



Experimental investigations of the performance of an electric contra-rotating fan as a function of speed ratio

Sebastian Hawner*, Andreas Hupfer 

University of the Bundeswehr Munich, Institute of Aeronautical Engineering, Werner-Heisenberg-Weg 39, Neubiberg, 85577, Germany

ARTICLE INFO

Communicated by Cummings Russell

Keywords:

Contra-rotating electric propulsor
CFD-based design tool
Speed ratio adjustment
Tip clearance effects

ABSTRACT

Contra-rotating ducted fans are a promising concept for expanding the design space of electric propulsion systems towards high subsonic flight speeds. However, experimental data for small-scale, contra-rotating stages remain limited, particularly under off-design conditions. This study addresses this gap by experimentally investigating the influence of rotor speed ratio adjustments on the off-design performance in terms of the stability behavior of such a small-scale stage, designed using an extended iterative and CFD-based tool chain. The experimental data are provided in the form of performance maps and radial flow measurements under design and off-design conditions. The results confirm that the design operating conditions, including a high stage total pressure ratio of 1.21, are achieved. This demonstrates that the iterative design approach is suitable for small-scale propulsion systems with pronounced three-dimensional effects, particularly secondary flows. Local flow-field measurements and CFD evaluations provide initial insights into mechanisms related to the stability limit and rotor-rotor interaction. Hub corner separation appears to increase with throttling and may contribute to extensive flow separation in the rear rotor. Selecting an appropriate rotor speed ratio seems to enable aerodynamic unloading of the stability-critical front rotor hub, thereby shifting the stage stability limit towards higher throttling levels. An increase in rear rotor speed by 20 % relative to the design speed ratio shifts the stability limit by approximately 5 % in normalized nozzle area, thereby extending the operational envelope of the propulsion system.

1. Introduction

To date, electric propulsion systems in aviation have been limited to relatively low flight speeds, primarily due to the use of open propellers. In addition, the currently prevalent battery technology restricts both flight duration and maximum power output because of its inherently low gravimetric energy density [1–3]. Recent advances in hybrid propulsion concepts, including stationary gas turbines and hydrogen fuel cells [1,2,4] as onboard power generators, make electric propulsion systems feasible for use cases that were previously considered unsuitable [5].

To overcome the limitations of open propellers, ducted fan stages can be employed. As summarized by Qian et al. [6], ducted rotors have so far been mainly used in eVTOL (electric vertical take-off and landing) applications, where contra-rotating configurations enable higher thrust at comparable compactness, albeit at slightly reduced efficiency. In combination with shrouding, such configurations may allow for higher total pressure ratios, thereby extending the feasible design space toward higher subsonic flight speeds.

In conventional gas turbines, contra-rotating rotors have not been widely adopted due to their high implementation complexity. In con-

trast, this concept can be readily realized in electric propulsion systems by using individual electric motors to drive each rotor independently. Additionally, this introduces a further degree of freedom in the form of the rotor speed ratio, which may extend the off-design operating range of the propulsion system.

The contra-rotating concept has been investigated in several studies with respect to various design parameters. Manas and Pradeep [7] describe contra-rotating fan stages as a promising concept because they can provide a higher pressure rise per unit engine length and a swirl-free outflow. However, their practical application remains challenged by mechanical complexity, noise, and stability issues under distorted inflow. They investigated stall inception characteristics as a function of speed ratio, highlighting the importance of rotor-speed matching. Mistry and Pradeep [8] identified an optimal axial spacing between the two rotors in terms of overall stage performance and observed two distinct stall regimes: stall at the rear rotor and, under increased throttling, stall at both rotors. Furthermore, the performance of the rear rotor can be significantly improved by adjusting the rotor speed ratio. Shahriyari et al. [9] investigated post-stall behavior and showed that the stage becomes more resistant to stall and surge when the rear rotor operates at a higher rotational speed than in the design condition. In contrast,

* Corresponding author.

E-mail addresses: sebastian.hawner@unibw.de (S. Hawner), andreas.hupfer@unibw.de (A. Hupfer).

<https://doi.org/10.1016/j.ast.2026.113930>

Received 30 April 2026; Received in revised form 9 September 2026; Accepted 28 September 2026

Available online 1 October 2026

1270-9638/© 2026 The Author(s). Published by Elsevier Masson SAS. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

Open access funding enabled and organized by Projekt DEAL.

Nomenclature*Formula symbols*

A	area (m ²)
$\hat{A}$	normalized area (–)
AS	axial spacing (m)
D	diffusion factor (–)
Ma	Mach number (–)
P	power (W)
TC	tip clearance (m)
T	temperature (K)
c	absolute speed (m/s)
c_p	specific heat capacity (J/kg/K)
d	blade thickness (m)
e	error (–)
i	blade count (–)
n	rotational speed (1/s)
p	pressure (Pa)
r	radius (m)
t/s	inverse of the solidity (–)
u	circumferential speed (m/s)
w	specific work (J/kg)
w	relative speed (m/s)
$\dot{m}$	mass flow rate (kg/s)
α	absolute flow angle (°)
β	relative flow angle (°)
η	efficiency (–)
λ	speed ratio (–)
π	pressure ratio (–)
ρ	density (kg/m ³)
τ	temperature ratio (–)

Indices

0	ambient
1	SAO nozzle
9	nozzle exit
1.1	front rotor inlet
1.2	inter-rotor plane
1.3	rear rotor outlet
D	Design
FN	fixed nozzle
ML	mean line
R	rotor
VN	variable nozzle
corr.	corrected
k	exaggeration
t	total value

Abbreviations

CFD	computational fluid dynamics
CFRP	carbon fiber reinforced plastics
SAO	smooth approach orifice
URANS	unsteady Reynolds-averaged Navier-Stokes

0	ambient	1.1	front rotor inlet
1	SAO nozzle	1.2	inter-rotor plane
9	nozzle exit	1.3	rear rotor outlet.

small-scale blades due to the challenges associated with detailed flow measurements. This lack of data seems to be particularly relevant for electrically driven contra-rotating ducted fans, where the independent control of both rotor speeds enables a systematic variation of the rotor speed ratio.

To address this gap, the project *ELAPSED (Electrified Aircraft Propulsion - safe, efficient, digitally linked)*, part of the dtec.bw research initiative, focuses on the investigation of electric, small-scale, contra-rotating ducted propulsion systems. It builds directly on previous projects conducted at the institute, in which this general approach was initially validated in [15]. In addition, in [16] it could be demonstrated through experimental and numerical studies that the performance of such small-scale blades is strongly influenced by secondary flows. To account for these effects, a CFD-based iterative design tool chain was developed and experimentally validated using a rotor-stator configuration [17]. While conventional design methods rely primarily on empirical correlations, the present approach directly integrates CFD to determine initially unknown parameters such as rotor efficiency.

This study therefore pursues three main objectives. First, the CFD-based iterative tool chain is applied to design an innovative propulsion system with a contra-rotating rotor-rotor stage and a high stage total pressure ratio, which serves as the basis for the subsequent investigations. The resulting design is assessed experimentally under design conditions and compared to the CFD predictions regarding the global performance specifications, such as the stage total pressure ratio. Second, the influence of speed-ratio variation on off-design performance is investigated experimentally as the main objective of this work. For this purpose, the general stability behavior of the system is first analyzed, including rotor-rotor interaction under throttled operating conditions. Based on this analysis, it is assessed whether the additional degree of freedom provided by the electric motors can shift the stability limit and thereby enlarge the operating range. Third, the influence of tip clearance is examined as an important design parameter of such small-scale rotors.

Together, these investigations address the lack of experimental data for small-scale contra-rotating stages by providing design and off-design data in the form of performance maps and radial flow measurements. These local flow-field measurements enable an initial identification of mechanisms related to the stability limit and global performance. While CFD is used only to validate the design tool with respect to global performance parameters, the experimental data provide a basis for future, more detailed comparisons between experimental and numerical results.

2. Design tool chain adaptation and utilization

A CFD-based design tool chain for ducted electric propulsors was previously introduced and validated in [17]. This original implementation was limited to rotor-stator configurations and was applied to a stage with a moderate total pressure ratio compared with the system considered in the present work. Here, the tool is extended and validated for a contra-rotating rotor-rotor stage, shown in Fig. 1, with a high stage total pressure ratio. The mechanical design is briefly outlined before describing the adapted design methodology based on Fig. 2.

2.1. Contra-rotating mechanical design

The measurement planes j are defined as follows:

Additional letter-based indices identify system components as shown in Fig. 1. The mass flow rate is determined using a calibrated SAO nozzle (smooth approach orifice) based on ambient conditions and pressure

Wang et al. [10] reported based on numerical studies that instability originates at the rear rotor, with tip clearance having a strong influence on stall margin and efficiency.

However, experimental investigations of small-scale, highly loaded contra-rotating ducted fans remain limited. Experimental investigations by Yu et al. [11] demonstrated that an increase in tip clearance by 1 % of blade height reduces efficiency by approximately 1.3 % to 2.9 %, with sensitivity increasing at higher blade loading, as confirmed by Vala et al. [12] and Ryu et al. [13]. Wu et al. [14] further analyzed the influence of tip clearance on noise and vibration in axial fans and compressors and highlighted the lack of experimental data for highly loaded

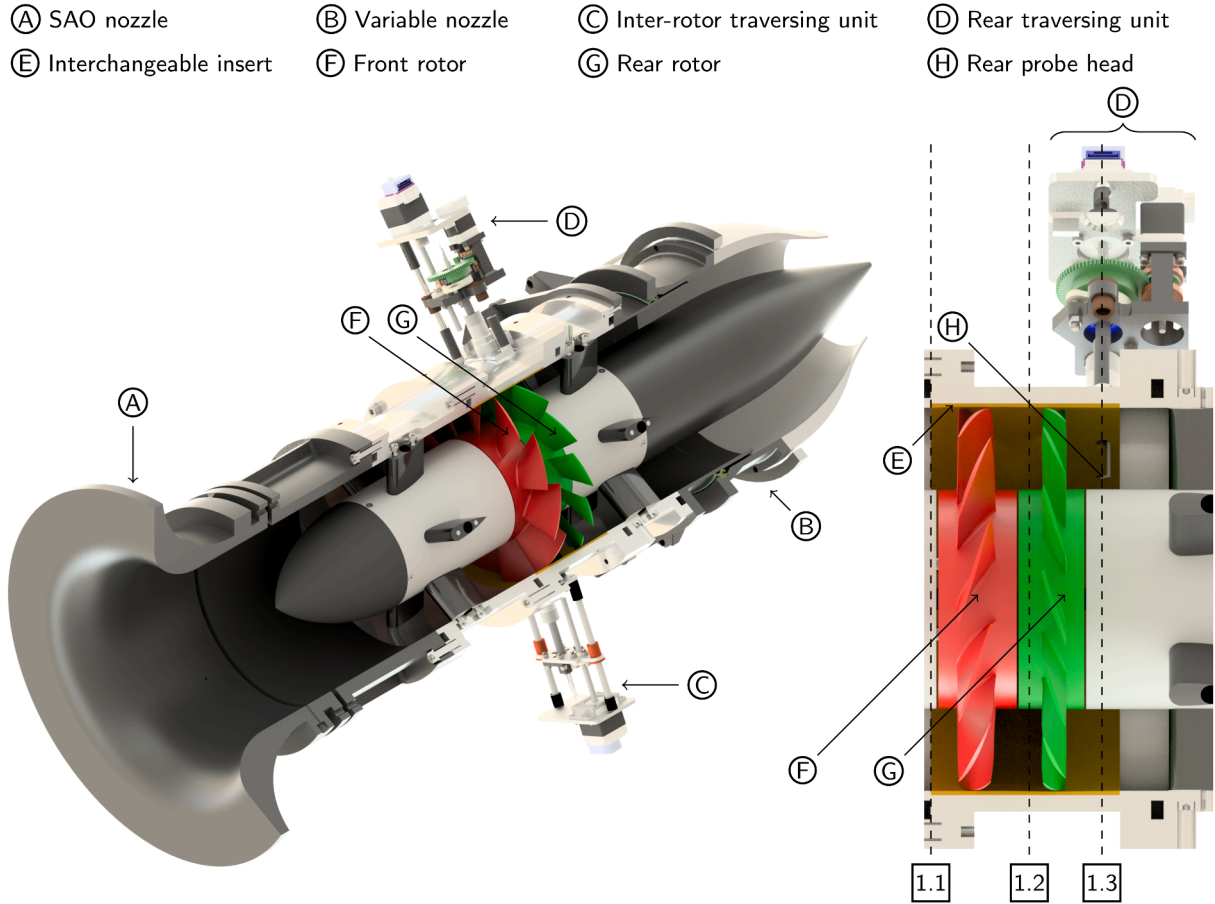


Fig. 1. Rendering of the investigated engine with the SAO nozzle, shown in shortened form, and stage numbering (right).

measurements at plane 1. The contra-rotating rotors between planes 1.1 and 1.3 form the fan stage. Rotor blades were initially manufactured from carbon fiber reinforced plastics (CFRP) [17]; here, they are replaced with blisks machined from aluminum EN AW-7075 to reduce manufacturing and assembly tolerances and ensure predictable strain-untwisting behavior. Finite element analyzes indicate significantly reduced geometric deviations during operation compared to CFRP blades. Tip clearance (TC) is adjusted using interchangeable 3D-printed casing inserts, marked by an *E* in Fig. 1. The rotors are driven by two brushless motors arranged face-to-face, each controlled and powered individually by an electronic speed controller. Water and compressed-air cooling are applied. The maximum shaft power of the front rotor is prescribed as 13.5 kW based on the characteristics of the available motors and the limitations of the power supply. This value accounts for system losses and is selected primarily according to the motor speed–power relationship.

