

CRANFIELD UNIVERSITY

Jesús Matesanz García

**Propulsion Integration Design and
Evaluation for Novel Aircraft
Configurations**

School of Aerospace, Transport and Manufacturing
Centre for Propulsion and Thermal Power Engineering

Ph.D. Thesis

Academic Year: 2022

Supervisor: Prof. D. G. MacManus

October 2022

© Cranfield University 2022. All rights reserved. No part of this publication may be reproduced without the written permission of the copyright owner.

Abstract

To comply with the future environmental requirements for the aviation industry, it is necessary to move towards more efficient aircraft and propulsive systems. Within this context, different novel aircraft concepts have been introduced to increase overall propulsive efficiency compared with the current technologies. A common characteristic between these concepts is the close integration of the propulsive system within the airframe. As a consequence, the impact of the propulsion integration on the aerodynamic performance of the aircraft is expected to increase in comparison with the conventional wing-mounted podded engines. However, with a few exceptions, for these new configurations the impact of the propulsion integration on the aerodynamic performance has not been sufficiently quantified. The aim of this research is to establish the methods for the aerodynamic design of the propulsion integration of the novel embedded propulsion systems. These methods are then used, for an example configuration, to quantify the impact of the propulsion integration in the overall aerodynamic performance and characteristics.

A systematic design methodology was developed for the aerodynamic analysis of embedded propulsion systems. This methodology includes the parametric definition of the geometry, the aerodynamic evaluation of the propulsor, and a tailored post-processing approach. An aft-mounted annular boundary layer ingestion propulsor for a medium-range single-aisle aircraft is used as a sample case study. A hierarchical approach with an increasing level of fidelity was applied to determine the modelling requirements for the embedded propulsion systems. This involved low order methods for drag prediction and computational fluid dynamics (CFD) methods. The CFD methods included two different fan approaches (one-dimensional and through-flow), as well as 2D axisymmetric and 3D models. To understand the limits of the design space, the design methodology was combined with a multi-objective optimisation (MOO) approach based on evolutionary algorithms.

In a preliminary analysis, power savings for the whole aircraft between 3-11% were predicted due to the integration of the aft-mounted propulsor. Compared with the CFD analyses, low order models for the prediction of the aerodynamic performance found in the open literature overpredicted the power savings in approximately a 50%, making them unsuitable for the evaluation of the aerodynamic performance in embedded systems. A comparison of the modelling fidelity of the different CFD approaches shown a reduction of approximately 2% of the power savings from the original 3-11%, when 2D axisymmetric models are applied instead of more representative 3D approaches. However, the 2D axisymmetric models had about 1% of

the computational cost of the 3D versions. The application of a more representative through-flow fan model also increased the predicted power savings by up to 1-1.5% when compared with a one dimensional fan.

The location of the aft-mounted propulsor was found to have a significant impact on the aerodynamic performance of the embedded propulsor and the predicted power savings. Relative to the overall benefits in the power consumption of $\sim 11\%$, variations of approximately 4-5% on the predicted power savings are observed with the change of the propulsor axial and radial location. Locations near to the fuselage centreline are preferred. Short aft-fuselage lengths with a low fan radius of the aft-mounted propulsor provided the highest thrust contribution and power savings. The more detailed design of the housing components (intake, nacelle and exhaust) of the aft-mounted propulsor has a second order impact in comparison with the propulsor location. At a fixed propulsor position, an increase of up to 1.5% of the power savings was obtained with the MOO of the aerodynamic design of the propulsion integration. From these changes, approximately one-third was obtained with the optimisation of the exhaust design, while the remaining benefits were obtained with the optimisation of the aft-fuselage, intake and nacelle geometries.

Keywords

Novel aircraft configurations, embedded propulsion, boundary layer ingestion, propulsion integration, multi-objective optimisation, computational fluid dynamics, aerodynamics

Acknowledgements

First of all, I would like to thank my supervisor Professor David MacManus for giving me this fantastic opportunity in the first place. This thesis would not have been possible without his guidance and patience during the last years. Also, I have to thank all the colleagues of the installations team. Your guidance and support have made this possible. In particular thanks to Fernando Tejero and Robert Christie for their help when it was needed most.

Secondly, I would like to thank my family without their unconditional support this would not have been possible. Thanks to my parents for always being there when I needed most. Also, thanks to my siblings Ana and Carlos for bringing me joy with a big dose of sarcasm and surrealism. Finally, thanks to my wonderful niece the little Maya, for giving me hope for the future.

Last, but not least, I must thank all the friends I made along my path. In particular, Pepo and Dimitris for these years living together sharing our joys, pains and support. Also, thanks to Anastasia for keeping me sane and stay on my side during the most difficult moments. I cannot forget about many of the other friends I made during these years. I have to enumerate some of them in no particular order: Alecos, Pavlos, Hanah, Fran, Nico, Priya, Avery, Marta, Tom, Luca, the twins Ioannis and Dimitris, María, Andrés, Ferran, Matteo, Elisa, Raul, Tala, Evangelia, Georgia, Marco, Dario, Lapo, Daniele, and Kawtar. I apologise if have forgotten anyone. All of you have made my life special in an unique way.

Last years have been tough and have required a big part of my energy. I cannot stop thanking all the people that shared my way and the people that have waited patiently for me to return to a normal life. Thanks for the Takomas (Caro, Viktortilla, Poguero, Buddy, Joel, and Violi) for keeping the music going. Thanks to my friends from Spain, for standing on my side even when we were not physically close. In particular to Miguel Vela, Sergio Haro, Pitu, Jul, Ellena, Make, Mig, Manu, Dani, Baskoh, Inés, Melonchito and Andrea.

After this, the only thing left to say is that on the good and in the bad times I know that I have been supported by all of you. Now life goes on, and the best is yet to come. I end this acknowledgements section sharing a poem from José Agustín Goytisolo. Life keeps pushing for us, we just have to live the remaining of it. This includes the sweet and the bitter moments. But, in any case, I know I have love and I have my friends for the way. You also have my unconditional love and support forever.

Palabras para Julia

Tú no puedes volver atrás
porque la vida ya te empuja
como un aullido interminable.

Hija mía es mejor vivir
con la alegría de los hombres
que llorar ante el muro ciego.

Te sentirás acorralada
te sentirás perdida o sola
tal vez querrás no haber nacido.

Yo sé muy bien que te dirán
que la vida no tiene objeto
que es un asunto desgraciado.

Entonces siempre acuérdate
de lo que un día yo escribí
pensando en ti como ahora pienso.

La vida es bella, ya verás
como a pesar de los pesares
tendrás amigos, tendrás amor.

Un hombre solo, una mujer
así tomados, de uno en uno
son como polvo, no son nada.

Pero yo cuando te hablo a ti
cuando te escribo estas palabras

pienso también en otra gente.

Tu destino está en los demás
tu futuro es tu propia vida
tu dignidad es la de todos.

Otros esperan que resistas
que les ayude tu alegría
tu canción entre sus canciones.

Entonces siempre acuérdate
de lo que un día yo escribí
pensando en ti
como ahora pienso.

Nunca te entregues ni te apartes
junto al camino, nunca digas
no puedo más y aquí me quedo.

La vida es bella, tú verás
como a pesar de los pesares
tendrás amor, tendrás amigos.

Por lo demás no hay elección
y este mundo tal como es
será todo tu patrimonio.

Perdóname no sé decirte
nada más pero tú comprende
que yo aún estoy en el camino.

Y siempre siempre acuérdate
de lo que un día yo escribí
pensando en ti como ahora pienso.

José Agustín Goytisolo (1992)

Contents

Abstract	v
Acknowledgements	viii
List of Contents	xv
List of Figures	xxix
List of Tables	xxxi
Nomenclature	xxxvii
1 Introduction	1
1.1 Background	1
1.2 Aims and objectives	3
1.3 Research roadmap	4
1.4 Thesis structure	6
1.5 Bibliographic contributions	7
2 Literature Review	9
2.1 Non-conventional aircraft configurations	10
2.1.1 Blended Wing Body concepts	13
2.1.2 Wing mounted distributed propulsion	15
2.1.3 Multiple aft-mounted propulsors	16
2.1.4 Aft-mounted annular propulsor	18
2.1.5 Challenges to the aero-propulsive design	19
2.2 Performance accounting for novel aircraft configurations	22
2.2.1 Momentum based methods	23
2.2.2 Energy based methods	26
2.3 Low order turbomachinery models	29
2.3.1 Non-throughflow approaches	30
2.3.2 Throughflow approaches	32
2.4 Propulsion integration aerodynamics	37
2.4.1 Propulsion integration for conventional podded aero-engines	37
2.4.2 Propulsion integration of novel aircraft configurations	41
2.5 Gap in knowledge	46
3 Methodology	47
3.1 Overview of the design strategy	47
3.2 Geometry design and generation	49

3.2.1	Intuitive Class Shape Transformations	50
3.2.2	Geometry parametrisation	51
3.2.2.1	2D aeroline parametrisation	52
3.2.2.2	Axisymmetric fuselage geometry	52
3.2.2.3	Aft-fuselage geometry	53
3.2.2.4	Intake geometry	54
3.2.2.5	Fan cowl geometry	56
3.2.2.6	Exhaust geometry	57
3.2.2.7	Turbomachinery domain	58
3.2.2.8	3D geometry parametrisation	59
3.2.2.9	Reference 3D airframe and position of the	61
3.3	Performance accounting	64
3.3.1	Modified Near-Field Method	64
3.3.2	Normalisation of the performance metrics	67
3.3.3	Intake performance evaluation	69
3.3.4	Exhaust performance evaluation	71
3.3.5	Correction of the force contribution with mass flow imbalances	72
3.3.6	Three-dimensional effects in the performance accounting	73
3.3.7	Low order performance accounting	74
3.3.7.1	Net Vehicle force from momentum deficit ingested	75
3.3.7.2	Net Vehicle Force from modified near-field method	77
3.4	Computational Fluid Dynamics methods	79
3.4.1	Boundary conditions and numerical solver	79
3.4.1.1	Boundary conditions for the 2D axisymmetric model	79
3.4.1.2	Boundary conditions for the 3D non-axisymmetric model	81
3.4.2	Turbomachinery models	82
3.4.2.1	One-dimensional fan model	82
3.4.2.2	Radial profiles at the stage exit	84
3.4.2.3	Hall-Thollet Model	86
3.4.2.4	Hall-Thollet model implementation	89
3.4.3	Mesh methodology	90
3.4.3.1	Structured mesh for the 2D axisymmetric configurations with semi-coupled one-dimensional propulsor	90
3.4.3.2	Structured mesh for the 2D axisymmetric configurations with coupled through-flow propulsor	91
3.4.3.3	Hybrid mesh for the 3D non-axisymmetric configuration with through-flow propulsor	93
3.5	Multi-objective optimisation	95
3.5.1	Framework	96
3.5.2	Convergence evaluation	100
3.5.3	Configuration of the optimisation algorithms	101
3.5.3.1	NSGA-II	101
3.5.3.2	IBEA	102

3.5.3.3	OMOPSO	102
4	Analysis of the propulsive requirements	103
4.1	Background	103
4.1.1	Power distributions for the aft mounted boundary layer ingestion propulsor	106
4.2	Cycle sizing methodology	107
4.3	Baseline wing-mounted power plant	111
4.4	Power savings with boundary layer	114
4.4.1	Independent power source	115
4.4.2	Power extracted from the wing mounted engines	120
4.5	Summary	123
5	Axisymmetric propulsion integration aerodynamics with semi-coupled turbomachinery	127
5.1	Case study	127
5.2	Optimisation of the aft-fuselage, intake and fan cowl geometries of an embedded propulsor	128
5.2.1	Definition of the optimisation problem	130
5.2.2	Multi-objective optimisation of the aerodynamic performance	132
5.2.2.1	Importance of the aerodynamic analysis	136
5.2.2.2	Aerodynamics of the selected designs	139
5.2.2.3	Influence of the design parameters on the aerodynamic performance	146
5.3	Effect of the system location	151
5.3.1	Definition of the study	151
5.3.2	Effect of the location of the propulsion system on the performance	154
5.3.3	Effect of the location of the propulsion system on the aerodynamics	160
5.3.4	Effect of the location of the propulsion system on the design parameters	167
5.4	Robustness assessment	173
5.4.1	Intake performance sensitivity to off-design conditions	174
5.4.2	Sensitivity to fan outlet conditions	180
5.5	Summary	185
6	Low order turbomachinery model for propulsion integration aerodynamics	187
6.1	NASA Rotor 4	188
6.2	Mixing Plane Model	188
6.3	Reference body force model implementation	189
6.4	Rotor 4 validation	191
6.4.1	Stage performance	191
6.4.2	Radial distributions	193

6.5	Conclusions	195
7	Axisymmetric propulsion integration aerodynamics with through-flow turbomachinery	197
7.1	Effect of the fan stage modelling in the aerodynamic performance	198
7.1.1	Effect of the fan model in the thrust contribution and propulsive efficiency	200
7.1.2	Effect of the fan model in the intake performance	205
7.1.3	Effect of the fan model in the exhaust performance	207
7.2	Optimisation of the exhaust of an embedded propulsor	209
7.2.1	Determination of the objective functions	209
7.2.2	Exhaust design optimisation	213
7.3	Optimisation of the integration of an embedded propulsion system	218
7.3.1	Impact of the design parameters in the aerodynamic performance	222
7.4	Summary	224
8	Non-axisymmetric propulsion integration	227
8.1	Design requirements	227
8.1.1	Aerodynamic constraints	229
8.1.2	Aircraft geometry constraints	233
8.2	Evaluation of an axisymmetric cowl design under 3D flow conditions	234
8.2.1	Comparison of the aerodynamic performance	237
8.2.2	Comparison of the exhaust performance	243
8.2.3	Comparison of the intake performance	245
8.2.4	Influence of the number of outlet guided vanes	248
8.3	Impact of the aft mounted propulsor location	251
8.3.1	Impact of the position on the aerodynamic performance	252
8.3.2	Impact of the position on the intake performance	258
8.3.3	Impact of the position on the exhaust performance	264
8.4	Impact of the propulsor dimensions on the performance	265
8.5	Non-axisymmetric cowl design	271
8.6	Summary	277
9	Conclusions and future work	281
9.1	Contributions to knowledge and conclusions	281
9.1.1	Methods for the design of embedded propulsion systems	282
9.1.2	Modelling requirements for embedded propulsion systems in novel aircraft configurations	284
9.1.3	Design of the propulsion integration of the aft-mounted boundary layer ingestion propulsor	286
9.2	Future work	288
A	Governing equations	305
A.1	Definition of the reference frame	305
A.1.1	2D axisymmetric reference frame	305

A.1.2	3D reference frame	306
A.2	Favre-averaged Navier-stokes equations	308
A.2.1	k- ω SST turbulence model	308
A.2.2	Wall functions in k- ω SST turbulence model	309
A.3	Equations for the properties of the air	310
B	Propulsive system sizing	313
B.1	Study of the boundary layer properties	313
B.2	Fan dimensions and capacity matching	318
B.3	Exhaust sizing under boundary layer effects	319
B.4	Fan stage scaling	320
B.5	Sizing of 2D axisymmetric configurations	321
B.6	Sizing of 3D configurations	323
B.7	Preliminary performance accounting based on low order models	324
B.8	Conclusions	334
C	Summary of uncertainty accounting	337
C.1	Uncertainty accounting methods	337
C.2	2D axisymmetric aircraft models	339
C.2.1	Reference axisymmetric fuselage	339
C.2.2	Decoupled propulsor 2D axisymmetric structured mesh	340
C.2.3	Coupled propulsor 2D axisymmetric structured mesh	341
C.3	3D aircraft models	342
C.3.1	DLR-F6 aircraft grid sensitivity study	342
C.3.2	Coupled propulsor 3D non-axisymmetric hybrid mesh	342
C.4	Turbomachinery models	343
D	Validation of the computational fluid dynamics methodology	345
D.1	Summary of previous validation cases	346
D.2	Nacelle aerodynamics validation	347
D.2.1	Validation cases	347
D.2.2	Reference metrics and accounting	347
D.2.3	2D axisymmetric validation	349
D.2.4	3D flow validation	352
D.3	DLR F6 validation	353
D.4	Conclusions	356
E	Additional considerations for the low order turbomachinery model	359
E.1	Geometry feature extraction	359
E.2	Hall-Thollet model calibration process	362
E.3	Radial distributions	363
F	Multi-objective optimization benchmark problems and algorithm selection	369
F.1	Reference functions	369

F.1.1	Zitzler-Deb-Thiele's function 1	370
F.1.2	Zitzler-Deb-Thiele's function 3	370
F.2	Selected algorithm configurations	371
F.3	Pareto consistency	372
F.4	Pareto convergence	374
F.5	Convergence evaluations	376
F.6	Application of the algorithms to a real case	381
F.7	Conclusions	383

List of Figures

1.1	Schematic of some of the main propulsion concepts for the novel aircraft configurations. Aft mounted embedded propulsion based on NASA STARC-ABL ¹ , MIT D8 ² and ONERA NOVA ³ . Wing-mounted embedded propulsion based on ONERA DRAGON ⁴ . Blended Wing Body with distributed embedded propulsion based on NASA N3-X ⁵	2
1.2	Scheme of the research roadmap including the most relevant stages for the evaluation of the impact of the propulsion integration on the aerodynamic performance for novel aircraft configurations	5
1.3	Map for the different contents of the thesis	6
2.1	Summary of the main novel aircraft concepts to date from an aerodynamic perspective. Comparison between the level of propulsion integration and the level of technology disruption of the overall airframe and propulsion system	11
2.2	Schematic of the changes on kinetic energy at the wake of the aircraft as defined by Rolt and Whurr ⁶	13
2.3	Summary of the main BWB configurations for different levels of propulsion integration. N2A and N2B from Kim et al. ⁷ , N3X adapted from the sample OpenVSP hangar configuration ⁸	14
2.4	Wing mounted distribution propulsion concept by ONERA: DRAGON (Distributed fans Research Aircraft with electric Generators by ONERA)from Schmollgruber et al ⁴	15
2.5	Multiple aft-mounted propulsors configurations. ONERA NOVA configuration from Wiart et al. ³ . MIT D8 schematic configuration from Uranga et al ⁹	16
2.6	Comparison between experimental and numerical results for the total pressure coefficient at the inlet reference plane for the MIT D8 cruise conditions. Adapted from Hall et al ¹⁰	17
2.7	Aft-mounted annular propulsor configurations. STARC ABL configuration from Gray et al ¹¹ . CENTERLINE concept from Seitz et al ¹²	18
2.8	Isentropic stagnation pressure relation contours around the STARC-ABL, based on Gray et al ¹³	19
2.9	Schematic of the governing distortion types for the different embedded propulsion systems	21

2.10	Summary of the different types of force book-keeping approaches found in the literature	23
2.11	Decomposition between the far field and the near field surfaces	24
2.12	Thrust and drag bookkeeping for installed turbofans with separate jet exhaust applying the modified near-field method ¹⁴ , adapted for Goulos et al ¹⁵	24
2.13	Evolution of the exergy balance for a blended wing body configuration with an embedded propulsor, as defined by Arntz and Atinault ¹⁶ . . .	29
2.14	Summary of the different categories of low order fan models found in the literature	30
2.15	Comparison of the lift-drag and Hall modified methods proposed by Thollet ¹⁷ with a mixing-plane URANS fan modelling approach. Evaluation of the swirl angle at the AIP for different sectoral distortion profiles for BLI configurations	35
2.16	Summary of the key design factors for the individual housing components of a conventional wing-mounted podded propulsor	38
3.1	Framework for ANDANTE (ANalysis and Design of Aircraft Novel Technologies), and coupling with the multi-objective optimization methodology	48
3.2	Available configurations on the present work based on the CFD methodology	49
3.3	Definition of the aero-engine installation housing components for an aft-mounted annular boundary layer ingestion propulsor	52
3.4	Axisymmetric fuselage geometry definition (fore-body in green, mid-body in red), and propulsion system location (black). Not to scale	53
3.5	Geometry parametrisation of the aft fuselage curve	54
3.6	Geometry parametrisation of the intake	55
3.7	Geometry parametrisation of the fan cowl	56
3.8	Hybrid intuitive Class-Shape transformation definition proposed for the fan cowls of embedded propulsion systems	57
3.9	Geometry parametrisation of the exhaust	58
3.10	Definition of the turbomachinery domain and the local iCST curve to match the housing components with the fixed stage geometry	59
3.11	Definition of the five azimuthal control lines of the propulsion integration design. Nomenclature defined from 0 to 180 degrees. Positive towards the starboard side of the aircraft and negative towards the port side. Left right symmetry on the plane containing $\Phi = 0.0^\circ$ and $\Phi = 180^\circ$	60
3.12	Sample azimuthal variation of a general parameter with a second order iCST definition. Control points in red	61
3.13	Comparison between the original DLR-F6 ¹⁸ and the DLR-F6 with the FX2B fairing ¹⁹	62

3.14	Definition of the propulsor location based on the 3D reference aircraft (DLR-F6 with FX2B ¹⁹), and definition of the geometry matching between the airframe and the aft-fuselage surface	63
3.15	Sketch of the intake scarf and engine inclination (Not to scale)	63
3.16	Modified Near-Field method domain and forces	65
3.17	Schematic of the reference thrust and the reference airframe net-vehicle force for a sample configuration (NASA Common Research Model ²⁰) for illustration purposes. Not to scale	68
3.18	Intake performance reference measure planes and rake distribution	69
3.19	View of the non-dimensional mass flow at the Fuselage station of the DLR-F6 ¹⁹	73
3.20	Schematic of the forces over the fuselage and the propulsor with the low order performance accounting based on the momentum deficit ingested	76
3.21	Schematic of the forces over the fuselage and the propulsor with the low order performance accounting based on the modified near-field method	78
3.22	Boundary conditions for the 2D axisymmetric model with one dimensional fan	80
3.23	Boundary conditions for the 2D axisymmetric model with throughflow turbomachinery model	80
3.24	Boundary conditions for the 3D model with throughflow turbomachinery model	81
3.25	Sketch of the stations of the flow on the aft mounted boundary layer propulsor	83
3.26	Workflow for the mass flow rate matching model	85
3.27	Verification of the mass flow matching condition at the fan	85
3.28	Example of the radial profiles distribution for the OGV outlet boundary	86
3.29	Definition of the body force components and the blockage factor	87
3.30	Hall-Thollet BFM workflow on the User Defined Function framework of ANSYS Fluent	89
3.31	Mesh domain and details for the 2D axisymmetric configuration with semi-coupled one dimensional propulsor	91
3.32	Mesh details for the 2D axisymmetric configuration with coupled throughflow low order turbomachinery model	92
3.33	Detailed view of the main regions of the 3D aircraft mesh	94
3.34	Methodology for the 3D hybrid mesh methodology	95
3.35	Sample of mirrored mesh control	96
3.36	Work flow for the multi-objective optimisation process for the evolutionary algorithms	97
3.37	Work flow for the multi-objective optimisation process for the particle swarm algorithm	98
3.38	Schematic example of a two-dimensional hypervolume, based on Deb ²¹ and Robinson ²²	101

4.1	Schematic of the aft mounted boundary layer powering possibilities: (A) Independent power source case, and (B) power extracted from the wing mounted propulsors	107
4.2	Workflow for the cycle matching and selection methodology	109
4.3	Sample operation maps for optimised cycle design at different technology levels (η_{fan} constant): (A) Optimal Specific Fuel Consumption in cruise conditions, (B) required Combustor Outlet Temperature in take-off conditions, and (C) required Compressor Discharge Temperature in take-off conditions	110
4.4	Schematic of the turbomachinery model for the wing podded propulsor in the configuration of reference (not to scale)	113
4.5	Selected baseline configuration based on the technology levels at take-off for the Case A (Tab 4.1)	114
4.6	Selected baseline configuration based on the technology levels at take-off for the Case B (Tab 4.1)	114
4.7	Power Saving Coefficient for the different TSPC and TS values on the Case A ($M_{cruise} = 0.78$) with an independent power source. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.415 and polytropic efficiency of 0.91	117
4.8	Power Saving Coefficient for the different TSPC and TS values on the Case B ($M_{cruise} = 0.75$) with an independent power source. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.45 and polytropic efficiency of 0.91	118
4.9	Change on the dimensions of the wing-mounted propulsor with the increase on TS , for the configurations with the independent power source for the BLI propulsor	119
4.10	Schematic of the turbomachinery model for the wing podded propulsor in the configuration with power extracted from the turbine (not to scale)	120
4.11	Power Saving Coefficient for the different TSPC and TS values on the Case A ($M_{cruise} = 0.78$) with the power extracted from the wing-mounted power-plants. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.415 and polytropic efficiency of 0.91	121
4.12	Power Saving Coefficient for the different TSPC and TS values on the Case A ($M_{cruise} = 0.75$) with the power extracted from the wing-mounted power-plants. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.45 and polytropic efficiency of 0.91	122
4.13	Changes on the wing-mounted power-plant requirements with the $TSPC$ and the TS for the Case A in cruise conditions ($M = 0.78$): (A) Bypass ratio, (B) Fan Pressure Ratio, and (C) Fan radius	124
5.1	Schematic of the differences between the one-dimensional fan and the through-flow fan model. Effects on the definition of the exhaust inlet conditions. Not to scale	129

