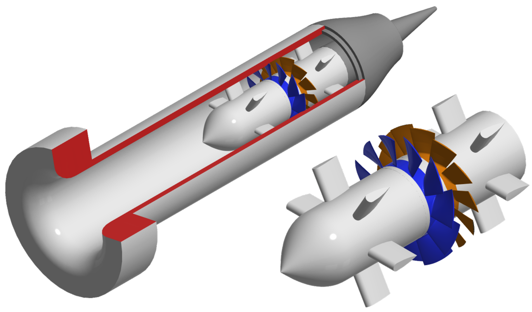

FIG 1. Contra-rotating fan

the higher aerodynamic load on the fan stage and for the entire operating range with different flight speeds. To compensate the efficiency disadvantages, the use of a fan stage consisting of two contra-rotating rotors, see Fig 1, is becoming a possible solution with electric engines. Implementing this approach in conventional gas turbines has so far failed due to its high mechanical complexity. Furthermore a contra-rotating stage is potentially more efficient than a conventional rotor-stator stage, or able to achieve

doi: 10.25967/630467

a higher pressure ratio with the same efficiency. In contrast to a conventional stage, the work performed on the fluid can be separated between the two rotors. Furthermore, the ability to easily and individually adjust the rotational speed of the two rotors opens up an additional degree of freedom to extend the operating range and to select efficient off-design operating points. One of the parameters influencing performance and flow stability during operation is the tip clearance. Various options have been developed in recent years. Due to the complexity and challenging operating conditions of gas turbines, only passive and slow-acting thermal systems have been used so far. An innovative and already patented gap clearance control has been developed at the Institute and has already been presented at the DLRK 2022 Ref. [4]. The objective of this work is to investigate the aerodynamic potential of a possible implementation of such a system on the developed CRF.

2. CONTRA-ROTATING FANS

2.1. Basic Principle

CRFs contain of two rotors rotating in opposite directions. The corresponding velocity diagrams of the two rotor blade rows are shown in Fig 2.

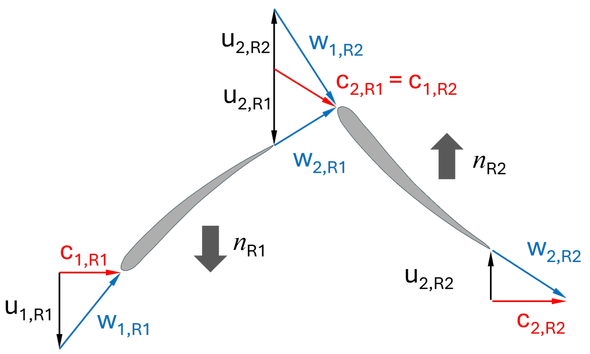

FIG 2. Velocity diagrams of a CRF

The stages rotate in opposite directions with their rotational velocities n_{R1} and n_{R2} creating circumferential component u_{R1} and u_{R2} in the direction of the rotation. These components change depending on the radial position. In this analysis, however, this position is kept constant, therefor the speeds $\mathrm{u}_{1,\mathrm{R1}}$ and $_{2,\mathrm{R1}}$ as well as $\mathrm{u}_{1,\mathrm{R2}}$ and $\mathrm{u}_{2,\mathrm{R2}}$ can be assumed to be the same. Furthermore, component c represents the absolute velocity, whereas w denotes the relative velocity. The advantages of the CRF compared to a conventional rotor-stator stage can be seen in Fig 2. The swirl created by rotor R1 will be compensated with an additional energy input by rotor R2. At first sight, the outcome is similar to a conventional rotor-stator stage. The function of the stator can be defined as increasing the static pressure with the reduction of the swirl, which subsequently results in a reduction of kinetic energy while increasing static energy. In case of a CRF, both rotors contribute to the total pressure increase. The distribution of the total pressure increase over two rotors offers the advantage of reducing the aerodynamic load of the first rotor, which usually leads to better overall efficiency of the stage, an higher overall stage pressure ratio, which enables higher flight Mach numbers, or a combination of both. There are several possibilities for the optimisation of a CRF. In terms of overall efficiency, it would make sense to optimise the load distribution. In this project, the focus is on increasing the overall pressure ratio with the aim of achieving higher flight Mach number as a result of a higher specific thrust.