Structural integration is realized via six struts containing wiring, instrumentation, and cooling lines. All components are sealed to prevent leakage into the main flow. Two thrust nozzle configurations are used. Fixed nozzles enable measurements at discrete operating points, ensuring comparability with CFD simulations. A variable nozzle, indicated by label *B* in Fig. 1, enables continuous throttle variation. It varies the minimum cross-section via axial displacement of the outer nozzle duct and is actuated by stepper motors. Apart from the mass flow rate measurement, two miniature five hole pressure probes are installed after each rotor in two traversing units, indication *C*, *D*, and *H*, discussed in detail below.

2.2. Design tool chain principle

The primary objective of the tool chain is to determine a propulsion system design for a given operating point. It provides a systematic procedure to translate prescribed specifications into the aerodynamic and geometric design of a contra-rotating ducted fan stage. Starting from the required operating conditions, the tool chain derives the required stage performance and determines the corresponding geometry including both rotors. Quantities that cannot be reliably prescribed in advance for this small-scale configuration, such as rotor efficiency, are obtained from CFD-based evaluations and updated iteratively.

The workflow, illustrated in Fig. 2 and previously validated with respect to its general functionality in [17], consists of three iterative steps: (1) aerodynamic design and performance calculation, (2) automated geometry generation, and (3) CFD analysis. The aerodynamic design is based on a quasi-one-dimensional approach, assuming steady operating conditions and uniform flow properties within each flow plane. Although the actual flow is characterized by superimposed, at least periodic, unsteady components, it is treated as time-averaged and quasi-steady for the initial calculations. A meanline analysis of streamlines on an axial flow surface at midspan is combined with a simplified radial-equilibrium approach to determine the velocity distributions and blade geometry. Input parameters are classified as fixed (design point specifications) or iterative (e.g., rotor efficiencies), which are updated until convergence is achieved. Table 1 summarizes the key parameters of the engine designed here that are required for the following discussion. The design point is defined at steady-state test bench conditions ($Ma_0 = 0$)

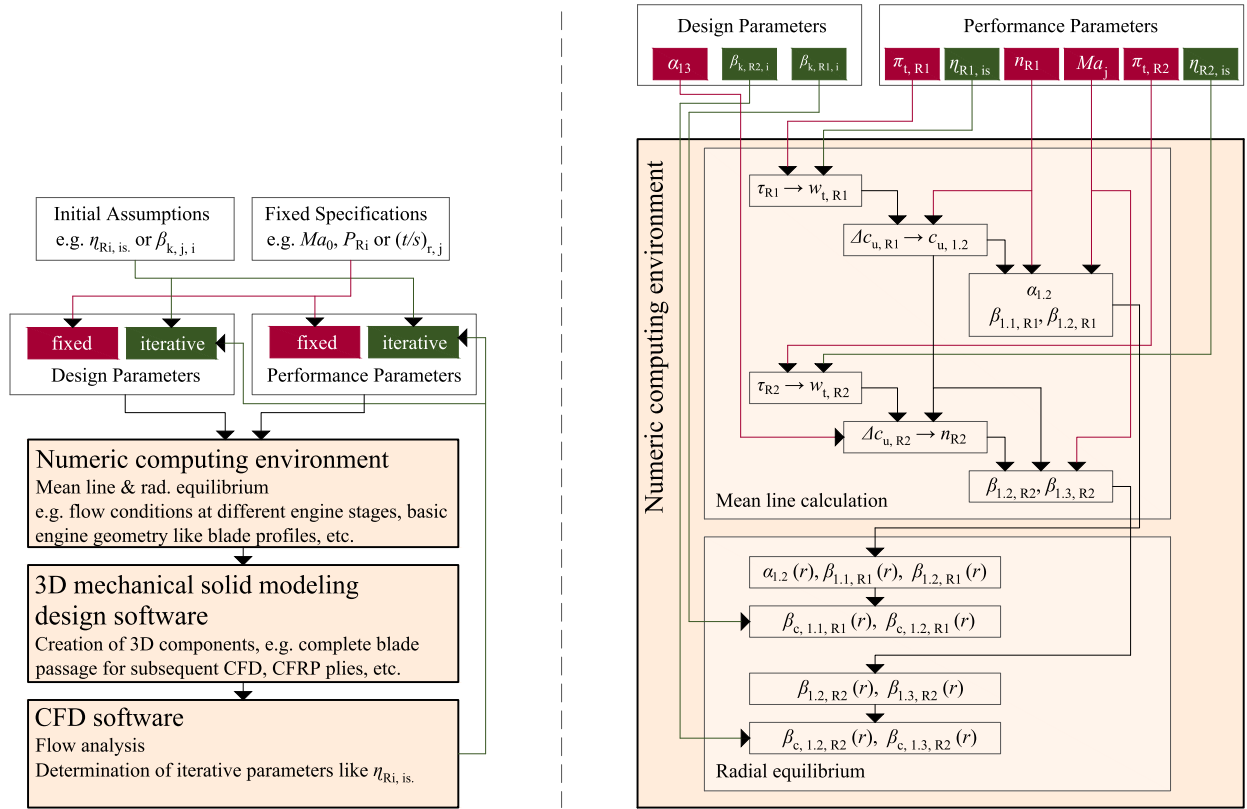


Fig. 2. Overview of the design tool chain (left) and the implemented engine and geometry calculation workflow in the numerical environment (right).

to match the experimental setup. The pressure ratios are defined as follows:

$$\pi_{t, R1} = \frac{p_{t, 1, 2}}{p_{t, 1, 1}}; \quad \pi_{t, R2} = \frac{p_{t, 1, 3}}{p_{t, 1, 2}}; \quad \pi_{t, R1 R2} = \frac{p_{t, 1, 3}}{p_{t, 1, 1}}. \quad (1)$$

In correspondence with the initial goal of enabling higher flight speeds, a relatively large total pressure ratio of $\pi_{t, R1 R2} = 1.21$ was chosen across the entire stage. Using the isentropic efficiency of the first rotor, $\eta_{R1, is}$, initially estimated and obtained iteratively, the specific work $w_{t, R1}$ can be calculated as follows:

$$w_{t, R1} = c_p \cdot (T_{t, 1, 2} - T_{t, 1, 1}) = c_p \cdot \frac{T_0}{\eta_{R1, is}} \cdot \left(\left(\frac{\pi_{t, R1}}{\pi_{t, R1}} \right)^{\frac{\kappa-1}{\kappa}} - 1 \right). \quad (2)$$

Based on the specific work, the mass flow rate $\dot{m}$ can then be calculated from the maximum shaft power P_{R1} . The outer radius is determined from the specified hub radius r_{hub} and the axial Mach numbers Ma at the respective planes. These Mach numbers were selected and adjusted such that constant hub and shroud radii were obtained, as shown in Table 1.

Assuming swirl-free inflow ($\alpha_{1, 1} = 90^\circ$), the flow component in circumferential direction $c_{u, 1, 2}$ is determined from the rotor's rotational speed n_{R1} :

$$c_{u, 1, 2} = \frac{w_{t, R1}}{2 \cdot \pi \cdot r_{1, 2, R1, ML} \cdot n_{R1}}. \quad (3)$$

An adiabatic inlet is assumed, such that $T_{t, 1, 1} = T_{t, 0} = T_0$, and the specific heat capacity of air is denoted by c_p . Both rotors impart work to the flow by deflecting it. To facilitate the following discussion, velocity triangles representing the flow conditions within the axial flow surface at the design point are introduced in Fig. 3. The rear rotor is designed such that the swirl generated by the front rotor is completely removed, resulting in axial outflow ($\alpha_{1, 3} = 90^\circ$):

$$\Delta c_{u, R2} = -c_{u, 1, 2} = -\Delta c_{u, R1}. \quad (4)$$

Rotor interaction is therefore directly coupled through the velocity triangles and Euler's turbine equation:

$$n_{R2} = \frac{-w_{t, R2}}{2 \cdot \pi \cdot r_{1, 2, R2, ML} \cdot c_{u, 1, 2}}. \quad (5)$$

The simplified radial equilibrium follows a free-vortex design ($c_u \cdot r = const$), which is used to determine the spatial distributions of the absolute and relative flow angles α and β . Blade metal angles are corrected by the exaggerations β_k to account for underturning effects. These corrections are iteratively adjusted. For the adopted free-vortex design, the specific enthalpy rise is constant along the span, resulting in the highest flow deflection near the hub. This region is therefore particularly susceptible to flow separation. The risk of flow separation is assessed using the diffusion factor D , which is determined spanwise from the relative velocity components and the pitch-to-chord ratio t/s . The following equation illustrates the calculation for the front rotor

$$D_{R1}(r) = 1 - \frac{w_{1, 2, R1}(r)}{w_{1, 1}(r)} + \frac{\Delta w_{u, R1} \cdot \left(\frac{t}{s} \right)_{R1}(r)}{2 \cdot w_{1, 1}(r)} \quad (6)$$

which should remain below approximately 0.6 [18]. For a given specific work input, D can be reduced by increasing axial Mach number, hub radius, or rotational speed. However, increasing hub radius or axial Mach number reduces blade height, thereby intensifying three-dimensional effects such as tip clearance losses. Rotational speed is limited by mechanical constraints. As a compromise, axial Mach numbers of approximately 0.2 and high rotational speeds are selected, resulting in a blade height of about 37 mm. Diffusion factors range from 0.21 to 0.47 over most of the span, increasing near the hub to 0.56 (below 15 % span), where flow separation is more likely. Due to the lower loading of the rear rotor its diffusion factor is less critical, ranging from 0.22 to 0.46 over the entire span.

The second step involves automated CAD-based geometry generation. For instance, the individual radial blade profiles are determined

Table 1

Key performance (left) and design (right) parameters; fixed values in regular font, iterative values in italics.

Parameter	Value	Unit
Ma_0	0	–
π_{t,R_1}	1.11	–
π_{t,R_2}	1.09	–
π_{SAO}	<i>0.997</i>	–
π_n	<i>0.995</i>	–
$\eta_{R_1, is}$	<i>0.838</i>	–
$\eta_{R_2, is}$	<i>0.806</i>	–
P_{R_1}	13.50	kW
P_{R_2}	<i>12.50</i>	kW
n_{R_1}	28500	rpm
n_{R_2}	<i>25410</i>	rpm
$Ma_{1,1}$	0.21	–
$Ma_{1,2}$	<i>0.19</i>	–
$Ma_{1,3}$	<i>0.18</i>	–
$\dot{m}$	<i>1.29</i>	kg/s
T	210	N

Parameter	Value R_1	(Final) Value R_2	Unit
$t/s(r_{hub})$	0.90	0.95	–
$t/s(r_{ML})$	0.90	1.00	–
$t/s(r_{shroud})$	0.90	1.05	–
r_{hub}	49.90	49.90	mm
r_{shroud}	<i>87.57</i>	<i>87.57</i>	mm
$d_{max}/s(r_{hub})$	0.100	0.185	–
$d_{max}/s(r_{shroud})$	0.035	0.047	–
i_B	11	13	–
AS	15		mm
TC	0.30	0.30	mm
$\beta_{k, hub}$	<i>10.80</i>	<i>6.50</i>	deg
$\beta_{k, ML}$	<i>6.80</i>	<i>4.00</i>	deg
$\beta_{k, shroud}$	<i>1.00</i>	<i>–1.75</i>	deg

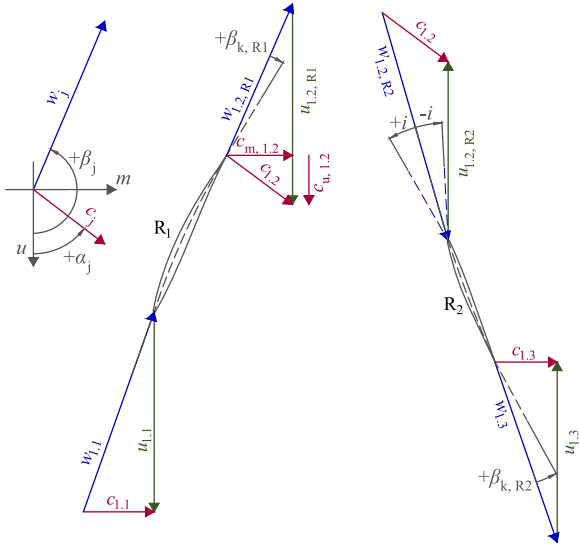


Fig. 3. Schematic illustration of the blade cascade and the corresponding velocity triangles.

from the blade metal angles, which in turn result from the radial equilibrium. Certain design choices were kept fixed, such as the use of the NACA 65 airfoil profile, although this could be modified in future investigations. To obtain the spatial geometry, these profiles are stacked according to their centroids. In addition to the calculated parameters, further design constraints are considered, excerpts of which have already been summarized in Table 1. Blade chord length is minimized to achieve a compact design, depending on blade count i and solidity. The rear rotor uses a higher blade count to mitigate unsteady and potentially incidence-affected off-design inflow. Different prime numbers are selected for blade counts to reduce interaction effects.

