Numerical investigation of the effects of fuselage upsweep in a propulsive fuselage concept

Anaïs Luisa Habermann¹ · Anubhav Gokhale^{2,3} · Mirko Hornung¹

Received: 2 April 2020 / Revised: 27 August 2020 / Accepted: 24 November 2020 / Published online: 6 January 2021
 © The Author(s) 2021

Abstract

In recent years, aircraft concepts employing wake-filling devices to reduce mission fuel burn have gained increasing attention. The study presented here aims at a detailed physical understanding of the effects of integrating a propulsive fuselage device on a commercial aircraft. Compared to an isolated, axisymmetric fuselage-propulsor configuration, a propulsive fuselage device experiences an increased circumferential inlet distortion due to three-dimensional geometric features of the aircraft. This study uses three-dimensional CFD simulations to investigate the effect of fuselage upsweep on the aero-propulsive performance of an aircraft configuration featuring a boundary layer ingestion device. It is shown that fuselage upsweep has a negative impact on the performance of a propulsive fuselage device as compared to an axisymmetric configuration. Increasing the upsweep angle by $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$ leads to an increase in required fuselage fan shaft power by 19%. Furthermore, it is demonstrated that the negative effects of fuselage upsweep on the propulsor's performance can be effectively mitigated by a circumferential variation in the propulsor nacelle thickness.

Keywords Boundary layer ingestion · Propulsive fuselage concept · Computational fluid dynamics

List of symbols

α	Angle of attack [$^\circ$]	F	Force [N]
α_{GC}	Longitudinal ground clearance angle [$^\circ$]	$f_{\eta,\text{BLI}}$	BLI efficiency factor [-]
$\alpha_{\text{SW,PFC}}$	Upsweep angle for propulsive fuselages [$^\circ$]	h	Height [m]
$\alpha_{\text{SW,Roskam}}$	Upsweep angle for conventional fuselages [$^\circ$]	h	Total enthalpy [$\frac{\text{kg m}^2}{\text{s}^2}$]
$\dot{m}$	Mass flow [$\frac{\text{kg}}{\text{s}}$]	I_i	Circumferential distortion intensity of the i th ring [-]
η	Efficiency [-]	I_{max}	Maximum circumferential distortion intensity [-]
κ	Specific heat ratio [-]	k	Turbulent kinetic energy [$\frac{\text{J}}{\text{kg}}$]
ω	Turbulent frequency [$\frac{1}{\text{s}}$]	l	Length [m]
Π_{fan}	Fan pressure ratio [-]	M	Mach number [-]
ρ	Density [$\frac{\text{kg}}{\text{m}^3}$]	P	Power [W]
τ	Shear stress [$\frac{\text{N}}{\text{m}^2}$]	p	Pressure [Pa]
Θ	Angle [$^\circ$]	PAV	Average total pressure [Pa]
A	Area [m^2]	PAVLOW	Average low total pressure [Pa]
c	Chord length [m]	R	Universal gas constant [$\frac{\text{J}}{\text{kg K}}$]
C_f	Skin friction coefficient [-]	Re	Reynolds number [-]
C_p	Pressure coefficient [-]	S	Area [m^2]
d	Diameter [m]	T	Temperature [K]
		t/c	Thickness-to-chord ratio [-]
		U	Velocity [$\frac{\text{m}}{\text{s}}$]
		y^+	Dimensionless wall distance [-]

✉ Anaïs Luisa Habermann
 anais.habermann@bauhaus-luftfahrt.net

¹ Bauhaus Luftfahrt e.V., Willy-Messerschmitt-Straße 1, 82024 Taufkirchen, Germany

² Technical University of Munich, Munich, Germany

³ Nanyang Technological University, Singapore, Singapore

Indices

∞	Freestream
axi	Axisymmetric

fan	Fuselage fan
fuse	Fuselage
g	Gross
GC	Ground clearance
iso	Isentropic
n	Net
p	Polytropic
s	static
SW	Upsweep parameter
<i>t</i>	Total
w	Wall
<i>x</i>	In streamwise direction

Abbreviations

AIP	Aerodynamic interface plane
AMI	Arbitrary mesh interface
BLI	Boundary layer ingestion
BWB	Blended wing body
CAD	Computer-aided design
CFD	Computational fluid dynamics
FF	Fuselage fan
FL	Flight level
FPR	Fan pressure ratio
HiSA	High-speed aerodynamic
NPF	Net propulsive force
NPP	Net propulsive power
PFC	Propulsive fuselage concept
RANS	Reynolds-averaged Navier–Stokes
SAE	Society of automotive engineers
SHM	SnappyHexMesh
TLAR	Top-level aircraft requirements

1 Introduction

Commercial aircraft concepts featuring a boundary layer ingesting (BLI) propulsive device, such as a fuselage fan (FF), have gained increasing attention in recent years. Prominent examples include the boundary layer ingesting aft-fuselage fan concept Fuse Fan by NASA [1], the Bauhaus Luftfahrt Propulsive Fuselage [2], the propulsive fuselage concepts (PFC) investigated in the European research projects DisPURSAL [3] and CENTRELINE [4], the NASA STARC-ABL [5], and the embedded BLI configuration SAX-40 [6], and the N3-X blended wing body (BWB) [7]. The expected fuel burn reduction potential of these concepts varies in the one digit range between 2 and 10% [3, 8–10] and even a predicted increase in fuel burn of 1.7% [11]. Top-level aircraft requirements (TLAR) as well as the aircraft and power train configuration and geometry of the concepts differ significantly.

An optimal and efficient integration of a BLI system on aircraft level requires a detailed physical understanding of

aero-propulsive effects caused by the interaction between the aircraft components. For a PFC, the incident air stream to the FF is distorted, since it has traversed along the fuselage, thereby forming a boundary layer. When considering the whole aircraft configuration, the flow is further distorted by an interaction between the wing downwash and the boundary layer and interference of air flow at component intersections, such as wing-fuselage, vertical tailplane-fuselage fan cowl-ing, and other three-dimensional features of the aircraft (see Fig. 1). Thus, from a holistic aircraft-level perspective, it is important to consider not only the aerodynamic optimization of a simplified axisymmetric fuselage-propulsor geometry, but also to estimate the impact of three-dimensional geometrical aircraft features on the aero-propulsive aircraft performance to ensure an optimal design.

For some aircraft concepts featuring boundary layer ingestion, two- and three-dimensional geometry optimization schemes have been developed and applied using computational fluid dynamics (CFD) simulations. In 2001, Rodriguez conducted a two- and three-dimensional multi-disciplinary design optimization of a BLI engine inlet of a BWB with a Reynolds-Averaged Navier Stokes (RANS) solver. The study aimed at the demonstration of the potential advantages of BLI for BWB with a focus on engine inlet performance [12]. Gray et al. investigated the effect of the aft-fuselage-propulsor fan pressure ratio (FPR) variation on the performance of the NASA STARC-ABL concept. They introduced a coupled-adjoint approach of a one-dimensional thermodynamic cycle coupled to an RANS CFD simulation. In their approach, the fuselage was modeled as two-dimensional and axisymmetric, neglecting the effect of the wing on the propulsor performance. The aim of the study was to improve the performance of the BLI propulsor by optimizing the propulsor shape for a single design variable (FPR) [13, 14]. Kenway and Kiris applied CFD on a simplified model of the NASA STARC-ABL concept studying the effect of wing downwash. They performed a three-dimensional adjoint-based aerodynamic shape optimization of the fuselage diffuser and nacelle inlet to minimize inlet distortion at the BLI propulsor. Their results showed that the optimal nacelle and aft-fuselage shape is sensitive to design flight conditions and wing downwash [15]. Building on their previous work, Yildirim and Gray and Gray et al. performed geometry optimizations of the three-dimensional NASA STARC-ABL aircraft configuration including vertical tailplane and wing [16, 17]. They aimed at optimizing the geometry for a range of target net forces and FF pressure ratios. In both three-dimensional studies by Kenway and Kiris as well as Yildirim and Gray, fuselage upsweep was not studied in particular, but the fuselage tail geometry was represented by shape variables in the optimization problem.