5.2	Definition of the degrees of freedom for the optimisation aft-fuselage, intake and nacelle geometries. Axial and radial location of the propulsor fixed	130
5.3	Comparison of the candidate objective functions for the optimisation of the aft-fuselage, intake and fan cowl aerodynamic design ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	131
5.4	Non-dominated designs from the optimisation of the aft-fuselage, intake and fan cowl aerodynamic design ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	133
5.5	Mach number contours for the Datum and Preliminary designs used as reference ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	135
5.6	Comparison of the TS and $TSPC$ values predicted between the CFD and the low order performance prediction approaches for three configurations: Preliminary design, Datum design and Optimum TS values ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	137
5.7	Comparison of the PSC predicted between the CFD and the low order performance prediction approaches for three configurations: Preliminary design, Datum design and Optimum TS values ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	138
5.8	Mach number contours for the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	140
5.9	Pressure coefficient distributions over the fuselage for the the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	141
5.10	Pressure coefficient distributions over the intake (A) and the fan cowl (B) for the the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	143
5.11	Friction coefficient distributions over the fan cowl (A) and the aft-fuselage (B) for the the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)	144
5.12	Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$) coloured by the Thrust Split value. Datum design as black dashed line. Values normalised with the optimisation limits	146
5.13	Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$) coloured by the intake performance: IPR_{ing} (A), and RDI (B). Datum design as black dashed line. Values normalised with the optimisation limits	147
5.14	Box plots for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$) organised by the quartiles of each objective function (TS , IPR_{ing} and RDI)	149
5.15	Definition of the different axial and radial locations of the propulsor for the study of the propulsor location effects. Definition of the reference angles based on a sample aircraft, not to scale	152

5.16	Candidate objective functions correlation for the 8 locations of the Design Space Exploration (Fig. 5.15). The Pearson Index is evaluated on the top section (upper right) of the correlation matrix of each location. The Spearman index is evaluated on the bottom section (lower left) of the correlation matrix	154
5.17	Pareto surfaces for the eight cases of the location design space exploration	155
5.18	Pareto surfaces projected into pairs of objective functions for the eight cases of the location design space exploration	156
5.19	Propulsor radial and axial location effects evaluated from the best performing ingested pressure recovery non-dominated designs of each optimization	157
5.20	Propulsor radial and axial location effects evaluated from the best performing Thrust Split non-dominated designs of each optimization	158
5.21	Propulsor radial and axial location effects evaluated from the best performing Radial Distortion Index non-dominated designs of each optimization	159
5.22	Mach number contours for the best TS non dominated designs of the locations L4, L5, and L6	161
5.23	Mach number contours for the best IPR_{ing} non dominated designs of the locations L4, L5, and L6	162
5.24	Mach number contours for the best RDI non dominated designs of the locations L4, L5, and L6	163
5.25	Effect of the change on radial and axial position: Mach number contours for the best TS non dominated designs of the locations L5, L8, and L6	165
5.26	Effect of the change on radial and axial position: Mach number contours for the best IPR_{ing} non dominated designs of the locations L5, L8, and L6	166
5.27	Aft fuselage control parameters of the non-dominated designs for the 25% highest values of TS for each of the locations of the DSE (Fig. 5.15)	168
5.28	Intake control parameters of the non-dominated designs for the 25% highest values of TS for each of the locations of the DSE (Fig. 5.15)	169
5.29	Aft fuselage control parameters of the non-dominated designs for the 25% highest values of TS for each of the locations of the DSE (Fig. 5.15)	170
5.30	Aft fuselage control parameters of the non-dominated designs for the 25% highest values of IPR_{ing} for each of the locations of the DSE (Fig. 5.15)	171
5.31	Intake control parameters of the non-dominated designs for the 25% highest values of IPR_{ing} for each of the locations of the DSE (Fig. 5.15)	172
5.32	Aft fuselage control parameters of the non-dominated designs for the 25% lowest values of RDI for each of the locations of the DSE (Fig. 5.15)	173

5.33	Intake control parameters of the non-dominated designs for the 25% lowest values of RDI for each of the locations of the DSE (Fig. 5.15)	174
5.34	Schematic of the effects of the variation of the BLR in the intake and nacelle operation	175
5.35	Effect of the BLR on the fan face capacity for the reference axisymmetric fuselage in mid-cruise conditions	176
5.36	Maps of sensitivity of the IPR_{ing} to the operation conditions for the maximum TS and IPR_{ing} designs of the locations L6 and L8	177
5.37	Maps of sensitivity of the RDI to the operation conditions for the maximum TS and IPR_{ing} designs of the locations L6 and L8	178
5.38	Effect of the exhaust inlet conditions on the aerodynamic performance for the design with the maximum TS of the location L6	181
5.39	Effect of the fan outlet conditions for the design with the maximum TS of the location L6. Variation from the reference uniform conditions at the fan stage outlet for the C_V , C_D , and GPF	182
5.40	Effect of the fan outlet conditions for the design with the maximum TS of the location L6. Mach number contours at the exhaust for the baseline Uniform conditions case and three different radial distributions ($h = 0.3 - 0.5 - 0.7$) at a fixed load ($P_{t,max}/\bar{P}_t = 1.05$)	183
5.41	Effect of the fan outlet conditions for the design with the maximum TS of the location L6. Change on the power saving coefficient (PSC) for two different propulsor power sources	184
6.1	NASA Rotor 4 geometry ²³ with the relative locations of the measurement rakes	189
6.2	Sample view of the NASA Rotor 4 ²³ domain for the mixing plane CFD approach	190
6.3	Stage Pressure Ratio (PR) of the NASA-R4 at 50-70-87.5-100% max rpm operation line. Experimental data from Hughes et al. ²³	192
6.4	Stage isentropic efficiency (η_{is}) of the NASA-R4 at 50-70-87.5-100% max rpm operation line. Experimental data from Hughes et al. ²³	192
6.5	NASA R4 normal operation point Fan Exit Rake pressure ratio, isentropic efficiency, and swirl angle radial profiles: (a) 87.5% max rpm operation line, and (b) 100% max rpm operation line. Experimental results from Hughes et al. ²⁴	194
7.1	View of the sample geometry applied for the comparison between the through-flow fan model and the one-dimensional fan model ($l_{aft}/l_{ps} = 0.7$ and $r_{hub}/r_{fus} = 0.1$)	199
7.2	Effect of the one dimensional fan model in the prediction of the Thrust Split and the Thrust Specific Power Consumption for the Datum configuration	201
7.3	Effect of the one dimensional fan model on the prediction of the Gross Propulsive Force and the Ideal Propulsive Force. Variation of the metrics based on the values with the through-flow configuration	201

7.4	Mach number contours for the reference Datum design at mid-cruise conditions (Tab. 5.1). Comparison between the one-dimensional fan approach with two levels of fan efficiency and the through-flow low order fan model in 2D axisymmetric models	203
7.5	Mach number contours for the reference Datum design at mid-cruise conditions (Tab. 5.1). Detail around the fan cowl. Comparison between the one-dimensional fan approach with two levels of fan efficiency and the through-flow low order fan model in 2D axisymmetric models	204
7.6	Pressure coefficient distributions over the fuselage (A) and the intake (B) for the the Datum design with the through-flow and the one-dimensional fan models	205
7.7	Effect of the one dimensional fan model on the prediction of the intake performance	206
7.8	Pressure ratio and non-dimensional mass flow radial distributions at the AIP for the through-flow fan model and the one-dimensional fan model	207
7.9	Effect of the one dimensional fan model in the prediction of the exhaust performance. Variation of the metrics based on the values with the through-flow configuration	208
7.10	Definition of the 9 degrees of freedom for the design of the exhaust and the nacelle after-body	211
7.11	Comparison of the candidate objective functions for the optimisation of the exhaust aerodynamic design	212
7.12	Pareto front of the multi-objective optimisation of the exhaust and nacelle after-body design	214
7.13	Mach number over the selected designs of the multi-objective optimisation of the exhaust and the nacelle afterbody. Datum design included for reference	216
7.14	Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$) of the aerodynamic design of the exhaust. Coloured by TS (A), and C_d (B). Datum design as black dashed line. Values normalised with the optimisation limits	217
7.15	Pareto front of the multi-objective optimisation of the aft-fuselage, intake and nacelle with the downselected non-dominated exhaust design ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$). Comparison with the Datum design and the downselected non-dominated design (Ex-C)	219
7.16	Mach number contours for the reference designs (Datum and Opt. C) compared with the best performing non-dominated designs for each objective function of the aft-fuselage, intake and fan cowl aerodynamic optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$)	221
7.17	Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$) of the aerodynamic design of the aft-fuselage, intake and fan-cowl. Coloured by the Thrust Split value. Datum design as black dashed line. Values normalised with the optimisation limits	222

7.18	Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$) of the aerodynamic design of the aft-fuselage, intake and fan-cowl. Coloured by the intake performance: IPR_{ing} (A), and RDI (B). Datum design as black dashed line. Values normalised with the optimisation limits	223
8.1	Pressure coefficient contours for the scaled DLR-F6 with FX2B body fairing ¹⁹ at cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$, $Re = 2.6 \times 10^7$) and nominal angle of attack ($C_L = 0.5$)	228
8.2	Side view of the DLR-F6 with FX2B body fairing and definition of the sample relative stations	230
8.3	Momentum deficit distributions at the reference sample stations for the DLR-F6 at cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$, $Re = 2.6 \times 10^7$) and nominal angle of attack ($C_L = 0.5$)	231
8.4	Distribution of mass flow per unit of area at the reference sample stations for the DLR-F6 at cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$, $Re = 2.6 \times 10^7$) and nominal angle of attack ($C_L = 0.5$)	232
8.5	Schematic of the length and ground clearance constraints for the propulsion integration in the DLR-F6 ¹⁹ airframe configuration	234
8.6	View of the baseline axisymmetric cowl positioned on the central axis of the DLR-F6 airframe	236
8.7	Effect of the model on the TS and $TSPC$ prediction ($M_\infty = 0.75$, $h = 32000\text{ft}$)	238
8.8	Pressure coefficient over the aft fuselage with detail of the intake duct region ($M_\infty = 0.75$, $h = 32000\text{ft}$). Surfaces unwrapped from the engine central axis. Axial coordinate normalised with the total length	239
8.9	Comparison of the modified gross propulsive force GPF^* and the ideal propulsive force IPF for the different fan models and dimensions ($M_\infty = 0.75$, $h = 32000\text{ft}$). Values referenced to the 3D configuration with through-flow fan model	240
8.10	Mach number distributions over the 2D axisymmetric aft mounted annular propulsor design ($M_\infty = 0.75$, $h = 32000\text{ft}$). Upper figure: View of the flow at the aft-fuselage and power plant (through-flow fan on top, one-dimensional fan on bottom). Lower figure: View of the flow over the power plant (through-flow fan on top, one-dimensional fan on bottom)	241
8.11	Mach number distributions over the three-dimensional aft mounted annular propulsor with axisymmetric cowl design ($M_\infty = 0.75$, $h = 32000\text{ft}$) and angle of attack 0°	242
8.12	Effect of the dimensionality and fan approach in the prediction of the exhaust performance ($M_\infty = 0.75$, $h = 32000\text{ft}$). Values referenced to the 3D through-flow configuration	245
8.13	Effect of the model on the intake performance prediction ($M_\infty = 0.75$, $h = 32000\text{ft}$) at nominal angle of attack ($C_L = 0.5$)	246

8.14	Non-dimensional mass flow and pressure ratio distributions at the AIP for the 2D axisymmetric case and the 3D flow with axisymmetric aft mounted propulsor ($M_\infty = 0.75$, $h = 32000\text{ft}$)	248
8.15	Effect of the number of OGVs on the aerodynamic performance for an axisymmetric intake and cowl in cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$) and nominal angle of attack ($C_L = 0.5$)	250
8.16	Effect of the number of OGVs on the intake performance for an axisymmetric intake and cowl in cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$) and nominal angle of attack ($C_L = 0.5$)	251
8.17	Definition of the different locations tested for the aft-mounted propulsor on the DLR-F6 aircraft	253
8.18	Effect of the propulsor position on the aerodynamic performance. Evaluation of the TS (left) and $TSPC$ (right) for an axisymmetric cowl design	254
8.19	Changes on the momentum flux deficit with the change on vertical position of the propulsor. Evaluated in a plane tangent to the highlight, and in the symmetry plane of the airframe	255
8.20	Changes on pressure coefficient over the fan cowl and aft fuselage surfaces with the change on the vertical position of the propulsor. Surfaces unwrapped from the central axis of the engine. Axial coordinate normalised with the total length of the surface	257
8.21	Changes on momentum flux deficit at the highlight plane with the change on the propulsor axial position	258
8.22	Effects of the axial position of the aft-mounted propulsor in the momentum flux deficit (symmetry plane of the airframe), and force contribution of the aft-fuselage surface (positive on the thrust direction)	259
8.23	Effect of the propulsor position on the exhaust performance. Evaluation of the IPR , DC_{60} , RDI , and CDI for an axisymmetric intake design	260
8.24	Variation of the total pressure recovery from the freestream value with the change on vertical position for a fixed $l_{aft}/l_{ps} = 0.8$. Sampled at the fan AIP	261
8.25	Variation of the total pressure recovery from the freestream value with the change on axial position for a fixed $r/r_{fus} = 0.75$. Sampled at the fan AIP	262
8.26	Change on the relative incidence of the flow with the aft fuselage geometry. Intake separation in location L1	263
8.27	View of the flow separation at the hub for the location L7. Contour of axial wall shear stress. Oil flow lines (clear color) to show the wall shear stress gradient on the surfaces	263
8.28	Effect of the propulsor position on the exhaust performance. Evaluation of the C_V^* and C_d for an axisymmetric cowl and exhaust design	264
8.29	Definition of the geometry parameters applied for the design space exploration of the effects of the nacelle and exhaust design of the 3D models	266

8.30	Changes on TS , $TSPC$ and C_d for the Design Space Exploration of the bulk geometry parameters of the nacelle and the exhaust. Baseline design for reference	267
8.31	Histogram for the degrees of freedom of the DSE. Parameters normalised with the optimisation limits. Colour division based on the TS values. Thrust split range split into three equal intervals	268
8.32	Projection of the pressure force and viscous force contribution over the fan cowl for the baseline design and the design with maximum TS of the Design Space Exploration. Length normalised with l_{nac}	269
8.33	Mach number contours for the for the baseline design and the design with maximum TS of the Design Space Exploration. Top and side views	270
8.34	Definition of the degrees of freedom and reference control lines for the non-axisymmetric design of the housing components of the embedded propulsor. In red parameters set constant azimuthally (axisymmetric). In blue parameters applied to each control line. Control lines defined with left-right symmetry by the plane containing $\Phi = 0.0^\circ$ and $\Phi = 180^\circ$	272
8.35	Momentum flux deficit ingested at the highlight plane (top) in the Baseline configuration ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$). Side view of the momentum flux deficit generated by the cowl and the losses and the non-ingested wake	273
8.36	Comparison between the Baseline (black) and the bespoke (red) design in the 5 control-lines ($\Phi = 0.0^\circ - 45^\circ - 90^\circ - 135^\circ - 180^\circ$), and front views of the intake (purple) and fan cowl (khaki)	275
8.37	Increase of the aerodynamic performance metrics from the Baseline design. Comparison between the Bespoke design and the design with the maximum TS of the DSE	276
8.38	Mach number contours over the Baseline and the Bespoke design. Side and top views of the flow in the engine symmetry axis	277
8.39	Intake pressure ratio (based on freestream total pressure) at the AIP for the Baseline and Bespoke configurations	278
A.1	Basic definition of the cylindrical coordinates system applied in the 2D axisymmetric models	306
A.2	Schematic of the origin of the 2D axisymmetric reference frame	306
A.3	Schematic of the 3D reference frames for the aircraft and the fan	307
B.1	Intake performance reference measure planes and rake distribution	315
B.2	Schematic of the velocity, displacement and momentum distributions in an incompressible boundary layer	316
B.3	Schematic of the momentum flux deficit potential integration method over a section of the fuselage	317
B.4	Schematic of the main stations of the flow on the aft mounted boundary layer propulsor	318

B.5	Momentum flux deficit variation with the boundary layer ingestion rate for the reference axisymmetric fuselage in mid-cruise conditions	322
B.6	Effect of the <i>BLR</i> on the fan dimensions for the 2D axisymmetric cases: (A) Fan face area variation with the boundary layer ingestion rate for the reference axisymmetric fuselage in mid-cruise conditions, and (B) fan face capacity variation with the boundary layer ingestion rate for the reference axisymmetric fuselage in mid-cruise conditions	322
B.7	Momentum flux deficit variation vs fan mass flow and radial requirements for a target Mach number of 0.5 at the fan face. Sampled from the DLR-F6 fuselage ²⁵ at cruise conditions. Values at reference stage at 60% of the fuselage length	324
B.8	Momentum flux deficit propagation at different lengths for a sample 3D aft-mounted boundary layer ingestion propulsor	325
B.9	Evolution of the thrust split with the boundary layer ingestion rate for different fan pressure ratios using the momentum deficit ingested approaches. Comparison between the adapted approach and the approach proposed by Seitz et al ²⁶	326
B.10	Evolution of the thrust split with the boundary layer ingestion rate for different fan pressure ratios using the modified near-field method approach	327
B.11	Evolution of the Thrust Specific Power Consumption with the boundary layer ingestion rate for different fan pressure ratios using: the adapted momentum deficit ingested approach (A), and the modified near-field method approach (B)	329
B.12	Comparison of the predicted values for the Thrust Split (A) and the Thrust specific Power consumption (B) for two propulsion integration geometries: a baseline design based on the preliminary design geometry (Tab. B.1), and a manually designed compact cowl	332
B.13	Mach number contours of the Preliminary and Modified propulsion integration designs	333
D.1	Definition of the experimental rake by Langley (1979) ²⁷	348
D.2	Definition of the force components on the validation process ²⁸	349
D.3	Drag coefficient of nacelle and sting section as a function of freestream Mach number and MFCR for the ARA cowl 1. Experimental results from Langley ²⁷ . Numerical reference from Robinson et al. ²⁸	351
D.4	Effect of incidence on drag rise for the PVD geometry, normalised against drag at $M = 0.8$ at $MFCR = 0.7$. Experimental results obtained from Robinson et al ²⁸ . Numerical reference from Robinson et al. ²⁸	353
D.5	Comparison between the original DLR-F6 ¹⁸ and the DLR-F6 with the FX2B faring ¹⁹	354
D.6	Sample of C_p and y^+ values for the angle of attack 0.0 case with the DLR-F6 with FX2B body fairing for the 3rd Drag Prediction Workshop ¹⁹	355

D.7	Validation of the DLR-F6 polar and based on the experimental results of ONERA wind tunnel for the Second Drag Prediction Workshop ¹⁸ . The analysis includes the structured mesh validation performed by Milea ²⁹ for comparison. The DLR-F6 with FX2B body fairing for the 3rd Drag Prediction Workshop ¹⁹ are also included for additional context	356
E.1	Blade geometry characteristics to be input on the BFM Fluent implementation. Length elements normalized with the reference blade tip radius	360
E.2	Outlet Guide Vane geometry characteristics to be input on the BFM Fluent implementation. Length elements normalized with the reference blade tip radius	361
E.3	Calibration process schematic, effects on the FPR and η_{is} . Experimental data from Hughes et al. ²³	362
E.4	Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 50% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit	364
E.5	: Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 70% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit	365
E.6	Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 87.5% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit	366
E.7	Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 100% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit	367
F.1	Global Pareto-optimal set defined for the Zitzler-Deb-Thiele's function 1 ($g(\mathbf{x}) = 1$)	370
F.2	Global Pareto-optimal set defined for the Zitzler-Deb-Thiele's function 3 ($g(\mathbf{x}) = 1$)	371
F.3	ZDT function 1 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 65 individuals	373
F.4	ZDT function 1 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 365 individuals	374
F.5	ZDT function 3 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 65 individuals	375
F.6	ZDT function 3 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 365 individuals	376
F.7	Convergence of the selected algorithms for one test with the DTZ function 1 ³⁰ . Evaluated for a seed size of 65 individuals (A), and a seed size of 365 individuals (B)	377

F.8	Convergence of the selected algorithms for one test with the DTZ function 3 ³⁰ . Evaluated for a seed size of 65 individuals (A), and a seed size of 365 individuals (B)	378
F.9	Distribution of the Hypervolume index at different generations for the 40 tests with the DTZ function 1	379
F.10	Distribution of the Hypervolume index at different generations for the 40 tests with the DTZ function 3	381
F.11	Comparison of the Hypervolume Index convergence rate for the different multi-objective optimisation algorithms	383
F.12	Comparison of the Pareto fronts for the different multi-objective optimisation algorithms	384

List of Tables

4.1	Operation condition for the engine performance optimisation	112
4.2	Assumed isentropic efficiencies for the turbine and compressor stages of the wing-mounted three-spool turbofan configuration	112
4.3	Optimum operating conditions for the baseline wing-mounted engine at cruise	113
5.1	Sample mid-cruise operating conditions.	128
5.2	Main dimensions of the cowl for the reference designs and the down- selected non-dominated designs of the multi-objective optimisation . .	135
6.1	NASA Rotor 4 design characteristics ²³	188
7.1	Summary of the test configurations for the comparison between the 2D axisymmetric approaches	198
8.1	Summary of the test configurations for the comparison between the 2D axisymmetric and the 3D modelling approaches	236
8.2	Summary of the geometry changes from the Baseline design for the design with the highest TS of the design space exploration	271
8.3	Summary of the geometry changes from the Baseline in the Bespoke design. Summary of the parameters modified axisymmetrically	274
8.4	Summary of the geometry changes from the Baseline in the Bespoke design. Summary of the parameters modified in each control-line . . .	274
9.1	Summary of the changes on the predicted performance with the mod- els applied. Changes on performance based on $TS \sim 0.22$ and $PSC \sim$ 13% . Exchange rate equal to 1% of the PSC value per 0.01 units of TS	286
B.1	Cowl dimensions for the low order performance evaluation	326
C.1	Grid sensitivity study for the 2D axisymmetric reference cylindrical fuselage	340
C.2	Grid sensitivity study for the 2D axisymmetric decoupled boundary layer ingestion approach	340
C.3	Grid sensitivity study for the 2D axisymmetric coupled boundary layer ingestion propulsor approach	341

C.4	Grid sensitivity study for the DL-F6 configuration scaled to reference length under cruise conditions	342
C.5	Grid sensitivity study for the 3D configuration with the through-flow fan model	343
C.6	Grid sensitivity study for the 2D axisymmetric BFM approach	344
C.7	Grid sensitivity study for the 3D sector BFM approach	344
C.8	Grid sensitivity study for the Mixing plane approach	344
D.1	Key geometry parameters of the nacelles tested in the validation . . .	347
D.2	Test conditions for the ARA cowl 1 configuration	350
D.3	Evaluation of the net performance values at selected design points for ARA cowl 1 ²⁷	350
D.4	Test conditions for the PVD configuration	352
D.5	Evaluation of the net performance values at selected design points for PVD ²⁸	352
F.1	Algorithm selected for the multi-objective methodology verification . .	372
F.2	Test array for the multi-objective optimization algorithm performance test	372

Nomenclature

Abbreviations

AIP	Aerodynamic interference plane	
ANDANTE	Analysis and Design of Aircraft Novel Technologies	
AR	Aspect ratio of the intake throat	
AR_{int}	Intake aspect ratio (L/D_{fan})	
BFM	Body Force Model	
BLI	Boundary Layer Ingestion	
BP	Bernstein Polynomial	
BPR	By-pass ratio	[-]
CDT	Compressor Discharge Temperature	[K]
COT	Combustor Outlet Temperature	[K]
CR	Contraction ratio of the intake	
CST	Class-Shape Transformation	
DSE	Design Space Exploration	
FPR	Fan Pressure Ratio	
HiCST	Hybrid Intuitive Class-Shape Transformation	
HPC	High Pressure Compressor	[-]
HPT	High Pressure Turbine	[-]
httr	Hub-to-tip ratio	

HV	Specific Calorific value	[J/(kg)]
IBEA	Indicator based Evolutionary Algorithm	
iCST	Intuitive Class-Shape Transformation	
IPC	Intermediate Pressure Compressor	[-]
IPT	Intermediate Pressure Turbine	[-]
LHS	Latin Hypercube Sampling	
LPC	Low Pressure Compressor	[-]
LPT	Low Pressure Turbine	[-]
MFD	Momentum Flux Deficit Method	
MNFM	Modified Near Field Method	
MOO	Multi Objective Optimisation	
MOO	Multi-Objective Optimization	
NSGA-II	Non-dominated Sorted Genetic Algorithm II	
OGV	Outlet Guided Vanes	
OMOPSO	Multi-Objective Particle Swarm Algorithm	
OPR	Overall Pressure Ratio	[-]
R4	NASA Rotor 4 configuration	
SFC	Specific Fuel Consumption	[mg/(kN s)]
UDF	User Defined Function	

Greek Symbols

α	Swirl angle	[rad]
Δ	Relative increment between two values	
δ	Local deviation angle	[rad]
$\delta^{\eta_{cal}}$	Reference calibration local deviation angle	[rad]
δ_{prop}	Propulsor angle	[rad]

η_{is}	Total-to-total isentropic efficiency	[-]
η_{poly}	Total-to-total polytropic efficiency	[-]
η_{prop}	Propulsive efficiency	[-]
λ	Fan nozzle pressure ratio	[-]
λ_{eff}	Effective fan nozzle pressure ratio	[-]
μ	Dynamic viscosity	[Pa·s]
Ω	Fan rotational speed	[rad/s]
Ω_{int}	Intake inclination	[rad]
$\overline{\tau}$	Shear tensor	[N/m ²]
Φ	Azimuthal control coordinate	[deg]
ϕ	Drag force component	[N]
ρ	Density	[kg/m ³]
θ	Azimuthal coordinate	[rad]
θ_i	Wall forces	[N]
φ	Non-dimensional axial coordinate	[-]
ξ	Non-dimensional vertical coordinate	[-]
ζ	Fineness ratio	[-]