Pitch-to-chord ratio directly influences the diffusion factor and is adjusted accordingly. Axial spacing AS is chosen as a compromise between aerodynamic performance, noise, and measurement accessibility. While a larger gap may promote wake dissipation and mixing downstream of the first rotor, it can also increase noise generation [8,19]. The design tip clearance is set to 0.3 mm ($\approx 0.8\%$ blade height), balancing performance and mechanical safety.

For later discussion, a potential incidence angle i at the leading edge of the second rotor is introduced in Fig. 3. Its sign convention was chosen such that a negative incidence angle corresponds to a clock-wise rotation of the relative flow.

2.3. Numerical setup

The flow is simulated using a compressible unsteady Reynolds-averaged Navier-Stokes (URANS) approach with the shear stress transport (SST) turbulence model of Menter [20]. The transient rotor-stator interface is used to capture rotor interaction effects through pointwise transfer of flow quantities within a sliding-mesh approach between the domains. Wall-resolved boundary layers are ensured by maintaining $y^+ < 1$ using prism layers. To reduce computational cost, a single blade passage is simulated. The two rotor domains include two blades each and are modeled as rotating domains. The blade passage's pitch within the stationary domains inlet, nozzle and downstream domain was chosen to 60° due to the number of struts, resulting in overall pitch ratios between the domains close to unity. The resulting blade passage is shown in Fig. 4. Close to design conditions, the simulation of a single blade passage appears to be a valid approach [21,22]. However, spike-type stall inception seems to require a sufficient number of blade passages; otherwise, stall may be predicted only at excessive throttling. In addition, the predicted number of rotating-stall cells and their propagation speed seem to depend on the forced periodicity imposed by the blade-passage approach [23,24]. Therefore, future simulations intended to support experimentally identified mechanisms may require full-annulus modeling to capture off-design phenomena such as rotating stall.

The inlet and downstream domains extend the blade passage to include an undisturbed flow region between the overall inlet and outlet, marked in orange, and the engine itself. These inlet and outlet boundaries are modeled as openings with the entrainment option, which defines the prescribed temperature and pressure values as either total or static quantities depending on whether the flow enters or leaves the domain. These boundary pressure and temperature are set according to the International Standard Atmosphere [25].

Since unsteady simulations are performed, convergence is assessed using the root-mean-square residuals and two additional criteria following common recommendations for transient calculations. The residuals are required to decrease to at least below 1×10^{-4} and, in design operation, below 5×10^{-5} . As a first criterion, the ratio between the arithmetic averages evaluated over two time intervals, corresponding to one full

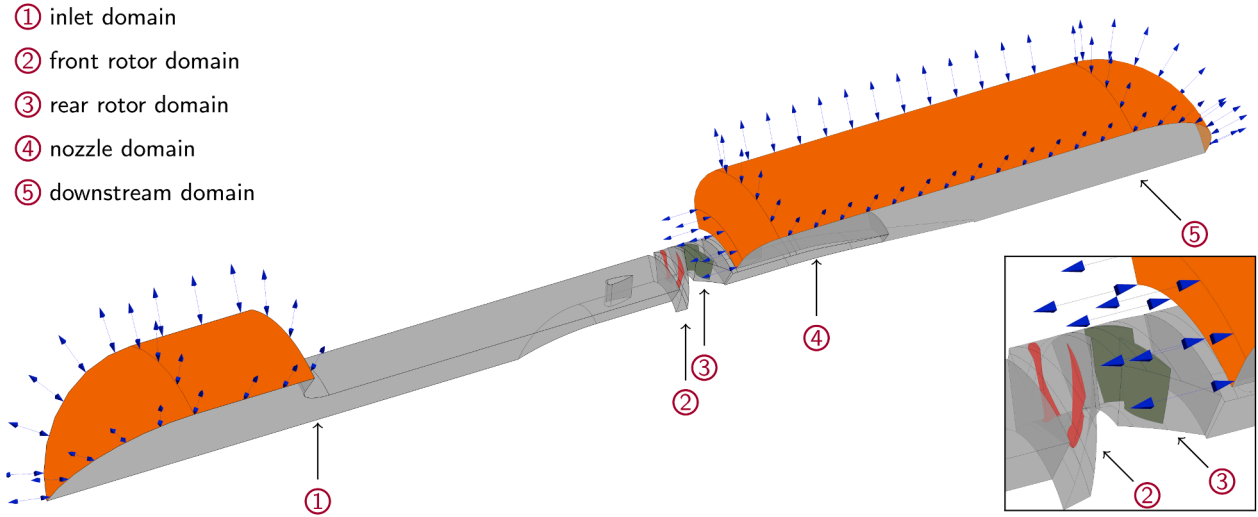


Fig. 4. Simulated blade passage with a zoomed view of the rotor domains in the framed box on the right.

and one half revolution of the front rotor, is evaluated for the mass flow rate and the total pressure rises across each rotor and must remain below 0.5 %. As a second criterion, the coefficient of variation of the mass flow rate, defined as the standard deviation divided by the mean over one complete front rotor revolution, must also remain below 0.5 %. Once these convergence requirements are satisfied, all output quantities are obtained by averaging the respective variable over the time steps corresponding to one complete revolution of the front rotor.

Based on the Richardson Extrapolation [26] mesh convergence is assessed using three grids with refinement ratios greater than 1.3 between their average element sizes h . The mesh study is first conducted with the highest temporal resolution of 750 time steps per front rotor revolution, while the influence of temporal resolution is investigated separately in a second step below. The finest grid, used as reference, contains approximately 23×10^6 elements, whereas the coarsest grid contains about 4×10^6 elements. The first layer thickness of 5×10^{-7} m is kept constant throughout the mesh study to ensure $y^+ < 1$ and thus avoid empirical wall functions by using a wall-resolved boundary-layer treatment that resolves the viscous sublayer. For all meshes, a growth rate below 1.3 is chosen, increasing toward the coarser meshes. In combination with this growth rate, the number of prism layers is set individually to achieve a smooth transition from the prism layers to the core mesh. In addition, the element size is refined on all surfaces. Within the design tool chain, only design operation is considered. In contrast, the mesh study is performed both at the design operating point and under throttled conditions using a normalized nozzle area of 70 %. The throttled case represents a more aerodynamically critical operating condition, with potentially increased loss mechanisms, stronger velocity or pressure gradients, potentially larger separation, and unsteady interaction effects. It is therefore expected to be more sensitive to mesh and time-step resolution and is included to ensure that the same mesh setup can be used for future detailed numerical off-design studies. In order to improve readability, a normalized indication relates a certain considered throttle condition in terms of simulated nozzle area A_9 with the design nozzle area $A_{9,D}$:

$$\hat{A}_9 = \frac{A_9}{A_{9,D}}. \quad (7)$$

To select an appropriate mesh, six parameters are evaluated each. Within Fig. 5, the corresponding relative deviations with respect to the finest mesh, e_r , and the extrapolated results, $e_{r,extra}$, are plotted as a function of the average cell size h . The upper plot shows the throttled operation, whereas the lower plot shows the design operation. Whereas during the previous design the total pressure ratios were specified indi-

Table 2

Apparent orders p and numerical uncertainties (GCI) of the medium sized mesh.

ϕ	$\Delta p_{t,R2,0}$	$\Delta p_{t,R1,0}$	$\Delta p_{t,R2}$	η_{R1}	η_{R2}	$\dot{m}$
$p_\phi, \hat{A}_9 = 70\%$	4.91	12.74	7.40	19.25	6.36	12.06
$p_\phi, \hat{A}_9 = 100\%$	12.18	11.94	6.05	4.68	4.88	3.76
$GCI_{I_\phi} \text{ in } \%, \hat{A}_9 = 70\%$	0.19	0.02	0.14	0.02	0.12	0.05
$GCI_{I_\phi} \text{ in } \%, \hat{A}_9 = 100\%$	0.04	0.10	0.07	0.64	0.18	0.22

vidually for each rotor, the total pressure increases $\Delta p_{t,0}$ evaluated here are related to the ambient conditions, taking the losses of the inlet into account. Expressed by the additional index 0, this anticipates that later the experimental data are also measured against the ambient as there is no measurement directly ahead of the front rotor.

Relative deviations with respect to the finest grid and Richardson-extrapolated solution remain below 0.5 %, indicating negligible discretization error on this mesh. For the coarsest mesh, deviations reach up to approximately 7 % under throttled conditions, whereas for the medium-sized mesh they remain below 2 %, even when the extrapolated solution is used as reference. This indicates that the medium grid is sufficient to capture the relevant solution features. Consequently, it is chosen for all further simulations as a trade-off between computing time and accuracy. In addition to the first prism layer thickness given above, a growth rate of 1.2, a wall refinement of 1 mm, reduced to 0.5 mm in the tip region, and an unstructured mesh outside the prism layers are used within the rotor domains. The respective apparent orders p and the numerical uncertainties, given in form of the grid convergence index GCI, of this mesh are summarized in Table 2.

Time-step sensitivity is also evaluated on the reference grid, corresponding to the finest grid, to provide the same basis for both studies. For this purpose, 33 to 750 time steps per revolution of the front rotor are investigated, as shown in Fig. 6. While deviations remain small under design conditions, coarse temporal resolution leads to increased errors under throttled operation. A significant increase in deviations is observed for 33 time steps per revolution while simulations with more than 250 time steps per revolution are considered sufficient. 375 time steps are selected as a trade-off, corresponding to a time step width of $5.6 \mu\text{s}$. Applying this time-step size to the selected medium grid results in deviations below 3 % and below 1.2 % under throttled and design conditions when the fine grid with the highest temporal resolution of 750 time steps per revolution is used as reference. Using this setup, convergence of an individual operating point according to the criteria given above is typically reached within 3000 to 4000 time steps.

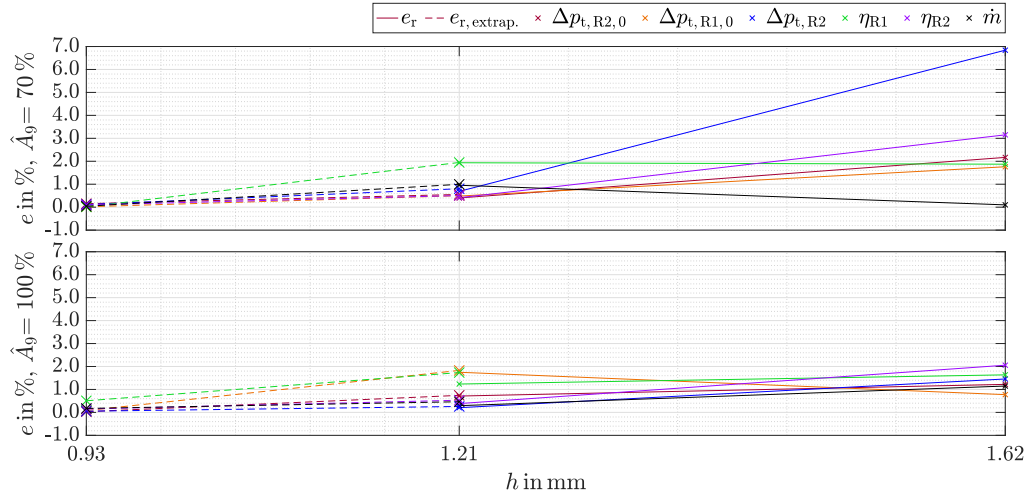


Fig. 5. Mesh convergence as a function of the average element size h : relative deviations with respect to the finest mesh, e_r , and the extrapolated solution, $e_{r,extrap}$, under design and throttled conditions (center, bottom).