Martinez Fernandez and Smith conducted three-dimensional RANS CFD simulations to investigate forces and

Fig. 1 Artist's view of the CENTRELINE aircraft configuration (top). Simplified fuselage-propulsor geometry with an upsweep of $\Delta\alpha_{SW,PFC} = 3.5^\circ$ (bottom)

moments acting on the fuselage-propulsor geometry of a tube-and-wing BLI concept compared to a non-BLI fuselage. Both geometries featured fuselage upsweep, but the isolated effect of fuselage upsweep was not studied [18]. In 2020, Ahuja and Mavris published a comprehensive study on the sensitivity of typical tube-and-wing airframe features on BLI performance for a configuration similar to the NASA STARC-ABL concept using RANS CFD [19]. These features included variations in fuselage shape and slenderness ratio, fuselage nose and tail cone design, sensitivity to wing and vertical tailplane design, and the shape of the fuselage leading up to the fuselage fan. They studied the effect of fuselage upsweep for a conventional fuselage without a propulsive fuselage device.

The present study focuses on a three-dimensional numerical investigation of the impact of fuselage upsweep on the performance of a PFC. Fuselage upsweep is a conventional measure in integrated aircraft design used to avoid tail strike during take-off rotation as a compromise between increased fuselage drag and added landing gear weight. When a fuselage upsweep is introduced, the landing gear height can be decreased while ensuring a specified maximum take-off rotation angle. However, the fuselage upsweep introduces distortion to the boundary layer airflow incident to the FF and, thus, affects the potential benefit of a BLI device integration on aircraft level. As was shown in previous studies [14, 15, 19], other components of the airframe, such as the vertical tailplane located in front of the FF, wing, and belly fairing, have an effect on FF inlet distortion and BLI performance. To separate the effect of fuselage upsweep from

other three-dimensional effects, it was chosen to study the influence of fuselage upsweep on the BLI performance in isolation. For a generic fuselage-propulsor arrangement, the effect is analyzed using the open-source CFD tool OpenFOAM [20]. Petrusson showed that the thickness-to-chord ratio of a propulsor cowling can have a significant influence on its surface force [21]. Thus, to investigate the potential of reducing the negative effect of fuselage upsweep through an adaption of the FF nacelle shape, the circumferential variation of the maximum thickness of the propulsor nacelle on the aerodynamic performance is additionally evaluated.

2 Geometry and study parameters

The BLI fuselage-propulsor configuration studied here is similar to the geometry of the fully annular podded, turbo-electric PFC investigated in the CENTRELINE project [4] (see Fig. 1). The baseline geometry consists of an axisymmetric fuselage with a length of 67 m and a diameter of 5.35 m. The FF face is located at 90% fuselage length. It features a tip diameter of 2.37 m and a hub diameter of 1.3 m (see Fig. 2). The baseline geometry, especially the FF duct and nacelle shape, was defined by an engineering judgment; it is not optimized for the given flow conditions. A detailed description of the parameterized geometry can be found in [22].

Fig. 2 Fuselage-propulsor geometry (fuselage nose and tail only). Blue: $\Delta\alpha_{\text{SW,PFC}} = 0^\circ$ (axisymmetric); orange: $\Delta\alpha_{\text{SW,PFC}} = 7^\circ$

2.1 Geometry parameterization for fuselage upsweep

When describing the geometry of a conventional fuselage, the fuselage upsweep angle $\alpha_{\text{SW,Roskam}}$ as defined by Roskam [23] and visualized in Fig. 2 can be employed. From an aircraft level perspective, fuselage upsweep is a means to ensure longitudinal ground clearance in take-off rotation. Conventionally, the longitudinal ground clearance angle is defined as the angle between the ground and a line that can be drawn between the main landing gear on the ground and that part of the fuselage which impacts with the ground first in the event of a tail strike. The landing gear length is, therefore, partly determined by a required minimum ground clearance angle. As in the early phases of conceptual aircraft design, neither the position nor length of the landing gear is defined, an alternative ground clearance angle is introduced for convenience. The ground clearance angle α_{GC} is defined here as the angle between the ground, at the axial position describing the end of the constant fuselage diameter, and the part of the aircraft which impacts with the ground first in the event of a tail strike. In conventional aircraft, this would correspond to the bottom of the fuselage tail, as it is closest to the ground at take-off rotation. However, for

the PFC, this angle needs to be defined with respect to the nacelle, as depicted in Fig. 2. The main landing gear of the configuration is assumed to provide an elevation of $h_{\text{GC}} = 2.5$ m to the fuselage, which is similar to landing gear lengths of existing aircraft, e.g., A330 (2.6 m [24]), A350 (2.54 m [25]). The distance h_{SW} between the point of first impact during a tail strike and the maximum fuselage wall diameter is chosen as the main design variable for upsweep. A new upsweep angle is defined to be used for fuselages featuring an FF, $\alpha_{\text{SW,PFC}}$. A change in the upsweep angle $\alpha_{\text{SW,PFC}}$ results in a change in h_{GC} and hence the ground clearance angle α_{GC} . Thus, the entire FF geometry “shifts” in response to a change in $\alpha_{\text{SW,PFC}}$ or α_{GC} . For a fair comparison of the results, the FF geometry is kept constant for the fuselage upsweep study. This ensures that the cylindrical symmetry of the propulsor and the aft cone is always maintained. The fuselage sigmoid curve until the propulsor hub is morphed to guarantee an azimuthally smooth transition between the fuselage center body and the propulsor. Consequently, the overall wetted area of the upsweep configurations deviates from the reference geometry only by up to 0.22% and the profile drag is nearly invariant for the parts of the geometry unaffected by upsweep. Furthermore, careful shaping

Table 1 Values of upsweep parameters for various aircraft

	$\alpha_{\text{SW,Roskam}} [^\circ]$	$\alpha_{\text{SW,PFC}} [^\circ]$	$\alpha_{\text{GC}} [^\circ]$	$h_{\text{SW}} [\text{m}]$	$\Delta\alpha_{\text{SW,PFC}} = \alpha_{\text{SW,PFC}} - \alpha_{\text{SW,PFC,axi}} [^\circ]$
A330-300	5.7	15.8	22.4	5.5	–
A350-1000	6.4	17.8	22.0	5.2	–
B787-9	5.9	16.9	22.9	4.5	–
PFC	0.0	6.70	18.8	1.3	0.0
	2.2	10.2	21.9	2.0	3.5
	2.9	11.2	22.8	2.2	4.5
	4.5	13.7	25.0	2.8	7.0

The values for commercial aircraft are approximated from [24–26]

Case	Top $(t/c)_1$	Bottom $(t/c)_2$
Reference Geometry	0.08	0.08
$(t/c)_1$ Variation	0.06	0.08
	0.10	
	0.12	
$(t/c)_2$ Variation	0.08	0.06
		0.10
		0.12
$(t/c)_1$ & $(t/c)_2$ Variation	0.06	0.12
	0.10	0.06
	0.12	0.06

Fig. 3 Nacelle thickness-to-chord ratio variation

Fig. 4 Fuselage fan stations

of the fuselage geometry prevents separation of the flow at the lower part of the fuselage.