Roman Symbols

$\dot{m}$	Mass flow rate	[kg/s]
$\mathbf{e}_{(x,y,z),(x,r,\theta)}$	Coordinate versor in Cartesian or Cylindrical coordinates	[-]
$\mathbf{f}$	Body force vector	[N/kg]
$\mathbf{n}$	Normal vector	[-]
$\mathbf{V}$	Velocity vector	[m/s]
$\mathbf{W}$	Relative velocity vector	[m/s]
b	Blockage factor	[-]

BLR	Boundary Layer Ingestion Rate	[-]
C	Class Function	[-]
c	iCST chord	[m]
C_f	Friction coefficient	[-]
C_L	Lift Coefficient	[-]
$C_{D,0}$	Drag coefficient at zero lift	[-]
C_D	Drag coefficient	[-]
C_d	Discharge coefficient	[-]
C_f	Friction coefficient at zero lift	[-]
C_p	Pressure coefficient at zero lift	[-]
C_V	Velocity coefficient	[-]
C_V^*	Modified velocity coefficient	[-]
CDI	Circumferential Distortion Index	[-]
D	Drag Force	[N]
D^*	Modified Drag Force	[N]
$DC_{(60)}$	Sectional distortion coefficient of a 60 degrees sector	[-]
e	Specific internal energy	[J/kg]
e_t	Specific total internal energy	[J/s]
F_i	Gauge forces	[N]
f_n	Normal force component	[N/kg]
f_p	Parallel force component	[N/kg]
f_{if}	Non-dimensional curvature radius of the nacelle	[-]
F_{RN}	Relative Net Thrust Force	[N]
FF	Form factor	[-]
GCI	Grid Convergence Index	[-]

GPF	Gross Propulsive Force	[N]
GPF^*	Modified Gross Propulsive Force	[N]
h_t	Specific total enthalpy	[J/kg]
$HV(Y')$	Hypervolume index	[-]
IPF	Ideal Propulsive Force	[N]
IPR	Intake Pressure Recovery	[-]
IPR_{ing}	Ingested Intake Pressure Recovery	[-]
$K_{i,n}$	Bernstein Polynomial coefficients	[-]
K_{Mach}	Compressibility correction factor	[-]
M	Mach number	[-]
M_r	Relative Mach number	[-]
$MFCR$	Mass Flow Capture Ratio	[-]
N_1, N_2	Class Shape control coefficients	[m]
N_{blades}	Number of blades	[-]
N_{seed}	Number of individuals in the seed	[-]
NVF	Net Vehicle Force	[N]
NVF_{ref-af}	Net Vehicle Force of the reference airframe	[N]
p	Static Pressure	[Pa]
P_t	Total Pressure	[Pa]
PSC	Power Saving Coefficient	[-]
Q	Capacity	[-]
q	Dynamic Pressure	[Pa]
R	Universal Gas Constant for the air as an ideal gas	[J/kg K]
r	Radial coordinate	[m]
RDI	Radial Distortion Index	[-]

Re	Reynolds number	[-]
s	Blade solidity	[-]
S_{ref}	Reference area	[m ²]
S_{wet}	Wetted area	[m ²]
T	Static Temperature	[K]
T	Thrust force	[N]
t	time	[s]
T^*	Modified Thrust Force	[N]
T_t	Total Temperature	[K]
TS	Thrust Split	[-]
$TSPC$	Thrust Specific Power Consumption	[m/s]
V	Velocity	[m/s]
W	Relative velocity vector	[m/s]
W_{prop}	Required power of the aft-mounted propulsor	[W]
x	Axial coordinate	[m]
y	Vertical coordinate	[m]

Subscripts

∞	Relative to the freestream
θ	Azimuthal component
BLI	Relative to boundary layer ingestion
$cowl$	Relative to the fan cowl
ex	Relative to the exhaust
fan	Fan station metric
fus	Relative to the fuselage
hi	Relative to the highlight

<i>ing</i>	Ingested amount
<i>max</i>	Maximum operation reference
<i>ps</i>	Blade pressure side
<i>r</i>	Radial component
<i>ref</i>	Reference value
<i>ss</i>	Blade suction side
<i>stage</i>	OGV station metric
<i>x</i>	Axial component

CHAPTER 1

Introduction

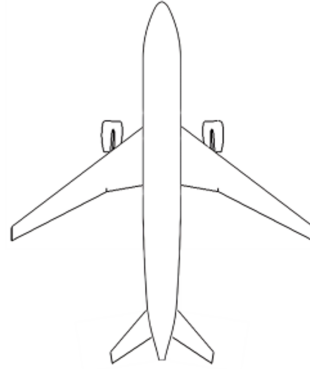
1.1 Background

National governments and international agencies have committed to reduce the net aviation CO₂ emissions by 75% for the year 2050 based on 2000 levels^{31,32}. One approach to achieve the aviation emission goals is to reduce fuel burn. This can be partially achieved by an increase of the propulsive efficiency, the application of aerodynamic improvements to reduce the overall aircraft drag, or the reduction of the total weight of the system. Within this context, different novel aircraft propulsion configurations have been proposed to obtain more efficient systems^{1,3-5,33,34}.

The most common feature in the new proposed aircraft concepts to date is the closer integration of the propulsive system within the airframe (Fig. 1.1). It is expected that the closer integration of the propulsive system will increase the aerodynamic coupling between the airframe and the propulsor. A consequence of the closer propulsion-airframe coupling is the increase of the impact of the propulsion integration design on the aerodynamic performance of the aircraft, compared with the podded engine configurations.

In the preliminary design stage, the housing components of the propulsive system (intake, nacelle and exhaust) have been traditionally designed and evaluated individually³⁵⁻³⁸. However, the increase on the aerodynamic coupling requires a combined analysis of the propulsion integration design on the aerodynamic performance. The closer integration of the propulsive system is expected to increase the distortion levels at the intake³⁹⁻⁴¹. This increase on the distortion can propagate through the turbomachinery stage and affect the propulsive system performance^{42,43}. Hence, the intake system design will condition the turbomachinery operation and the exhaust

Conventional tube-and-wing podded propulsion



Propulsion concepts for novel aircraft configurations

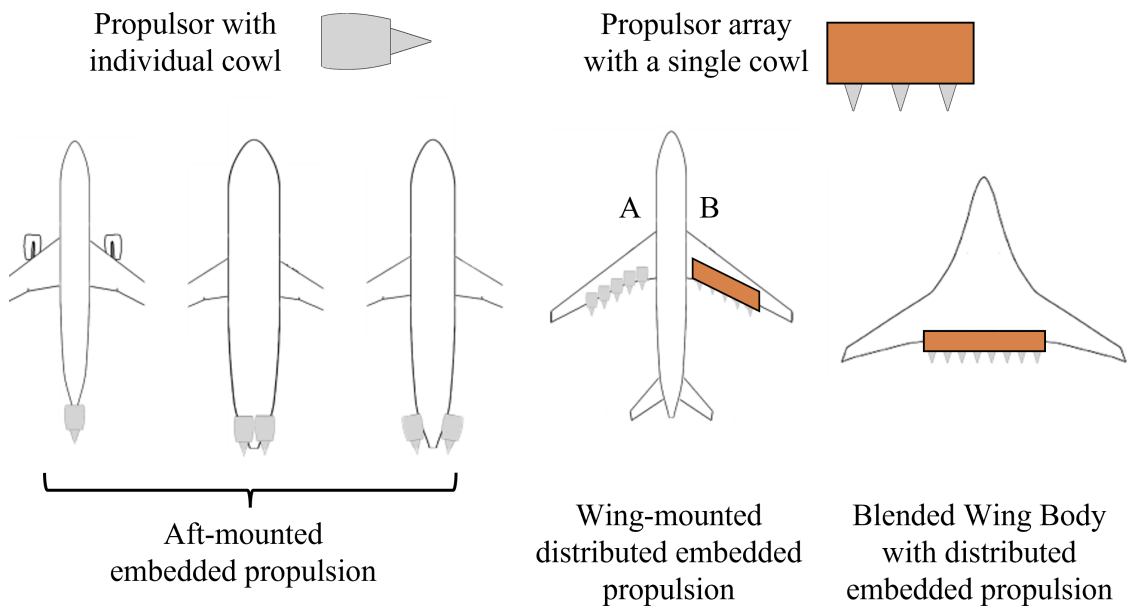


FIGURE 1.1: Schematic of some of the main propulsion concepts for the novel aircraft configurations. Aft mounted embedded propulsion based on NASA STARC-ABL¹, MIT D8² and ONERA NOVA³. Wing-mounted embedded propulsion based on ONERA DRAGON⁴. Blended Wing Body with distributed embedded propulsion based on NASA N3-X⁵

design requirements. Also, the fan cowl design is expected to have a significant role in the overall propulsive performance of the aircraft. Thus, a holistic analysis of the propulsion integration aerodynamics is required to determine the impact of the novel propulsive systems on the aerodynamic performance.

Some important work has been carried on the design of the propulsion integration

of novel embedded propulsion configurations. This work has evolved from preliminary design with decoupled aerodynamics^{1,44} to a detailed aerodynamic analysis of the embedded configuration in more advanced design stages for specific configurations^{13,41}. However, the increase in the complexity of the aerodynamic design may require a detailed aerodynamic design from the early design stages. Hence, for the preliminary design stage, the impact of the design of the individual housing components on the aerodynamic performance needs further attention. Thus, to enable the future design of such novel embedded systems, an evaluation of the current modelling and design capabilities is required.

1.2 Aims and objectives

The scientific aim of this research is to establish the methods for the aerodynamic design of the propulsion integration of novel embedded propulsive configurations. This work provides a foundation for the preliminary design of these configurations and captures the main model requirements to enable future work. The main research objectives to achieve the aim of the project are:

1. Identify the key design aspects of the propulsion integration design for the novel configurations and develop an appropriate parametrisation methodology for the new geometries
2. Develop a numerical framework to enable the evaluation of the aerodynamic performance of the propulsion integration for novel aircraft configurations
3. Define a robust methodology to evaluate the aerodynamic performance of the aero-engine installation in novel aircraft configurations. This includes system performance, contribution to the vehicle's propulsion, and individual housing components (intake, nacelle, exhaust) performance
4. Develop an adapted multi-objective optimization methodology for the individual installation elements and the full installation. Integrate the optimization methodology with the design numerical framework
5. Select and develop a simplified turbomachinery model, and integrate it with the existing numerical methodology
6. Apply the combined methodology to evaluate the impact of the propulsion integration design on the performance of the novel configurations

7. Study the impact of the aerodynamic modelling in the evaluation of the aerodynamic performance of novel aircraft configurations

1.3 Research roadmap

The research strategy can be divided into different groups for each of the individual objectives of the project (Fig. 1.2). The general methodological approach is initially defined to obtain the necessary information to evaluate the impact of the propulsion integration design on the aerodynamic performance. This methodology is then updated for different levels of complexity based on the feedback obtained from each of the data campaigns. The process can be divided into three levels: the geometry design requirements, the aerodynamic model requirements, and the data extraction and analysis.

First, a geometry generation strategy is developed to enable the definition of the propulsion integration design with an accessible parametric methodology (Chapter. 3.2). A Hybrid Intuitive Class-Shape Transformations (HiCST)³⁸ approach is used to define the 2D axisymmetric and the 3D architectures. The 2D parametric definition of the different housing components of the aero-engine installation is necessary to extend the definition to the 3D non-axisymmetric geometries.

Second, an aerodynamic modelling work-flow was applied using a compressible viscous solver (Chapter 3.4). Initially, a semi-coupled approach with a simplified one dimensional propulsor model is used to characterize the aerodynamic characteristics of the propulsion integration. The models are later upgraded to a fully coupled approach with the definition of throughflow turbomachinery model (Chapter 3.4.2.3).

In parallel with the aerodynamic modelling, a performance accounting strategy is implemented (Chapter 3.3). The accounting methodology is based on well established methods^{14,45} and adapted to the new requirements of the novel aircraft configuration and embedded propulsive systems. The accounting methodology is developed to include the housing components, the propulsor and the whole system performance. The trade between the different figures of merit is evaluated on different chapters of the thesis (Chapters 5 and 7).

Finally, the developed strategies and capabilities are used for data extraction campaigns (Chapters 5,7, and 8). The data extraction is based on Design Space Explorations (DSE) and Multi-Objective Optimizations (MOO) (Chapter 3.5). The

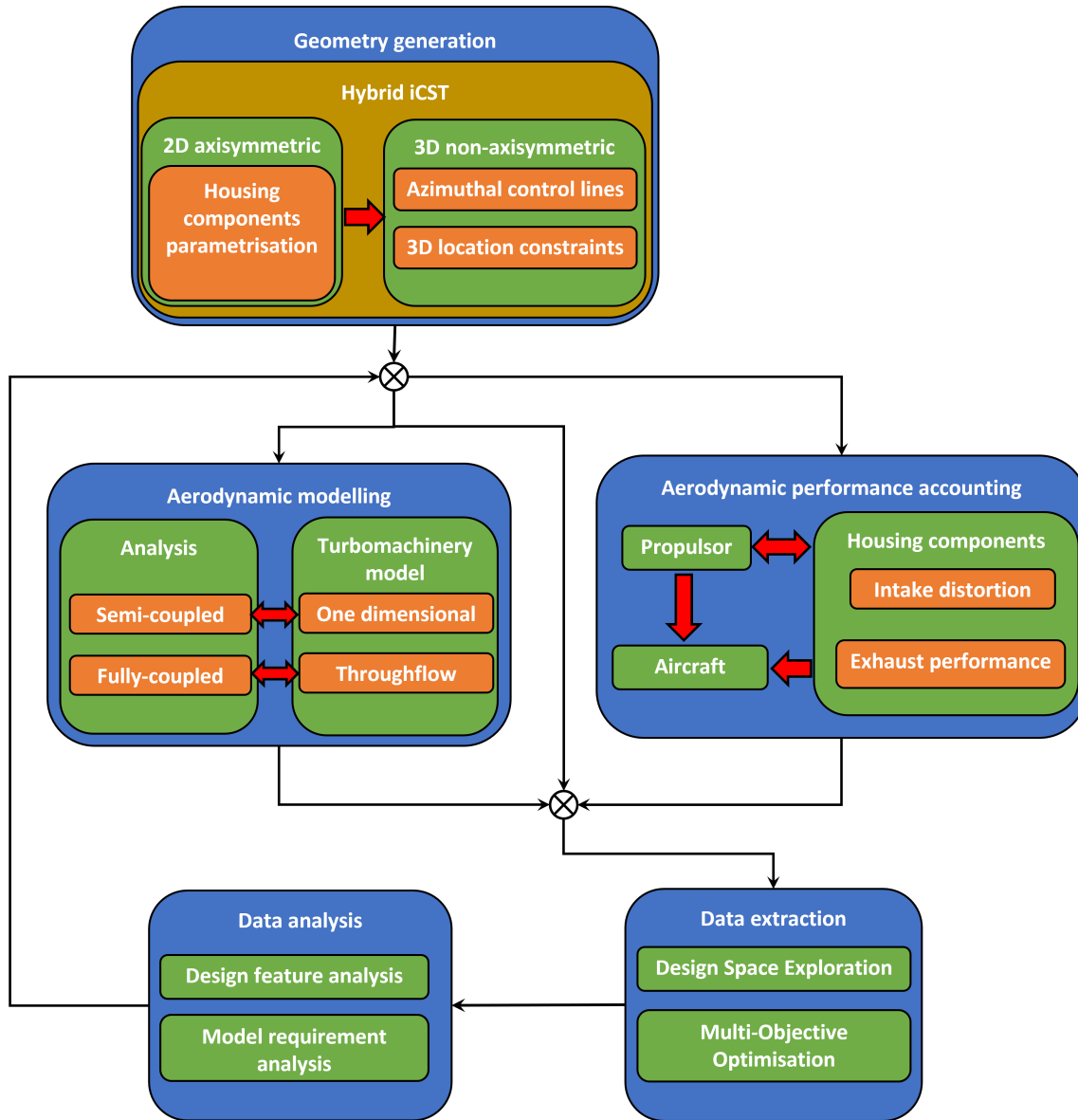


FIGURE 1.2: Scheme of the research roadmap including the most relevant stages for the evaluation of the impact of the propulsion integration on the aerodynamic performance for novel aircraft configurations

data obtained is afterwards analysed to obtain the main characteristics of the novel aircraft configurations and improve the design methodology.

To analyse the design methodologies an aft-mounted annular Boundary Layer Ingestion (BLI) propulsor is selected as a case study. The propulsive system is applied to a medium-size single aisle aircraft. This design based on the NASA STARC-ABL⁴⁰ and the Bauhaus Luftfahrt CENTRELINE¹² and Propulsive Fuselage²⁶ concepts.

1.4 Thesis structure

The thesis is divided into three thematic blocks (Fig. 1.3). The first block is defined to establish the methods for the design and performance evaluation of the novel embedded systems. This block includes: the identification of the problem and the design requirements through the literature review (Chapter 2), the design strategy defined in the methodology (Chapter 3), the analysis of the propulsive and cycle requirements for the case study (Chapter 4), and the sizing of the propulsive system (Appendix B).

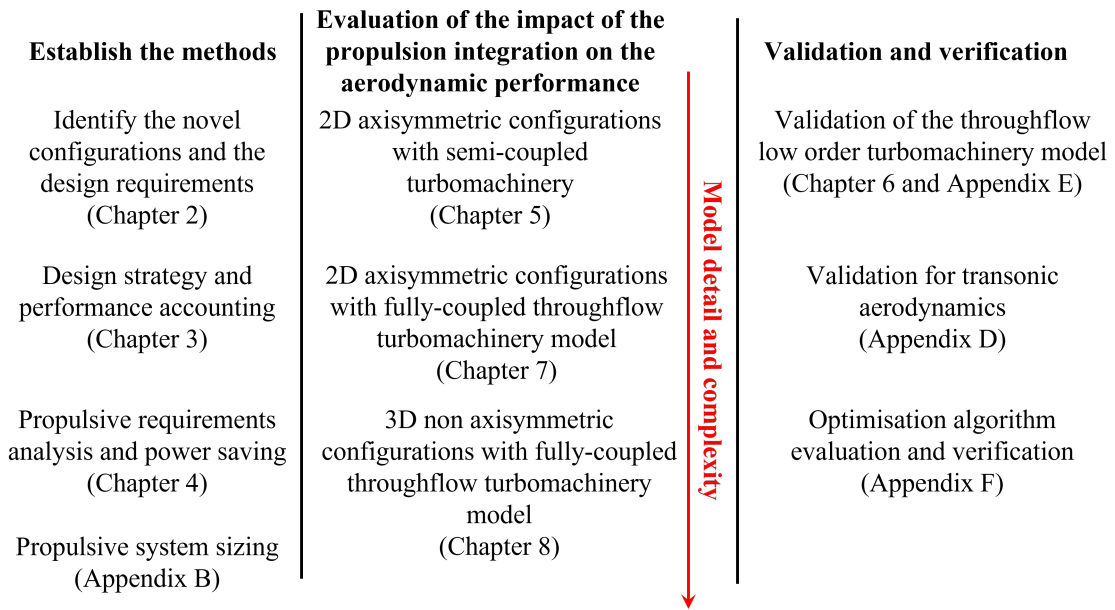


FIGURE 1.3: Map for the different contents of the thesis

The second block comprises the results on the evaluation of the impact of the propulsion integration design on the aerodynamic performance. The approach was addressed with a systematic increase on the complexity of the model and the physical phenomena captured by the model. This includes: a 2D axisymmetric approach with a semi-coupled turbomachinery model (Chapter 5), a 2D axisymmetric approach with a fully coupled propulsive system through a low order fan model (Chapter 7), and a 3D non-axisymmetric approach with the fully coupled turbomachinery model (Chapter 8).

Finally, the last block summarises the validation and verification of the models. The validation of the low order turbomachinery model is included on the main body of

the thesis in Chapter 6 because of its relevance and contribution. The rest of the validation and verification approaches have been gathered on the Appendices section. These appendices include the validation of the CFD methodology for transonic aerodynamics (Appendix D), and the verification of the optimisation algorithms applied (Appendix F).

1.5 Bibliographic contributions

On the course of this project the author has directly and indirectly contributed to the general knowledge about propulsion integration aerodynamics for embedded and conventional propulsion systems. Part of the contribution has been disseminated on a set of publications. That contributions are listed here:

- Matesanz-García, J., Piovesan, T., and MacManus, D. G. Aerodynamic “Optimization of the Exhaust System of an Aft-Mounted Boundary Layer Ingestion Propulsor.” *International Journal of Numerical Methods for Heat and Fluid Flow*, Available online 15 December 2022, DOI: 10.1108/HFF-06-2022-0362⁴⁶
- Tejero, F., Matesanz-García, J., Swarthout, A., and MacManus, D. “Towards the Design and Optimisation of Future Compact Aero-engines: Intake/Fan-cowl Trade-off Investigation.” *International Journal of Numerical Methods for Heat and Fluid Flow*, Available online 9 December 2022, DOI: 10.1108/HFF-06-2022-0366⁴⁷
- Matesanz-García, J., Christie, R., Tejero, F., MacManus, D., and Heidebrecht, A. “An Automated Approach for the Aerodynamic Design of Close-Coupled Propulsion/Airframe Configurations.” *1st Aerospace Europe Conference, Bordeaux, France, 2020.*⁴⁸
- Matesanz-García, J., Piovesan, T., and MacManus, D. G. Aerodynamic “Optimization of the Exhaust System of an Aft-Mounted Boundary Layer Ingestion Propulsor.” *FP06-2022-Matesanz-Garcia. 56th 3AF International Conference on Applied Aerodynamics, Toulouse – France. 2022.*⁴⁹
- Tejero, F., Matesanz-García, J., Swarthout, A., and MacManus, D. “Towards the Design and Optimisation of Future Compact Aero-engines: Intake/Fan-cowl Trade-off Investigation.” *FP04-AERO2022-Tejero. 56th 3AF International Conference on Applied Aerodynamics, Toulouse – France. 2022*⁵⁰

- Sanchez Moreno, F., MacManus, D., Tejero, F., Matesanz-García, J., and Sheaf, C. “Robustness of Optimisation Algorithms for Transonic Aerodynamic Design.” 9th European Conference for Aeronautics and Space Sciences (EUCASS-3AF 2022), Lille – France. 2022. ⁵¹
- Swarthout, A., MacManus, D., Tejero, F., Matesanz-García, J., Boscagli, L., and Sheaf, C. ”A Comparative Assessment of Multi-Objective Optimisation Methodologies for Aero-Engine Nacelles”. 33rd Congress of the International Council of the Aeronautical Sciences (ICAS). Stockholm– Sweden. 2022. ⁵²

CHAPTER 2

Literature Review

The aim of this chapter is to provide an overview of the main aerodynamic design requirements for the propulsion integration of the novel aircraft configurations. This chapter is divided into five sections. The idea behind each section is to identify and introduce the main challenges on the design of the propulsion integration for the embedded systems. A summary of the different approaches to analyse these challenges is also provided based on the open literature.

This chapter starts with the identification of the main novel aircraft configurations proposed to date for long and medium range configurations (Section 2.1). The common aspects and main challenges of the aerodynamic design of the propulsive system for these novel configurations are also identified. The second part of this chapter comprises a discussion of the current performance evaluation techniques (Section 2.2). The advantages of the main methods for the performance evaluation are then discussed to determine the most adequate approach for the present work. In the third part of this chapter, an overview of the different turbomachinery modelling approaches for the embedded propulsor design is provided (Section 2.3). The benefits of the individual methods are evaluated to determine the approaches for the present work. Later, the advances on the methods for the aerodynamic design of the propulsion integration are evaluated (Section 2.4). This evaluation starts with the design methods for conventional wing podded propulsive systems. The evaluation of these methods is used to provide the foundation for the aerodynamic analysis of the embedded propulsion systems. Later, the analysis is extended to the current aerodynamic design techniques for the propulsive system of novel aircraft configurations. Finally, the information of the literature analysis is summarised to determine the current challenges for the aerodynamic design of the propulsion integration of

the novel propulsion systems (Section 2.5). These challenges are compared with the scope of this work to identify some of its main scientific contributions.

2.1 Non-conventional aircraft configurations

On the path to sustainability, national and international agencies have committed to reduce the environmental impact of the aviation industry. The European Commission has targeted a reduction of 75% of the net aviation CO₂ emissions per passenger-kilometre for the year 2050 based on 2000 levels^{31,32}. At the same time, relative to the same technology reference reductions of the 90% on NO_x emissions and the 65% on noise levels are aimed³¹. A similar target was proposed by NASA on their N+3 plans for the next generation of low emission aircraft⁵³. In these plans, compared with the 2000 levels the target is to reduce the fuel burn in a 50 to 60% , the emission of NO_x in a 75%, and the noise levels in 42 dB.

To achieve these ambitious goals, significant changes in the aviation industry are required. One approach to achieve the CO₂ and NO_x emission goals is the reduction of the fuel burn and overall energy consumption of the aircraft. This can be partially achieved by the increase of the propulsive efficiency, the introduction of aerodynamic improvements to reduce the overall vehicle drag, the reduction of the total weight of the system, or changes on the energy sources and fuels applied in the aviation industry. However, these goals are unlikely to be achieved by the modification of the current aircraft and propulsion architecture. For this reason, in recent times multiple novel aircraft concepts have been proposed to obtain more environmentally efficient systems^{1,3-5,33,34}.

Each of the individual novel aircraft concepts provides some degree of innovation to reduce the overall energy consumption of the vehicle. These innovations do not only come from the increase of the aerodynamic efficiency but from different levels of technology improvements. A clear example is the change on the energy source applied to the vehicle propulsion. These novel energy sources vary from the use of alternative fuels such as natural gas⁵⁴ and hydrogen⁵⁵, to the application of electric^{1,12} or hybrid-electric systems^{12,56}. For this reason, in the preliminary design of these novel concepts it is common to find a multi-disciplinary analysis to determine the viability of the novel concept^{4,12,26,34,57-59}. However, once this viability has been addressed at the beginning of the preliminary design stages, a detailed analysis of each of the new technologies involved is required. Since the present work is mainly focussed on the aerodynamics of the propulsion integration of such novel

systems, the analysis of the innovation and technological requirements of the novel aircraft concepts will be studied from the aero-propulsive point of view. Also, while the concepts introduced in the present work are applicable for short-range and small aircraft configurations, the review of the existing literature and the core of this work is focussed on medium and long range transonic aircraft configurations.