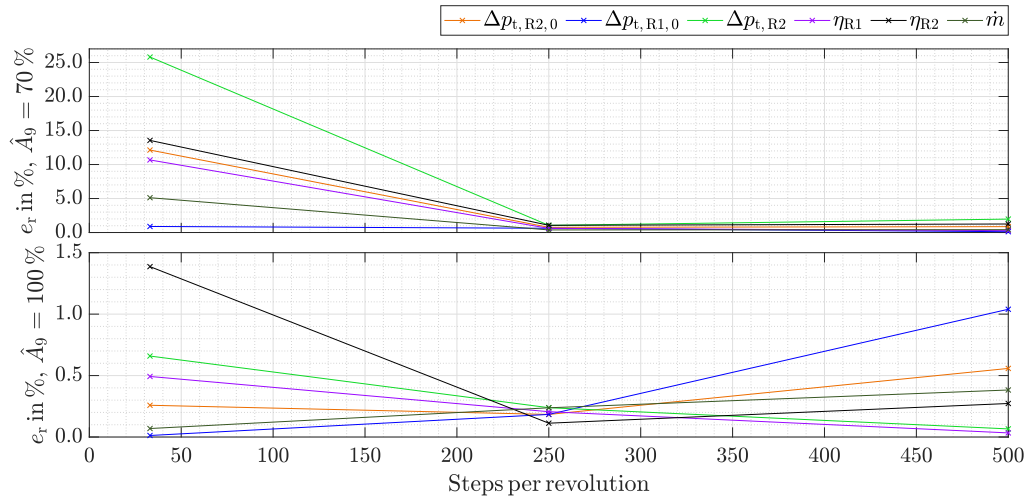


Fig. 6. Convergence study regarding the number of time steps per front rotor revolution: relative deviation e_r related to the simulation results carried out with 750 steps.

In general, the convergence study was conducted twice: once for the first design iteration and once for the final iteration presented here. Apart from that, the general setup, including the use of unsteady simulations and a single blade passage, remains the same throughout the iterative design procedure and the validation simulations in design and off-design operation discussed below. However, compared with the initial design simulations described in [27], the simulations discussed here are based on the actual hardware geometry, with a slightly larger rotor outer radius of approximately 0.2 mm due to manufacturing tolerances, while the tip clearance remains as initially specified.

The iterative design finally converged to the parameters listed in Table 1, including a mass flow rate of 1.29 kg/s and a thrust of 210 N. The aluminum rotors manufactured in accordance with the final design are shown in Fig. 7.

Design convergence was assessed by comparing selected simulation results of the current iteration with those of the previous iteration. This criterion was applied to all quantities, regardless of whether they were iteratively transferred from the previous iteration, such as the rotor efficiencies, or prescribed as fixed targets, such as the total pressure rises across the rotors. For convergence, all deviations had to remain below 0.5 % which was achieved after six iterations.

Despite unsteady inflow, both rotors achieve comparable isentropic efficiencies. The rear rotor requires smaller deviation corrections due to its lower loading. Recapitulating, the objective of this study is not to obtain an optimal design but to investigate whether the stability limit of a contra-rotating stage can be shifted toward higher throttling by adjusting the rotor speed ratio. Additionally, the applicability of the design tool chain to small-scale propulsion systems should be revised experimentally.

3. Experimental configuration and instrumentation

The experimental setup is based on a static test rig and remains unchanged compared to the initial validation of the tool chain [17]. A LabVIEW-based measurement program operates the National Instruments Data Acquisition (DAQ) system and records all measurement data at 15 Hz. It also controls measurement sequences, enabling automated measurements over a wide range of operating conditions. In particular, throttle curves and performance maps are obtained from measurements at steady-state operating points. Each operating point is defined by the nozzle area, the corrected front rotor speed, and the speed ratio. To ensure comparability, corrected quantities of rotor speeds, mass flow, and

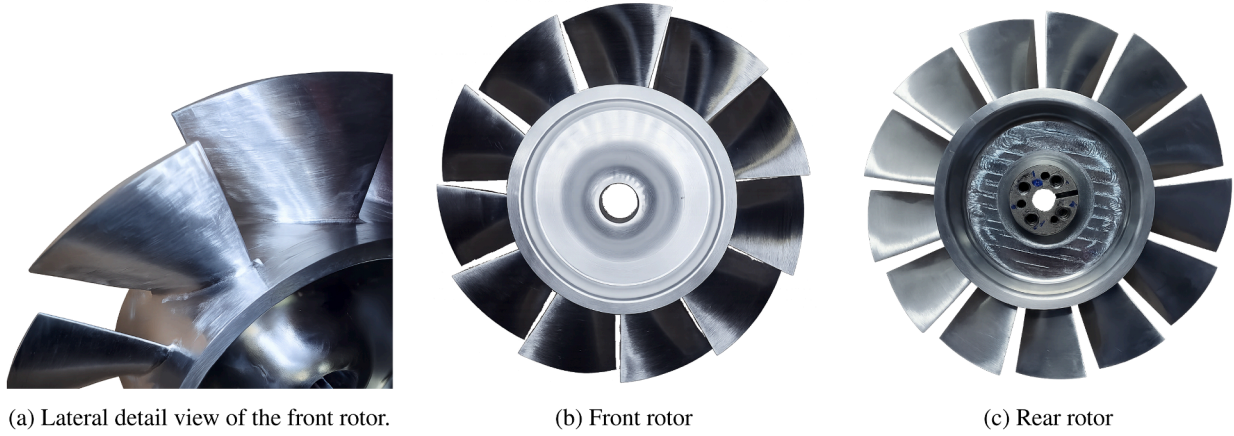


Fig. 7. Front and rear blisks machined from aluminum. The front rotor rotates clockwise, while the rear rotor rotates counterclockwise.

Table 3

Measurement ranges, maximum deviations, and resolutions of the pressure sensors.

Position/Plane	Sensor type	Sensor range	Max. deviation in Pa	Resolution in Pa
Ambient/0	absolute	1.6 bar	± 400.0	39.1
SAO nozzle/1	differential	4000 Pa	± 10.0	1.0
Inter-rotor plane/1.2	differential	16 kPa	± 40.0	3.9
Rear rotor outlet/1.3	differential	40 kPa	± 100.0	9.8

thrust are used based on the same reference ambient conditions as in [17]. Both rotor speeds are controlled via closed-loop regulation. While the corrected front rotor speed is set according to its design value, the rear rotor speed is adjusted through the speed ratio λ_E , which introduces an additional degree of freedom:

$$\lambda_E = \frac{n_{\text{cor., R2}}}{n_{\text{cor., R1}}} = \frac{n_{R2}}{n_{R1}} = \lambda_D \cdot \lambda. \quad (8)$$

In design operation the adjustment λ is set to unity, corresponding to the design speed ratio λ_D :

$$\lambda_D = \frac{n_{R2,D}}{n_{R1,D}} = 0.892. \quad (9)$$

For off-design operation, the corrected front rotor speed is kept constant, while the rear rotor speed is varied via λ .

Pressure data are acquired using digital transducers with an accuracy of 0.25 % full scale and an effective 12-bit resolution. The pressure sensors employed in the measurement setup are summarized in Table 3, which lists the sensor type, maximum deviation, and resolution. The measurement ranges were selected according to the highest values expected at the corresponding measurement positions. All pressure sensors are calibrated using a stand-alone routine based on a multi-range pressure controller and calibration device with an accuracy of 0.02 % full scale and a control stability of 0.003 % full scale. Ambient pressure is recorded with an absolute-pressure sensor, whereas differential-pressure sensors are used for all remaining pressure measurements to improve measurement accuracy.

The mass flow rate is determined using an SAO nozzle in accordance with DIN EN ISO 5801:2017 [28], whose diameter was measured with an accuracy of 0.1 mm. Downstream of each rotor, total pressure and velocity profiles are measured using calibrated miniature five-hole probes. To minimize flow blockage, only the conical probe head, with a diameter of just 1.5 mm, is inserted into the flow path. Test measurements with only one probe at a time confirmed that the measurement results are independent of whether the probe is located in the flow channel. Both probes are mounted on radial traversing systems with stepper motor actuation. Including the installation uncertainty of the traversing unit, the absolute radial positioning accuracy of the probe is estimated to be

within ± 1 % of the blade height. Because the positioning accuracy of the motor is substantially higher and the estimated uncertainty mainly arises from the initial probe installation, the radial probe positions remain reproducible throughout the measurement campaign. For the rear probe, the yaw angle is additionally adjusted in a closed-loop manner prior to measurement to minimize inflow misalignment.

Two thrust-nozzle configurations are employed: fixed nozzles, denoted by FN, for selected discrete nozzle areas, and a variable nozzle, denoted by VN, to obtain finely resolved throttle curves. In both cases, the nozzle area is set first, and the individual operating points are then measured as steady-state operating points once speed and mass flow rate have stabilized.

For each steady-state operating point, synchronous measurements are taken at 19 radial positions to determine radial distributions of total pressure, with increased resolution near the hub and shroud. The measured distributions $p_{t,1.2}(r)$ and $p_{t,1.3}(r)$ are directly comparable to the corresponding CFD results, since the axial probe positions coincide with the axial domain boundaries of the CFD setup. Based on these profiles, local total pressure ratios, such as the front rotor total pressure ratio $\pi_{R1,0}(r)$, can be derived. Compared to Eq. (1), all experimental total pressure ratios are calculated relative to ambient pressure and therefore include the inlet losses, since no pressure measurement is available directly upstream of the front rotor. Area-averaged values of total pressure and total pressure ratios across the stage and individual rotors are then calculated from these profiles using annulus-area weighting, where each radial measurement position is assigned to and weighted by the corresponding annular section of the flow area:

$$\pi_{t,R1,0} = \frac{p_{t,1.2}}{p_{t,0}}; \quad \pi_{t,R2,0} = \frac{p_{t,1.3}}{p_{t,0}}; \quad \pi_{t,R2} = \frac{\pi_{R2,0}}{\pi_{R1,0}}. \quad (10)$$

To describe the throttling state, the actual nozzle areas of the fixed and variable nozzles are normalized by the design nozzle area $A_{9,D}$ and introduced as:

$$\hat{A}_{9,FN} = \frac{A_{9,FN}}{A_{9,D}}, \quad \hat{A}_{9,VN} = \frac{A_{9,VN}}{A_{9,D}}. \quad (11)$$

Under throttled conditions, the nozzle area is varied in fine increments of 1.25 %, whereas near unthrottled operation, larger increments of up to 10 % are used.

The experimental results are primarily obtained using a baseline insert corresponding to a tip clearance TC_0 of 0.3 mm in standstill (≈ 0.8 % blade height). Additional inserts allow variations of the tip clearance up to 1 mm (≈ 2.7 %). For consistency with the CFD model, identical tip clearance values are applied to both rotors. A finite element analysis indicates a blade elongation of 0.01 mm to 0.05 mm at design speed. While this effect is considered qualitatively in the analysis, it is not included in the CFD model and may require future consideration.