To study the effect of additional upsweep, three geometries were generated with $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ, 4.5^\circ$ and 7° compared to the axisymmetric geometry. Table 1 presents values of the upsweep variables for various existing aircraft with a size similar to that of the PFC geometry. The minimum and maximum values for the PFC geometry correspond to the axisymmetric and $\Delta\alpha_{\text{SW,PFC}} = 7^\circ$ upsweep case, respectively. It can be shown that existing aircraft have an average value of $\alpha_{\text{GC}} = 22.5^\circ$, which corresponds to $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$ for the PFC configuration.

2.2 Geometry parameterization for nacelle thickness variation

To alleviate the detrimental effects of upsweep on the fuselage-propulsor performance, the geometry of the FF can be adapted to the changed flow conditions. These non-axisymmetric adaptations mainly rely on an azimuthal variation of the outer FF nacelle profile. The azimuthal variation of the nacelle maximum thickness is investigated in the present study, serving as an example for a geometric optimization of the fan. Figure 3 (right) depicts this variation. To study the effects of variation of FF performance with change in nacelle exterior surface shape alone, the thickness-to-chord ratio of the FF nacelle (t/c) is varied, while the interior contour of the nacelle through all propulsor stations (1, 2, 13, and 18, see Fig. 4) remains constant across all configurations. As a

result, the airflow duct inside the propulsor is identical in every configuration. This ensures a fair comparison of the effects of the change in nacelle thickness on pressure and viscous drag and their correlation to FF performance.

The thickness-to-chord ratio at the top $(t/c)_1$ and bottom $(t/c)_2$ of the FF nacelle is varied independently and the external surface of the nacelle varies continuously as it transitions from the top to bottom profile. In this study, nacelle thickness variation is performed on a fuselage-propulsor geometry with upsweep angle $\alpha_{\text{SW,PFC}} = 10.2^\circ$ ($\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$), as this geometry most closely represents the upsweep of present aircraft. The table in Fig. 3 lists the values of the uppermost and lowermost surface thickness-to-chord ratio of the eight evaluated geometries. (t/c) is varied independently for the top and bottom surface from 0.06 to 0.12, with the reference case value being 0.08.

3 Simulation setup

The three-dimensional numerical simulations are performed using the open-source CFD software OpenFOAM v1706 [20], which models fluid flows by solving the RANS equations. The bidirectional influence between FF propulsion system and airframe is modeled by a fan model based on physical principles. The geometry is represented by a symmetrical half-model.

3.1 Solver

The High-Speed Aerodynamic solver (HiSA) [27] is employed for this study. The solver models compressible transonic and supersonic flows using a density-based approach, which allows for the resolution of numerical discontinuities, such as the formation and propagation of shock waves [28]. HiSA utilizes two flux interpolation schemes, the Harten–Lax–van Leer Contact solver and the Advection Upstream Splitting Method, to improve the computation of the face fluxes over a wider range of speed regimes.

The Runge–Kutta time-stepping scheme is implemented in the solver, which improves stability and accuracy while incrementing the speed of solution [28]. This approach involves solving the system of equations using a generalized minimum residual method with lower–upper symmetric Gauss–Seidel preconditioning. Finally, to best approximate the flow domain, the solver allows for the definition of non-reflective farfield boundary conditions, which is essential when modeling transonic external flows with shocks. Furthermore, it is ensured that truncating the numerical domain does not influence the development of the flow [29].

A second-order least-squares approach is used to calculate the gradient terms. It leads to faster convergence and a more stable simulation compared to a Gauss linear scheme. Divergence terms are evaluated using a second-order Gauss scheme with linear interpolation. An orthogonal scheme is used for the computation of surface normal gradients and Laplacian schemes, which can be used on meshes of low non-orthogonality. Gradient limiters for pressure, temperature, velocity, and the turbulent quantities are defined to improve stability and convergence. The $k\text{-}\omega$ -SST turbulence model is applied, which provides the best accuracy for predicting transonic flows in combination with the HiSA solver (see Sect. 3.4).

Instead of fully resolving the boundary layer, wall functions are employed to limit the required computational resources (see Sect. 3.2).

3.2 Numerical grid

The open-source computer-aided design (CAD) software FreeCAD [30] is used to generate the required half-models of the three-dimensional geometries described in Sects. 2.1 and 2.2. The geometries are cut at their x – z -symmetry plane to capture non-axisymmetric effects while saving computational resources. The mesh is generated from a surface descriptive file (e.g., .stl file) using OpenFOAM's native mesh generator, SnappyHexMesh (SHM) [31]. The mesh generation utility allows us to generate a hex-dominant structured mesh. SHM utilizes an additional background mesh to generate the mesh around the geometry contained in the .stl file and it allows the generation of a “boundary layer” mesh, i.e., densely packed cells near the nacelle and fuselage wall to capture the boundary layer. In the near wall region of the fuselage and the nacelle, a boundary layer containing a structured hexagonal mesh is created. The mesh downstream of the fuselage aft cone is refined to capture the wake and flow changes due to the propulsor.

To reduce computational effort, a grid study is performed on the two-dimensional axisymmetric reference fuselage-propulsor geometry to test the applicability of wall functions. A control case with fully resolved boundary layers, i.e., complete turbulence resolution and an average

dimensionless wall distance $y^+ < 0.4$ is used as a benchmark. The cases are compared by the error in the calculation of total surface forces, fan force, and FF shaft power (see Sect. 4.2) with respect to the control case. The results indicate less than 1% error when using a combination of the following wall functions: `kqRWallFunction`, `nutUWallFunction`, `omegaWallFunction`, and `alphatWallFunction` with an average y^+ of ~ 60 . Hence, the above-mentioned wall functions are employed and the height of the first cell closest to the wall is set to $7.5 \cdot 10^{-4}$ m for all geometries considered in this study, which correspond to the aforementioned y^+ of 60. The simulation domain extends by $5 \cdot l_{\text{fuse}}$ in streamwise direction and $20 \cdot d_{\text{fuse}}$ in axial direction with a total number of ~ 5.5 – 6 million cells.

3.3 Fuselage fan model

In principle, a compressor fan imparts energy to an incident flow, which results in an increase in total pressure and total temperature. The total pressure increase is subsequently converted to an increase in flow velocity by a succeeding contracting duct, thereby generating thrust. The part of the podded propulsor examined here, which influences the fluid flow incident from the fuselage, consists of a fan rotor and stator, as well as a duct bounded by the fuselage contour (duct inner wall) and the lower nacelle contour (duct outer wall). In addition, the fuselage aft cone acts as a part of the nozzle wall.

A simplified fan model has to meet several requirements: as the design of a BLI fan is highly dependent on the distorted inlet flow conditions and therefore cannot be specified at an early stage of the conceptual aircraft design, its main characteristics have to be described in the CFD setup without detailed knowledge of the FF geometry and its properties. Furthermore, the model has to function independently of the prescribed fan characteristics such as peak input shaft power and fan diameter. This implies that the model must function with minimal and geometry independent input parameters. Finally, the model must be simple enough to allow an aircraft-level simulation with reasonable computational effort. Practically, this means that the FF needs to be modeled without actually simulating it as a physical element.