Based exclusively on aerodynamic and propulsive criteria, the main novel aircraft concepts to date have been classified based on their level of propulsion integration and their level of aerodynamic technology disruption (Fig. 2.1). To provide a clear view of the different concepts, they have been divided into four basic non-exclusive groups: the Blended Wing Body concepts⁵, the multiple-aft mounted propulsor concepts^{3,9}, the wing mounted distributed propulsion concepts⁴, and the aft-mounted annular propulsor concepts^{1,12}.

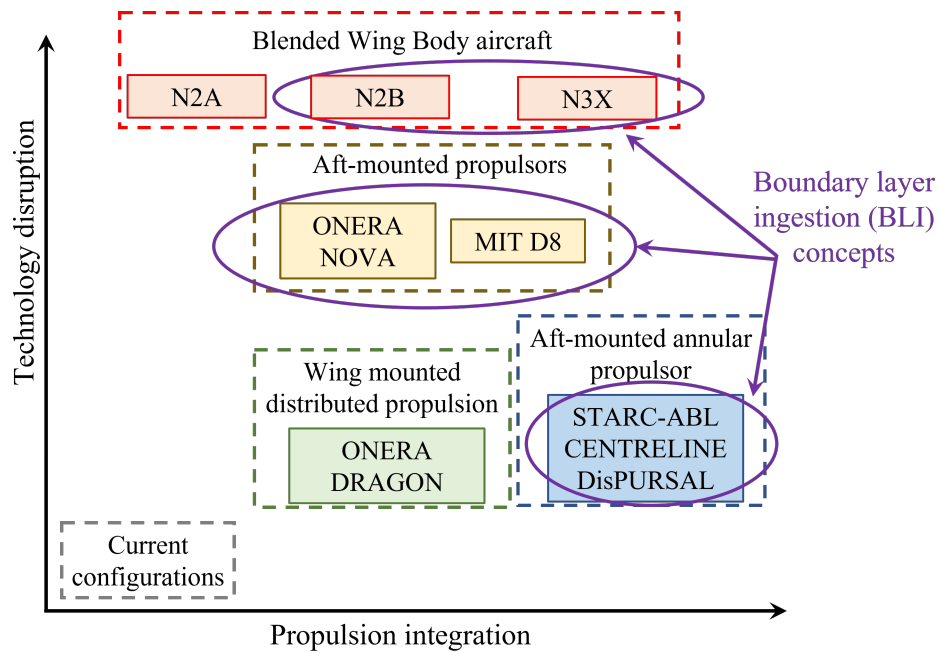


FIGURE 2.1: Summary of the main novel aircraft concepts to date from an aerodynamic perspective. Comparison between the level of propulsion integration and the level of technology disruption of the overall airframe and propulsion system

A more detailed analysis of the individual concepts within the defined groups of novel aircraft configurations is later included in this section. First, it is of interest to understand some of the key aspects and common characteristics of the novel concepts. From the aero-propulsive point of view, the most shared characteristic between the different designs is the closer integration of the propulsion system within the airframe. At top-level analysis, two non-exclusive trends can be identified: more distributed propulsion (DP) systems and boundary layer ingestion (BLI).

In aircraft distributed propulsion, the propulsive elements of the vehicle are distributed span-wise or around the fuselage in such a form that provides benefits to the efficiency of the overall system^{5,34}. These benefits can come from the increase of the aerodynamic or propulsive efficiency or from other sources such as the power or weight reduction⁵. Possibly the most representative cases are the distributed fans over the Blended Wing Body configurations such as the NASA N2B⁷ and the N3X^{41,60} (Fig. 2.3) concepts, and the under-wing multiple propulsors of the ONERA DRAGON configuration⁴ (Fig. 2.4).

The concept of boundary layer ingestion (BLI) in propulsion aerodynamics normally refers to the use of an embedded propulsion system that ingests partially or completely the boundary layer of the main body. In aviation, this concept was investigated as early as 1947 when Smith and Roberts⁶¹ evaluated the benefits of the boundary layer ingestion over the aircraft wings. Nevertheless, the general interest in this technology for the aircraft propulsive system dissipated during the following years probably due to the challenges for its application⁶². However, the potential aerodynamic gains⁶³ derived from the BLI have increased the interest for this technology to meet the environmental targets of the future aviation industry. Representative cases of BLI configurations are the blended wing body N3X and N2B configurations of NASA^{41,60} (Fig. 2.3), the multiple aft-mounted fuselage propulsor configurations such as the ONERA NOVA³ and the MIT D8⁹ (Fig. 2.5), and the aft mounted annular propulsor configurations such as the NASA STARC-ABL configuration¹¹ and the CENTRELINE¹² (Fig. 2.7).

While origin for the benefits of the BLI technology is complex to define, a common explanation is that the aerodynamic benefits come from the minimisation of the difference between the kinetic energy of the wake and the free-stream conditions^{6,64} (Fig. 2.2). As a simplified summary, the presence of a propelled aerodynamic body within the free-stream has an associated profile of kinetic energy difference in the wake of the body. The propulsor jet stream has a positive contribution to the wake kinetic energy, while the airframe wake presents a deficit from the freestream momentum. This deficit of the freestream kinetic energy is what results on the aerodynamic losses. The propulsor then has to compensate the overall kinetic energy loss to maintain steady flight. On boundary layer ingestion systems, the propulsive system is located in such a way that it partially re-energises the region of kinetic energy deficit generated by the airframe. This compensates the loss in mechanical energy at the wake. At the same time, this reduces the amount of energy required to maintain the steady flight condition. Thus, an adequate distribution of the re-energisation in the wake results in an improvement of the aerodynamic efficiency⁶⁴.

This is commonly achieved by the relocation of the propulsion systems closer to the parts of the airframe that produce the largest amount of kinetic energy loss in the wake, such as the fuselage boundary layer.

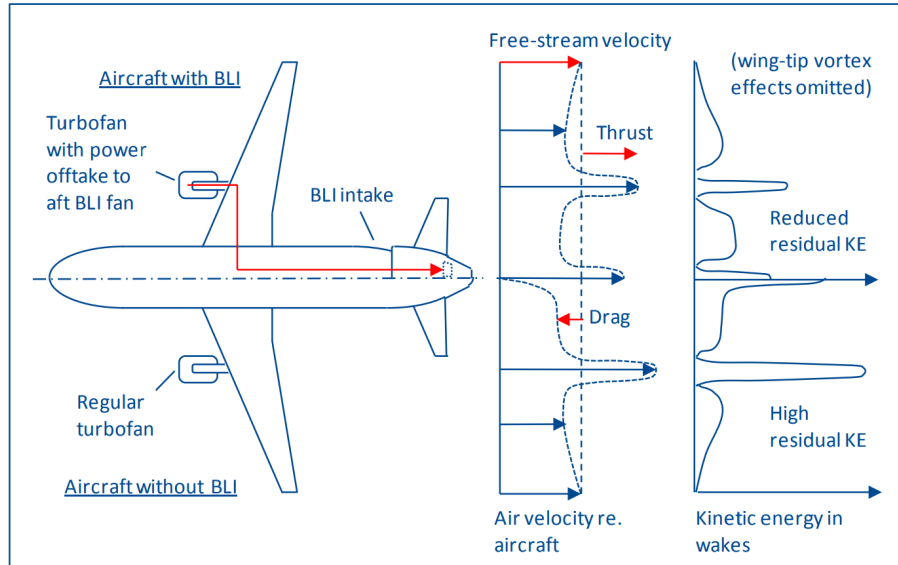


FIGURE 2.2: Schematic of the changes on kinetic energy at the wake of the aircraft as defined by Rolt and Whurr⁶

As mentioned before, the distributed propulsion and BLI concepts are non-exclusive. Thus, a combined application of both ideas is present in multiple novel aircraft configurations. The blend of these concepts enables the balance of the aerodynamic benefits with other restrictions derived from the particular novel aircraft designs. The remaining part of this section is focused on how these concepts are applied for the different families of novel aircraft configurations.

2.1.1 Blended Wing Body concepts

Arguably the most transformational novel aircraft concept is the Blended Wing Body (BWB) configuration^{7,41,60,65,66} (Fig. 2.1), also referred to as Hybrid Wing Body (HWB). This concept abandons the conventional tube-and-wing aircraft idea for a lift-generating wide body configuration. The BWB is expected to have notable benefits in comparison with the conventional aircraft configurations, such as an increase on the overall aerodynamic efficiency and the shielding from the noise propagation in the engines⁴¹. This configuration has also been proposed as a solution for the space limitations for alternative power sources in aviation, such as hydrogen⁵⁵ or turboelectric systems^{60,67}.



FIGURE 2.3: Summary of the main BWB configurations for different levels of propulsion integration. N2A and N2B from Kim et al.⁷, N3X adapted from the sample OpenVSP hangar configuration⁸

The BWB concept is in continuous evolution. During the past years, many different BWB configurations have been proposed^{7,41,60,65,68}. Focused on the propulsion aerodynamics point of view, three representative cases can be selected to illustrate the different aero-propulsive concepts for the BWB configurations (Fig. 2.3). The first case is the NASA N2A configuration. The N2A included the novel aerodynamic characteristics of the BWB concept while a more conventional approach to the propulsion was maintained with conventional podded engines over the main body of the BWB^{7,69}. This configuration was proposed in the early conceptual design of the BWB configurations, but was abandoned in favour of a more integrated distributed propulsion^{7,70} such as the NASA N2B concept^{7,71} and the earlier silent aircraft (SAX-40) proposed by Boeing⁶⁸. Finally, most of the BWB work has converged to the NASA N3X configuration⁶⁰. This configuration includes a set of distributed embedded turbo-electric fans with a single common nacelle driven by wing tip mounted gas turbines. The row of embedded propulsors are mounted in the rear of the main body of the BWB in such way that they ingest a part of the boundary layer in the suction side of the vehicle.

This configuration introduces some challenges from the aerodynamic point of view. The use of the lifting fuselage and the merged wing-body requires detailed analysis in terms of the overall vehicle aerodynamics across all the flight envelope. The strong coupling between the propulsive system and the airframe will require a detailed aerodynamic analysis of the effect of the change on the operating conditions on the distributed boundary layer ingestion fans. This may increase the sensitivity of the propulsive system to changes on the operating conditions and under specific manoeuvres of the aircraft such as take-off and landing. Hence, the BWB requires a holistic approach to the airframe and propulsive system design from the aerodynamic and structural point of view.

2.1.2 Wing mounted distributed propulsion

For medium and long range vehicles, the current most relevant wing mounted embedded distributed propulsion concept is the ONERA DRAGON (Distributed fans Research Aircraft with electric Generators by ONERA)⁴. This concept approaches the vehicle design with a more conservative approach in terms of airframe technology compared with the BWB configurations. In this configuration the conventional tube-and-wing airframe distribution is maintained and an array of ducted distributed fans is introduced on the pressure side of the wing (Fig. 2.4). The electric fans are driven by two rear mounted gas generators. Benefits of 7% in the specific fuel consumption are expected at mission level for this configuration compared with a conventional podded architecture.

The wake recovery concept can be applied to maximise the benefits of the distributed propulsion concepts. For this reason it is common to find both concepts together in novel configurations (NASA N3X). Thus, the novel configurations cannot solely be characterised by any of these systems. Nevertheless, it can be argued that the application of the boundary layer ingestion as a characteristic requires the ingestion of a substantial part of the overall airframe wake deficit. On conventional tube and wing configurations, around 60-70% of the drag share is attributed to viscous drag, and around half of the viscous drag of the system is associated with the fuselage¹². Hence, the application of the BLI nomenclature to the wing mounted distributed propulsion configurations is questionable. For this reason, in this analysis these concepts were removed from the BLI category (Fig. 2.1). Regardless, these configurations are still highly embedded propulsion systems and the determination of their aerodynamic performance requires further study.



FIGURE 2.4: Wing mounted distribution propulsion concept by ONERA: DRAGON (Distributed fans Research Aircraft with electric Generators by ONERA)from Schmollgruber et al⁴

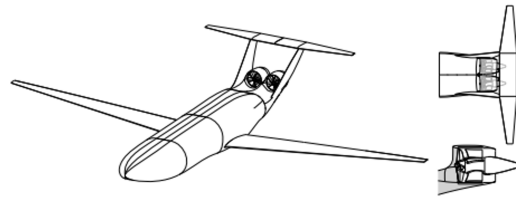
While the level in technology disruption of the airframe is not comparable to the BWB configurations (Fig. 2.1), important challenges are still present in the design of the embedded distributed propulsion systems. The integration of the ducted propulsors on the pressure side of the wing is expected to increase the interaction between the propulsor and the airframe in the lift generation of the vehicle. This requires a combined aerodynamic analysis of the different operating points of the aircraft and under specific manoeuvres. Furthermore, the jet stream from the ducted wing fans is expected to interact with the ailerons and the flaps. This will modify the flight operation with respect to conventional uncoupled systems. Finally, the change in the operating angle of attack is expected to change the local boundary layer distribution on the pressure side of the wing. Thus, the propulsion integration must be designed to maintain the aerodynamic efficiency under these variable conditions.

2.1.3 Multiple aft-mounted propulsors

Another popular concept within the novel aircraft configurations is the introduction of multiple aft mounted propulsors on the fuselage. These configurations are designed to ingest a substantial part of the boundary layer of the fuselage while operating as the main propulsion sources for the aircraft^{3,9}. This is an evolution of the aft mounted podded propulsors commonly found in business jet applications^{64,72}, towards a more integrated propulsion design. The two main novel concepts within this family are the ONERA NOVA (Nextgen Onera Versatile Aircraft)³ and the MIT D8 configuration^{2,9,10} (Fig. 2.5). These configurations also introduce significant innovations in the airframe design as the lifting fuselage of the NOVA and the double bubble fuselage on the D8, while the fuselage and wing concept is maintained.



ONERA NOVA



MIT D8

FIGURE 2.5: Multiple aft-mounted propulsors configurations. ONERA NOVA configuration from Wiart et al.³. MIT D8 schematic configuration from Uranga et al.⁹

The application of these concepts as the primary propulsion system requires of a substantially larger fan diameter when compared with configurations with more

distributed propulsion systems (Fig. 2.5). This results in the ingestion of the largest part of the fuselage boundary layer with a clear sectoral flow distortion profile at the fan face^{3,10} (Fig. 2.6). Hence, a strong coupling between the fuselage design and the final propulsive performance is expected. Thus, a concurrent design of the whole configuration is required. Also, the increase on the fan radius will result in aerodynamic penalties due to the increase in the drag. A possible solution for this is moving towards a more embedded system. However, this can derive on higher aerodynamic coupling and a further increase in intake distortion.

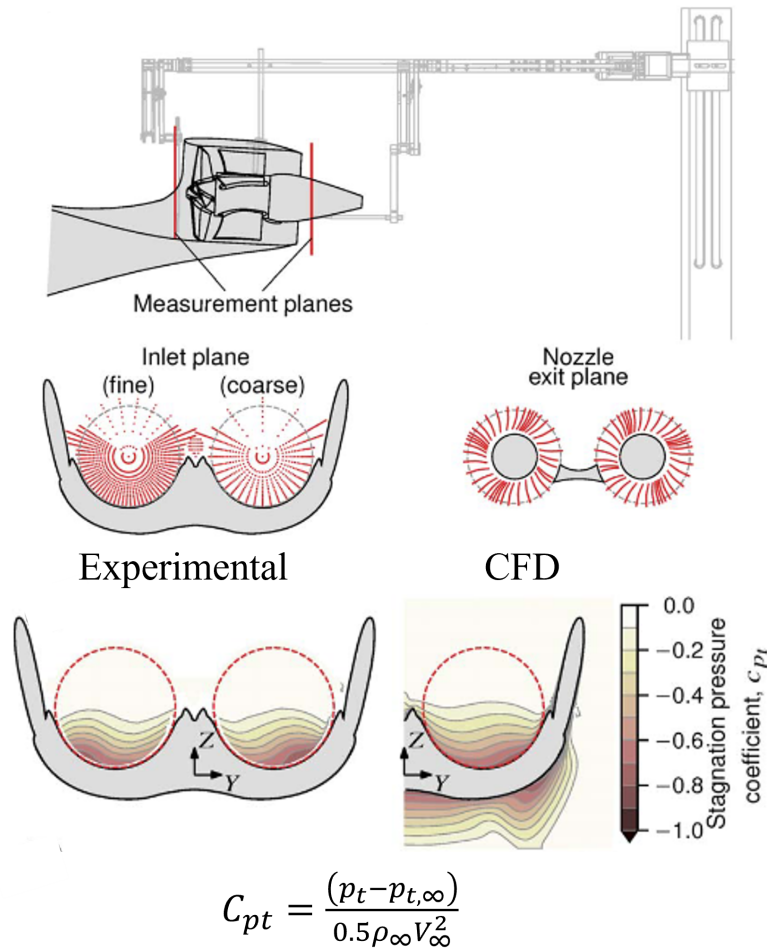


FIGURE 2.6: Comparison between experimental and numerical results for the total pressure coefficient at the inlet reference plane for the MIT D8 cruise conditions. Adapted from Hall et al¹⁰

Another configuration from this group that is worth acknowledging is the distributed aft-fuselage boundary layer ingestion concept proposed by Tse and Hall⁷³. In this configuration the main aft mounted propulsors are replaced by an array of smaller propulsors in a similar way than the previous two concepts to maximize the aerodynamic efficiency and reduce the induced drag of the nacelle. This solves some

of the problems of the larger aft mounted engines, but incorporates the additional problems of the more distributed propulsion.

2.1.4 Aft-mounted annular propulsor

A popular boundary layer ingestion concept is the introduction of a single aft-mounted annular propulsor around the fuselage^{11,12,74,75}. This concept aims to reduce the power input required for conventional wing mounted engines in a tube-and-wing aircraft configuration by a re-distribution of the thrust through a boundary layer ingestion fan. The benefits of the fuselage wake ingestion are expected to reduce the overall thrust requirement and lead to a reduction of the power required by the propulsive system. This is also expected to reduce the wing mounted engine size with a further reduction of the drag and structural weight in the wings. Nevertheless, the weight reduction requires further study due to the new structural requirements for the aft mounted systems.

Two such concepts are the STARC-ABL¹¹ and the CENTRELINE concept¹² (Fig. 2.7). Both configurations propose a turbo-electric system for the aft-mounted propulsor partially driven by the wing mounted fans. A predecessor to the CENTRELINE is the Propulsive Fuselage concept proposed by the DisPURSAL project⁷⁴. This concept introduced an aft mounted gas turbine driven by a core that extracted the mass flow from an S-duct at the end of the fan stage.



FIGURE 2.7: Aft-mounted annular propulsor configurations. STARC ABL configuration from Gray et al¹¹. CENTERLINE concept from Seitz et al¹²

All these concepts maintain the conventional aircraft structure, with the introduction of a vertical T-tail to compensate the loss of space for the horizontal stabiliser due to the presence of the aft-mounted propulsor. The use of the annular propulsion system around the fuselage produces a mainly radial distortion profile at the intake due to the incident fuselage boundary layer. However, it is notable that the presence

of the vertical tail and the propagation of the wake of the aircraft wings and other non-uniform elements of the airframe will introduce additional sectoral distortion profiles into the fan face^{13,76} (Fig. 2.8). Thus, the intake performance is expected to be strongly dependent on the airframe design. For this reason, a combined analysis of the airframe and the propulsive system is required.

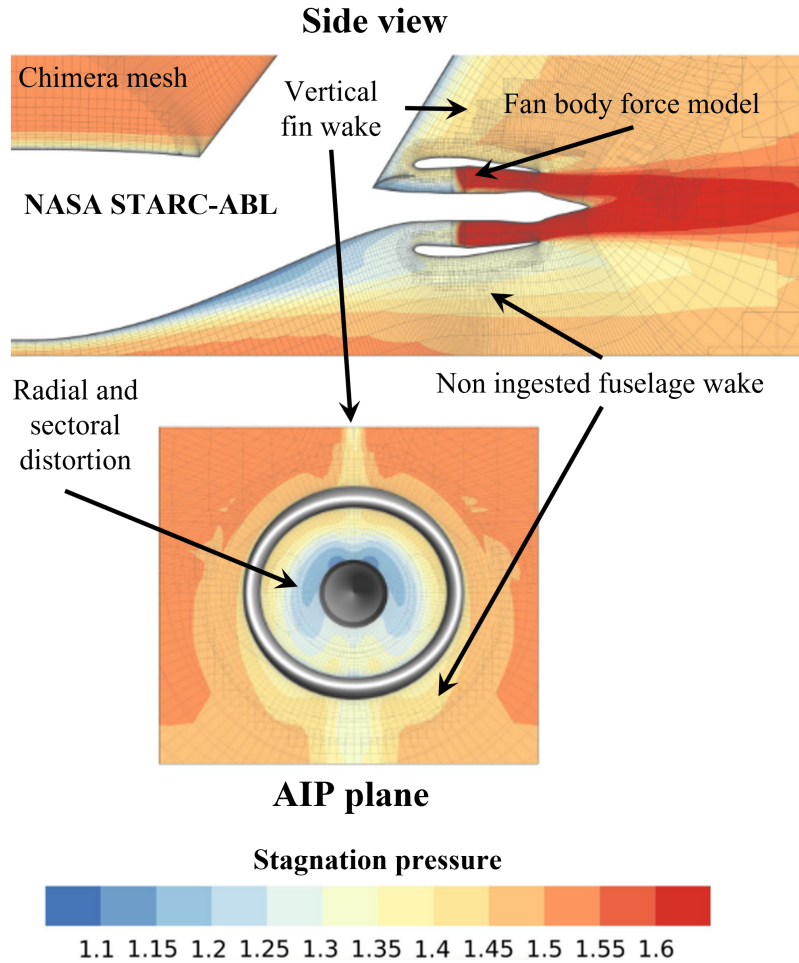


FIGURE 2.8: Isentropic stagnation pressure relation contours around the STARC-ABL, based on Gray et al¹³

2.1.5 Challenges to the aero-propulsive design

While the proposed novel medium and long range aircraft configurations present different characteristics and levels of innovation (Fig. 2.1), several common characteristics can be identified from the aero-propulsive point of view. The identification

of such characteristics is necessary to determine the main challenges for the aerodynamic design of the propulsive system and its integration. A summary of the main aerodynamic challenges is:

1. **Airframe-propulsor aerodynamic coupling.** The transition to more embedded forms of propulsion results in an increase of the aerodynamic coupling between the propulsive system and the airframe. On embedded configurations, the incident flow on the propulsor is highly conditioned by the airframe design at every operating condition. This requires a concurrent approach to the propulsion integration and vehicle design. Thus, to achieve an efficient aerodynamic design methodology both airframe and propulsive system have to be considered simultaneously
2. **Flow distortion and non-uniformities.** The transition towards more embedded propulsive systems also leads to an increase on the perceived flow distortion by the turbomachinery. Part of the distortion can be mitigated by an effective airframe and propulsion integration aerodynamic design. However, some level of distortion is always expected due to the nature of the embedded systems. Each of the embedded systems has a governing type of flow distortion (Fig. 2.9). The limits of the operation of the turbomachinery stage will be determined by that governing distortion. Sectoral distortion types are expected for the embedded arrays of propulsors as in the NASA N3X, the distributed wing mounted propulsor and the multiple aft mounted propulsor concepts. For the aft mounted annular propulsor, radial distortion profiles are expected. However, these governing mechanisms are not exclusive and some combination between them and local distortion types could be expected under some operating conditions. Hence, to maintain stable operating conditions at the fan face, it is essential to understand the effect of the aerodynamic design of the propulsor and the airframe in the development of the flow distortion
3. **Turbomachinery design and distortion propagation.** The distortion at the intake is expected to propagate through the turbomachinery stage affect the propulsive system performance^{43,77}. Hence, the fan operation and design are highly dependent on the aerodynamic conditions at the intake fixed by the airframe and the propulsor integration design. The flow distortion at the intake can be reduced to mitigate the effects on the performance by an adequate aero-propulsive design. However, since some levels of distortion are always expected, it becomes essential to design distortion tolerant fans⁷⁸ to reduce the penalties of the distortion the system performance. This becomes

an additional line of research for the novel aircraft configurations. Also, the distortion propagation by the turbomachinery will affect the operation of the exhaust system with potential penalties on the overall system aerodynamic performance. Thus, to accurately model the aerodynamic performance of the system in early design stages, it becomes essential to introduce a representative model of the turbomachinery

4. **Efficiency accounting and force decomposition.** The determination of the split of the aerodynamic forces is an essential part of the aero-propulsive design. This is of great importance to separate the features of the aerodynamic design that contribute to the reduction of the aerodynamic losses. However, in embedded system the high level of integration and the coupling between the airframe and the propulsive system complicate this analysis. The decomposition of the force contribution is obscured because of the interaction between the different elements of the propulsion integration. Conventional accounting methodologies could be insufficient to extract all the information derived from the aerodynamic design. Thus, an interest for more complex and detailed forms of force analysis has emerged^{16,75,79}. However, further development is still required to finally introduce these accounting methodologies on the aero-propulsive aerodynamic design⁸⁰. Under this context, it becomes critical to evaluate the advantages and disadvantages of all the force accounting methods for any of the aircraft design phases

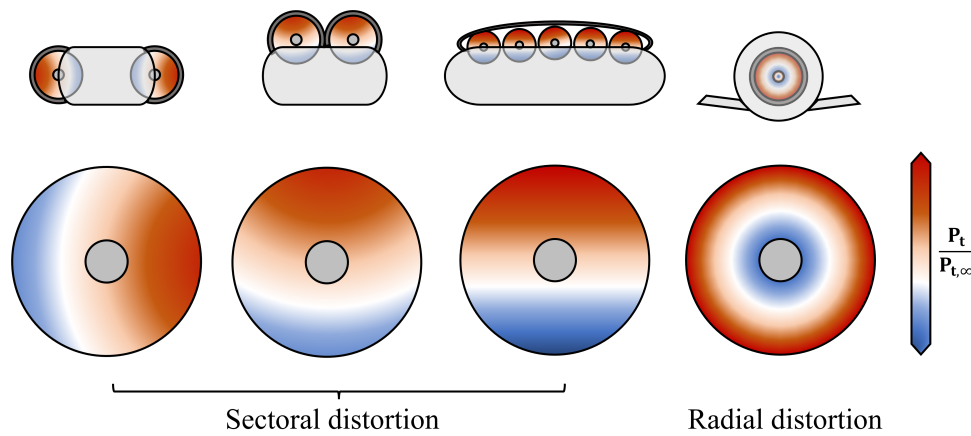


FIGURE 2.9: Schematic of the governing distortion types for the different embedded propulsion systems

The aforementioned challenges need to be addressed at some level to evaluate the aerodynamic performance of the embedded propulsive system. In the present work, some solutions to account for these challenges while modelling the aerodynamic

performance are presented. In the next sections of this chapter, the main approaches taken in previous work to study the aerodynamic performance are presented. This review of the different approaches to date is applied to identify the individual gaps in the current knowledge of the aerodynamic performance evaluation of the embedded system. They are also applied to establish the current baselines for the methods discussed later in this thesis.