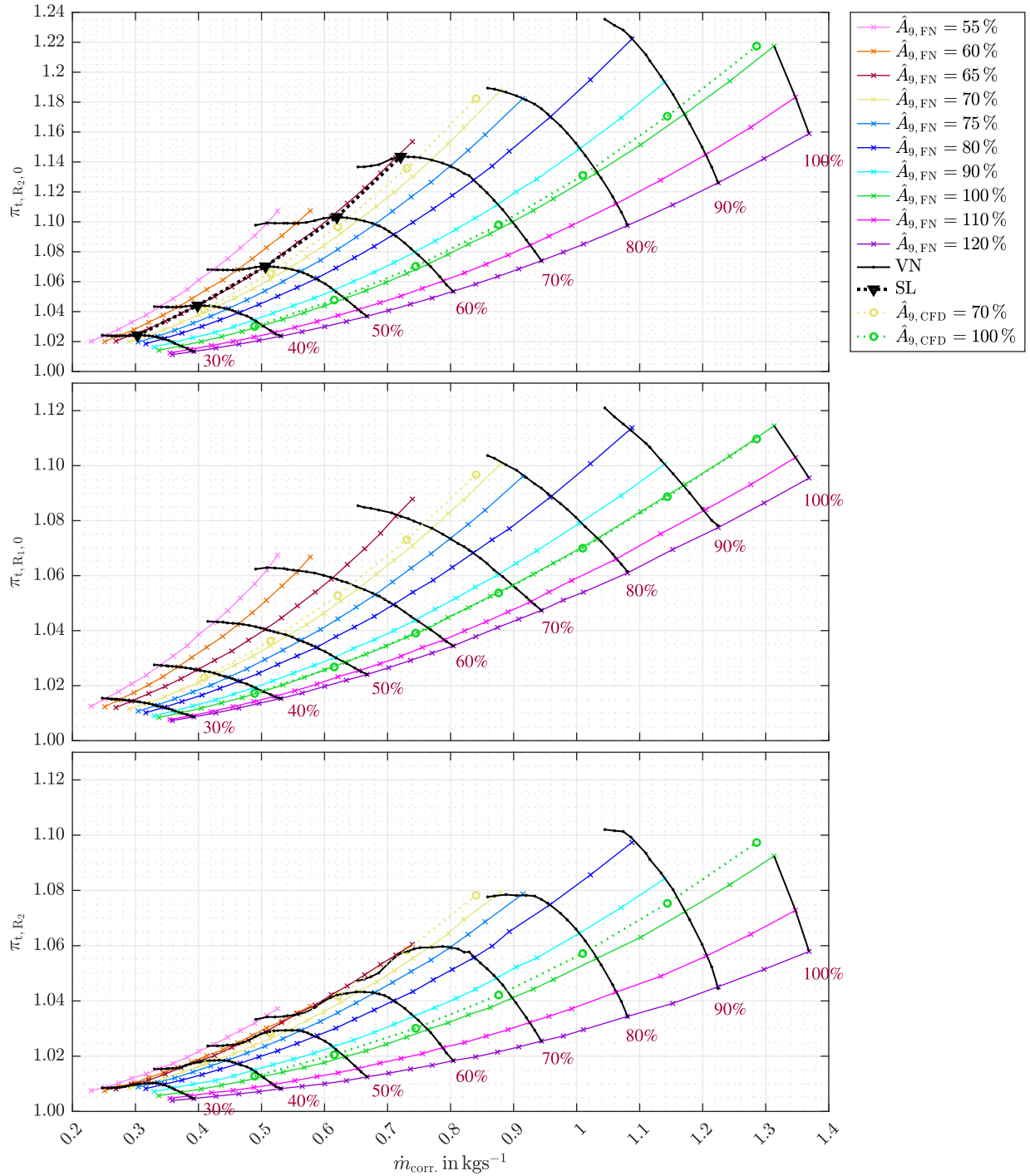


Fig. 8. Experimental performance map in design operation with $\lambda = 1$; total pressure ratio π_t across the entire stage and the front and rear rotor against the corrected mass flow $\dot{m}_{\text{corr}}$.

Although this is not the main focus of this work, thrust is measured with a six-component force and moment sensor with a capability of 1000 N in each direction and a rated accuracy of 0.1 % of full scale, corresponding to a maximum deviation of ± 1 N. Before operation, the setup is aligned horizontally and zeroed to compensate for installation-induced loads. The main objective is to determine at which operating point along the throttle curves of the individual rotors and the overall stage the stability limit occurs as a function of speed ratio. For this purpose, the stability limit is first defined as the operating boundary based on the pressure-rise characteristic obtained from the measured steady performance maps, rather than by directly measuring rotating stall or a surge

boundary. At the stability-limit operating points the highest pressure rise is reached with increasing throttling, before the pressure rise decreases again under further throttling. This pressure rise is expressed by the total pressure ratio across the respective rotor or the overall stage. Accordingly, all operating points at lower throttling up to this nozzle setting are considered to be within the stable operating range of the respective rotor or stage.

An assessment based on rotor efficiencies is not yet possible here. This would require either a direct measurement of shaft power, which is not available in the present setup, or measured motor and controller efficiency maps to subtract their losses and infer the actual shaft power

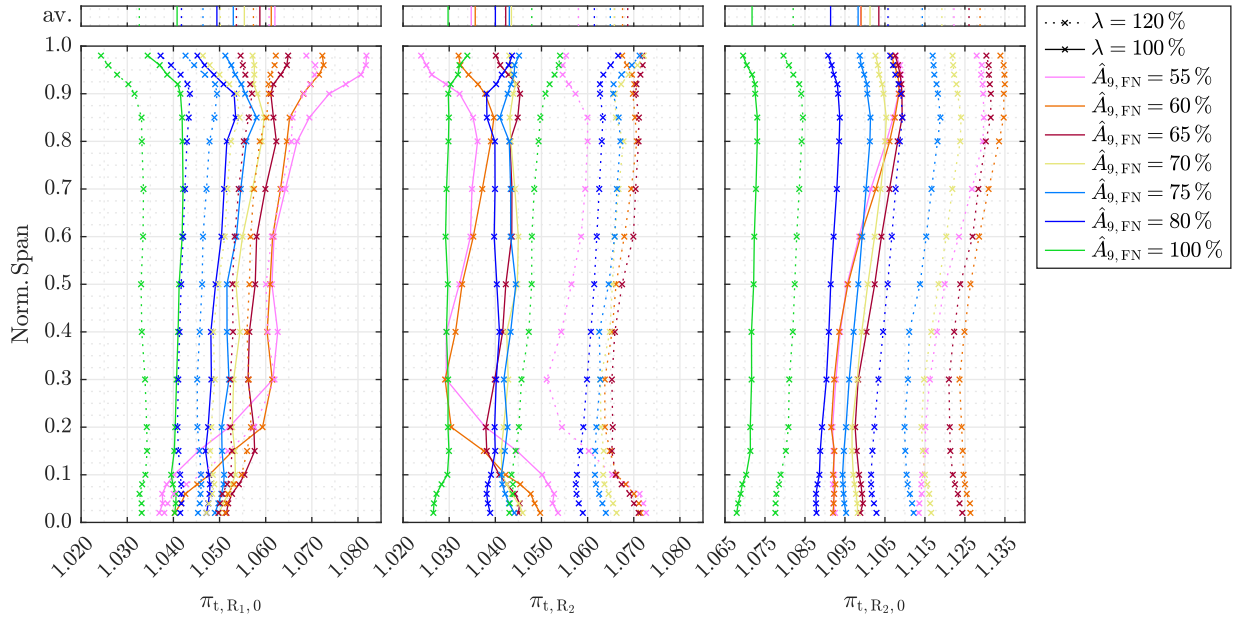


Fig. 9. Total pressure ratio π_t of the front rotor, rear rotor, and entire stage (left, center, right) at 60% engine speed as a function of the normalized nozzle area (colors as in Fig. 8), utilizing the fixed nozzles.

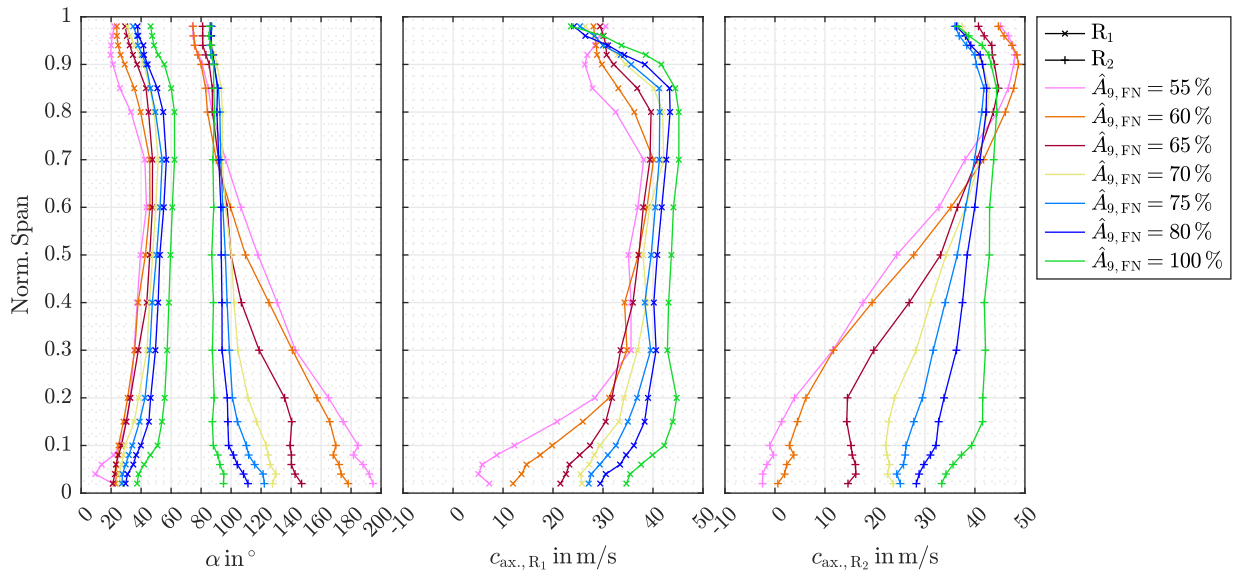


Fig. 10. Absolute flow angle α after the front and the rear rotor, respectively, at 60% corrected engine speed as a function of the normalized fixed nozzle area $\hat{A}_{9, FN}$, indicated with the respective colors chosen for Fig. 8.

from the measured electrical input power to the controller. This extension is left for future work. The present analysis therefore focuses on whether a shift of the stability limit with speed ratio can be identified.

The presented experimental data were obtained in one coherent measurement campaign. To assess reproducibility, large parts of the performance maps were repeated with slightly different settings, e.g., different sensors or exchanged pressure probes. The small deviations between these campaigns remain within the measurement uncertainty discussed below.

4. Experimental investigations and validation

The experimental investigations are divided into three parts. First, the overall system performance is analyzed to validate the design tool chain and to derive possible mechanisms limiting the operating range from the experimental findings. Second, the rotor speed ratio is varied

to assess its potential for improving throttled operation. Finally, the influence of tip clearance is investigated to evaluate its relevance for the numerical design process.

4.1. General performance

The general performance characteristics are determined experimentally using performance maps. Initially, the system is operated at the design speed ratio and design tip clearance. To maintain this clearance despite the blade elongation discussed above, the insert providing a standstill clearance of 0.4 mm is used for speeds above 72.5%. As shown in Fig. 8, highly resolved throttle curves are measured using the variable nozzle at intervals of 10% corrected engine speed. In addition, fixed nozzles with normalized areas between 55% and 120% are used to determine the system characteristic lines, imposed by the duct-nozzle system and defined by the corresponding nozzle exit areas. Due to experimen-

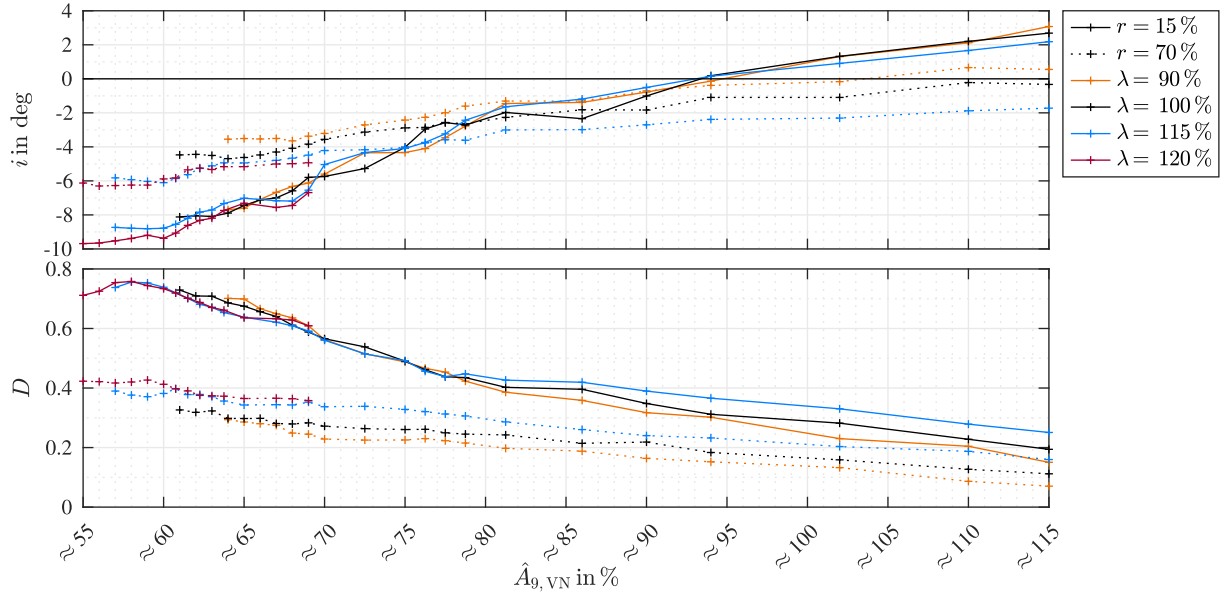


Fig. 11. Rear rotor inlet incidence i and diffusion factor D (15% and 70% span) at 60% speed as functions of normalized variable nozzle area $\hat{A}_{9,VN}$ and speed ratio adjustment λ .