Commercial CFD software such as ANSYS® Fluent® allow the implementation of a fan boundary condition to simulate the effects of a fan under various conditions. These are based on simple thermodynamical relations, but also require an additional input of certain fan characteristic curves while being relatively accurate [32]. To allow an implementation without the requirement of additional inputs, the model used here is based on the following simple thermodynamic relations, which are applied to every cell of the FF inlet and outlet boundary patches through an arbitrary mesh interface (AMI) interpolation. The errors incurred due to this simplification

will be consistent across all simulations and hence cancel out in a comparative study employing the same model for all simulations.

In the present study, the combined effect of FF rotor and stator on the fluid flow is modeled as a “black box” using customized boundary conditions at the patches representing the FF rotor inlet at station 2 and the FF stator outlet at station 13 (see Fig. 4). Its effect is represented by a total pressure and temperature jump across these two patches. Mass flow continuity is ensured using velocity as a derived variable.

The FF PR is chosen as an input parameter for the fan model. The losses associated with the fan are modeled by the use of a polytropic efficiency $\eta_{p_{fan}}$. The polytropic efficiency of the combined FF rotor and stator stage is defined as 92.5% in a first instance and the FPR is fixed to 1.4 for all studies following the design of the PFC investigated in the CENTRELINE project [4]. The FPR of 1.4 is the outcome of a pre-sizing study aiming at the best power ratio between the electric FF and the turboelectric underwing podded engines [4]. The polytropic fan efficiency is approximated to be 1% smaller than the cruise fan efficiency of the underwing podded ultra high bypass ratio geared turbofan engines [33].

Total pressure and total temperature for a transonic compressible flow are described by:

$$p_t = p_s \cdot \left(1 + \frac{\kappa - 1}{2} M^2\right)^{\frac{\kappa}{\kappa - 1}} \quad (1)$$

$$T_t = T_s \cdot \left(1 + \frac{\kappa - 1}{2} M^2\right), \quad (2)$$

where p_s is the static pressure, κ is the ratio of specific heat capacities, and T_s is the static temperature and the Mach number is defined as: $M = \frac{U}{\sqrt{\kappa R T_s}}$, with U being the magnitude of the flow velocity and R the universal gas constant. Across the fan stage, the total pressure of the flow is increased by a factor $\prod_{fan}$, which is equal to the fan pressure ratio FPR [34]. Hence, the total pressure at the fan outlet can be calculated using the total pressure at the fan inlet:

$$p_{t13} = p_{t2} \cdot \prod_{fan}. \quad (3)$$

The change in total temperature over the fan stage is defined by the FPR and the polytropic efficiency [34, 35]:

$$T_{t13} = T_{t2} \cdot \prod_{fan}^{\frac{\kappa - 1}{\kappa \eta_{p_{fan}}}}. \quad (4)$$

Mass flow continuity is maintained throughout the entire propulsor by making sure that the following condition is satisfied [34]:

$$\dot{m}_2 = \dot{m}_{13} \Leftrightarrow \rho_2 A_2 U_{x_2} = \rho_{13} A_{13} U_{x13}, \quad (5)$$

Table 2 Freestream conditions for test case 2308 of the Onera M6 wing

Quantity	Value	Symbol
Mach number [-]	0.84	Ma_∞
Reynolds number [-]	$14.6 \cdot 10^6$	Re_∞
Angle of attack [°]	3.06	α
Velocity $\left[\frac{m}{s}\right]$	291.25	U_∞
Density $\left[\frac{kg}{m^3}\right]$	1.1402	ρ_∞
Pressure [Pa]	98,193	p_∞
Temperature [K]	300	T_∞
Root chord [m]	0.8105	c_{root}

A is the cross-sectional area of the particular propulsor station.

3.4 Validation

The HiSA solver has been successfully validated for compressible, transonic flow conditions [27]. To evaluate the solver prediction capabilities for pressure and skin friction for three-dimensional transonic flows, the flow around the well-studied Onera M6 wing is computed using the same simulation setup as applied in the presented study (Sect. 3.1).

For the validation, the mesh provided by the CFL3D Test/Validation archive is used [36]. Freestream conditions corresponding to case 2308 of the Onera S2MA wind tunnel test are provided by NASA Langley Research Center Turbulence Model Numerical Analysis [37] and documented in [38] (see Table 2). Experimental results for the pressure distribution are documented for the original wind tunnel tests in [39]. The skin friction distribution is compared against numerical results computed with the CFD codes FUN3D and CFL3D developed at NASA Langley Research Center [38].

The solver predicts the pressure distribution as well as the skin friction distribution in x -, y -, and z -direction well across the wingspan, as depicted in Figs. 5 and 6. Among other reasons, discrepancies in the numerical results for the skin friction distribution (FUN3D, CFL3D and OpenFOAM) might arise from uncertainties concerning the freestream conditions used in the original studies as well as the slightly different meshes.

4 Study results

The results of fuselage upsweep as well as nacelle thickness-to-chord ratio variation are evaluated based on an aero-propulsive performance characteristic, the BLI efficiency factor $f_{\eta, BLI}$ (see Sect. 4.2). As expected, fuselage upsweep impairs the fuselage-propulsor performance significantly. Strategic

Fig. 5 Comparison of pressure coefficient and skin friction distribution over Onera M6 wing determined experimentally, predicted by different CFD solvers and by the OpenFOAM HiSA solver at 20, 65

and 96% wing span. Experimental data from [39], CFL3D, FUN3D, and USM3D data from [38]

Fig. 6 Pressure distribution on upper side of Onera M6 wing. Left: pressure coefficients computed by OpenFOAM HiSA with indicated probe sections used in Fig. 5. Right: pressure contours computed by USM3D on family 4 prism/hex L1 grid. Adapted from [38]

Table 3 Freestream conditions

Quantity	Value	Symbol
Mach number [-]	0.8	Ma_∞
Reynolds number [-]	$4.4 \cdot 10^8$	Re_∞
Gas constant $\left[\frac{J}{kg \cdot K}\right]$	287.05	R
Specific heat ratio [-]	1.4	κ
Velocity $\left[\frac{m}{s}\right]$	237.23	U_∞
Density $\left[\frac{kg}{m^3}\right]$	0.3796	ρ_∞
Pressure [Pa]	23842	p_∞
Temperature [K]	218.81	T_∞
Turbulent kinetic energy $\left[\frac{J}{kg}\right]$	0.0844	k
Turbulent frequency $\left[\frac{1}{s}\right]$	53046	ω

alteration of the outer FF nacelle surface geometry can, however, efficiently alleviate the effect.

4.1 Freestream conditions

The flow field is initialized with the following freestream values, which correspond to cruise flight conditions $Ma = 0.8$ at FL350 (10,668 m) and ISA+0K (see Table 3). At the x - z -symmetry plane of the domain, a symmetry plane boundary condition is applied.