2.2 Performance accounting for novel aircraft configurations

An adequate accounting of the different force contributions of the aircraft and the embedded propulsor is essential to determine the aerodynamic performance of the different designs. As mentioned before, the strong coupling between the airframe and the propulsor aerodynamics complicates the separation of the force contributions of the different elements. Hence, it is not possible to determine the aerodynamic performance in isolation, but the system has to be considered as a whole. In addition, the increase of the complexity of the aerodynamics around the embedded systems may require a re-evaluation of conventional force accounting methods. For this reason, there is growing interest in the development of novel accounting methods such as the energy based methods^{16,79,81}, and the improvement of more conventional methodologies such as the momentum based approaches^{14,15,80,82–84}. Each of these different accounting methods introduce different advantages for the force accounting and decomposition. However, for most of these methods the greater the amount of information that is obtained from the analysis the greater the complexity (Fig. 2.10). Also, the methods that allow to obtain a greater level of information are generally far-field methods that require a higher numerical grid resolution and computational cost. Hence, it is not currently possible to determine the best general force accounting method. Furthermore, the selection of the force bookkeeping approach has to be evaluated for each individual case. In this section a discussion of some of the most relevant accounting methodologies found in the literature is provided. The advantages and challenges of each method are also discussed to try to determine the most suitable approach for the aerodynamic analysis of the present work.

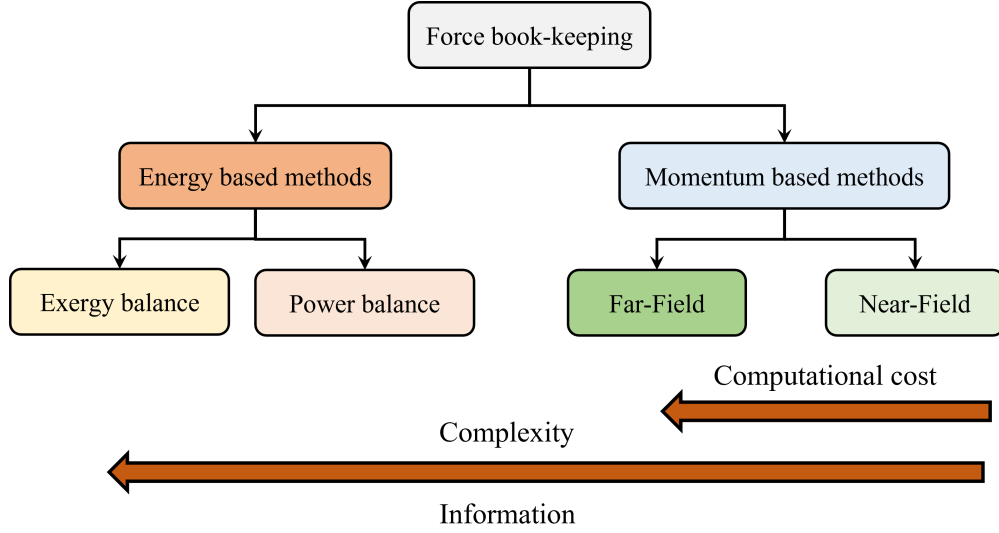


FIGURE 2.10: Summary of the different types of force book-keeping approaches found in the literature

2.2.1 Momentum based methods

The momentum based approaches obtain the forces over the body of interest based on the momentum equation of the Navier-Stokes formulation (Eq. 2.1). From the momentum balance the forces that affect the body of interest can be extracted by two approaches: the Near-Field approach and the Far-Field approach.

$$\begin{aligned}
 \mathbf{F}_w &= \iint_{S_{w_i}} [P\mathbf{n} - \bar{\bar{\boldsymbol{\tau}}} \cdot \mathbf{n}] \cdot d\mathbf{S} \\
 &= - \iint_{S_o, S_{out}, S_{ff_i}} [\rho\mathbf{V}(\mathbf{V} \cdot \mathbf{n}) + P\mathbf{n} - \bar{\bar{\boldsymbol{\tau}}} \cdot \mathbf{n}] \cdot d\mathbf{S}
 \end{aligned} \tag{2.1}$$

The Near-Field approach is based on the pressure and viscous force integration over the surfaces of the body of interest, S_{w_i} (Fig. 2.11). This corresponds to the left hand side of the momentum equation (Eq. 2.1). This approach has two clear advantages when applied to computational fluid dynamics. First, it is a simple and straightforward method to obtain the overall forces over a body that only requires the integration of the pressure and viscous forces over the surface. Second, it requires a relatively lower computational cost than the other methods, since the mesh refinement can be focused on the region of interest around the body. However, this method only allows the decomposition of the drag forces between the pressure and the friction contribution. This can be an inconvenient when a detailed decomposition of the drag contributors is required.

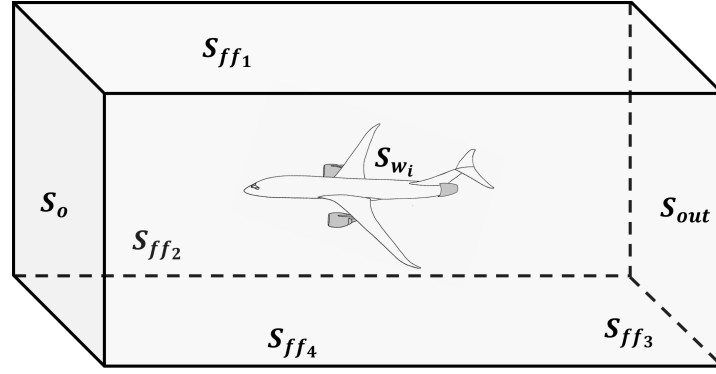


FIGURE 2.11: Decomposition between the far field and the near field surfaces

To account for the contribution of individual parts of the body, some method to split the forces is required. A good example of this is the Modified Near-Field Method proposed by AGARD 237¹⁴. This method has been commonly applied for the aerodynamic analysis of conventional podded engines^{15,35,50}. The main advantage of this near-field method is that allows the separation of the thrust and drag components of a podded aero-engine configuration by the introduction of reference control volumes in the domain (Fig. 2.12). Additional flow field data can be applied to complete of the force definition. However, this is not required for the net force contribution of the propulsive system and the vehicle. Therefore, this method provides sufficient flexibility to be adapted for the embedded propulsor requirements.

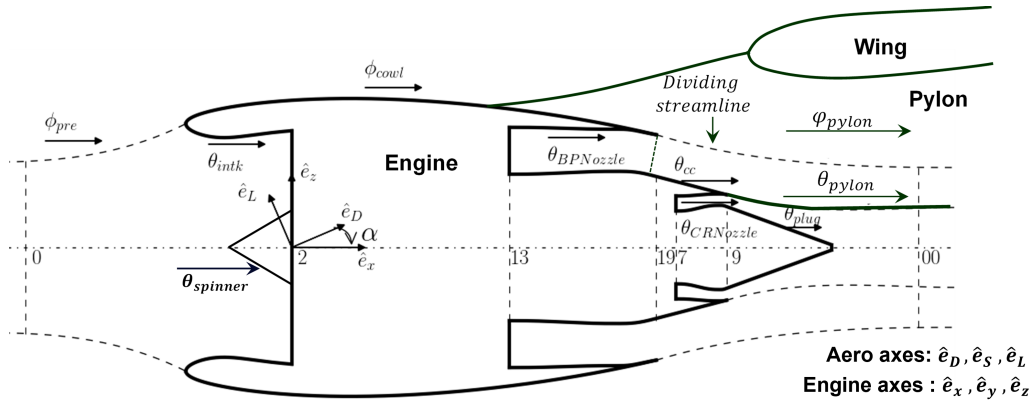


FIGURE 2.12: Thrust and drag bookkeeping for installed turbofans with separate jet exhaust applying the modified near-field method¹⁴, adapted for Goulos et al¹⁵

The Far-Field methods are proposed as a more complete alternative to the Near-Field formulation. These methods obtain the forces acting on the body of interest, S_{wi} , as the integral of the momentum (Eq. 2.1) at the far field regions of the fluid domain (Fig. 2.11). This formulation can be simply applied by the integration of the outer forces of the domain or it can be reduced to the downstream sample

plane (S_{out}) with the divergence theorem⁸¹ (Eq. 2.2) based on the reference values upstream (S_o). This reduction of the integration surfaces is the starting point for the wake-reduced far-field methods^{83,85–88}. An advantage of these methods is that they allow the phenomenological decomposition of the drag terms. Hence, the application of these methods provides a substantially higher amount of information about the physical origin of the force contributions over the aerodynamic body of interest. However, from the computational point of view, these methods are considerably more expensive than the Near-Field method equivalent because they require a higher mesh definition on the regions downstream the body of interest. Besides, the selection of the aerodynamic reference plane applied to extract the value (S_{out}) can have an effect on the final force values obtained. Thus, a detailed analysis of the sensitivity to the location of this plane and the dimensions of the volume is required. Furthermore, the phenomenological decomposition of the forces and their contribution to the lift and drag of the object is not always required from the engineering point of view, but it could be of interest for a more detailed analysis.

$$\mathbf{F}_w = - \iint_{S_{out}} [\rho (\mathbf{V} - \mathbf{V}_o) (\mathbf{V} \cdot \mathbf{n}) + (P - P_o) \mathbf{n} - \bar{\bar{\tau}} \cdot \mathbf{n}] \cdot d\mathbf{S} \quad (2.2)$$

While the rule of thumb “higher force decomposition information higher computational cost” can be applied, some notable exceptions can be found in the literature. Paparone and Toagaccini⁸⁴ developed a far-field approach for CFD with a reduced grid dependence. This approach allowed an adequate decomposition of the viscous and wake drag components based on the entropy variations on the flow. However, this approach requires the decomposition of the domain into shock wave and viscous regions (boundary layer and wake) by the definition of additional metrics. This adds an additional layer of complexity to the method and may require additional calibration to properly determine the critical values for the viscous and shock wave selectors⁶⁴. Another notable exception is the application of the partial-pressure field method proposed by Schmitz⁸². This Near-Field method allows the decomposition of the pressure drag term into “form drag” and “vortex drag”. This provides additional information about the drag phenomenology while maintaining a reduced computational cost. Nevertheless, this method still requires further development and understanding to be generally applicable for the force bookkeeping for the aerodynamic analysis of the propulsion integration.

2.2.2 Energy based methods

The energy based methods have emerged as an alternative to the conventional momentum based approaches. These approaches are based on the conservation of the integral energy in the control volume. The energy based approaches allow a clear identification of the energy sources, sinks and irreversibilities that affect the vehicle performance. An advantage of such methods is that they do not require a thrust-drag split of the forces, as conventional momentum based methods require. Thus, these formulations have become popular on the aerodynamic analysis of embedded systems and novel aircraft configurations⁸⁰. To date, there are two main formulations for the energy based force accounting: the Power Balance method^{75,79,89,90}, and the Exergy Analysis method^{16,81}.

The Power Balance method was first formulated by Drela⁷⁹ for a generic computational fluid dynamics approach. Drela's formulation analyses the mechanical energy conservation within a control volume. This approach includes the coupling effect between the propulsion system and the overall vehicle aerodynamics. On the propelled aircraft the wake energy is conserved, this wake energy is divided between the kinetic wake energy that is extracted at the sample plane downstream (S_{out}) and the dissipated kinetic energy within the volume. This formulation can be summarised in equation 2.3. The total mechanical power supplied is divided between the shaft power (P_S), the net pressure-volume power (P_V), and the net propulsor mechanical energy flow rate into the control volume (P_K). The total mechanical power supplied is equal to the mechanical energy flow rate out of the control volume ($\dot{\epsilon}$), and the viscous dissipation rate (Φ).

$$P_S + P_V + P_K = \dot{\epsilon} + \Phi \quad (2.3)$$

This method has some direct advantages for the analysis of embedded propulsion systems. The main advantage is that the method removes the ambiguity of the thrust and drag definitions required for conventional momentum based force accounting. The power dissipation in the vehicle and the propulsive system are unified through an integral analysis of the system efficiency without force domain separation. This is of particular interest in the embedded propulsion systems where the separation between the thrust and drag domains is not as straightforward as in configurations with less aerodynamic coupling between the airframe and the propulsion system. Another advantage of this method is the decomposition of the wake energy sources and losses. This allows an intuitive interpretation of the aerodynamic efficiency of

the aircraft-propulsion system. However, this requires some additional understanding of the metrics to allow an adequate interpretation. Finally, some of the terms of the power based analysis (Eq. 2.3) can be used to compare the benefits of different configurations. For example, Uranga et al⁸⁹ and Blumethal et al⁹⁰ applied the energy source decomposition to quantify the benefits of boundary layer ingestion systems in comparison to conventional propulsion through the comparison of their energy flow rate into the control volume (P_K).

Nevertheless, while this method appears to be tailored for the aerodynamic analysis of embedded propulsion systems, some challenges are present for its application on the preliminary design stage. The Power Balance method is a far-field equivalent method. Thus, it requires additional computational effort to accurately model the far field region and to capture the wake dissipation. Additional sensitivity studies to the control volume integration and the sample plane are required to determine the mesh sensitivity of the method. Additionally, there is not a direct equivalence between the power balance and the net force over the vehicle-propulsor system. Therefore, to obtain the force imbalance in the system a momentum integration is still required. For this reason, in applications where the force imbalance is still required (e.g., determine the excess thrust of a configuration to maintain steady flight), the Power Balance method can only provide additional information but not identify the changes on the thrust requirement. Finally, it is necessary to highlight that, to date, there is a notable absence of open source high fidelity experimental benchmark cases for the validation of this method. Nevertheless, this is expected to change in the near future. Uranga et al⁸⁹ applied the power balance method to subsonic wind tunnel experiments with the MIT D8 configuration. However, due to the limitations of the wind tunnel data the kinetic energy dissipation was extracted from the kinetic energy fluxes.

The second main energy based method is the Exergy Analysis proposed by Arntz¹⁶. Similarly to the Power Balance method, this method is based on the energy conservation equation in combination. However, Arntz's formulation not only includes the contribution of the energy balance, but also combines the second law of thermodynamics. This allows the split of the total energy of the system between two main components: the exergy and the anergy. The exergy can be defined as the potentially recoverable part of the total energy, while the anergy is defined as the unrecoverable energy or the destroyed exergy. Hence, the identification of the exergy of the system provides a clear idea of the potential of aerodynamic improvement from the energy perspective and identifies the quality of the energy involved in a closed system. Any change in the system conditions will produce an exergy imbalance that

can be used to identify the changes on the system efficiency. The Exergy Analysis also allows phenomenological decomposition of the exergy and late the drag contribution. Arntz's method is characterised by equation 2.4. In this formulation the exergy supplied by the propulsive system ($\dot{\epsilon}_{prop}$) and the thermal exergy supplied by heat transfer ($\dot{\epsilon}_q$) are equal to the sum residual mechanical exergy that flows out of the domain ($\dot{\epsilon}_m$), the residual thermal energy that escapes the domain ($\dot{\epsilon}_{th}$), the capacity of accumulation of the aircraft ($W\dot{\Gamma}_h$), and the thermal, viscous and shock anergy contributions ($\dot{A}_\Phi$, $\dot{A}_{\nabla T}$, $\dot{A}_w$).

$$\dot{\epsilon}_{prop} + \dot{\epsilon}_q = W\dot{\Gamma}_h + \dot{\epsilon}_m + \dot{\epsilon}_{th} + \dot{A}_\Phi + \dot{A}_{\nabla T} + \dot{A}_w \quad (2.4)$$

This formulation proposes an improvement from the original Power Balance method and allows the identification of the limit theoretical values. The formulation also provides a correlation between the drag with a phenomenological decomposition based on the exergy and anergy contribution. This provides a more complete definition of the system efficiency when compared with the power balance method. This definition was applied to embedded propulsion systems by Arntz and Atinault¹⁶, where a 50% improvement on the aerodynamic performance was identified for a 2D simplified blended-wing-body architecture (Fig. 2.13). However, Arntz's method required some improvements to increase the phenomenological decomposition of the losses on the system and compare it with far field analysis techniques. Aguirre⁸¹ proposed some modifications to Arntz's formulation to increase the available information and its application. Aguirre also formulated a wake-reduced approach to extend the application of the exergy analysis for the experimental analysis of complex embedded propulsive systems.

The Exergy Analysis formulation modified by Aguirre⁸¹ defines a promising starting point for the general application of this method for aerodynamic analysis. Nevertheless, this approach presents also some inconveniences that need to be addressed. First, similar to the Power Balance method, the Exergy Analysis is based on far-field methods. Thus, it requires a higher computational cost when compared with conventional near-field approaches. This has been addressed by Aguirre⁸¹ with the wake reduced formulation. The formulation reduces the dependence on the sample downstream plane, but some corrections are still required. In comparison with Drela's formulation, the exergy analysis provides a clearer definition of the drag. However, the interpretation of the different terms requires further study to correctly understand the exergy and anergy contributions. On the experimental side, Aguirre's formulation potentially will allow the increase of the available experimental data

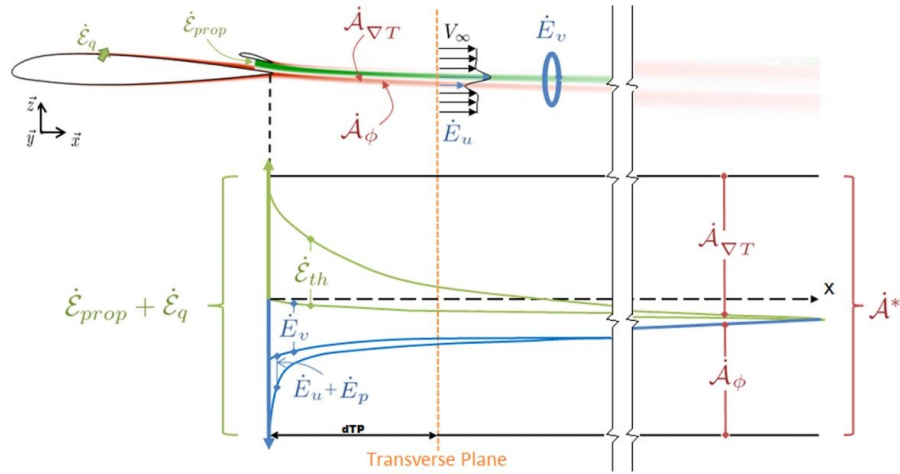


FIGURE 2.13: Evolution of the exergy balance for a blended wing body configuration with an embedded propulsor, as defined by Arntz and Atinault¹⁶

for the validation of the methods. However, relevant experimental benchmarks for validation are not widely available at this time.

2.3 Low order turbomachinery models

As mentioned earlier, the closer coupling between the propulsive system and the airframe is expected to increase the flow distortion levels at the propulsor intake. This distortion will propagate through the turbomachinery stage and penalise the overall system performance^{42,43}. The aerodynamic design of the propulsion integration has to account for this increase in the distortion. The intake design also has to mitigate and provide consistent flow conditions across the different operation points of the aircraft. To minimise the penalties of the distortion propagation, the turbomachinery stage must be redesigned^{78,91}. The redesign of the turbomachinery stage is beyond the scope of this project. However, to assess the aerodynamic performance of the embedded systems some type of representative turbomachinery model is still required. Moreover, for close-coupled systems the turbomachinery is expected to have an influence on the aerodynamics and the distortion levels at the intake. Cao et al⁹² found that the application of a representative fan model in CFD simulations mitigated the predicted distortion levels for short intakes at high flow incidence.

As stated above, the evaluation of the aerodynamic performance of the close-coupled propulsor-airframe systems necessitates a representative turbomachinery model. However, the full characterisation of the turbomachinery stage in CFD requires a substantial amount of computational resources^{17,93}. For this reason, several low order turbomachinery models have been proposed to represent the turbomachinery effects with lower computational requirements (Fig. 2.14). Each of these models provides some advantages in the accuracy or the computational requirements. In general, these two factors are inversely correlated. Thus, for the accurate evaluation of the propulsive system performance the selected turbomachinery model will require a compromise between both. In this section, a general analysis of the different models is provided. The advantages of each individual group of models will be compared to assist in the selection of the most appropriate approach for the present work.

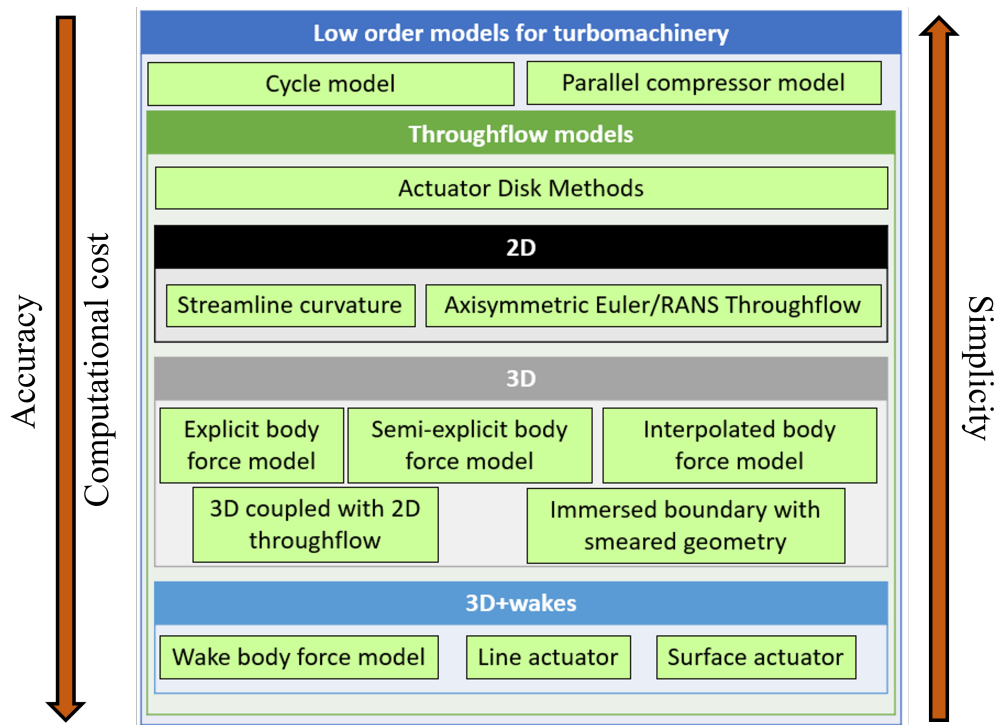


FIGURE 2.14: Summary of the different categories of low order fan models found in the literature

2.3.1 Non-throughflow approaches

The simplest approach for the definition of the turbomachinery effects in CFD models is the definition of fixed boundary conditions based on the stage performance maps. Based on the required fan operation point, the required mass flow at the fan face is imposed, while the outlet conditions at the exhaust inlet boundary

are defined by the stage pressure ratio and the defined fan efficiency. This approach has been widely applied for the aerodynamic analysis of conventional podded propulsors^{35,36,38,94,95}, and also for the preliminary design of novel aircraft configurations^{11,59,64,96–98}. However, due to the simplicity of this method some relevant information about the impact of the turbomachinery on the performance could be missed. For instance, the fan inlet conditions can be used to match the mass flow requirements of the fan, but they do not represent the mass flow redistribution due to the fan. This is not a problem for cases with homogeneous incident flows at the fan. However, for cases under high distortion this could misrepresent the actual flow at the fan face as shown by Cao et al⁹² and Boscagli et al⁹³. Another problem is the characterisation of the fan operation under distortion. Gunn et al⁴² and Defoe et al⁴³ found in separate studies that notable changes on the fan performance and operating point are expected for turbomachinery under flow distortion. It can be argued that future turbomachinery design could mitigate these losses as shown by Castillo-Pardo and Hall⁷⁸ and Hall et al.⁹¹. However, the fan cycle model should take into consideration these effects. Despite these inconveniences, the simplicity of the model still provides an interesting starting point to obtain fast results in the preliminary design stages. Finally, some representation of the fan outlet total pressure and temperature distribution may be required. These distributions are largely affected by the fan work redistribution⁴². However, some preliminary characterisation of the magnitude distributions at the OGV outlet can be included to improve the model. Goulos et al⁹⁹ noticed an increase of 0.08-0.19% in the overall velocity coefficient and an increase 0.45-0.72% in the discharge coefficient for a large turbofan with separated exhaust by the modification of the turbomachinery outlet profiles. Thus, there is a potential of improvement for these models by the application non-uniform fan outlet conditions.