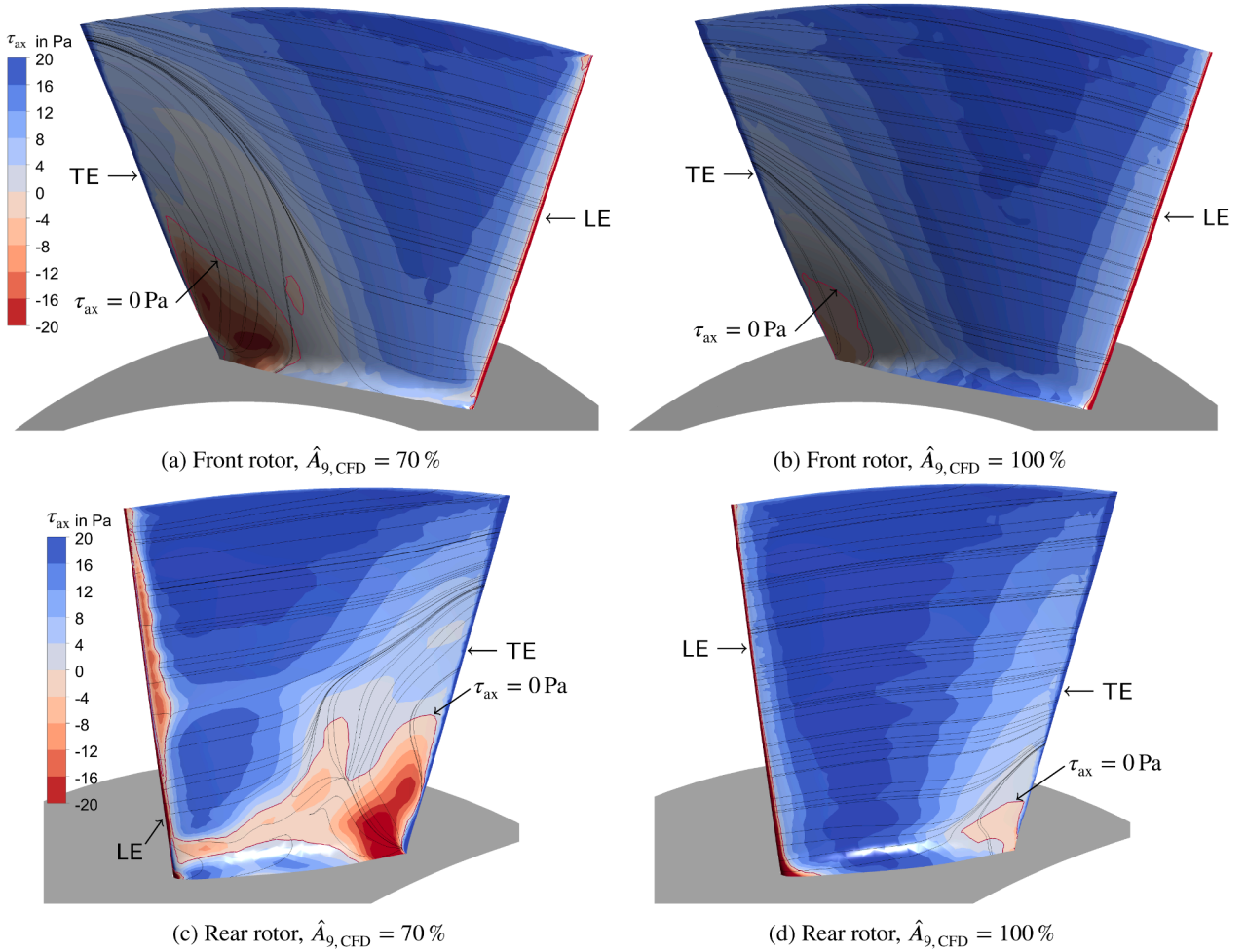


Fig. 12. Axial wall shear stress component τ_{ax} and wall shear streamlines on the suction sides at two different throttling positions at 60% corrected engine speed. The $\tau_{ax} = 0$ Pa contour highlights the extent of the hub-connected region of reversed axial wall shear. LE and TE denote the leading and trailing edges, respectively.

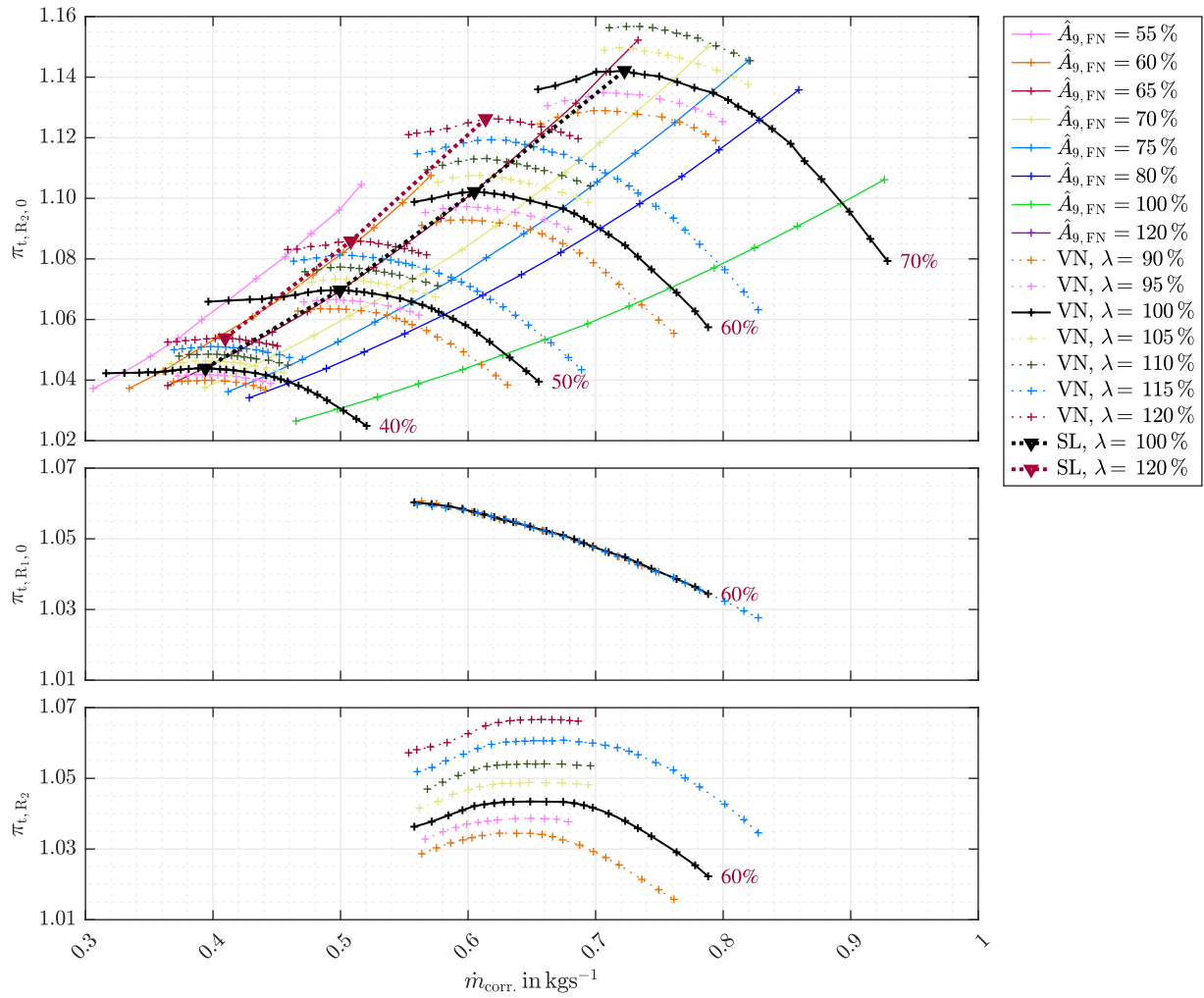


Fig. 13. Performance map as a function of the speed ratio adjustment λ , limited to 60% engine speed in case of the rotors' individual maps (center and bottom).

tal safety constraints, the maximum throttling increasingly has to be limited at higher speeds to avoid severe vibrations and damage. Therefore, the stability limit is reached only up to 70% corrected front rotor speed.

Error bars are omitted in the following figures for clarity. The temporal scatter at each steady-state operating point is instead quantified by the relative standard deviation of the recorded samples. As an example, at a corrected front rotor speed of 60%, the relative standard deviation of the mass flow rate reaches a maximum of 0.16% under strongly throttled conditions, while it is approximately 0.12% close to the design operating point. A similar trend is observed for the total pressure rise. The relative standard deviation of the total pressure difference across the front rotor increases from 0.26% close to the design operating point to 1.12% under throttled off-design conditions. For the total pressure rise across the stage, the corresponding values range from 0.10% to 1.10%. While the relative standard deviations depend only weakly on speed, the relative measurement uncertainties decrease with increasing speed because the absolute sensor uncertainties remain approximately constant. Based on the sensor uncertainties given in Table 3 and the specifications for measuring the SAO nozzle diameter, described above, the relative uncertainties of the corrected mass flow rate and the total pressure rises across the front rotor and stage range from 3.23% to 0.25%, from 2.88% to 0.21%, and from 4.27% to 0.28%, respectively, over the investigated speed range.

The performance map shows typical throttle characteristics of axial turbomachinery. The stability limit, marked by SL in Fig. 8, of the

stage is located close to the 65% nozzle operating line. Though there is only a moderate reduction in stage total pressure ratio at even higher throttling, fewer operation points were measured within these throttle conditions at higher speeds to prevent damaging the system. The rear rotor reaches its stability limit earlier, at normalized nozzle areas of approximately 70% to 75%, followed by a pronounced reduction in total pressure ratio. In contrast, the front rotor remains stable down to approximately 60% nozzle area. Thus, for the investigated operating range, the rear rotor appears to stall first, largely independent of engine speed, which is consistent with previous findings, e.g. in [10].

The experimental data are compared with CFD results at the design nozzle and, as above, at 70% normalized nozzle area, summarized in form of the relative deviations e_ϕ in Table 4. Under design conditions, simulation and experiment are in good agreement, confirming that the tool chain enables a design that meets the intended global specifications. However, the experiment shows a systematic shift toward higher mass flow rates. Similar, the measured corrected thrust of 227 N at design speed and design nozzle is higher than the simulated thrust. At 70% nozzle area, deviations increase and simulation results seem to be shifted toward higher throttling. Since the trends of the deviations e_ϕ do not follow the speed-dependent decrease in experimental uncertainty, the observed differences are unlikely to be caused solely by measurement uncertainty. Instead, they suggest that the numerical model is less accurate under throttled off-design conditions. Nevertheless, since the experimental uncertainties are of the same order of magnitude as the deviations between simulation and experiment, a reduction of the ex-

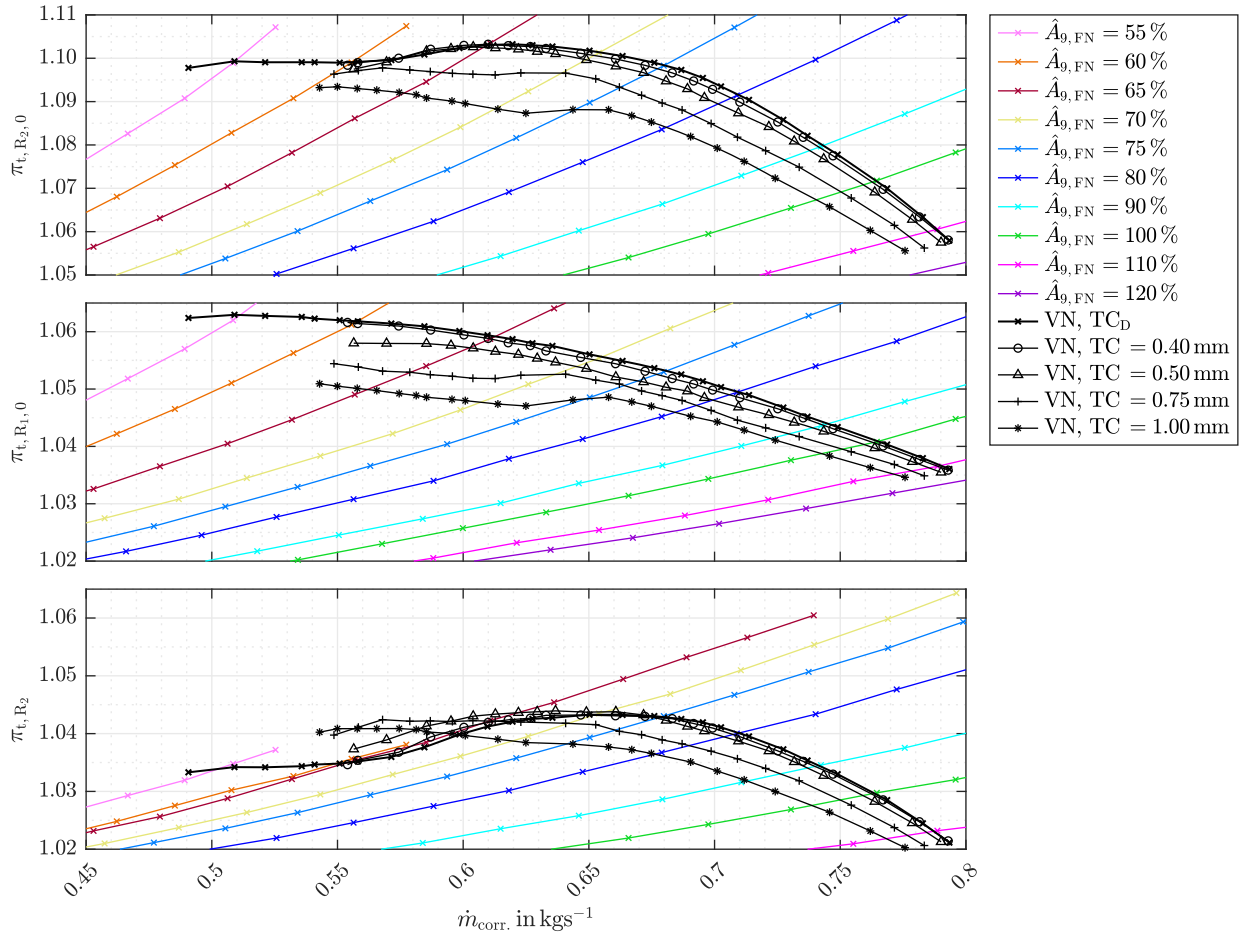


Fig. 14. Dependency of the throttle curve characteristics of the entire stage, front and rear rotor (top, center and bottom) at 60 % engine speed on the tip clearance.