4.2 Performance characteristics

To compare the aero-propulsive performance of the geometric variations with each other, the BLI benefit is evaluated with a metric that combines the effect of the propulsor on the airframe as well as the effect of the altered inflow on the

propulsion system—the BLI efficiency factor as introduced in [40]. It relates the product of the net propulsive power (NPP) to the isentropic expended propulsor shaft power:

$$f_{\eta, BLI} = \frac{NPP}{P_{shaft, iso}} \forall P_{shaft, iso} > 0. \quad (6)$$

The NPP is the product of net axial force acting on the bare fuselage and propulsor arrangement ($\hat{=}$ net propulsive force (NPF)) and the freestream flight velocity: $NPP = NPF \cdot U_\infty$. The NPF is the sum of all axial forces acting on the configuration—pressure and viscous forces on all surfaces (fuselage, propulsor nacelle, propulsor duct, and aft body) $F_{surface}$ and the absolute value of the fan stage volume force F_{fan} . All surface forces are calculated by integration of the pressure and viscous forces on the surfaces. The contribution of the pressure and momentum flux of fan face and fan exit to the NPF is calculated from mass flow averaged values at propulsor stations 2 and 13 (see Fig. 4). A negative NPF represents a force in drag direction:

$$NPF = |F_{fan}| - F_{surface} = \dot{m}(U_{13} - U_2) + A_{13}p_{s,13} - A_2p_{s,2} - \sum_i \iint_{S_i} (pn_x + \tau_w) dS_i. \quad (7)$$

In the present study, the isentropic shaft power $P_{shaft, iso}$ is replaced by the actual shaft power $P_{shaft} = P_{shaft, actual}$, which is the isentropic shaft power corrected by the polytropic efficiency of the FF. As the polytropic fan efficiency is kept constant across all studied cases, the resulting trends, however, will be similar. The actual propulsor shaft power is determined from mass flow averaged propulsor station enthalpy:

$$P_{shaft} = \dot{m}(h_{13} - h_2). \quad (8)$$

In addition, net and gross thrust are defined as:

$$F_g = \dot{m}U_{18} + A_{18}(p_{s,18} - p_{s,\infty}) \quad (9)$$

$$F_n = F_g - \dot{m}U_\infty. \quad (10)$$

For a given flight condition, the fuselage and FF nacelle force in drag direction should either be minimized for a given expended propulsor shaft power or a minimum shaft power should be expended for a given NPF.

For non-axisymmetric fuselage-propulsor configurations, circumferential distortion is added to the radial distortion inherent to the ingested boundary layer. To quantify the effect, the distortion coefficient defined by the Society of Automotive Engineers (SAE) in the “Gas Turbine Engine Inlet Flow Distortion Guidelines” is calculated at the Aerodynamic Interface Plane (AIP), which is here placed 30 cm in front of the FF face [41]. The coefficient is based on the

Fig. 7 Exemplary total pressure distribution for upsweep case $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$, ring $i = 11$

calculation of the intensity of circumferential distortion indicated by regions of low total pressure. For a number of concentric rings, n , the intensity is calculated for a number of equidistantly spaced points in circumferential direction, m . In the present study, $n = 12$ rings were evaluated with $m = 100$ samples for a half ring (see Fig. 9, left). The total pressure distribution studied here follows a one-per-rev pattern (the region of low pressure is separated only by one region of high pressure) and, thus, the following definition for the intensity I_i of the i th ring applies [41]:

$$I_i = \left(\frac{\text{PAV} - \text{PAVLOW}}{\text{PAV}} \right)_i \quad (11)$$

The following equations are used to calculate the average total pressure PAV_i and the average low total pressure PAVLOW_i of each ring as visualized in Fig. 7 [41]:

$$\text{PAV}_i = \frac{1}{360} \int_0^{360} p_t(\Theta)_i d\Theta \quad (12)$$

$$\text{PAVLOW}_i = \int_{\Theta_{1i}}^{\Theta_{2i}} p_t(\Theta)_i d\Theta. \quad (13)$$

$p_t(\Theta)_i$ is the linear fit between the data points in circumference. Θ_i is the circumferential extent of the low-pressure region with Θ_{1i} and Θ_{2i} being its lower and upper limits [41]. The maximum value of the intensities calculated for each ring $\max(I_i)$ is the maximum circumferential intensity I_{max} .

All performance parameters are calculated by an automated post-processing of the CFD results.

4.3 Fuselage upsweep study

The performance of the baseline case is compared to an isolated fuselage, which is shaped like the axisymmetric fuselage-propulsor geometry, but without an FF and nacelle present. For this case, the NPF is decreased by the

Table 4 Circumferential distortion intensity for upsweep cases

$\Delta\alpha_{\text{SW,PFC}}$	I_{max}
0.0	0.0069
3.5	0.0343
4.5	0.0409
7.0	0.1505

presence of the FF from a force in drag direction of -20.32 to -0.97 kN, even though the total surface force in drag direction is increased from -20.32 to -25.82 kN. The additional surface force derives mainly from the added nacelle wetted surface.

With increasing fuselage upsweep, the lower portion of the propulsor is more deeply embedded in the fuselage boundary layer, as shown in Fig. 8. The boundary layer thickness is increased due to a larger rate of change of area at the lower fuselage tail (see Fig. 9). Conversely, the upper part of the propulsor sees a bigger amount of the boundary layer. This leads to an azimuthal imbalance in the ingested airflow, both in terms of the local fluid flow properties, such as inlet Mach number, as well as local mass flow. The propulsor faces a flow with two different gradients; radial outwards due to the boundary layer and azimuthal due to the fuselage upsweep. Consequently, distortion increases with increasing fuselage upsweep (Fig. 10 and Table 4). A non-zero circumferential distortion for the axisymmetric case is due to numerical errors. Distortion increases by $\sim 19\%$ between $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$ and 4.5° . The circumferential distortion intensity is similar to the results reported by Gray et al. [17]. The fuselage geometry investigated in their study features an $\alpha_{\text{SW,PFC}}$ of $\sim 10^\circ$, which is close to the value of this study's $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$ case ($\alpha_{\text{SW,PFC}} = 10.2^\circ$). Gray et al. conducted three-dimensional numerical optimization studies for the NASA STARC-ABL fuselage-propulsor configuration including wing, vertical tailplane and fuselage upsweep at $\text{Ma} = 0.785$ and FL370. For resulting FPR values ranging between 1.23 and 1.31, the distortion coefficient of the optimized geometries ranges between 0.030 and 0.044. A significantly lower degree of distortion would be expected for the results of the present study, as wing and vertical tailplane are not accounted for and the FF features a higher FPR. However, the effect might be offset due to added distortion from the unoptimized fuselage and propulsor geometry.

For high upsweep angles, the flow downstream of the propulsor is highly distorted due to the high inflow distortion, as shown in Fig. 11. Even though the assumption of constant FPR and $\eta_{p_{\text{fan}}}$ prescribes an axisymmetric influence of the fan on the flow, a high degree of distortion cannot be corrected even by mass flow averaging. A single-stage FF as modeled in this study is not sufficient to completely negate the large variation in the stagnation pressure incident to the

Fig. 8 Comparison of Mach number distribution with and without a BLI propulsor for $\Delta\alpha_{SW,PFC} = 3.5^\circ$. Boundary layer profiles at $x/l_{fuse} = 79\%$, 88% , and 89% (FF highlight)

FF, as depicted in Fig. 10. The downstream flow distortion is detrimental to the FF performance, as the major benefit of a BLI fan is obtained by the recovery of the fuselage-induced

momentum deficit by wake-filling [35]: The asymmetry of downstream flow reduces the amount of momentum

Fig. 9 Boundary layer velocity profiles in front of the fuselage fan ($x/l_{\text{fuse}} = 88\%$) for $\Delta\alpha_{\text{SW,PFC}} = 0, 3.5$, and 7°

Fig. 10 Stagnation pressure distribution at AIP. Sample points for distortion coefficient calculation depicted for $\Delta\alpha_{\text{SW,PFC}} = 0^\circ$ (left)

Fig. 11 Mach number distribution around nacelle and aft cone for $\Delta\alpha_{\text{SW,PFC}} = 7^\circ$

recovered circumferentially. Thus, the benefit of integrating a propulsive device is reduced.