2.2. Mechanical Setup And Design Aspects

This section contains a brief description of the CRF developed at the institute (see Fig 1) and its test bench. Main design parameters of the test engine for parameter validation can be found in Tab. 1. Generally, the project investigates to achieve high fan pressure ratios at high cruising speeds with a electric CRF. In order to limit time for setting up, reduce costs and increase flexibility, many of the required components for the test engine were manufactured at the institute. A standard SAO nozzle (smooth approach orifice) is used as an inlet into the propulsor to measure the mass flow. Due to the low fluid temperatures, the use of carbon-fibre reinforced polymer (CFRP) in the rotor area is possible. The development and manufacturing process of the blades (see Fig 3) is well described in detail in Ref. [7].
With their low weight, they also enable the high dynamics of the electric motors. The radius of the rear rotor is now 4.2% smaller than intended. This was also taken into account in the CAD and accordingly for the CFD simulations. A tip clearance of 0.30 mm is provided in the design point. For the following numerical investigations, the TC over the rotor stages was taken into account according to Fig 4. The casing design allows different TCs to be set by using inserts with different diameter or the active clearance control system. Both blade rows are powered by individual brushless motors, which are positioned in a face-to-face configuration and mounted to casings that build the hub. Each rotor is connected to the duct by six struts, which also contain all feed and control cables for the motors. The duct is completed by a adjustable convergent nozzle. The main ducting components surrounding the

Parameter	Value	Unit
$\mathrm{PR}_{\mathrm{R1}}$	1.11	-
$\mathrm{PR}_{\mathrm{R2}}$	1.09	-
$\eta_{\mathrm{R1,is}}$	0.838	-
$\eta_{\mathrm{R2,is}}$	0.806	-
n_{R1}	28500	rpm
n_{R2}	25410	rpm

TAB 1. Design parameter of the CRF (Ma_0=0)

FIG 3. CFRP blades manufactured at the institute Ref. [7]

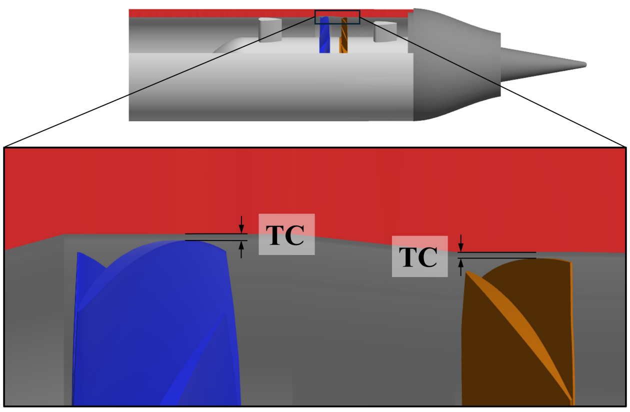

FIG 4. TC design over Rotor 1 and 2

rotating parts are manufactured from aluminium 7050 in order to accommodate blade-off events. The front SAO nozzle and the thrust nozzle are 3D printed to reduce manufacturing complexity.