Another simplified method to evaluate the turbomachinery effects is the Parallel Compressor Model¹⁰⁰. This model allows to maintain the cycle analysis, but at the same time can account for the effects of the distortion transfer through the turbomachinery stage. This correction is achieved by the definition of multiple compressors operating in parallel and discharging to the same static pressure with different levels of inlet total pressure values or distortion. Under these conditions, the compressor operating at lower total pressure (higher distortion) has to introduce more work to achieve the equivalent discharge static pressure. The operating point of the compressor is then obtained from the mean operating point between the compressors. This method can be used for turbomachinery stages operating under sectoral, radial and local distortion¹⁰¹ (Fig. 2.9). Also, in recent times (2022) Silva et al¹⁰²

developed the Modified Parallel Compressor method. The method is an extension of the conventional parallel compressor theory for CFD that matches the fan work distribution across the blade span. This method also improves the characterisation of the total pressure profiles at the fan boundary compared with simple aspirated boundary conditions. Nevertheless, none of these approaches allow to fully match the turbomachinery exit profile for cases under clean flow or distortion. Also, they require a more detailed knowledge of the fan operation under different conditions. Thus, either a reference fan map has to be applied or a reference operating point needs to be imposed based on on technology assumptions⁹⁶.

2.3.2 Throughflow approaches

The main inconvenience of the non-throughflow low order turbomachinery models is that the changes on the stage outlet conditions are decoupled from the conditions at the stage inlet. To couple the turbomachinery stage effects with the propulsion integration aerodynamics a throughflow turbomachinery model is required (Fig. 2.14). Thollet¹⁷ proposed a classification of the throughflow methods into four groups: explicit, interpolation-based, semi-explicit, and implicit methods. This classification is used as reference in the present work and the characteristics and advantages of each method are evaluated approximately in ascending order of complexity.

The bulk majority of the throughflow low order turbomachinery methods rely on the formulation defined by Marble¹⁰³. Marble proposed the representation of the effects of the turbomachinery by the addition of body forces (source terms) to the Euler or Navier-Stokes equations, based on an axisymmetric representation of a blade row based with an infinite number of blades. With this approximation, Marble was able to derive the thermodynamic relations that linked the blade forces to the rise of total enthalpy and the production of entropy inside the blade row. This formulation enabled the definition of different low order models such as the streamline curvature and the body force model approaches.

The streamline curvature approaches provide a cost effective representation of the turbomachinery effects based on the stage geometry in a two-dimensional stage. For this reason, these formulations are commonly applied in the preliminary turbomachinery design¹⁰⁴. The streamline curvature methods model the flow turning based on the Euler equations through the blade camber. Radial equilibrium across the blade is assumed and the losses within the blade have to be specified by an explicit of implicit correlation¹⁰⁵. These formulations can characterise the flow transfer and

change on stage efficiency for fan inlet radial distortion¹⁰⁴. However, these formulations cannot be applied to model mass redistribution under three-dimensional flow distortion. In addition, these models are unable to capture the effects of reverse flow in turbomachinery. This reduces the operability on the near stall regions of the operation¹⁷. Also, some of the streamline curvature methods have problems when operating under transonic flows. This can be a challenge to model the flow on the near tip region of the blade or some blade blockage effects. As a solution to the limitations of the streamline curvature, other two-dimensional throughflow approaches have been proposed based either on the Euler or Navier-Stokes equations¹⁰⁶. Some of these approaches can model the effects of transonic flow, mass redistribution and reverse flow on the blade row. However, some correlations are still required to represent the blockage and viscous losses on the blade row for most of these models¹⁷. Thus, for 2D axisymmetric turbomachinery stages the change in the flow properties can be modelled with different levels of accuracy. Nonetheless, a more generic definition of the low order turbomachinery model will be required to also include three-dimensional flow effects.

There are several three-dimensional low order throughflow turbomachinery models available in the literature. The simplest group of models under this category are the interpolated body force models. In this approach, the body forces that produce the change on enthalpy, flow rotation and entropy inside the passage are obtained by interpolation from a database^{17,107}. These models are based on prior computations or experiments to construct the database. Hence, they can be matched to a higher order reference to provide a more accurate model of the turbomachinery response with a fraction of the original computational cost. However, the dependence on the database is also the most significant limitation of such models. The operating range of the models is prescribed by the quality and extension of the higher order database, which is normally much more expensive to obtain. This also presents an inconvenience for the analysis of close coupled systems, where the distortion patterns at the fan boundaries can have significant variations between operating points or between propulsion integration aerodynamic designs.

A step beyond the interpolation based body force models are the semi-explicit body force model formulations^{41,108,109}. These formulations define the changes on the flow turning, enthalpy rise and losses by a combination of explicit physical formulations and some correlations based on experimental data. As an illustrative example, Kim and Liou⁴¹ proposed a body force model approach to couple the turbomachinery design with the aerodynamic performance of the NASA N3X configuration (Fig. 2.3). This body force model formulation relied on an explicit formulation of the flow

turning mechanism, but applied Marble's formulation¹⁰³ to extract the losses based on a correlation with the required entropy distribution. The advantage of these models is that the flow turning is defined directly by a physical model dependent on the blade geometry. However, they need a calibration reference for the implicit loss formulation. This may require extensive calibration of the models before operation. Thus, the universality of the methodology is reduced.

The explicit body force model formulations are presented as the most generic application of the throughflow approaches. These formulations rely completely on explicit physical definition of the change on properties within the turbomachinery stage. Hence, the losses on the stage performance are no longer an input, but a direct consequence of the physical model and the flow conditions in the stage. The first model of this kind was proposed by Gong¹¹⁰. Gong divided the body force contributions into two main sources relative to the local relative velocity: the normal force that defines the flow turning and the parallel force that determine the losses of the system. The only inconvenience of Gong's formulation is that the parallel force component requires calibration to match the higher order turbomachinery reference. This formulation has been maintained for multiple body force model definitions. Hall¹¹¹ introduced an inviscid formulation for this model that neglected the parallel force terms. This formulation was proposed directly for the evaluation of embedded propulsion systems and did not require calibration. Thollet¹⁷ proposed a viscous formulation of Hall's model in which he included a parallel force separated into two terms: the friction losses and the deviation losses. The deviation losses are applied to calibrate the model based on the reference. Further validation under highly distorted flows was applied to this model by Benichou et al¹¹². Thollet also proposed some modifications to Gong's model to increase the accuracy of the loss model¹⁷. Thollet also proposed a lift-drag analogy model for the parallel and normal force contributions. However, these two approaches require a more complex calibration than the Hall-Thollet model. Thollet compared the performance of these models for the operation under highly distorted flows with higher order references as URANS (Fig. 2.15) and obtained encouraging results with an adequate representation of the radial and sectoral distortion distribution¹⁷.

Another explicit formulation of the body force model is the Immersed Boundary Method with Smeared Geometry (IBMSG) proposed by Cao et al¹¹³. In this formulation the turning of the flow is defined by the definition of a camber surface and a deviation that the flow is forced to follow. The camber surface is smeared into each of the cells of the domain to guide the flow rotation. An immersed wall boundary condition is the applied to the smeared geometry. For the losses, Cao¹¹³ proposed

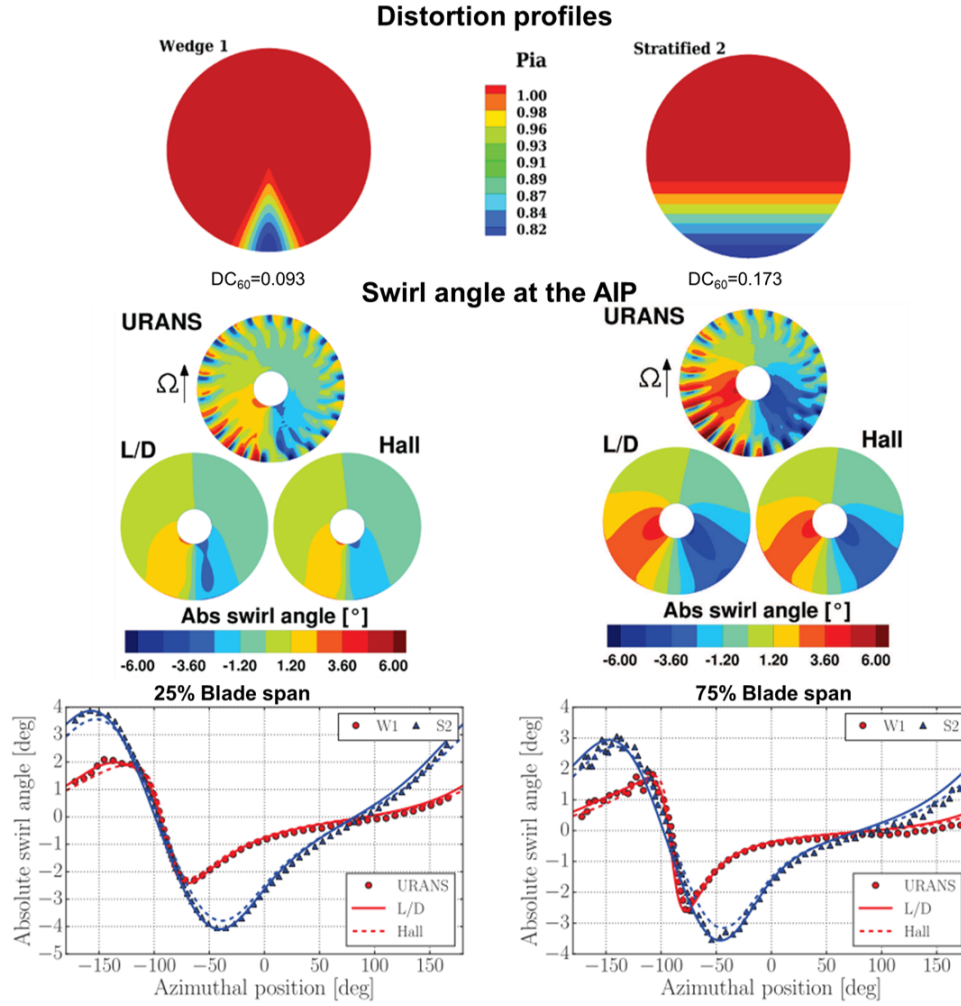


FIGURE 2.15: Comparison of the lift-drag and Hall modified methods proposed by Thollet¹⁷ with a mixing-plane URANS fan modelling approach. Evaluation of the swirl angle at the AIP for different sectoral distortion profiles for BLI configurations

an explicit formulation of the parallel force based on three calibration variables. The main advantages of this implementation are the control of the flow angle, the simplification of the normal force component, and the simplicity of the implementation within existing CFD solvers. The main complication of this approach is the complex calibration based on three coefficients for the different loss sources.

A notable exception to the application of Marble's formulation in low order turbomachinery models are the actuator disk methods. These methods are based on the Rankine-Froude Blade-Element momentum theory¹¹⁴. In this formulation, surfaces of discontinuity on the fluid domain are applied in replacement of the blade elements. On these discontinuities the fluid properties are modified according to the turbomachinery effects. A clear advantage of these approaches is that they can be adapted

to represent different levels of complexity in the stage model. Different enthalpy and entropy rise profiles can be applied, as well as circumferential and radial variations to adjust the turning losses. Also, the application of a surface of discontinuity requires a notably lower relative computational cost than some other throughflow approaches that require higher refinement on the turbomachinery region. For these reasons, this approach has been applied for preliminary evaluations of the embedded propulsion systems^{61,90,115,116}. However, the actuator disk approaches present some inconveniences for a more detailed evaluation of the close-coupled propulsion systems. The representation of the turbomachinery effects are bounded to the prescribed changes of the magnitudes of interest at the discontinuity surfaces. Also, the actuator disk approach cannot model some physical phenomena of interest for the turbomachinery performance, such as the redistribution of the mass flow, the blockage effects or the emergence of shock waves on the blades. Thus, these models are limited in their representation of the changes in the stage performance to strong variations of the distortion at the intake. In addition, since the method was originally developed to model propeller and helicopter blades, some convergence issues may be observed with blades with low diameter-to-chord ratios and high pressure ratios¹⁷.

Other throughflow low order turbomachinery models can be found in the literature to increase the accuracy or reduce the computational cost. For the reduction of the computational cost, some approaches that combine three dimensional flow with a 2D axisymmetric turbomachinery formulation can be found (both body forces based or streamline curvature based). These formulations can be cost effective, but the accuracy of the turbomachinery response is substantially reduced. Nevertheless, in cases in which the accuracy of the turbomachinery response is not paramount, these models can enable a wide range of computational studies with fewer computational resources. On the other hand, less cost effective and more complex approaches have also been proposed to define additional physical phenomena derived from the turbomachinery and the unsteady effects of the turbomachinery stage. This is the case of the 3D throughflow methods with wakes¹⁷. These methods have a reduced computational cost compared with the unsteady full stage geometry approaches. However, they have a much higher cost compared with the other throughflow methods. For this reason, they are not considered in the present work.

2.4 Propulsion integration aerodynamics

Up to this point, the different novel aircraft configurations have been introduced, the challenges for the aerodynamic designs of such configurations have been evaluated, and a discussion of the current methods for the evaluation of the aerodynamic performance and the aero-propulsive coupling has been provided. This has been used to identify some of the most adequate methodologies to develop a strategy for the aerodynamic design of novel aircraft configurations. In this section, a discussion of the different studies to date about the impact of the propulsion integration on the aerodynamic performance is provided. The aim of this section is to determine the current gap in the knowledge about the aerodynamic design of embedded propulsion systems. For further context, the state of the art of the aerodynamic design of the propulsion integration of conventional podded systems is also analysed. This analysis is useful to map the current knowledge on the aerodynamic design of the propulsion integration to the novel requirements and identify possible common characteristics.

2.4.1 Propulsion integration for conventional podded aero-engines

The propulsion integration design of conventional wing-mounted aero-engines for long and medium range aircraft is also moving towards more closely coupled airframe propulsor designs. This is due to the evolution of the technology towards ultra-high bypass ratio engines (UHBR) to maximize the propulsive efficiency and reduce the fuel burn¹¹⁷. The closer integration is then a requirement derived from the increase on the engine radius, since the engine must be moved closer to the wing to keep the ground clearance requirement¹¹⁸. At the same time, there is a growing interest on the application of a more compact engine cowl to reduce the nacelle drag increased by the rise on the fan diameter¹¹⁹. This created new challenges for the fan-cowl, the intake and the exhaust design (Fig. 2.16). The level of aerodynamic coupling of these configurations is not comparable with the expected for the embedded propulsive systems. However, to contextualise the requirements for the design of the novel embedded propulsors, it is of great interest to understand how the challenges in the design of the propulsion integration of wing-mounted podded propulsors are addressed.

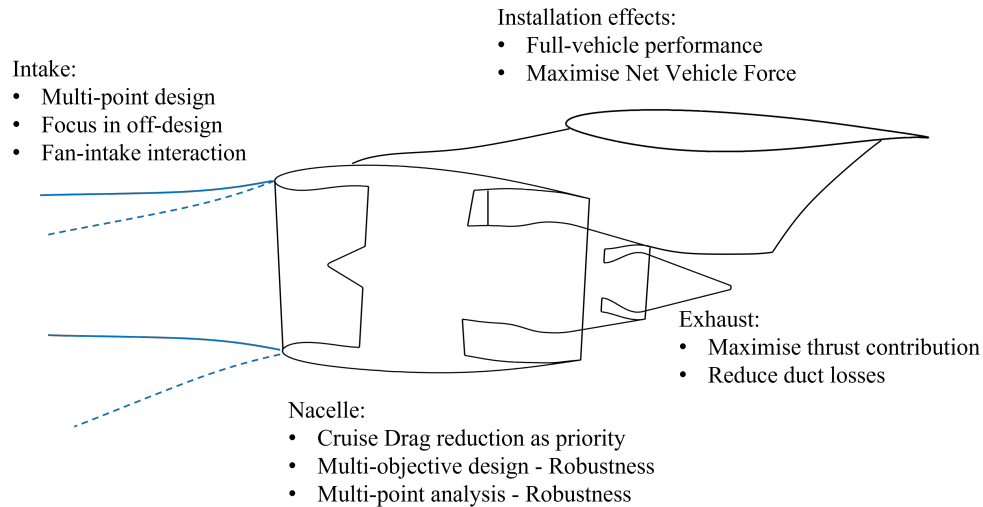


FIGURE 2.16: Summary of the key design factors for the individual housing components of a conventional wing-mounted podded propulsor

The intake design is a critical part of the propulsion integration. The intake will determine the fan operating conditions across the flight envelope and the geometric constraints for the nacelle drag reduction. The aerodynamic design of the intake is normally evaluated based on the performance under off-design critical condition¹²⁰ (e.g. maximum take off angle of attack or crosswind). On the transition to more compact cowlings systems, the intake length over fan radius is reduced substantially in comparison with previous engine architectures^{38,119}. This can reduce the intake operation range if the detailed design is not considered from the preliminary design stage. Also, the intake-turbomachinery coupling increases with the reduction of the intake length^{92,93}. Thus, it is essential to evaluate the viability of the intake design from the early design stages. This involves maintaining an acceptable level of flow distortion at conditions such as take-off at maximum angle of attack and crosswind, where the possible flow separation at the intake can increase the sectoral distortion. The threshold for the distortion limits is determined by the fan design and its response to the non-uniformities of the flow. These requirements can also be extrapolated to embedded propulsion systems, where the increase on the turbomachinery-intake coupling and the rise of the sensitivity to the operating conditions are expected to play a major role. Historical cases illustrate the importance of considering the intake design from the preliminary design stages for more embedded systems. A clear example is the GR4 Tornado¹²¹, where an inadequate design of the embedded intake lead to important engine problems at medium Mach number (0.7) operation, and engine stall for supersonic conditions. Hence, to avoid costly redesigns on later design stages, the intake performance must be addressed

for all the critical operating points on the preliminary phases of the design.

On the nacelle side, notable benefits in the aerodynamic performance of the propulsion integration can be obtained through the reduction of the cowl length¹¹⁹ and the detailed design of the nacelle geometry^{35,38}. These benefits on the aerodynamic performance translate directly to benefits on the fuel consumption¹²². Similarly to the intake operation, the nacelle aerodynamic performance has to be evaluated for a wide range of operating conditions. Nevertheless, to obtain significant aerodynamic benefits for the medium and long range aircraft configurations the most important operating point is during cruise operation³⁵. However, the designs need to demonstrate robust operation for changes on the operation condition. For this reason, the sensitivity to changes in the Mach number and the mass flow capture ratio from the cruise conditions are also evaluated. Hence, the nacelle design should be addressed from a multi-objective point of view based on the changes on the operating conditions²². Off-design conditions such as wind-milling and end-of-runway are also critical in terms of nacelle operation⁹⁴. Under these conditions, stable operation without aggressive flow separation in the nacelle must be achieved. Thus, it is also of interest to consider the multiple operating points in the multi-objective design process of the nacelle.

Under these conditions, the manual aerodynamic design of the nacelle is no longer achievable. More advanced design techniques such as multi-objective optimisation are required to evaluate the nacelle performance under these wide range of operating conditions^{22,35,95}. However, the computational cost required to evaluate the performance of the nacelle design installed on the aircraft can be prohibitive. Thus, it is convenient to approach the design problem from a hierarchical perspective with different degrees of fidelity. A progressive increase on the multi-objective optimisation design process from 2D axisymmetric nacelles to 3D isolated was performed by Tejero et al (2019³⁵, 2020¹²³, and 2022⁵⁰). Tejero et al¹²⁴ also evaluated the effect of the installation position on the aerodynamic performance of nacelles optimised by the isolated nacelle. Magrini et al⁹⁵ applied a similar approach to progress from 2D axisymmetric to 3D isolated and installed nacelle designs. The hierarchical approach provides significant advantages for the computational cost reduction. This approach could be of interest for some of the aerodynamic design of embedded propulsion systems, such as the aft-mounted annular propulsor (Fig. 2.7) where the radial flow component may dominate over the annular flow effects. For other embedded propulsion systems, some dimensionality simplification could be applied (e.g. the distributed propulsion concepts on the wing, or blended wing body configurations) but the flow mechanisms and fan performance are expected to be highly

three-dimensional. Also, for the embedded propulsive systems due to the propulsor-airframe coupling the isolated analysis is not feasible. Thus, a substantial increase in the complexity of the design process and computational cost is to be expected.

The exhaust architecture is also a critical part of the propulsion integration design. The increase of the bypass ratio (BPR) for the future generations of conventional wing-mounted power-plants is accompanied with a reduction on the fan pressure ratio to reduce the specific thrust and increase the propulsive efficiency¹²⁵. Goulos et al³⁶ noted that the increase on the BPR will lead to an increase of the gross propulsive force to net propulsive force ratio, with an increase on the requirement of gross propulsive force. The gross propulsive force is linearly dependent on the aerodynamic performance of the exhaust. Thus, it is essential to maximise this metric by the reduction of the viscous losses in the exhaust walls. The conventional exhaust design has been based on axisymmetric exhaust geometries. This simplifies the computational evaluation of the exhaust design. However, the exhaust design conventionally requires a high level of detail that leads to a high dimensional design space¹²⁶. Goulos et al³⁶ developed a 2D axisymmetric design methodology for the exhaust design. Goulos et al¹⁵ also studied the impact of the installation position on the exhaust aerodynamics for a down-selected design based on the aforementioned design methodology. In the same work, Goulos developed a thrust-drag bookkeeping methodology to account for the exhaust effects. Otter et al¹²⁷ proposed a non-axisymmetric exhaust design methodology to improve the gross propulsive force contribution of installed configurations with reported benefits of around 0.6% to 0.8% of the net vehicle force. However, similar to the nacelle analysis, the optimisation of the installed exhaust systems requires excessive computational cost.

The main limitation for the further optimisation of the installed power-plant aerodynamic performance is the increase of the computational cost. For this reason, some work on multi-fidelity analysis has been carried to mitigate the computational requirements in works such as Hueso-Rebassa et al¹²⁸ and Tejero et al¹²⁹. In the latter work, Tejero et al combined the multi-fidelity approach with dimensionality reduction techniques to optimise the net vehicle force of the system. Another approach applied to reduce the computational cost of multi-objective optimisation of the propulsion integration is the application of surrogate models. Goulos et al¹²⁶ applied Gaussian Processes Regression (Kriging Interpolation) trained with an initial CFD design space exploration to optimise a separated-flow exhaust system. Tejero et al also applied Kriging Interpolation to build surrogates for the nacelle multi-objective optimisation of 2D axisymmetric¹³⁰ and 3D isolated configurations¹³¹. The application of similar design techniques could simplify the design process for the

embedded propulsion systems. However, for the application of such techniques to embedded propulsion systems it is necessary to first understand the coupling and dependencies between the different housing components. Therefore, a detailed analysis of the impact of the housing components on the aerodynamic design is required. Also, the interaction between these housing components should be identified.

In summary, for an integral analysis of the propulsion integration of both the conventional wing-mounted and novel embedded power-plants, the robust operation under different conditions should be addressed. However, the computational cost increases substantially when more detailed aerodynamic effects are considered. A hierarchical approach with different levels of fidelity is not always possible for the design of embedded propulsive systems. However, for configurations such as the aft-mounted annular propulsor (Fig. 2.7) it could provide relevant information about the design space at a reduced computational cost. Additionally, single-point analysis can provide relevant information if the intake, nacelle and exhaust performance is evaluated simultaneously. Multi-objective analysis and optimisation provides an interesting framework to allow a systematic analysis of the effect of the propulsion integration design on the performance of the individual housing components. For the embedded propulsive systems, the airframe-propulsor coupling must be addressed, while on conventional configurations it can be overlooked for some housing components. Nevertheless, for final aerodynamic the effects of the integrated power-plant the overall efficiency must be accounted. Finally, some design techniques for the conventional podded configurations could be applied for the aerodynamic design of embedded configurations with a reduced computational cost. However, first a better understanding of the effect of the individual housing components of the embedded configurations is required.

2.4.2 Propulsion integration of novel aircraft configurations

A core part of the work on the propulsion integration of embedded systems has focussed on the analysis of boundary layer ingestion systems (BLI). This includes part of the BWB concept, the multiple aft-mounted propulsors and the aft-mounted annular propulsor configurations (Fig. 2.1). It is a complex task to encompass all the work performed on the investigation of such configurations during the last years. Diamantidou et al⁶² and Mendegozzo et al¹³² carried out intensive literature analysis of the different approaches in the model of the BLI systems. Both works were able to identify and summarise the main contributions to the modelling and the evaluation of such configurations from a multi-disciplinary perspective. The scope

of this section is somehow less ambitious, and the literature will be focused on the identification of the studies that cover the aerodynamic analysis of the propulsion integration of such novel systems. It was found that the approaches for the analysis of the impact of the propulsion integration have been following a natural progression based on the design stages of the novel aircraft concepts. The design process has evolved from the quantification of the system benefits with decoupled aerodynamics in the conceptual design stage to the detailed aerodynamic design of the embedded configurations on more advanced preliminary design stages. Nevertheless, the actual quantification of the impact of the aerodynamic design of the propulsion integration is yet to be addressed. Some progress has been made towards this direction in recent years. A summary of the progress towards that goal is presented in this section.