This effect can be seen in the increase in the negative NPF (increase of force in drag direction) and $f_{\eta, \text{BLI}}$ with an increase in upsweep angle. This penalty on performance is non-linear and increases drastically beyond the intermediate angles, unlike the increase in shaft power. Figure 12 compares the total surface force and the FF force of the four configurations. The trends indicate a sudden reduction in fan

force beyond an intermediate value of upsweep caused by flow separation in the fan duct. The flow separation is caused by a reduced mass flow in front of the lower part of the FF. The flow is accelerated around the lip of the nacelle and for the case with $\Delta\alpha_{\text{SW,PFC}} = 7.0^\circ$, and this leads to flow separation inside the nacelle between stations 1 and 2 (Fig. 11). A three-dimensionally optimized nacelle profile could avoid this undesired flow behavior.

For all configurations, the wetted area remains approximately the same. Thus, the increase in total surface force is due to a vertical shift of the nacelle position. With higher upsweep, the upper part of the nacelle is subject to faster flow, since the boundary layer thickness is smaller. Correspondingly, the viscous and pressure force of the FF nacelle is increased. The lower portion of the nacelle experiences a slower airflow. This leads to an insufficient local mass flow at station 1 and causes a separation in front of the fan face, as described above. The separation causes an increase in surface forces on the inner lining of the nacelle, which contributes to the higher total force in drag direction (see Fig. 12, left). Finally, the downstream flow distortion induces the formation of a shock on the aft-fuselage cone, which leads to an increased pressure force.

As indicated in Fig. 12, left, the FF nacelle contributes nearly 40% of the total surface force. A sharp increase in total force beyond an intermediate value of upsweep corresponds to the deterioration of flow around the nacelle and its effect on the fuselage flow downstream of the FF.

Even a small increase in P_{shaft} is significant for a BLI fan driven by a turboprop power source, such as the investigated configuration. Due to efficiency losses in the electric system, an increased power demand of the FF leads to an even higher power generation demand from the main engines [8]. Hence, it is imperative to limit the increase in P_{shaft} . Compared to the axisymmetric geometry, shaft power increases significantly as the propulsor forces the airflow near the lower fuselage contour to conform to its azimuthal symmetry. As the upsweep angle increases, P_{shaft} continues to increase with a diminishing slope, i.e., less increase in P_{shaft} for higher angles of upsweep (Fig. 12). A part of the fan power is expended in achieving the specified azimuthally constant FPR and $\eta_{p, \text{fan}}$, which forces the airflow to conform to the cylindrical symmetry of the propulsor. This produces a sharp increment in shaft power as soon as flow asymmetry occurs. The main cause of this sharp increase in power is an increase in mass flow and flow velocity. However, the increase in shaft power at higher upsweep angles is due to a higher relative change of the mass averaged flow temperatures and velocities through the fan stage. The mass flow values drop after $\Delta\alpha_{\text{SW,PFC}} = 3.5^\circ$ due to poor flow at the lower part of the fan despite the overall increase in axial direction.

The BLI efficiency does not strongly reflect the initial sharp increase in P_{shaft} . As the shaft power increases by 23%,

Fig. 12 Performance metrics for fuselage upsweep variation

from $\Delta\alpha_{SW,PFC} = 0$ to 7° , the NPF increases by 1073%, even though the total increase in force is only 10.4 kN. 10.4 kN equal 51.4% of the force on a fuselage, which is shaped like the axisymmetric case, but without an FF present. This indicates that the propulsor can tolerate a certain amount of flow asymmetry. However, there is a sudden reduction of performance as the upsweep continues to increase. The trends depicted by $f_{\eta,BLI}$ suggest that the loss of performance due to upsweep can be depleted successfully if the upsweep angle is not too large. Thus, the practical implementation of a BLI propulsor in such a configuration is only viable for intermediate upsweep angles.

From an aerodynamic point of view, fuselage upsweep should be avoided to keep the propulsor inlet distortion to a minimum. However, fuselage upsweep might be considered in two cases:

1. If measures to ensure ground clearance, such as the increase of the landing gear length, entail second-order effects on aircraft level, which outdo the positive effect of a BLI device integration, or
2. if the FF nacelle can be optimized in such a way that the increase in shaft power can be alleviated.

Second-order effects may include weight penalties, change in airport infrastructure, and maintenance accessibility. To estimate the effect of the introduction of fuselage upsweep on the landing gear length, the x-position of the main landing gear is assumed as 50% of the total fuselage length and a maximum take-off rotation angle of 10° . For the axisymmetric configuration, the maximum FF nacelle thickness is decisive in defining the fuselage point, which touches the ground first. In this case, the main landing gear length is 3.53 m. For any geometry with $\Delta\alpha_{SW,PFC} > 3.5^\circ$, the end of

center fuselage geometry touches the ground first in case of tailstrike. For these cases, the main landing gear length is 2.84 m. Thus, by introducing fuselage upsweep, the landing gear length can be reduced by 19.5%.

4.4 Nacelle thickness variation study

Ten different nacelle profiles with varying uppermost and lowermost thickness-to-chord ratios (t/c) are investigated (Table 3). The thickness variation study is conducted on a geometry with $\alpha_{SW,PFC} = 10.2^\circ$, i.e., $\Delta\alpha_{SW,PFC} = 3.5^\circ$. This fuselage upsweep most closely resembles the geometry of existing aircraft and has a relatively low impact on the FF performance characteristics (Sect. 4.3).

The velocity field in front of the FF nacelle is directly affected by a variation in the nacelle thickness-to-chord ratio: a reduction of the upper nacelle thickness suppresses the shock formation on top of the nacelle. However, the nacelle profile for $(t/c)_1 = 6\%$ leads to a higher surface force in drag direction due to the c_p distribution caused by a faster flow at the upper surface. Consequently, the benefits of removing the shock on the upper side of the nacelle are reduced. The shape of the nacelle lip varies with a change in the nacelle thickness due to the chosen geometry parameterization strategy. In effect, the suction force and, consequently, the fan force are lower for a smaller $(t/c)_1$ (see Fig. 13). This leads to an overall reduction in FF performance, since its wake-filling effect is reduced. The increase in NPF with an increased thickness-to-chord ratio of the upper nacelle profile indicates improved effectiveness of the nacelle. Thus, despite an increase in nacelle force, the BLI performance of the configuration is improved for thicker nacelle profiles on the upper surface, as shown in Fig. 15. Increasing the upper thickness to 12%, however, leads to flow separation at the

Fig. 13 Forces for upper (left) and lower (right) nacelle thickness variation

Fig. 14 Mach number distribution for nacelle thickness variation examples. The baseline case is depicted in the left bottom corner

inner side of the nacelle lip. Thus, an intermediate thickness-to-chord ratio of the upper nacelle surface ($(t/c)_1 = 10\%$) seems to be the most promising (see Fig. 14).

Varying the thickness on the lower part of the nacelle has a low impact on the surface forces. The maximum surface force variation is only 3% as compared to 6.5% when varying $(t/c)_1$. This is caused by the slower incident flow on the lower surface, which, in turn, leads to a smaller absolute force on the lower nacelle surface. An increase in nacelle thickness facilitates the existing distortion of the inlet flow. Within the propulsor, the effect of increasing the thickness is almost the same as that of an increase in upswep. A flow separation within the duct leads to a pronounced asymmetry of the flow behind the nacelle. As discussed in the previous section, this is detrimental to the BLI performance. Both NPF and $f_{\eta, \text{BLI}}$ drop in response to an increase in $(t/c)_2$. The shaft power begins to drop between $(t/c)_2 = 8\%$ and

Fig. 15 Aero-propulsive performance for thickness-to-chord ratio variations. Negative NPF can be interpreted as force in drag direction. Annotations: $(t/c)_1 | (t/c)_2$

10% (Fig. 15). A reason for this can be the continuous circumferential transition from the top to the bottom nacelle profile. The increased thickness-to-chord ratio of the lower profile entails an increase in the nacelle profile thickness along the horizontal x - y -plane. Hence, the beneficial effects of a thicker nacelle in the relatively higher speed flow are reflected in a reduced shaft power.