2.3. Tip clearance and active tip clearance control

The high dynamics of electric engines lead to rapid TC changes and the desired high flight velocities in particular require a higher pressure ratio with a expected greater influence on the TC. The work of Ref. [8] for example investigates the influence of different TCs of a contra-rotating compressor. Overall, the efficiency and stall margin of the rotors appear to be very sensitive to the clearance between the blade tips and the casing. A minimum TC is required to avoid damage caused by collision of the rotor tip with the casing. The operating point with the smallest gap is crucial for determining the minimum TC. However, the TC must not be larger than necessary to achieve the greatest possible efficiency. A variety of factors influence the gap width, including expansion, bending and displacement due to:

- Centrifugal forces (all rotating parts)

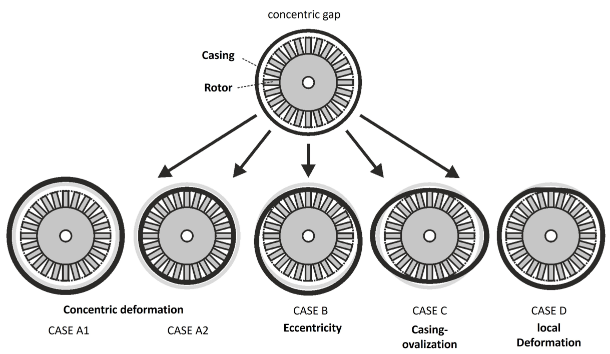

FIG 5. Possible casing deformations Ref. [1]

- Aerodynamic forces and pressure
- Gyroscopic and inertial forces from maneuvers
- Rotor imbalance (dynamic and static)
- Uneven heating

In addition, there are other irreversible changes, e.g. creep processes, erosion and deformations.

The sum of all influences results in different gap changes, see Fig 5.

For the test bench setup build within the ELAPSED project at the institute mainly centrifugal forces must be taken into account. In Ref. [7], the expansion of the CFRP blades as a result of rotation was deeply investigated. The investigations have shown that the blades have significant change in length effecting the TC. The possibility of installing different inserts described in the previous section allows to install the corresponding TC on the test stand for experimental investigations depending on the rotational speed. The objective of this research is to investigate the potential for implementing an active TC control on the developed CRF. With the innovative integration of auxetic structures it is possible to increase or decrease the casing diameter and consequently, the TC, through the application of axially-applied forces. The principle is presented in Fig 6 and also discussed in Ref. [4].

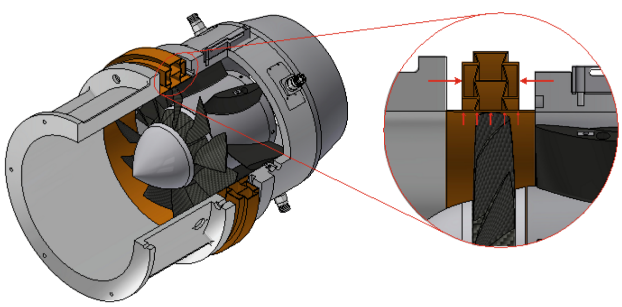

FIG 6. Active TC control system

3. NUMERICAL STUDY

The numerical simulations are performed using ANSYS CFX 2023 R2. The aim of the simulations is to analyse and evaluate the influence of different tip clearances over the rotors of the CRF described before. The simulation and its boundaries will be described in the following section. The rotors are designed with an uneven number of blades, specifically 11 and 13, due to the aerodynamic and acoustic considerations. However, this poses new problems for numerical analyses. Since there is no common divisor other than 1, it is not possible to divide the domains into periodic domains. As shown in Ref. [9] the deviation between complete 360° and a single blade passage is negligible. A single blade passage is thus created as a negative from the CAD and to significantly reduce the computational effort it is modeled by combining the rotational periodicity at the domain boundaries in the circumferential direction. The domain interface was modelled using the Profile Transformation approach with the corresponding pitch angles depending on the number of rotor blades.