First, on the conceptual design stage of the different embedded propulsive system concepts a decoupled aerodynamic approach was mostly applied. These decoupled approaches were presented at different levels. Seitz and Gologan⁴⁴ proposed a fully decoupled approach to quantify the benefits of the aft-mounted annular BLI propulsors based on analytical correlations for the aerodynamics of the embedded propulsor. This approach included the effects of the amount of momentum flux deficit ingested by the BLI system. Similarly, Welstead and Felder¹ applied a decoupled propulsor aerodynamics approach for the STARC-ABL system to quantify the benefits on the BLI propulsor based on the boundary layer ingested. Welstead and Felder found benefits of up to 15% of the cruise specific fuel consumption by the application of the aft-mounted annular propulsor. Felder et al⁶⁰ and Gladin et al¹³³ applied a similar approach for the conceptual design of the N3X configuration. With the uncoupled approach, Gladin et al¹³³ identified benefits of between 2.4 and 6% of the specific fuel consumption depending on the operation fan pressure ratio. Additional uncoupled approaches to quantify the BLI benefits for the BWB configurations were also proposed by Valencia et al¹³⁴ and Goldberg et al¹³⁵. Drela² proposed a preliminary analysis of the benefits of the D8 configuration with an uncoupled aerodynamics multi-disciplinary approach. Drela claimed benefits of up to 33% in fuel burn when compared to an equivalent wing podded configuration. All these methods provide the groundwork to understand the benefits of the BLI systems. However, the application of the uncoupled methodology neglects the actual interaction between the airframe and the propulsor. Hence, a more detailed aerodynamic analysis of the aero-propulsive coupling is required.

After the conceptual design stage, more detailed aerodynamic studies have emerged to provide a better view of the importance of the aerodynamic coupling between the airframe and the propulsor. With the rise of the fidelity of the aerodynamic analysis,

a reduction of the predicted benefits of the embedded propulsor configurations was observed. Pandya et al¹³⁶ performed a CFD analysis of the benefits of the MIT D8 configuration with a coupled airframe-propulsor analysis and an actuator disk to evaluate the propulsor effects. Pandya et al¹³⁶ obtained only a 9% benefit on the required mechanical energy from the potential 33% proposed by Drela². This was later confirmed by low-speed wing tunnel experiments carried by Uranga et al¹³⁷, with a benefit of 8.4% on the mechanical power. However, Uranga et al¹³⁷ also claimed that the benefits in the power reduction could rise to a 15% for a full sized configuration.

The aerodynamic interaction between the propulsor and the airframe was found to have a notable impact on the estimation of the system performance by separated works such as Gray et al⁹⁶ and Ahuja^{59,138}. Ahuja studied the effect of the level of detail of the fuselage geometry in the evaluation of the propulsor aerodynamics of different aft-mounted BLI propulsion concepts using CFD. Gray et al⁹⁶ developed a fully coupled design methodology to account for the propulsor-fuselage interaction of the STARC-ABL configuration. Furthermore, Gray et al⁹⁶ insisted on the need to account the effects of the aerodynamic design of the propulsion systems when the benefits of the novel configurations are accounted. Gray et al extended this work to optimise the aerodynamic performance of the propulsion system of the STARC-ABL under different levels of complexity^{11,13}. Gray et al.¹¹ proposed a CFD based fully-coupled aerodynamic single-objective optimisation approach with a one-dimensional cycle model to model the propulsor. Compared with the uncoupled analysis of Welstead and Felder¹ that estimated benefits of 15% of power consumption, the optimisation approach Gray et al quantified a benefit between 6% to 9% on the power requirements with respect to a reference podded configuration. Later, Gray et al¹³ refined the methodology to account to the intake distortion and included the turbomachinery effects by an implicit body force model. By the application of the intake distortion as a constraint in the optimisation process, Gray et al observed a 1.2% penalty of the optimum power saving when compared to an unconstrained case. Thus, just by the evaluation of the aerodynamic performance of the propulsion integration, the expected power savings reduce from 15% to 5-8% approximately. This highlights the importance of a detailed aerodynamic evaluation of the propulsion integration design of the embedded systems to accurately quantify the power savings of the novel configurations.

For highly embedded configurations such as the distributed propulsion in the BWB configurations, special attention has been given to the intake design from the early stages of the concept. Lee et al⁹⁸ proposed an adjoint optimisation of an embedded

BLI intake in isolation and evaluated the intake performance for different off-design conditions. Kim and Liou⁴¹ also proposed a fully coupled optimisation of the propulsion integration the embedded system with a low order propulsor model to account for the intake-turbomachinery interaction. This study combined a high number of degrees of freedom and demonstrated the potential of adjoint solvers for high dimensional problems such as the propulsion integration of novel propulsion systems.

At the same time, while the fan cowl and the intake have been the main focus on the general research, the importance of the exhaust aerodynamics on the performance has been largely neglected. This could be a direct effect of the complexity on the model of the distortion transfer through the turbomachinery. Thus, an significant amount of the research has been focused on studying the turbomachinery^{43,77,78}. Nevertheless, a few exceptions have been found. Kim and Liou^{7,41} optimised the fan cowl and the intake for the N2B and N3X configuration, accounting for the effects on the exhaust with and without distortion transfer. Seitz et al¹¹⁵ proposed a 2D axisymmetric optimisation methodology for the CENTRELINE concept that included changes on the exhaust geometry and distortion transfer with a simplified explicit momentum source term to model the distortion transfer through the fan. Finally, a slightly more focused analysis on the design of the turbomachinery and exhaust systems for an aft-mounted propulsor configuration was carried by Lee et al¹³⁹. Lee et al proposed a methodology for the design of the turbomachinery based on a hierarchical approach to include the distortion transfer. In this process Lee et al proposed a redesign of the tail cone to improve the turbomachinery thrust performance. Besides these studies, to the author's knowledge, there has not been any detailed analysis of the impact of the aerodynamic design and integration of the exhaust on the aerodynamic performance of the embedded propulsion systems.

At this stage, the majority of the analysis about the propulsion integration aerodynamic design have moved towards a fully coupled airframe-propulsor approach based on computational fluid dynamics for the definition of the system performance. The importance of the intake distortion has been accounted and some of the main efforts have been focused on the optimisation of the intake performance^{7,13,41,98}. Also, it was observed how some level of representation of the turbomachinery must be included to quantify the effects of the propulsor. This varied from semi-coupled one-dimensional cycle analysis^{11,96} to fully coupled throughflow models^{13,41}. The increase on the aerodynamic model fidelity presents new challenges for the evaluation of the embedded propulsors, due to the increase in the computational cost. To compensate for the rise of the cost, the studies are being focussed on more narrow aspects of the propulsion integration aerodynamics. In the development of these

advanced design techniques, the quantification of the importance of the individual components on the embedded configurations has been mostly neglected. Similarly, the interaction between the different housing components has not been addressed in detail.

For this reason, some reduced computational cost approaches have been applied to the preliminary design of the novel configurations. Habermann et al⁹⁷ proposed a 2D axisymmetric approach to model the aerodynamic performance of the CENTRE-LINE concept (Fig. 2.7). Habermann et al combined the 2D axisymmetric model with a parametric definition of the geometry to perform a design space exploration of the effects of the bulk parameters of propulsion integration design (fan dimensions, position and fan pressure ratio) on the performance. To achieve this with a reduced computational cost, the design process was aided by a surrogate model approach based on Gaussian Processes Regression. Habermann et al concluded that all the parameters applied in the study affected the aerodynamic performance either in the net vehicle force or in the change on the intake distortion. This approach was later extended by Seitz et al¹¹⁵ to include the distortion transfer. Habermann et al⁹⁷ also highlighted the need of a 3D evaluation to understand the off-design operability of the configuration. This was also performed by Habermann et al⁷⁶ with an evaluation of the effect of the 3D distortion and the upsweep angle of the tail cone propulsor. Habermann et al observed an increase on the sectoral distortion from the radial distortion 2D axisymmetric baseline. Ahuja⁵⁹ proposed a methodology for the conceptual design of BLI systems based on surrogate models with Artificial Neuronal-Networks trained with 3D CFD simulations. However, this last approach could not be extended to detailed aerodynamic design of the propulsion integration.

Two trends can be identified from the aerodynamic analyses of the propulsion integration to date. The first trend is the detailed fully coupled analysis to capture the bulk aerodynamic effects^{13,41}. This has allowed the definition of the methods for detailed optimisation of very particular designs. However, this approach has failed to provide relevant guidelines to the propulsion integration design and the importance of the individual housing components to model the performance (intake, nacelle and exhaust). Furthermore, the approaches to date have failed to apply a combined optimisation approach between the performance of the main components. Thus, it is still required to identify the importance of the coupling between the housing components and the importance of each component on the overall aircraft performance. The second trend is to perform more detailed analyses of the main contributors to the aerodynamic performance, focus on individual parameters but

with a reduced computational cost^{59,97}. This offers interesting opportunities to perform more detailed aerodynamic studies. The studies to date have not quantified the importance of the design of the housing components for the preliminary design. This may have neglected important aspects of the aerodynamic design that can be applied to generate design guidelines for future embedded propulsion design. Thus, a detailed analysis is required to identify the importance of the individual components. Additionally, the interaction between the components must be accounted.

2.5 Gap in knowledge

After the evaluation of the studies to date about the aerodynamics of embedded propulsion, it is evident that the impact of the propulsion integration design on the aerodynamic performance of the novel aircraft concepts, with a few exceptions¹³, has not been sufficiently quantified. The design of the propulsion integration has been studied in different levels of fidelity from one-dimensional to full 3D CFD models. However, the validity of the different degrees of modelling fidelity has not been properly addressed. Hence, a comparative analysis of the modelling requirements for embedded propulsion systems is needed. Furthermore, the design requirements of each of the housing components of the propulsion integration (intake, fan-cowl, and exhaust) have not been properly evaluated. This is necessary in order to provide guidelines for the design of the propulsion integration in future embedded systems. Thus, to create the groundwork for the design of future embedded propulsion configurations, it was considered essential to develop a methodology that accounts for the effect of the propulsion integration in the performance.

The close aerodynamic coupling between the housing components requires the integral design of the propulsion integration. Under the complex aerodynamic conditions of the embedded propulsion systems, the manual design of the propulsion integration is not always feasible. Thus, it becomes crucial to define an optimisation approach to define the aerodynamic design of the propulsion integration. Also, the aerodynamic interaction between the housing components requires a multi-objective design approach. While some combined analyses of the thrust contribution and the intake performance in embedded propulsion systems have been carried out in works such as Gray et al¹³ and Habermann et al⁹⁷, the combined performance of the different housing components has not been studied in the context of a multi-objective optimisation. Furthermore, the aerodynamic design of the exhaust systems of the embedded systems have been largely neglected.

CHAPTER 3

Methodology

The methods applied to determine the aerodynamic impact of the propulsion integration of novel aircraft concepts are described in this chapter. First, the overall strategy behind the design methodology is described in Section 3.1. Subsequently, the individual methods for each stage of the design strategy are defined. The definition of the geometry generation methodology is presented in Section 3.2. The definition of the aerodynamic performance methodology follows in Section 3.3. Then, the numerical methodology for the aerodynamic modelling of the designs is described in Section 3.4. Finally, the definition of the Multi-Objective Optimization (MOO) framework is presented in Section 3.5.

3.1 Overview of the design strategy

The present work evaluates the impact on the aerodynamic performance of the propulsion integration of the novel embedded propulsive systems. This is achieved through the representative case study based on an aft mounted annular propulsor installed on a medium-range single-aisle aircraft¹⁴⁰. The evaluation of the aerodynamic performance of such propulsive configurations requires of a systematic methodology for the design of the propulsion integration. For this purpose, a dedicated software based on a structured design strategy was developed (Fig. 3.1). The software named ANDANTE (ANalysis and Design of Aircraft Novel Technologies) bases the design and evaluation of the embedded propulsor design on four independent steps:

- Parametric geometry generation
- Automated meshing methodology

- Compressible, viscous computational fluid dynamics
- Adapted post-processing methodology for the embedded propulsor

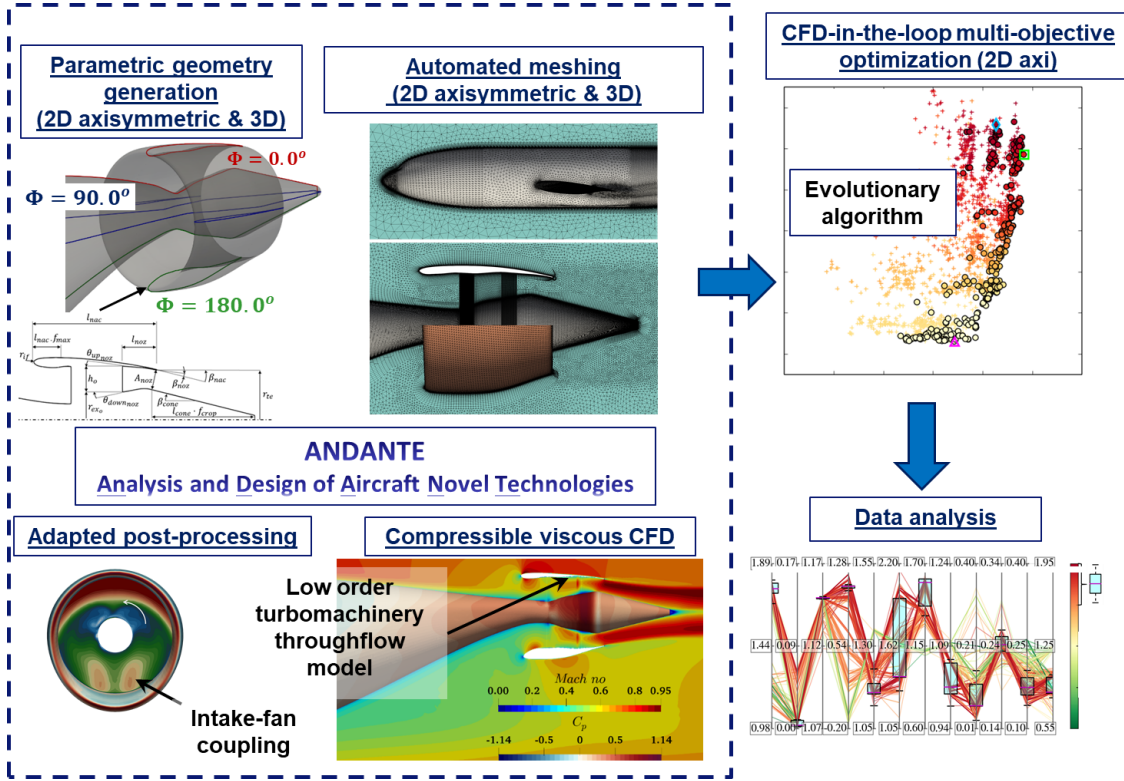


FIGURE 3.1: Framework for ANDANTE (ANalysis and Design of Aircraft Novel Technologies), and coupling with the multi-objective optimization methodology

The application of the defined systematic design strategy allows the comparison between different propulsion integration aerodynamic designs. To utilise this capability the approach described in ANDANTE is coupled with a parametric based Design Space Exploration (DSE) methodology and a multi-objective optimisation methodology (MOO) based on evolutionary and particle swarm algorithms. All the methods for the individual stages of the design methodology in ANDANTE are defined in this chapter, as well as the base methodology for the DSE and MOO.

The complexity and detail of the CFD models varies across the different chapters of the present work (Fig. 3.2). This evolution on the complexity allows a more detailed design and evaluation of the propulsion integration aerodynamics. The different configurations have been grouped based on their dimensionality and the turbomachinery approach applied. The details of the configurations and differences between them are defined in this chapter. They can be divided then into:

- 2D axisymmetric of 3D non-axisymmetric designs

- Semi-coupled one-dimensional turbomachinery model or fully-coupled low order fan model

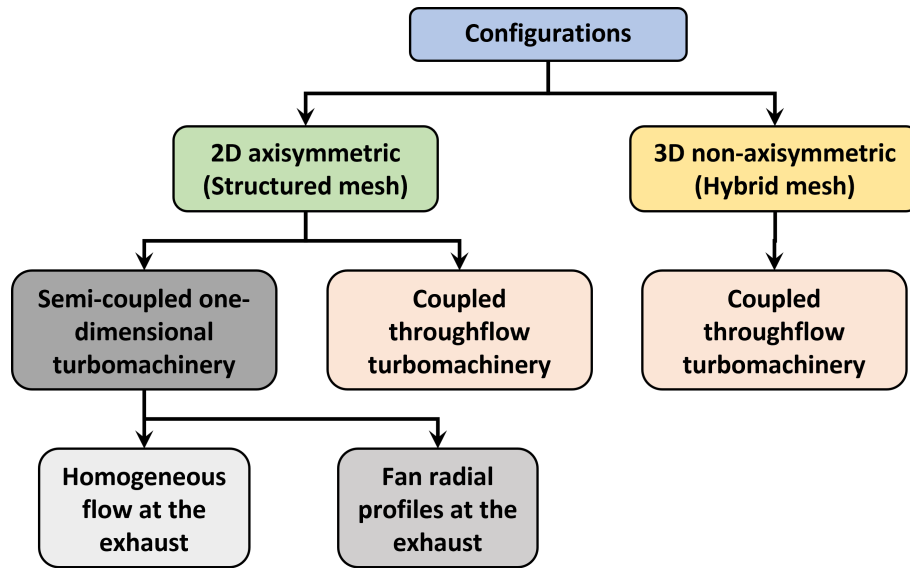


FIGURE 3.2: Available configurations on the present work based on the CFD methodology

3.2 Geometry design and generation

The geometry generation methodology plays a significant role on the development of aerodynamic design methods. The selected approach must provide a systematic process to define the different geometry characteristics of the designs with sufficient accuracy and flexibility. A reduced number of constraints or parameters are desired to minimise the complexity of the model. At the same time, the geometry generation has to be robust and the prescribed parametric space should represent the propulsive system design requirements. Thus, it is required a methodology that fulfils both requirements. To achieve this purpose, the intuitive Class-Shape Transformations method^{38,141–143} (iCST) is applied. This approach has already been successfully applied for the geometric design of the housing components (intake, nacelle and exhaust) of the aero-propulsive installation^{35,36,38} of conventional podded engines. For this work, the geometry parametrisation has been adapted for the design requirements of the embedded propulsion systems.

3.2.1 Intuitive Class Shape Transformations

An intuitive Class Shape Transformation (iCST) methodology is used for the geometry generation^{38,141,143}. The iCST methodology is a modification of the original Class-Shape Transformation (CST) method proposed by Kulfan¹⁴². In this methodology, the target geometries are generated by the definition of a set of geometric constraints. These constraints are controlled by a simple geometric parametrisation.

All the CST based methodologies (Eq. 3.1) are based on the application of a Class Function ($C(\varphi)$) as a baseline geometry that is modified by a shape function ($S(\varphi)$). An additional offset on the trailing edge of the curves can be applied for further modification of the final curve ($\Delta\xi_{te}$). The Class Function is selected to obtain an initial geometric shape similar to the desired geometry (Eq. 3.2). This is achieved by a change on the control parameters (N_1 and N_2) to define polynomials of different order. The Shape Function (Eq. 3.3) changes the the Class function through the application of controlled polynomial distributions. On the current approach, the Shape Function (Eq. 3.3) is defined as a combination of Bernstein Polynomials, BP, (Eq. 3.4) that are adjusted by the modification of the Bernstein Polynomial coefficients ($K_{i,n}$).

$$\xi(\varphi) = S(\varphi)C(\varphi) + \varphi\Delta\xi_{te}; \quad \xi = \frac{y}{c}; \quad \varphi = \frac{x}{c} \quad (3.1)$$

$$C_{N_2}^{N_1} = \varphi^{N_1} [1 - \varphi]^{N_2} \quad for \quad 0 \leq \varphi \leq 1 \quad (3.2)$$

$$S(\varphi) = \sum_{i=0}^n [bp_i K_{i,n} (\varphi^i (1 - \varphi)^{n-1})] \quad (3.3)$$

$$BP(\varphi) = \sum_{i=0}^n [K_{i,n} (\varphi^i (1 - \varphi)^{n-1})]; \quad K_{i,n} = \frac{n!}{i!(n-i)!} \quad (3.4)$$

In the iCST method, the combination of the Class Function and the Shape Function determines the non-dimensional vertical coordinate, ξ , for a given non-dimensional axial coordinate, φ , (Eq. 3.5). A k^{th} number of constraints on the position of those variables can be imposed. However, this only enforces specific radial points for an axial coordinate. To increase the control over the geometry generation, the derivatives of the geometry can also be defined. Therefore, the geometric constraints of the iCST method can be defined by the n^{th} derivative of ξ at a given φ (Eq. 3.6). With the constraints defined, a linear set of equations (Eq. 3.7) is constructed where **A**, **B** and **X** are defined as the matrices of coefficients, constant terms, and solutions, respectively. Each radial position and derivative constraint adds a new equation to

the system. With all the constraints defined, the iCST curve geometry is obtained by solving the linear system¹⁴³.

$$\xi^{(k)}(\varphi) = \left[\sum_{i=0}^n \left[bp_i K_{i,n} \left(\varphi^i (1 - \varphi^i)^{n-1} \right) \right] C(\varphi) + \varphi \Delta \xi_{te} \right]^{(k)} \quad (3.5)$$

$$\frac{d}{d\varphi} BP_{i,n}(\varphi) = n [BP_{k-1,n-1}(\varphi) - BP_{k,n-1}(\varphi)] \quad (3.6)$$

$$\mathbf{A} \cdot \mathbf{X} = \mathbf{B} \quad (3.7)$$

3.2.2 Geometry parametrisation

As it was mentioned in the previous section, the iCST method requires the definition of a set of geometric constraints to construct the geometries. For propulsion integration design, some parametrisations of the geometry have been proposed for each of the individual housing components^{35,36,38,143}. These definitions are used as reference in the present work. Nevertheless, the design of embedded propulsion systems require of an adapted parametric definition. Thus, a novel definition of the parameters is proposed in the present work.

The applied iCST methodology allows the definition of both 2D axisymmetric and 3D non-axisymmetric propulsion integration designs. For this, the geometry construction process is defined in two levels to control the radial and azimuthal variation of the geometries. First, the parametrisation of a two-dimensional aeroline (Section 3.2.2.1) is defined for each housing component (Fig. 3.3). This definition of the geometry constraints (or parameters) is prescribed at fixed azimuthal locations of the propulsive system denominated control lines. Second, the individual parameters of the control lines are applied as constraints for an azimuthal iCST curve to generate a non-axisymmetric 3D surface (Section 3.2.2.8). The intermediate 2D aerolines between the control azimuthal locations are a result of the azimuthal variation of the geometric constraints. As a complement to the parameters of each control azimuthal line, bulk geometric constraints are applied to modify the location and geometric characteristics of the propulsive system.

3.2.2.1 2D aeroline parametrisation

The constraints required for the iCST methodology are defined through a geometry parametrisation that captures the main characteristics of each of the housing components of the embedded propulsion system. The number of parameters was adjusted to provide an adequate and robust representation of the geometry with a reduced number of input requirements. For the reference geometry of the aft-mounted annular BLI propulsor^{26,40} the geometry of the propulsion integration is divided into four housing components: aft fuselage, intake, fan cowl, and exhaust (Fig. 3.3). As a baseline, previous parametrisations of the housing components for conventional wing-mounted power-plants are applied. This are the intake parametrisation defined by Christie et al^{38,143}, the nacelle parametric definition of Tejero et al³⁵, and the exhaust parametrisation proposed by Goulos et al³⁶. In the next sections, these parametrisations are extended for the definition of the geometry requirements of embedded systems.