Table 4
Relative deviations e_ϕ between simulation and experimental results.

n_{COR} in %	e_ϕ in %, $\hat{A}_9 = 70\%$				e_ϕ in %, $\hat{A}_9 = 100\%$			
	e_m in %	$e_{\Delta p_{t, R2, 0}}$	$e_{\Delta p_{t, R1, 0}}$	$e_{\Delta p_{t, R2}}$	e_m in %	$e_{\Delta p_{t, R2, 0}}$	$e_{\Delta p_{t, R1, 0}}$	$e_{\Delta p_{t, R2}}$
40	-4.69	-5.15	-6.19	-3.71	-2.74	-2.74	-4.62	0.13
50	-5.33	-5.02	-5.93	-3.80	-2.80	-2.69	-6.32	2.63
60	-5.05	-4.79	-5.00	-4.46	-2.59	-2.39	-4.54	0.67
70	-5.07	-4.35	-5.23	-3.21	-2.32	-1.99	-4.98	2.11
80	-4.32	-2.45	-2.41	-2.32	-1.96	-0.86	-3.90	3.05
90	-	-	-	-	-2.22	-0.64	-4.39	3.93
100	-	-	-	-	-1.93	0.20	-4.98	6.27

perimental uncertainties is required for a more detailed validation of the numerical model.

Radial total pressure profiles at 60% corrected engine speed are shown in Fig. 9 to identify the mechanisms limiting stability. The profiles indicate a strong influence of three-dimensional effects, particularly near the casing. In unthrottled operation, the front rotor shows a reduced total pressure ratio in the outer span region, which is partly compensated by the rear rotor. Under throttled conditions, this behavior reverses: the rear rotor performance decreases significantly in the outer span region, locally throttling the front rotor. However, since the front rotor's loading is small in the outer radial region it can tolerate this local throttling.

Near midspan, the rear rotor pressure ratio increases only up to nozzle areas of approximately 70 % to 75 %. At stronger throttling, a pronounced reduction occurs, consistent with the observed stability limit

of the rear rotor. Near the hub, the front rotor also shows a local reduction in total pressure ratio for nozzle areas below 70 %, with the affected span increasing as the nozzle area decreases. This suggests that the stage stability limit is influenced by both rear rotor deterioration in the mid and outer span and front rotor limitations near the hub.

To further analyze the flow mechanisms, spanwise distributions of absolute flow angle and axial velocity are shown in Fig. 10. In design operation, the rear rotor outflow is nearly axial, confirming the intended swirl removal. With increasing throttling, the axial velocity decreases and the front rotor outflow angle changes accordingly. At small nozzle areas, the inner span region exhibits strongly reduced axial throughflow and even local reverse flow downstream of the rear rotor. The apparent pressure rise near the hub of the rear rotor is therefore likely influenced by friction at the rotating hub rather than by useful axial work input.

The resulting incidence angle i at the rear rotor inlet becomes increasingly negative with throttling due to the reduction in axial velocity. As shown in Fig. 11, this effect is particularly pronounced near the hub. While the incidence is close to 0° at the design point for both radii, the pronounced shift toward negative values mainly occurs in the hub region, where values of up to -8° are reached.

At the same time, the rear rotor diffusion factor, calculated as shown in Eq. (6) from the measured velocity components and the geometric parameter of the blade in terms of the solidity, exceeds 0.6 for nozzle areas below 70 %, indicating increased loss generation and a higher risk of separation [18]. Although the front rotor incidence and diffusion factor cannot be measured directly, the design calculations indicate even higher diffusion factors near the front rotor hub. These observations

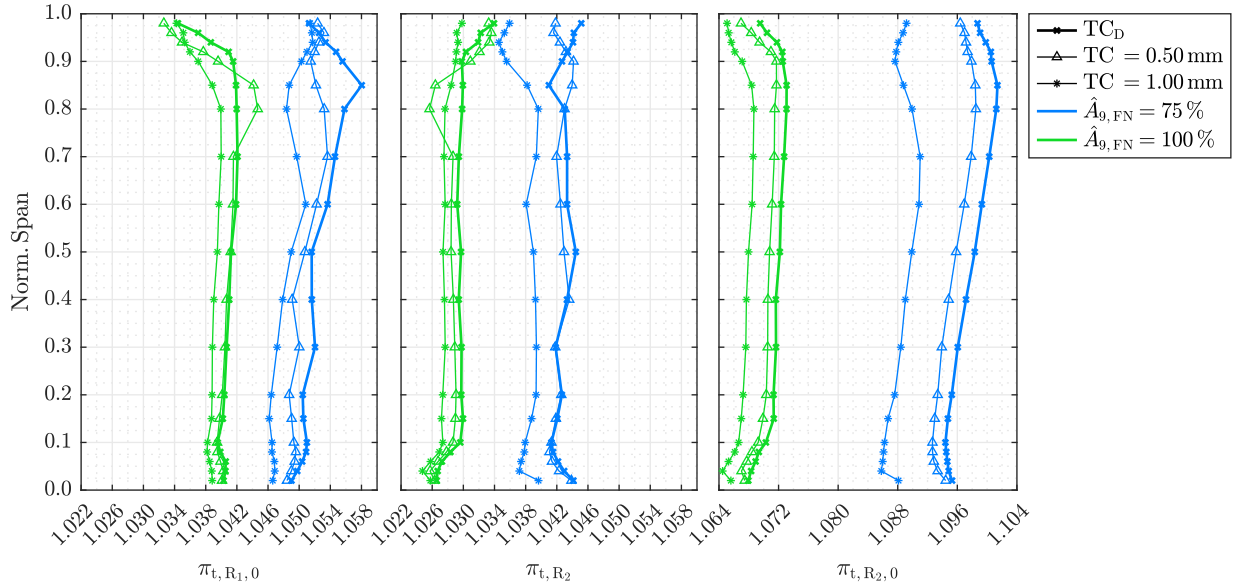


Fig. 15. Spanwise total pressure ratio π_t of the front rotor, rear rotor, and stage (left, center, right) at 60 % engine speed as a function of normalized fixed nozzle area $\hat{A}_{9, FN}$ and varying tip clearances.

seem to indicate that hub corner separation may occur at the front rotor, may propagate downstream, and may cause a larger radial extent of the rear rotor to separate. Although more detailed empirical prediction methods are available, diffusion factors above 0.6 can generally be used as an indicator [29,30].

Despite the CFD limitations discussed above, the hypothesis of hub corner stall is further supported by the flow-field analysis of the available CFD simulations. Fig. 12 shows the axial component of the wall shear stress, τ_{ax} , together with wall shear streamlines on the suction sides of both rotors. Negative values of τ_{ax} indicate a reversal of the axial wall shear component, while the $\tau_{ax} = 0$ Pa contour highlights the extent of the corresponding connected region. At the design condition with $\hat{A}_{9, CFD} = 100\%$, only a comparatively small region of reversed axial wall shear is observed near the hub of the front rotor. At the throttled operating point with $\hat{A}_{9, CFD} = 70\%$, this region increases substantially in both streamwise and radial extent. The wall shear streamlines additionally exhibit a pronounced deflection toward the hub-suction-side corner, indicating an increasingly three-dimensional near-wall flow and a substantial growth of the hub corner separation.

An even stronger change is observed on the rear rotor. While the region of reversed axial wall shear remains confined to the immediate hub vicinity at the design condition, it extends over a considerable portion of the suction surface at $\hat{A}_{9, CFD} = 70\%$ with higher values. This pronounced increase in the separated region of the rear rotor is consistent with the previously discussed hypothesis that the deterioration of the front rotor hub flow propagates downstream and adversely affects the inflow to the rear rotor.

In addition, a pronounced region of reversed axial wall shear develops downstream of the rear rotor leading edge at the throttled operating point. This observation is consistent with the experimentally observed increase in rear rotor incidence toward smaller nozzle areas. The increased incidence promotes a stronger adverse loading of the suction-side boundary layer near the leading edge and may therefore contribute to the development of the observed leading-edge separation.

Thus, the CFD results support the former experimental interpretation that the deterioration of the flow toward the stability limit appears to result from the combined effects of an increasingly separated hub-corner flow and an incidence-induced leading-edge separation. This hypothesis is further examined through speed-ratio variation. However, a more

comprehensive numerical investigation would require additional operating points along the throttling characteristic, particularly in the vicinity of the experimentally observed stability limit. Furthermore, the numerical design setup, especially the blade-passage approach, precludes a conclusive assessment of whether the suspected corner stall constitutes a stall-inception mechanism and whether it may subsequently develop into rotating stall.

4.2. Speed ratio variations

The independent electric drives allow the rotor speed ratio to be varied. In these experiments, the front rotor speed is kept constant. In contrast, the rear rotor speed is adjusted according to the speed-ratio parameter λ , set between 90 and 120 %. Changing the rear rotor speed affects both the circumferential speed at the rear rotor inlet and the axial flow component. Depending on which effect dominates, the rear rotor incidence may increase or decrease. For the present system, increasing the speed ratio leads to more negative incidence at both investigated radial positions as shown in Fig. 11. The rear rotor diffusion factor also increases, particularly at 70 %, because of the higher rear rotor work input. At 15 % span, however, the sharp increase in diffusion factor is shifted toward smaller nozzle areas.

Increasing the rear rotor speed is expected to increase the overall work input. Consequently, the operating points of the stage shift along their respective system characteristic line, defined by the respective nozzle exit area, toward higher pressure rises and mass flow rates as shown in Fig. 13. This contribution of the additional rear rotor work input cannot be fully separated from the aerodynamic effect of speed ratio adjustments based on the present data. However, the shift becomes disproportionately stronger under highly throttled operating conditions. Therefore, the overall stability limit shifts toward smaller nozzle areas when the speed ratio is increased, despite the associated increase in rear rotor loading and incidence. As shown in Fig. 13, operation with $\lambda = 120\%$ shifts the stage stability limit by approximately 5 % in normalized nozzle area, close to the 60 % nozzle.

This behavior is consistent with the hypothesis that the overall stability limit may be related to the pronounced growth of hub corner separation at the front rotor rather than by the rear rotor incidence angle. Increasing the rear rotor speed appears to unload the front rotor, which may delay the onset of hub separation at the front rotor for higher speed

ratios, as shown in the left plot of Fig. 9. Although the rear rotor operates with higher incidence and diffusion factor, it may benefit from the improved inflow conditions resulting from the delayed front rotor separation. Consequently, the entire stage may remain stable at smaller nozzle areas.

4.3. Tip clearance effects

The influence of tip clearance is investigated at 60 % corrected engine speed and design speed ratio using four further casing inserts. For a tip clearance of 0.4 mm, the overall characteristics of the throttle curves remain nearly unchanged, both across the entire stage and across the individual rotors. Increasing the tip clearance reduces the mass flow rate and total pressure ratio, indicating a decrease in aerodynamic efficiency. For larger clearances, a pronounced drop in total pressure ratio occurs between approximately 70 % and 75 % normalized nozzle area as shown in Fig. 14.