3.1. Setup

According to ANSYS, this approach utilises a technique to submit circumferentially averaged data between two adjacent domains. Moderate pitch angles are assumed to be uncritical Ref. [10]. Accordingly, the subsequent simulations discussed in following subsection generally base on the knowledge of Ref. [3]. Fig 7 shows an overview of the single blade passage simulated in Ansys CFX. It contains one-sixth of the total 360 degrees due to the selected number of struts. Thereby each rotor domain contains of two blades, ensuring a pitch ratio that is nearly unity. The focus of the numerical study is not just the design operating line, but also throttled conditions, which are highly influenced by transient effects, a dynamic simulation is carried out based on the transient rotor-stator approach.
In the rotating rotor domain, the shroud's wall is configured as contra-rotating. In contrast, the hub is defined as a rotating wall. To account for friction effects, the no-slip wall approach is used for all wall boundaries. Furthermore, all walls are set to be adiabatic. The application of inflation layers ensures a normalised wall distance y+ <= 4, to include boundary layer effects. Additionally, the mesh is refined at all surfaces. Turbulence is mathematically modeled by applying a separation factor model instead a Shear Stress Transport (SST) Model. All outer boundaries are specified by the entrainment and opening pressure and temperature option to define them as total or static values in case the surface is flowed through into or out of the passage, respectively. Finally, the engine speed is set by the angular velocity of the rotor domain. A mesh convergence study was carried out to find suitable values for element size, refinement size and first layer thickness of the inflation layers, see also Ref. [3].

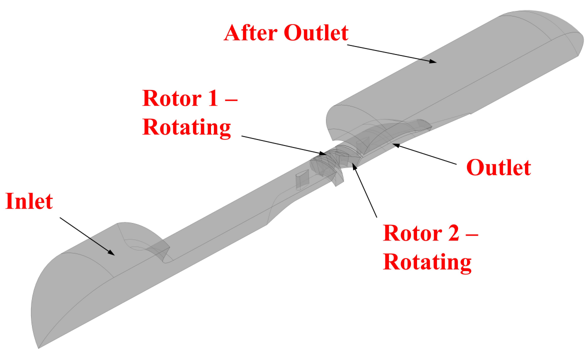

FIG 7. The entire domains used for numerical simulation

TC [mm]	TC/h_{R1}[%]
0.3	0.82
0.5	1.37
1.0	2.74
1.5	4.11
2.0	5.49

TAB 2. Tip clearances used for the numerical study

3.2. Results

In total, more than 100 simulations have been completed. As expected before, the TC has a significant influence on performance and flow stability. Fig 8 illustrates the total pressure ratio of the CRF at 60% and 100% speed with varying TCs and throttle positions. In general, the curves show the typical curve of a turbomachine, see Ref. [11]. At the design point of 100% speed, the pressure ratio is approximately 1.2 (red dot in Fig 8). For the purpose of this analysis, the behaviour at 100% engine speed will be the primary focus. The simulations at 60% rpm are mainly used for subsequent experimental validations. By increasing the TC from 0.3 mm (0.8% blade height) to 2.0 mm (5% blade height), the overall pressure ratio is reduced by 25% from 1.2 to approximately 1.15. An important aspect regarding the CRF is the distribution of the engine pressure ratio on the two rotors, which is illustrated in Figures Fig 9 and Fig 10. At the design point, the pressure ratio is 1.11 for rotor 1 and 1.075 for rotor 2. An increase in the TC to 2 mm results in a reduction in PR_1 to 1.09 (a decrease of 22%) and PR_2 to 1.05 (a decrease of 30%). It can be observed that the proportion of the total pressure increase of rotor 1 is generally higher, while the influence of the TC on the performance is lower. Rotor 2 shows a greater sensitivity to misaligned flow conditions and has a higher pressure loss compared to the design TC. With regard to flow stability, it can be observed that rotor 1 exhibits enhanced stability with a smaller TC. With rotor 2, a larger gap appears to be effective in preventing instability, although this may be at the expense of performance. It can also be seen that a small

TC is required to achieve a higher mass flow at the same back pressure (or nozzle position). The curves showing 0.3 mm and 0.5 mm TC reach stall conditions, but a higher resolution of the nozzle position seems to be necessary. It is very likely that a smaller TC will become unstable a little later. The results show that an individual consideration of both rotors is necessary to optimize the overall stage. The stability behavior of rotor 2 also appears to be strongly influenced by the interaction between the casing boundary layer and the gap width of rotor 1. In order to be able to make more precise statements on this, further simulations are planned. The numerical results confirm the potential of the CRF in terms of a large operating range with high efficiency and stability. There is particular potential in the combination of an innovative tip clearance control system with flexible rotor speed variation and adjustable nozzle geometry.