The variation in nacelle thickness on the bottom surface has a bigger impact on the performance of the propulsor-fuselage configuration. $f_{\eta, \text{BLI}}$ varies by 89% due to changes in the lower profile thickness as compared to a 65% variation for the upper profile thickness. Therefore, in particular, the adaption of the lower nacelle profile to the distorted flow conditions can be effective.

Based on the best-performing nacelle profiles for the independent variation of the top and bottom nacelle profile thickness-to-chord ratio, a geometry is generated with $(t/c)_1 = 10\%$ and $(t/c)_2 = 6\%$. The combination of the positive effects of both alterations show that the shaft power and surface forces are close to those of the axisymmetric case. The other performance characteristics also indicate the best performance of the geometry among all the thickness variation profiles considered, as shown in Fig. 15. However, BLI efficiency and NPF cannot be fully recovered to the values of the axisymmetric case, since only the top and bottom profile thicknesses are adapted. In addition, the maximum circumferential distortion intensity is reduced by $\sim 22\%$ and $\sim 8\%$ compared against the $\Delta\alpha_{\text{SW, PFC}} = 3.5^\circ$ baseline case for $(t/c)_1 = 8\% / (t/c)_2 = 6\%$ and $(t/c)_1 = 10\% / (t/c)_2 = 6\%$, respectively. In theory, a complete, three-dimensional optimization of the outer nacelle surface may result in a complete negation of the negative effect fuselage upsweep has on the fuselage-propulsor performance. However, distortion might not be reduced to the level of the axisymmetric case by a mere shape optimization.

4.5 Limitations

The results presented here have a limited validity for other PFC due to the underlying assumptions of the applied methods.

First, all geometries, baseline, and all derivations thereof are not optimized for the flow conditions examined in this study. Thus, it is not possible to attribute all occurring effects to the introduction of fuselage upsweep only. The applied geometry, especially the unchanged FF geometry, affects the fluid flow negatively (e.g., shocks on outer nacelle and separation inside FF duct). Furthermore, the performance of the FF is highly dependent on the introduced inlet flow distortion. Distortion due to the build-up of a boundary layer on the fuselage is altered by interference with wing downwash, interference of different aircraft components, blockage of the

FF by the vertical tailplane, etc., which are not considered in this study.

Another uncertainty arises from the applied CFD simulation setup. Even though the approximation of near wall effects with wall functions seems to result in only a minor error (see Sect. 3.1), a full resolution of the boundary layer would be preferable. Due to a lack of detailed fan characteristics, the implemented fan model simplifies the influence of a compressor on a fluid to its fundamental effects. The whole fan stage, rotor and stator, is modeled as a single “black box”. FPR and polytropic fan efficiency serve as input parameters to the fan boundary conditions. However, in reality, the characteristics of the fan blades would lead to a non-uniform FPR distribution across the FF face and the polytropic fan efficiency would be the outcome of a designed fan stage operating at a certain condition. Furthermore, three-dimensional effects of the fan are not modeled by the simplified model, i.e., swirl introduction to downstream flow. In addition, the implemented method leads to an overprediction of the upstream suction effect.

The mitigation of the negative fuselage upsweep effects by a nacelle shape optimization is based on studies on the isolated variation of either upper or lower nacelle thickness-to-chord ratio. Due to the applied parameterization approach, the results of the study are counterintuitive. If along with the nacelle thickness, which shapes the outer part of the nacelle, the position of the nacelle highlights and the shape of the inlet (stations 1–2) had been adapted, the optimum for the lower and upper nacelle thickness-to-chord ratio could have been different.

As the largest share of long-range total mission fuel is burned during cruise, the study presented here focuses on an assessment of the propulsor-fuselage performance for this specific operating condition. However, if the FF is operated at other times, such as take-off, a compromise of optimum geometries for the different operating conditions has to be found.

Despite the limitations of the study, it can be shown that the shaping of the fuselage-propulsor geometry has a significant impact on the aero-propulsive performance of a PFC. Furthermore, the introduction of fuselage upsweep has a strong effect on the required shaft power, which is decisive for a compromise between landing gear and fuselage shape adaption on aircraft level.

5 Conclusion

The aerodynamic shape of the fuselage and propulsor of a PFC has a significant impact on its aero-propulsive performance. From an aircraft-level perspective, all components of an aircraft have to be designed in such a way that all external constraints are met while minimizing fuel consumption.

Consequently, it may be necessary to ensure sufficient ground clearance during take-off by introducing a fuselage upsweep instead of an increased landing gear length. The impact of such a three-dimensional geometric variation on the bidirectional aero-propulsive performance of a PFC was investigated through three-dimensional RANS CFD simulations. It was shown that fuselage upsweep distorts the air-flow incident to the FF significantly and, consequently, has a detrimental effect on its performance. Changing from an axisymmetric to a non-axisymmetric fuselage alone increases the required FF shaft power. The effect is intensified with increasing upsweep. Alteration of the FF nacelle geometry as a means to mitigate the negative effect of the fuselage upsweep was assessed. By varying the thickness-to-chord ratio of the uppermost and lowermost nacelle profile, the performance of the fuselage-propulsor configuration was restored to the performance of the axisymmetric baseline geometry.

To fully answer the question whether the negative effect of fuselage upsweep can always be sufficiently mitigated by an optimized aerodynamic design of the fuselage-propulsor configuration, it will be necessary to conduct a three-dimensional geometrical optimization for all cases. Furthermore, the interaction of other aircraft components, such as wing or vertical tailplane, with the fuselage boundary layer and thus, the altered distortion of the air flow incident to the FF and, consequently, the PFC performance, has to be taken into account. The behavior of the BLI device for all operating conditions has to be assessed. An aircraft-level design study is required to select which compromise, an elongation of the landing gear length or a fuselage-propulsor design variation, will minimize the negative impact on mission fuel burn.

Acknowledgements We would like to thank Rebecca Zahn for her contribution to the study and Arne Seitz for his valuable feedback. This work was conducted within the CENTRELINE project, which has received funding from the European Union's Horizon 2020 research and innovation programme under Grant Agreement no. 723242.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit <http://creativecommons.org/licenses/by/4.0/>.