This drop shifts toward larger nozzle areas as tip clearance increases, indicating an earlier onset of performance deterioration. The radial profiles in Fig. 15 show that the rear rotor is particularly sensitive to tip-clearance variation, especially in throttled operation. Larger tip clearance strongly reduces the total pressure ratio in the outer span region of the rear rotor. The front rotor is affected more indirectly: reduced rear rotor performance causes additional throttling of the front rotor. Under throttled conditions, this initially local effect expands over a larger portion of the span as tip clearance increases. Overall, tip clearance has a strong effect on stage performance and stability. It affects not only the rear rotor directly, but also the front rotor through altered stage matching. Accurate measurement of the static tip clearance is therefore essential for comparison with CFD. Blade elongation during operation slightly reduces the effective clearance; however, the estimated reduction remains small compared with the investigated clearance variations. Furthermore, the slightly increased clearance used for speeds above 72.5 % does not appear to alter the fundamental characteristics of the throttle curves.

5. Conclusion

Based on the investigations presented in this work, the main conclusions can be summarized as follows:

1. The good agreement between the initial specifications of global design parameters and experimental data at the design point demonstrates that the implemented design methodology for small-scale systems is also applicable to contra-rotating fan stages. Without further optimization of the design parameters, the targeted stage total pressure ratio and mass flow rate are achieved by the iterative adjustment of the blade geometry, despite pronounced three-dimensional effects.
2. The experimental results confirm that the rear rotor reaches its stability limit first, despite the higher loading of the front rotor. Near the hub, strong reductions in axial velocity and even local flow reversal are observed under highly throttled conditions. This behavior is interpreted as a pronounced growth of the front rotor hub corner separation toward the stability limit at approximately 65 % normalized nozzle area, supported by initial evaluations of the available CFD data. Although initially confined to a limited radial region, this disturbance seems to propagate downstream and significantly affect the rear rotor. Due to the absence of a flow-straightening stator, the rear rotor may be particularly sensitive to distorted inflow, leading to a widening of the affected span and a reduction in overall performance.
3. The effect of speed-ratio adjustment is assessed based on the shift of the stability line. For the stage investigated here, increasing the rotor speed ratio increases the rear rotor incidence but simultaneously shifts the stability limit toward smaller nozzle areas. For a

20 % increase in rear rotor speed, this shift amounts to approximately 5 % normalized nozzle area, thereby extending the stable operating range. The observed shift of the stability limit is consistent with the previous hypothesis that the stability behavior is related to the pronounced growth of hub corner separation at the front rotor. Increasing the rear rotor speed appears to reduce the aerodynamic loading of the front rotor, thereby delaying the onset of hub separation. However, whether this extension of the stable operating range is also beneficial in terms of efficiency remains to be investigated in future work.

4. The influence of tip clearance is also significant. Even small deviations from the design value lead to noticeable reductions in performance, particularly under throttled conditions. Accurate control of tip clearance is therefore essential for meaningful comparison with numerical simulations.
5. Mesh and time-step sensitivity are assessed under both design and off-design conditions, as off-design operation is expected to be more sensitive to spatial and temporal resolution. The throttled case is included to ensure that the same mesh setup can be used for future detailed numerical off-design studies supporting the experimental findings presented here. However, the agreement between simulation and experiment deteriorates under throttled operation, and systematic deviations remain, especially in the simulated mass flow rate. Predicting off-design performance may therefore require further refinement of the numerical approach, in particular full-annulus simulations to capture off-design phenomena such as stall inception mechanisms or rotating stall.

Future studies should consider the effect of blade deformation on tip clearance and determine rotor efficiencies experimentally by accounting for motor and controller losses, particularly to assess the potential for enhanced efficiency under throttled conditions with adjusted speed ratio. In addition, further CFD analyses may provide deeper insights into the local flow phenomena and support the interpretations derived here.

Overall, the results demonstrate that adjusting the rotor speed ratio can provide an effective means of extending the operating range and stability margin of the contra-rotating fan stage investigated here within the tested operating range. For electrically driven contra-rotating fans with independently controlled motors, the speed ratio thus represents an additional actively controllable operating parameter that may allow the aerodynamic loading to be redistributed between the rotors and the stability margin to be adapted to the current operating condition. The results further highlight the importance of hub-region aerodynamics for small-scale fan stages with low blade heights, where endwall-related three-dimensional effects can affect a substantial fraction of the annulus. Consequently, both the aerodynamic design of the hub region and the operating-point-dependent redistribution of rotor loading should receive particular attention in such systems.

CRediT authorship contribution statement

Sebastian Hawner: Writing – review & editing, Writing – original draft, Visualization, Validation, Software, Resources, Methodology, Investigation, Formal analysis, Data curation, Conceptualization; **Andreas Hupfer:** Writing – review & editing, Validation, Supervision, Resources, Project administration, Investigation, Funding acquisition.

Funding

This publication is funded by the project *Electric Aircraft Propulsion - safe, efficient, digitally linked (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

Declaration of competing interest

The authors declare the following financial interests/personal relationships which may be considered as potential competing interests:

Sebastian Hawner reports financial support was provided by European Union. If there are other authors, they declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

Data will be made available on request.

References

- [1] W. Shan, S. Bao, S. Lin, L. Kang, Hybrid electric propulsion design and analysis based on regional aircraft mission, *World Electr. Veh. J.* 16 (4) (2025). <https://doi.org/10.3390/wevj16040212>
- [2] B.A. Adu-Gyamfi, C. Good, Electric aviation: a review of concepts and enabling technologies, *Transp. Eng.* 9 (2022). <https://doi.org/10.1016/j.treng.2022.100134>
- [3] R.C. Bolam, Y. Vagapov, A. Anuchin, Review of electrically powered propulsion for aircraft, in: 2018 53rd International Universities Power Engineering Conference (UPEC), 2018, pp. 1–6. <https://doi.org/10.1109/UPEC.2018.8541945>
- [4] M. Soleymani, V. Mostafavi, M. Hebert, S. Kelouwani, L. Boulon, Hydrogen propulsion systems for aircraft, a review on recent advances and ongoing challenges, *Int. J. Hydrog. Energy* 91 (2024) 137–171. <https://doi.org/10.1016/j.ijhydene.2024.10.131>
- [5] L. Sun, X. Liu, Q. Tao, J. Yang, Aerodynamic performance of ducted fans for large-scale electric propulsion aircraft under variable operating conditions, *Phys. Fluids* 37 (3) (2025) 035184. <https://doi.org/10.1063/5.0260520>
- [6] Y. Qian, Y. Luo, X. Hu, Z. Zeng, Y. Zhang, Improving the performance of ducted fans for VTOL applications: a review, *Sci. China Technol. Sci.* 65 (2022) 2521–2541. <https://doi.org/10.1007/s11431-021-2110-x>
- [7] M.P. Manas, A.M. Pradeep, Stall inception in a contra-rotating fan under radially distorted inflows, *Aerosp. Sci. Technol.* 105 (2020) 105909. <https://doi.org/10.1016/j.ast.2020.105909>
- [8] C. Mistry, A.M. Pradeep, Effect of variation in axial spacing and rotor speed combinations on the performance of a high aspect ratio contra-rotating axial fan stage, *Proc. Inst. Mech. Eng. A: J. Power Energy* 227 (2) (2013) 138–146. <https://doi.org/10.1177/0957650912467453>
- [9] M.J. Shahriyari, H. Khaleghi, M. Heinrich, A model for stall and surge in low-Speed contra-rotating fans, *J. Eng. Gas Turbines Power* 141 (8) (2019) 081009. <https://doi.org/10.1115/1.4043251>
- [10] Y. Wang, W. Chen, C. Wu, S. Ren, Effects of tip clearance size on the performance and tip leakage vortex in dual-rows counter-rotating compressor, *Proc. Inst. Mech. Eng. G: J. Aerosp. Eng.* 229 (11) (2015) 1953–1965. <https://doi.org/10.1177/0954410014562483>
- [11] X. Yu, J. Hou, G. An, B. Liu, Experimental study on the blade tip clearance sensitive effect in compressors with different loading levels, *Phys. Fluids* 36 (6) (2024) 065140. <https://doi.org/10.1063/5.0213355>
- [12] J.R. Vala, D.K. Patel, A.P. Darji, L. Natrayan, Investigation of varying tip clearance gap and operating conditions on the fulfilment of low-speed axial flow fan, *Int. J. Turbo Jet-Engines* 42 (2) (2025) 305–315. <https://doi.org/doi:10.1515/tjj-2024-0067>
- [13] M. Ryu, L. Cho, J. Cho, The effect of tip clearance on performance of a counter-rotating ducted fan in a VTOL UAV, *Trans. Jpn. Soc. Aeronaut. Space Sci.* 60 (1) (2017) 1–9. <https://doi.org/10.2322/tjsass.60.1>
- [14] Y. Wu, X. Zhang, F. Yang, S. Spence, Review of unsteady aerodynamic problems and control strategies for the blade tip flow of axial compressors, *Prog. Aerosp. Sci.* 152 (2025) 101063. <https://doi.org/10.1016/j.paerosci.2024.101063>
- [15] T. Ebus, M. Dietz, A. Hupfer, Experimental and numerical studies on small contra-rotating electrical ducted fan engines, *CEAS Aeronaut.* 12 (2021) 559–571. <https://doi.org/10.1007/s13272-021-00517-7>
- [16] T. Ebus, M. Dietz, A. Hupfer, Detailed performance studies on a small electric-powered contra-rotating ducted fan engine, *J. Propuls. Power* 0 (0) (0) 1–12. <https://doi.org/10.2514/1.B39046>
- [17] S. Hawner, L. Rduch, H. Baumhöfener, A. 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) (2024). <https://doi.org/10.3390/aerospace11070527>
- [18] S. Lieblein, F.C. Schwenk, R. Broderick, Diffusion factor for estimating losses and limiting blade loadings in axial-flow-compressor blade elements, in: *NACA-RM-E53D01*, 1953. <https://api.semanticscholar.org/CorpusID:117788867>
- [19] B. Roy, K. Ravibabu, P.S. Rao, S. Basu, A. Raju, P.N. Murthy, Flow studies in ducted twin-rotor contra-rotating axial flow fans, *Turbo Expo: Power Land Sea Air Vol. 1: Turbomachinery* (1992) V001T01A131. <https://doi.org/10.1115/92-GT-390>
- [20] F.R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA J.* 32 (8) (1994) 1598–1605. <https://doi.org/10.2514/3.12149>
- [21] J. Wang, N.P. Kruyt, Computational fluid dynamics simulations of aerodynamic performance of low-pressure axial fans with small hub-to-tip diameter ratio, *J. Fluids Eng.* 142 (9) (2020) 091202. <https://doi.org/10.1115/1.4047120>
- [22] D. Schlüter, Prediction and Investigation of Rotating Stall in Axial Flow Compressors by means of High- and Low-Fidelity Methods, Ph.D. thesis, Technische Universität München, 2022. <https://mediatum.ub.tum.de/1618418>
- [23] G. Pullan, A.M. Young, I.J. Day, E.M. Greitzer, Z.S. Spakovszky, Origins and structure of spike-type rotating stall, *J. Turbomach.* 137 (5) (2014). <https://doi.org/10.1115/1.4028494>
- [24] F. Crevel, N. Gourdain, S. Moreau, Numerical simulation of aerodynamic instabilities in a multistage high-speed high-pressure compressor on its Test-Rig—Part I: rotating stall, *J. Turbomach.* 136 (10) (2024). <https://doi.org/10.1115/1.4027967>
- [25] International Civil Aviation Organization, Manual of the ICAO Standard Atmosphere, Extended to 80 Kilometres (262 500 Feet), International Civil Aviation Organization, 3rd ed., 1993
- [26] I. Celik, U. Ghia, P. Roache, C. Freitas, H. Coleman, P. Raad, Procedure for estimation and reporting of uncertainty due to discretization in CFD applications, *J. Fluids Eng.* 130 (7) (2008) 078001. <https://doi.org/10.1115/1.2960953>
- [27] S. Hawner, A. Hupfer, Experimental design validation of an electric ducted and contra rotating fan engine, in: *Proceedings of the 26th International Symposium on Air Breathing Engines*, 2024, pp. 1663–1682. Paper 139.
- [28] ISO/DIN 5801:2017, Fans - Performance testing using standardized airways (ISO 5801:2017); German version EN ISO 5801:2017 (2018). <https://doi.org/10.31030/2637235>
- [29] J. Li, L. Ji, An improved prediction model for corner stall in axial compressors with dihedral effect, *Chin. J. Aeronaut.* 33 (5) (2020) 1433–1443. <https://doi.org/10.1016/j.cja.2019.12.031>
- [30] V.M. Lei, Z.S. Spakovszky, E.M. Greitzer, A criterion for axial compressor hub-corner stall, *J. Turbomach.* 130 (3) (2008) 031006. <https://doi.org/10.1115/1.2775492>