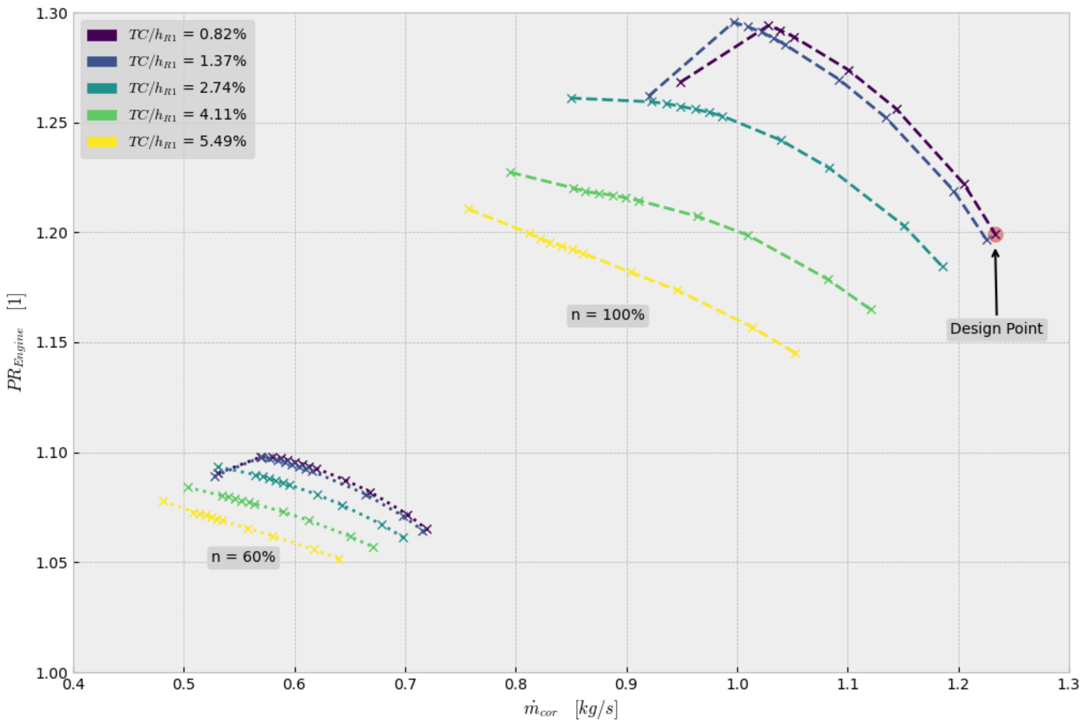

FIG 8. Performance map of the entire stage at 60% and 100% speed against corrected mass flow