References

1. Bolonkin, A.: A high efficiency fuselage propeller ("Fusefan") for subsonic aircraft. In: SAE Technical Paper, no. 1991-01-5569, pp. 1063–1075 (1999)
2. Steiner, H.-J., Seitz, A., Wieczorek, K., Plötner, K., Isikveren, A.T., Hornung, M.: Multi-disciplinary design and feasibility study of distributed propulsion systems. In: 28th International Congress of the Aeronautical Sciences (2012)
3. Isikveren, A.T., Seitz, A., Bijewitz, J., Mirzoyan, A., Isyanoc, A., Grenon, R., Atinault, O., Godard, J.-L., Stückl, S.: Distributed propulsion and ultra-high by-pass rotor study at aircraft level. *Aeronaut. J.* **119**(1221), 1327–1376 (2015)
4. Seitz, A., Bijewitz, J., Habermann, A., Goraj, Z., Kowalski, M., Castillo Pardo, A., Hall, C., Meller, F., Merkler, R., Petit, O., Samuelsson, S., Della Corte, B., van Sluis, M., Wortmann, G., Dietz, G.: Concept validation study for fuselage wake-filling propulsion integration. In: 31st Congress of the International Council of the Aeronautical Sciences (2018)
5. Welstead, J.R., Felder, J.L.: Conceptual design of a single-aisle turboelectric commercial transport with fuselage boundary layer ingestion. In: 54th AIAA aerospace sciences meeting (2016)
6. de La Rosa, E., Hall, C., Crichto, D.: Special session—towards a silent aircraft challenges in the silent aircraft engine design. In: 45th AIAA Aerospace Sciences Meeting and Exhibit (2002)
7. Armstrong, M.J., Ross, C.A., Blackwelder, M.J., Rajashekara, K.: Trade studies for NASA N3-X turboelectric distributed propulsion system electrical power system architecture. In: SAE International Journal of Aerospace, Vol. 5, No. 2012-01-2163, pp. 325–336 (2012)
8. Seitz, A., Habermann, A.L., van Sluis, M.: Optimality considerations for propulsive fuselage power savings. Part G J. *Aerosp. Eng.* (2020). <https://doi.org/10.1177/0954410020916319>
9. Hardin, L., Tillman, G., Sharma, O., Berton, J., Arend, D.: Aircraft system study of boundary layer ingesting propulsion. In: 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit. Reston, Virginia (2012)
10. Welstead, J., Felder, J., Guynn, M., Haller, B., Tong, M., Jones, S., Gray, J.S., Ordaz, L., Quinlan, J., Mason, B., Schnulo, S., Sil, A.: Overview of the NASA STARC-ABL (Rev. B) Advanced Concept (2018)
11. Giannakakis, P., Maldonado, Y.-B., Tantot, N., Frantz, C., Belleville, M.: Fuel burn evaluation of a turbo-electric propulsive fuselage aircraft. In: AIAA Propulsion and Energy 2019 Forum. Reston, Virginia (2019)
12. Rodriguez, D.L.: A multidisciplinary optimization method for designing boundary layer ingesting inlets. Dissertation, Stanford University (2001)
13. Gray, J.S., Mader, C.A., Kenway, G.K.W., Martins, J.R.R.A.: Modeling boundary layer ingestion using a coupled aeropropulsive analysis. *J. Aircr.* **55**(3), 1191–1199 (2017)
14. Gray, J.S., Martins, J.R.R.A.: Coupled aeropropulsive design optimisation of a boundary-layer ingestion propulsor. *Aeronaut. J.* **30**, 1–17 (2018)
15. Kenway, G.K., Kiris, C.C.: Aerodynamic shape optimization of the STARC-ABL concept for minimal inlet distortion. In: AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference (2018)
16. Yildirim, A., Gray, J.S., Mader, C.A., Martins, J.: Aeropropulsive design optimization of a boundary layer ingestion system. AIAA Aviation Forum, Reston, Virginia (2019)
17. Gray, J.S., Mader, C.A., Kenway, G.K.W., Martins, J.R.R.A.: Coupled aeropropulsive optimization of a three-dimensional boundary-layer ingestion propulsor considering inlet distortion. *J. Aircr.* **30**(12), 1–12 (2020)

18. Martínez Fernández, A., Smith, H.: Effect of a fuselage boundary layer ingesting propulsor on airframe forces and moments. *Aerosp. Sci. Technol.* **100**, (2020)
19. Ahuja, J., Mavris, D.N.: Sensitivity of boundary layer ingestion effects to tube and wing airframe design features. In: AIAA Scitech 2020 Forum. Reston, Virginia (2020)
20. Weller, H.G., Tabor, G., Jasak, H., Fureby, C.: A tensorial approach to computational continuum mechanics using object-oriented techniques. *Comput. Phys.* **12**(6), 620–631 (1998)
21. Petrusson, A.: Aerodynamic evaluation of nacelles for engines with ultra high bypass ratio. Master's thesis, Chalmers University of Technology (2017)
22. Habermann, A.L., Zahn, R., Seitz, A., Hornung, M.: Multidimensional parametric study of a propulsive fuselage concept using OpenFOAM. In: AIAA AVIATION Forum, virtual (2020a)
23. Roskam, J.: Airplane design part III: layout design of cockpit, fuselage, wing and empennage: cutaways and inboard profiles, 3rd ed. DAR Corporation, Lawrence, Kansas (2002)
24. Airbus S.A.S.: A330 Aircraft Characteristics Airport and Maintenance Planning (1993)
25. Airbus S.A.S.: A350-900/-1000 Aircraft Characteristics Airport and Maintenance Planning (2016)
26. Norwegian Air.: Norwegian Boeing 787-8/9 Dreamliner Maintenance Briefing. (2018). <https://www.norwegian.com/contentassets/833bf9fa7a654c1db6713032fae29da8/nor-2018-18---v02---09jul18---787-briefing.pdf>
27. HiSA.: 13/08/2020. <https://hisa.gitlab.io/>
28. Heyns, J.A., Oxtoby, O.F., Steenkamp, A.: Modelling high-speed flow using a matrix-free coupled solver. In: 9th OpenFOAM Workshop (2014)
29. Blazek, J.: Computational Fluid Dynamics: Principles and Applications. Elsevier, Amsterdam (2007)
30. Riegel, J., Werner, M., van Havre, Y.: Free CAD, 2001–2019. <http://www.freecadweb.org>
31. OpenFOAM: User Guide: snappyHexMesh, , 13/08/2020. <https://www.openfoam.com/documentation/guides/latest/doc/guide-meshing-snappyhexmesh.html>
32. Król, A., Król, M.: Study on numerical modeling of jet fans. *Tunn. Undergr. Sp. Technol.* **73**, 222–235 (2018)
33. Bijewitz, J., Seitz, A., Hornung, M.: Power plant pre-design exploration for a turbo-electric propulsive fuselage concept. In: 2018 Joint Propulsion Conference. Cincinnati, Ohio (2018)
34. Boyce, M.P.: Gas Turbine Engineering Handbook, 4th edn. Elsevier, Amsterdam (2012)
35. Kaiser, S., Grenon, R., Bijewitz, J., Prendinger, A., Atinault, O., Isikveren, A., Hornung, M.: Quasi-analytical aerodynamic methods for propulsive fuselage concepts. In: 29th Congress of the International Council of the Aeronautical Sciences (2014)
36. CFL3D Test/Validation archive, 13/08/2020. https://cfl3d.larc.nasa.gov/Cfl3dv6/cfl3dv6_testcases.html#onera
37. NASA Langley Research Center Turbulence Model Numerical Analysis, 13/08/2020. https://turbmodels.larc.nasa.gov/onera-wingnumerics_val.html
38. Diskin, B., Anderson, W.K., Pandya, M.J., Rumsey, C.L., Thomas, J., Liu, Y., Nishikawa, H.: Grid convergence for three dimensional benchmark turbulent flows. In: AIAA Aerospace Sciences Meeting, Reston, Virginia (2018)
39. Schmitt, V., Chapin, F.: Pressure distributions on the ONERA-M6-Wing at Transonic Mach Numbers. In: Experimental Data Base for Computer Program Assessment (1979)
40. Habermann, A.L., Bijewitz, J., Seitz, A., Hornung, M.: Performance bookkeeping for aircraft configurations with fuselage wake-filling propulsion integration. *CEAS Aeronaut. J.* **11**(2), 529–551 (2020b)
41. SAE International.: Gas turbine engine inlet flow distortion guidelines. In: Aerospace Recommended Practice, ARP1420, Rev. C. (2002)

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.