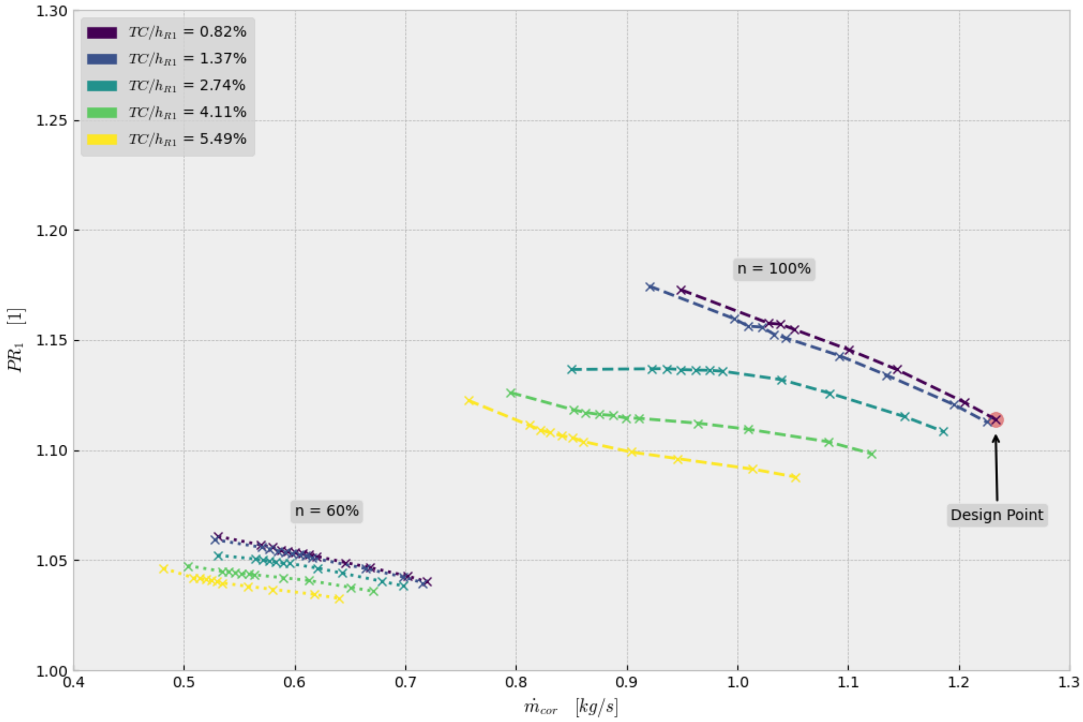

FIG 9. Performance map of rotor 1 at 60% and 100% speed against corrected mass flow

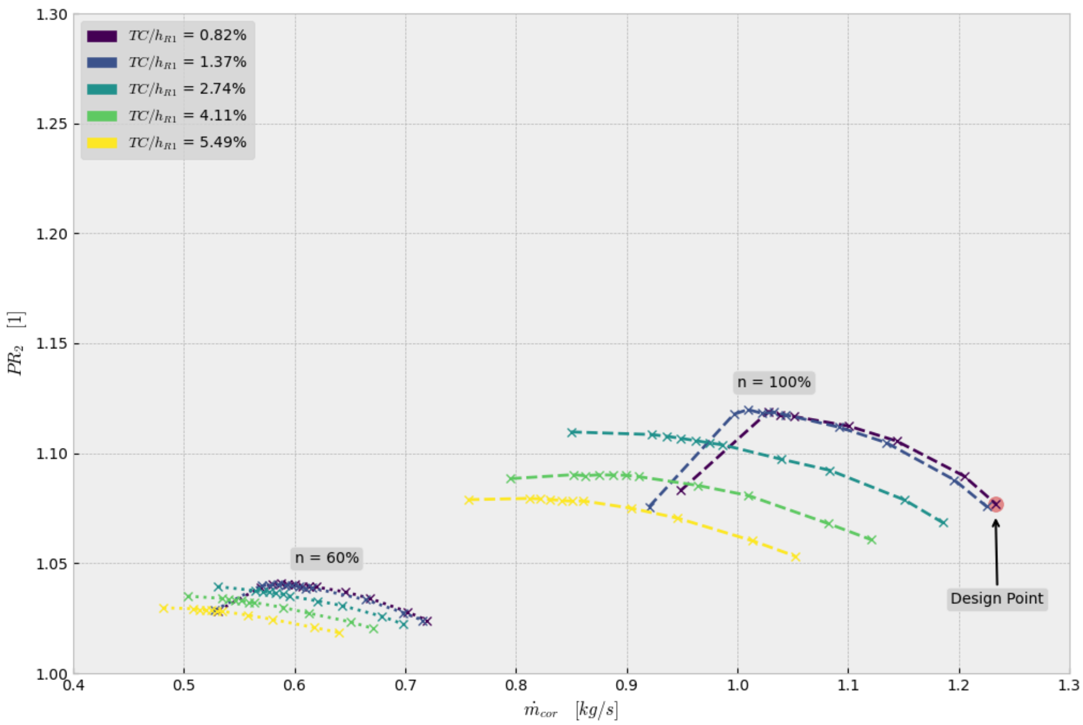

FIG 10. Performance map of rotor 2 at 60% and 100% speed against corrected mass flow

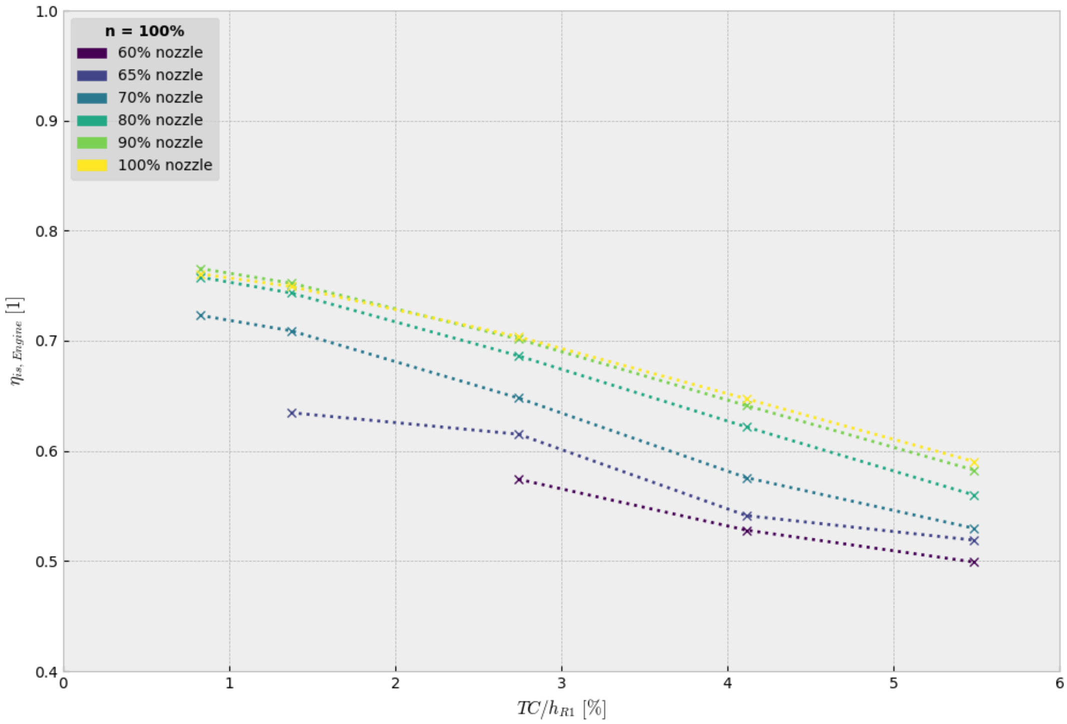

FIG 11. Isentropic efficiency $\eta_{\text{is, Engine}}$ for different throttle positions over TC for 100% speed

4. SUMMARY AND OUTLOOK

The study presents a numerical investigations of a CRF engine. The effects of different TCs during various operational parameter were examined and analysed. In detail, five different TCs at 60% and 100% of rotational velocity were investigated. In order to generate the performance map of the entire stage up 11 different throttle positions were simulated and evaluated. The following results can be identified as a conclusion of this study:

- The negative effect of increasing the TC on the performance is greater on rotor 2 than on rotor 1
- Unstable flow conditions occur in both rotors. However, rotor 2 seem to be the main factor which made the stage stall when the TC gets smaller.
- A reduction in the TC on Rotor 1 appears to have a stabilising effect on the flow.

In order to validate these results, the next step is to carry out measurements on the existing test bench, which has already been set up at the institute. It has been already used to validate an iterative development tool for contra-rotating electric fan engines (Ref [3] and Ref [7]). As mentioned in the previous section, the design of the casing allows to adjust the TC through different inserts. A main challenge is the change in length of the blade at different rotational velocities during operation through centrifugal forces as also described in Ref [7] and the precise measurement of the tip clearance during operation. As previously stated, the engine can be operated at up to 60% of the design rotational velocity in its current configuration. Initial measurements showed a satisfactory correlation between measurements and CFD-simulations up to a nozzle position of 75%. The numerical results demonstrated the sensitivity of the performance and flow stability in relation to the TC. Consequently, precise knowledge of the TC at the specific operating point is therefore essential for validating the CFD results. In this study the TC was changed with the same value. Beside the experimental validation of the presented CFD results, a study of two different TCs above the rotors seems promising for further investigation.

5. ACKNOWLEDGMENTS

This publication is funded by the research project ELAPSED within the cluster dtec.bw – digitalization and technology research center of the federal armed forces of Germany which we gratefully acknowledge. The cluster dtec.bw is funded by the European Union - NextGenerationEU.

Finanziert von der Europäischen Union
NextGenerationEU

Contact address:

niels.herter@unibw.de

References

[1] Andreas Hupfer. Spalte in Fluggasturbinen - Ursachen, Folgen und Optimierungsstrategien am Beispiel des Turboverdichters. Verlag Dr. Hut, München, Deutschland, 2019. ISBN: 978 3 8439 4164 8.

[2] Universität der Bundeswehr München. Electric Aircraft Propulsion - die Zukunft der Flugzeugantriebe, 28.08.2024. https://dtecbw.de/home/forschung/unibw-m/projekt-elapsed.

[3] Sebastian Hawner and Andreas Hupfer. Entwicklung eines CFD-Auslegungstools für ein gegenläufiges, elektrisches Fantriebwerk und Anwendung für eine Parameterstudie im Designpunkt. In Deutsche Luft- und Raumfahrtkongress 2022 (DLRK), Stuttgart, Germany, Sept. 2022.

[4] Niels Herter and Andreas Hupfer. Anwendung einer innovativen Spalthaltungsmaßnahme an einem elektrischen Fantriebwerk. In Deutsche Luft- und Raumfahrtkongress 2022 (DLRK), Stuttgart, Germany, Sept. 2022.

[5] Publications Office of the European Union. Communication from the commission to the european parliament, the european council, the council, the european economic and social committee and the committee of the regions, 2019. https://eur-lex.europa.eu/legal-content/EN/TXT/?qid=1576150542719&uri=COM%3A2019%3A640%3AFIN.

[6] European Commission, Directorate-General for Mobility and Transport, Directorate-General for Research and Innovation. Flightpath 2050: Europe's vision for aviation – maintaining global leadership and serving society's needs, Publication Office, 2011.

[7] Sebastian Hawner, Lukas Rduch, Henry Baumhöfener, and Andreas Hupfer. Development of Carbon Composite Blades within the Context of the Experimental Validation of a CFD-Based Design Tool for Contra-Rotating, Electric Fan Engines. Aerospace, 11(7):527, 2024. DOI: https://doi.org/10.3390/aerospace11070527.

[8] Yangang Wang, Weixiong Chen, Canghai Wu, and Siyuan Ren. Effects of tip clearance size on the performance and tip leakage vortex in dual-rows counter-rotating compressor. Proceedings of the Institution of Mechanical Engineers, Part G. Journal of Aerospace Engineering, 229(11):1953–1965, 2015.

[9] Tobias Ebus, Markus Dietz, and Andreas Hupfer. Expermimental and numerical studies on small contra-rotating electrical ducted fan engines. CEAS Aeronautical Journal, 12:559–571, 2021. DOI: https://doi.org/10.1007/s13272-021-00517-7.

[10] Ansys Inc. Introduction to CFX – Chapter 12 – Moving Zones, Tutorials. 2009.

[11] Hans Rick. Gasturbinen Und Flugantriebe. VDI-Buch. Springer, Berlin, Germany, 2013 edition, Jan. 2014.