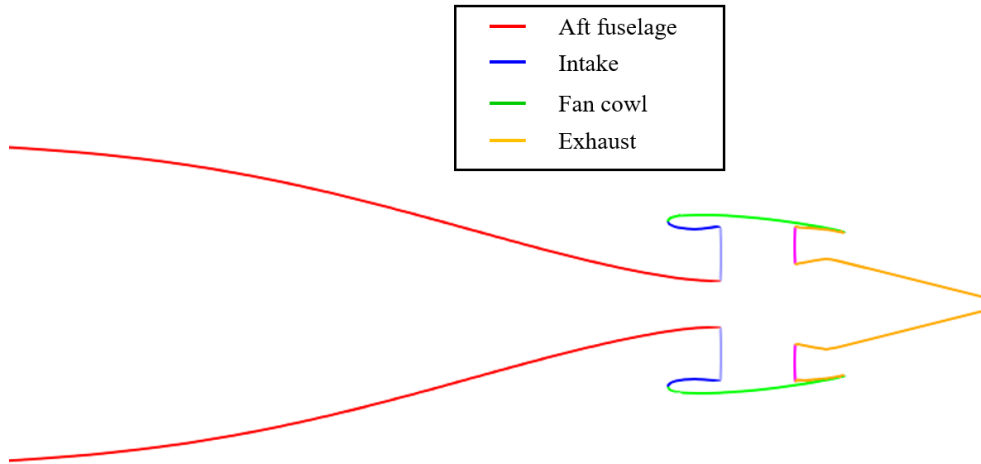


FIGURE 3.3: Definition of the aero-engine installation housing components for an aft-mounted annular boundary layer ingestion propulsor

3.2.2.2 Axisymmetric fuselage geometry

Besides the iCST geometry construction, reference geometric bodies are applied for the non-modified parts (fuselage forebody and mid-body) of the propulsion integration design. For the two dimensional aeroline parametrisation, a reference axisymmetric fuselage geometry is applied (Fig. 3.4). This axisymmetric fuselage is also used on the 2D axisymmetric studies of the present work. For 3D studies, a reference

airframe is applied. This airframe is defined in Section 3.2.2.9. The axisymmetric fuselage is divided into three sections:

- Fuselage fore-body. This section is defined by a semi-elliptical nose. The minor axis is defined by the fuselage radius (r_{fus}). The major axis is defined as a factor (e_{rat}) of the fuselage radius. For the axisymmetric studies of the present work e_{rat} is set to three.
- Fuselage mid-body. This section of the fuselage consist of a straight cylindrical section based on the radius of the fuselage. The extent of the cylindrical section is defined as a factor (f_{eng}) of the reference aircraft length (l_{ac}) from the tip of the elliptical fore-body.
- Fuselage after-body (aft-fuselage). This section of the fuselage is defined by the iCST methodology. The geometry extents from the end of the mid-body to the propulsor hub at the fan face plane (l_{aft}). As a reference the length of the afterbody (l_{ps}) is applied. This metric is also used as a constraint for the maximum length of the propulsive system

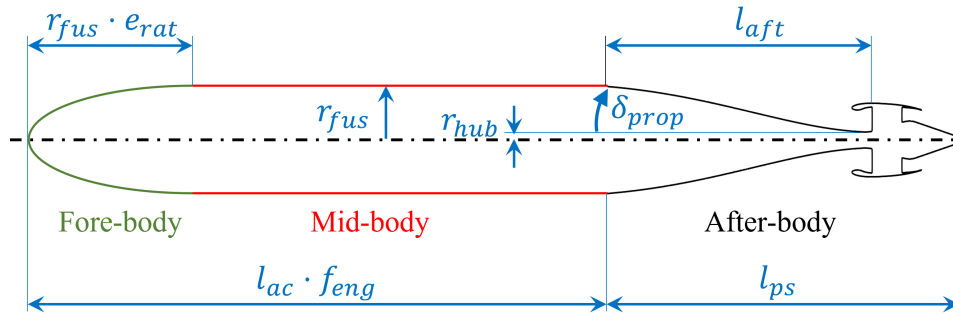


FIGURE 3.4: Axisymmetric fuselage geometry definition (fore-body in green, mid-body in red), and propulsion system location (black). Not to scale

3.2.2.3 Aft-fuselage geometry

A Hybrid iCST (HiCST) approach³⁸ is used for the construction of the aft-fuselage geometry (Fig. 3.5). In the HiCST method, the geometry is divided into a set of individual iCST curves with shared geometric constraints on the common boundaries. Two iCST curves are used for the aft-fuselage geometry based on a line Class Function. The first curve starts from a reference radial location at the end of the fuselage mid-body, point P1 in Fig. 3.5. Curvature continuity between the reference

fuselage mid-body and the aft-fuselage (P1) is imposed by matching the first and second derivatives of the radial component. The curve is extended to a point (P2) at the intake duct where the local slope is imposed (θ_{curv}). The radial and axial location of this point (P2) is defined by an area constraint (A_{thr}) from the intake throat. To apply the area constraint a two dimensional rolling ball method¹⁴⁴ is applied, assuming an axisymmetric distribution of the geometry. The second iCST curve is defined from the end of the first aft-fuselage curve (P2) to the hub location of the fan (P3). The shape of this curve is controlled by θ_{curv} and the slope at the fan face θ_{ff} (Fig. 3.5). The length of both curves is controlled by the aft-fuselage length (l_{aft}). This length is used to prescribe the axial location of the propulsor with respect to the end of the fuselage mid-body. The radial position of the propulsor is defined by the hub radius (r_{hub}). This parameter is commonly normalized as the hub-to-tip ratio ($httr$) with the radius of the fan tip, r_{fan} (Eq.3.8). Finally, the propulsor angle, δ_{prop} , (Eq. 3.9) is derived from the hub location, the length of the aft fuselage, and the fuselage radius (Fig. 3.5).

$$httr = \frac{r_{hub}}{r_{fan}} \quad (3.8)$$

$$\delta_{prop} = \text{atan} \left(\frac{r_{fus} - r_{hub}}{l_{aft}} \right) \quad (3.9)$$

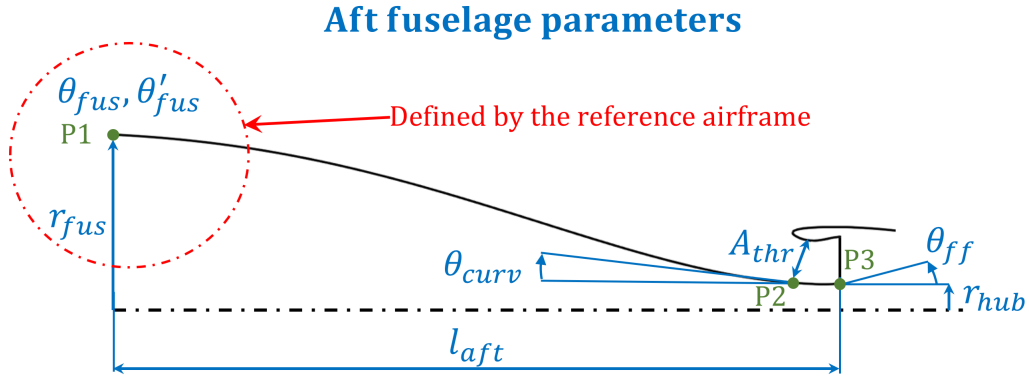


FIGURE 3.5: Geometry parametrisation of the aft fuselage curve

3.2.2.4 Intake geometry

The intake parametrisation (Fig. 3.6) was defined based on the work of Christie et al.³⁸ and adapted for the design requirements of the embedded propulsion system. The intake geometry is defined by a single iCST curve based on an airfoil Class

Function (Fig. 3.3). Contrary to conventional podded intakes where the flow incidence is mainly governed by the angle of attack, on embedded systems the upcoming flow is also defined by the design of the aft fuselage section. The aft fuselage will determine the flow orientation at the highlight plane of the propulsor as well as the local velocity of the boundary layer. Thus, higher control of the intake orientation is required to accommodate the flow at the intake. For this purpose, an intake inclination (Ω_{int}) control was included to correct the highlight orientation (Fig. 3.6). The intake inclination modifies the highlight radial position and the intake curvature distribution. The highlight radial position is modified based on the intake length (Eq. 3.10). The intake curvature distribution is adjusted by the rotation of the relative coordinate system where the iCST curve is created. The intake inclination is combined with the slope of the intake at the fan face (β_{int}) to control the flow orientation into the intake duct. Additionally, three curvature constraints are defined for the intake curve (r_{il} , k_{thr} , and k_{fan}). To reduce the number of parameters, the curvature at the throat (k_{thr}) and the fan face (k_{fan}) are automatically computed to minimize the curvature variation along the intake geometry using a least squares optimization. Finally, the lengths of the intake and the throat are defined by three normalized parameters. The length of the intake l_{int} is normalized with the fan diameter to define the intake aspect ratio, AR_{int} (Eq. 3.11). The throat axial and radial position are defined as a distance to the highlight point by the aspect ratio, AR (Eq. 3.12), and the contraction ratio, CR (Eq. 3.13), respectively.

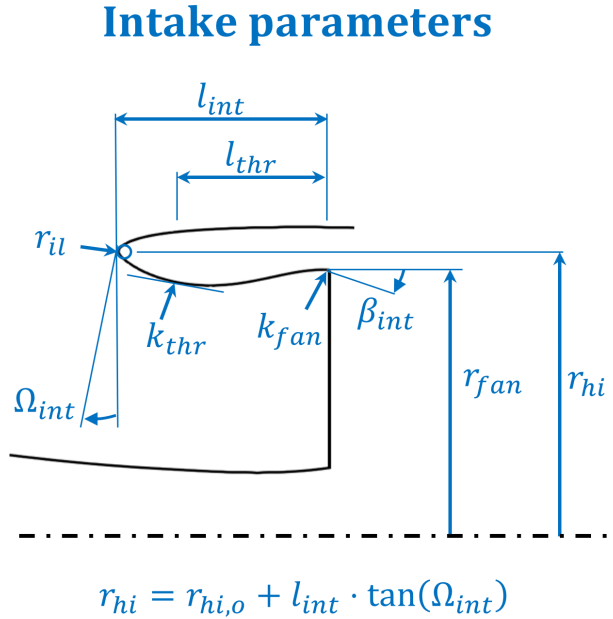


FIGURE 3.6: Geometry parametrisation of the intake

$$r_{hi} = r_{hi_o} + l_{int} \cdot \tan(\Omega_{int}) \quad (3.10)$$

$$AR_{int} = \frac{l_{int}}{2 \cdot r_{fan}} \quad (3.11)$$

$$AR = \frac{l_{int} - l_{thr}}{r_{hi} - r_{thr}} \quad (3.12)$$

$$CR = \left(\frac{r_{hi}}{r_{thr}} \right)^2 \quad (3.13)$$

3.2.2.5 Fan cowl geometry

The parametrisation of the fan cowl (Fig. 3.7) is based on the definition proposed by Tejero et al.³⁵. However, a novel definition of the fan cowl geometry is proposed based on a HiCST approach. The fan cowl is split into two iCST curves at the maximum radius location for the nacelle forebody and the afterbody geometries (Fig. 3.8). This location is defined by the maximum cowl radius value r_{max} and the relative position of the maximum radius with respect the nacelle length (f_{max}).

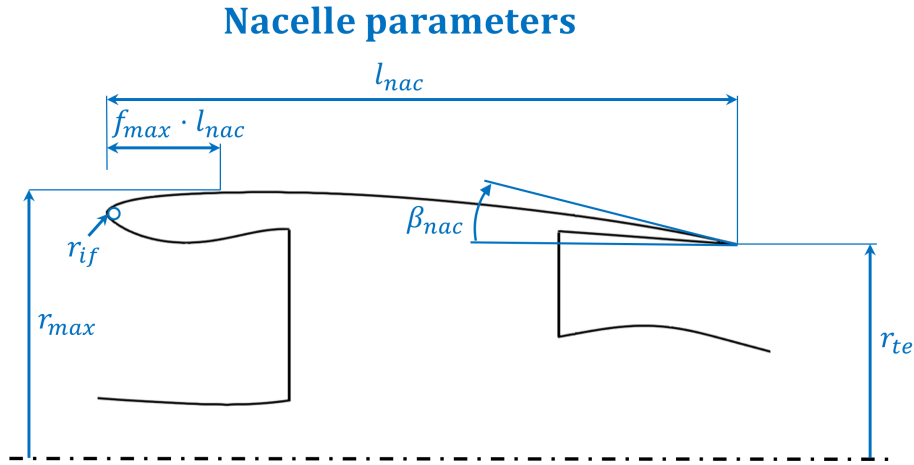


FIGURE 3.7: Geometry parametrisation of the fan cowl

The nacelle forebody iCST curve is set to be independent of the afterbody geometry. The nacelle afterbody iCST is based on an airfoil Class Function constrained by the curvature radius at the highlight, r_{if} , and a fixed slope of value zero at the maximum radius location. The curvature at this point is automatically calculated to obtain the minimum variation of the curvature radius in the forebody with a least squares optimisation.

The nacelle afterbody iCST is based on a straight line Class Function. The geometry is constrained by the inclination at the trailing edge or boat-tail angle (β_{nac}) and

the curvature at the trailing edge. The curvature at the trailing edge is obtained by a least squares minimisation of the curvature variation on the nacelle afterbody. The curvature at the maximum radius point is enforced to be the same as the value applied on the forebody constraint.

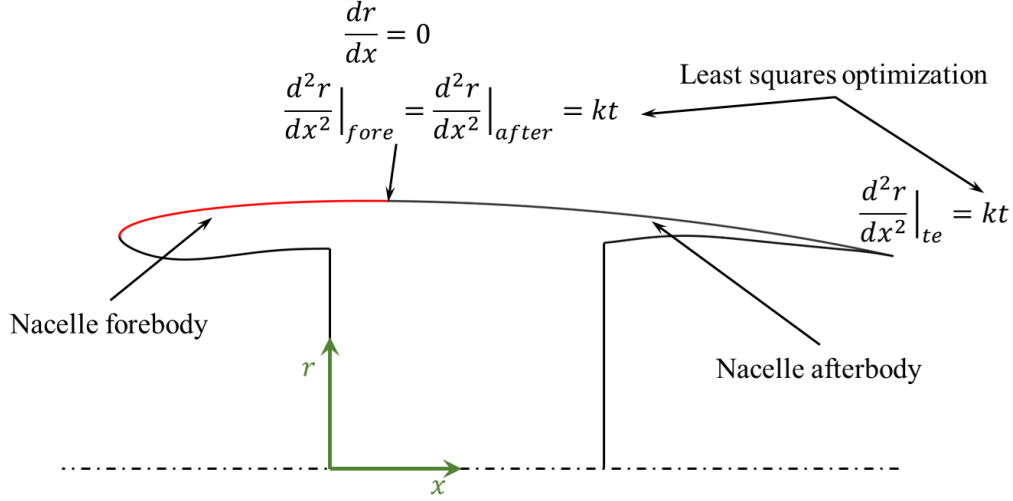


FIGURE 3.8: Hybrid intuitive Class-Shape transformation definition proposed for the fan cowls of embedded propulsion systems

The dimensions of the nacelle forebody and afterbody iCST curves (Fig. 3.8) are defined by the nacelle length l_{nac} and the radial location of the highlight (r_{hi}) and the nacelle trailing edge (r_{te}). The maximum radius of the nacelle can be defined as an increment from the highlight position (Eq. 3.14). Also, the radius of curvature at the intake can be normalized with the nacelle forebody dimensions to define f_{if} (Eq. 3.15).

$$\Delta r_{max} = r_{max} - r_{hi} \quad (3.14)$$

$$f_{if} = \frac{r_{if} \cdot f_{max} \cdot l_{nac}}{(\Delta r_{max})^2} \quad (3.15)$$

3.2.2.6 Exhaust geometry

The parametrisation of the exhaust system is inspired on the work of Goulos et al³⁶. However, the number of parameters has been reduced to define a single stream convergent exhaust system with a conical plug (Fig. 3.9). The exhaust duct is divided into two iCST curves based on a line Class Function: the upper exhaust surface curve (E1-E2) and the lower surface curve (E3-E4).

The geometry of the upper exhaust (E1-E2) duct iCST curve is controlled by two angles: the angle of the exhaust at the trailing edge (β_{noz}), and the slope angle at

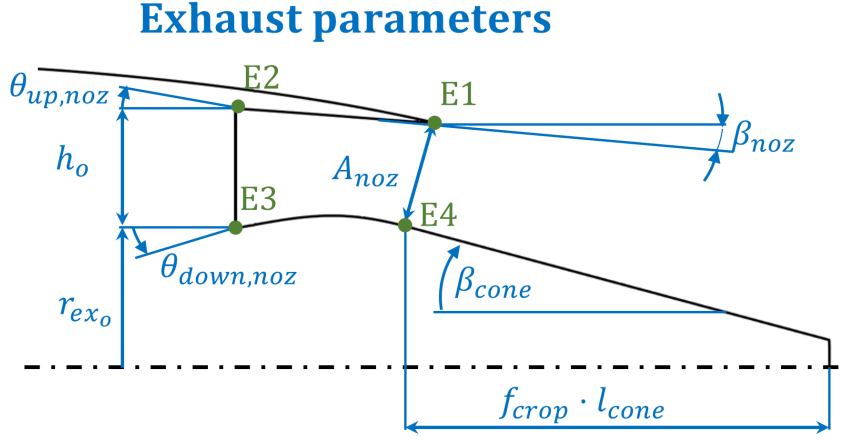


FIGURE 3.9: Geometry parametrisation of the exhaust

the OGV outlet ($\theta_{up,noz}$). This curve is constrained vertically by the radial position of the trailing edge (r_{te}) and the position of the tip of the OGV outlet ($h_0 + r_{exo}$). The length of the upper exhaust duct iCST curve is defined with the nozzle length (l_{noz}).

The constraints of the lower exhaust duct iCST curve (E3-E4) depend on the geometry of the upper exhaust duct. A rolling ball method¹⁴⁴ based on a 2D axisymmetric nozzle outlet area definition (A_{noz}) is applied. Hence, the length of the lower exhaust iCST curve is defined by A_{noz} , the nacelle trailing edge position (r_{te} and l_{nac}), and the angular constraints of the upper and lower nozzle curves at the discharge plane (β_{noz} and β_{cone}). The only remaining constraint for the lower fuselage iCST is the angle at the OVG hub $\theta_{down,noz}$.

In addition to the two iCST curves, a conical exhaust plug is included to the exhaust definition (Fig. 3.9). The geometry of the plug is defined as a straight line from the lower exhaust iCST curve with a prescribed slope (β_{cone}). The cone is cropped at a given length. That length is defined as a fraction of the total cone length (f_{crop}).

3.2.2.7 Turbomachinery domain

Two definitions of the turbomachinery domain are used in the present work:

- **Semi-coupled one-dimensional propulsor approach.** This approach does not include the internal design of the turbomachinery. This is not required because the effects of the turbomachinery are defined through a one-dimensional model. The limits of the excluded turbomachinery domain are defined from

the fan face to the OGV outlet (Fig. 3.10). The length of this section (l_{stage}) is the remaining of the nacelle length (l_{nac}) after subtracting the intake and the nozzle lengths (l_{int} and l_{noz}).

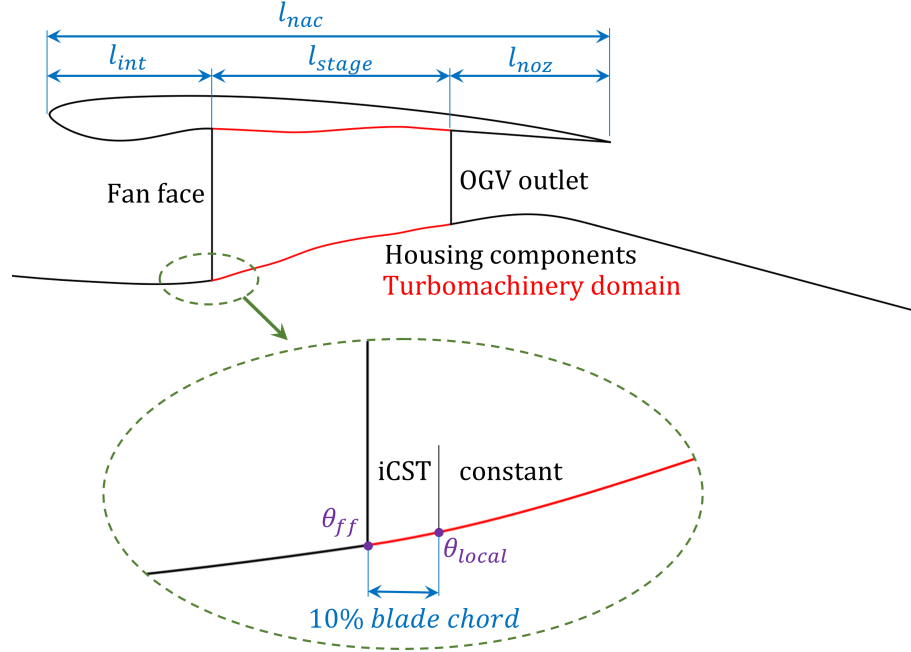


FIGURE 3.10: Definition of the turbomachinery domain and the local iCST curve to match the housing components with the fixed stage geometry

- **Fully-coupled throughflow turbomachinery approach.** This approach includes the turbomachinery domain without the discrete blade and vane geometries (Fig. 3.10). The domain is defined from the reference fan configuration of the present work²³, but can be modified for other designs. This domain includes the scaled fan and OGV domain of the configuration. The domain is extended 10% of the blade chord upstream of the fore most fan blade coordinate and 10% of the blade chord from the aft most vane coordinate. The hub-to-tip ratio is kept constant. To match the geometry transition from the aft fuselage curve, the intake and the exhaust geometries to the propulsor domain, a second order iCST curve (Fig. 3.10) is applied based on the local slope constraints (θ_{ff} , β_{int} , θ_{upnoz} , and $\theta_{downnoz}$).

3.2.2.8 3D geometry parametrisation

The parametrisation of the housing components of the aero-engine installation has been defined for two-dimensional aerolines. An additional step is required to define

three dimensional geometries based on this parametrisation. The approach is based on an azimuthal variation of the 2D aeroline parametrisation. A fixed number of lines at defined azimuthal locations (Fig. 3.11) is selected. The 2D parametrisation is enforced in those lines. These lines are used as control lines for the rest of the azimuthal definition of the geometry.

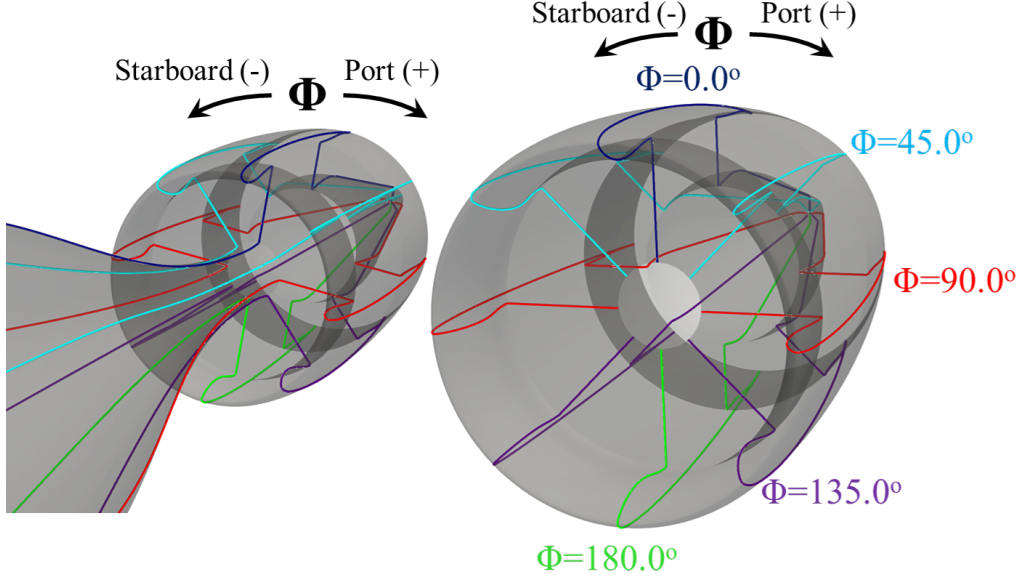


FIGURE 3.11: Definition of the five azimuthal control lines of the propulsion integration design. Nomenclature defined from 0 to 180 degrees. Positive towards the starboard side of the aircraft and negative towards the port side. Left right symmetry on the plane containing $\Phi = 0.0^\circ$ and $\Phi = 180^\circ$

Five individual control lines are applied (Fig. 3.11) in the starboard (positive) side of the aircraft. These control lines are defined in 45° intervals between 0.0° and 180.0° . The symmetry plane of the airframe (XZ as defined in Appendix A) is applied as the symmetry plane for the housing components. Thus, the five control lines are replicated on the port side of the aircraft (negative). This approach supports the reduction of the number of control lines to cut down the number of design parameters (20 per aeroline). The minimum required number of control lines is three, with lines at 0.0° , 90.0° , and 180.0° .

Each of the individual parameters of the intermediate aero-lines are defined with a second order azimuthal iCST curve (Fig 3.12). Curvature continuity is enforced at the control points. Two approaches are applied for the different parameters based on the geometry requirements. The first approach (Ap. 1) uses three iCST curves for the five control points (Fig 3.12). The limits of these curves are defined at azimuthal positions of 0.0° , 90.0° , and 180.0° . At these points, the azimuthal derivative of the

parameter is set to zero (i.e. local maximum or minimum), while the intermediate control lines (45.0° and 135.0°) are just used as position constraints. The second approach (Ap. 2) uses five iCST curves for the five control points (Fig 3.12). The azimuthal derivative of the parameter is set to zero at the limits of each curve.

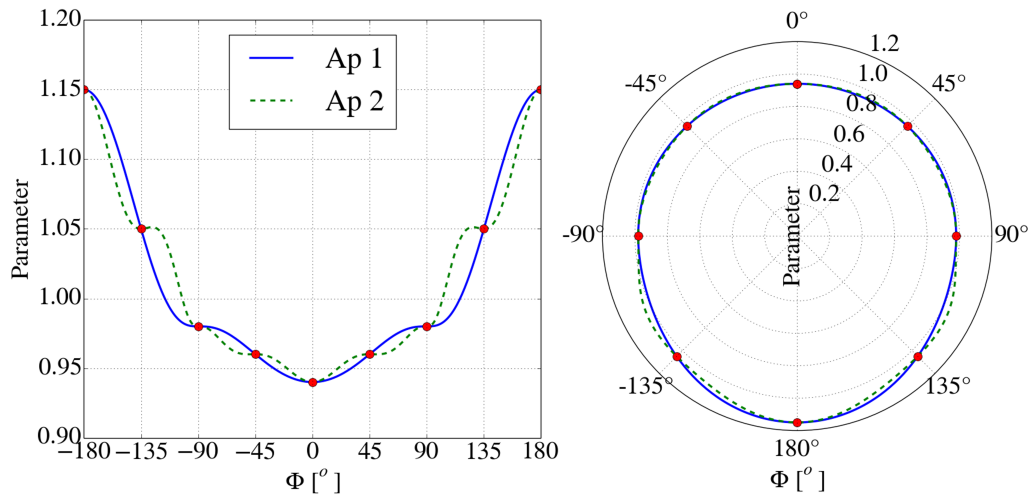


FIGURE 3.12: Sample azimuthal variation of a general parameter with a second order iCST definition. Control points in red

The application of five control lines provides a more detailed representation of the 3D geometry of the intake, nacelle and exhaust. However, in some conditions it was observed to over-constrain the aft-fuselage geometry. For this reason, a semi-coupled approach of the control lines of the aft fuselage and the remaining housing components is proposed. In this approach, azimuthal distribution of the geometry of the aft fuselage is controlled with three control lines (0.0° , 90.0° , and 180.0°), which are coupled with the control lines of the intake at the same azimuthal location. The aft-fuselage geometry is generated based on these three control lines. Then the remaining housing components (intake, nacelle and exhaust) are defined using the five control lines definition.

3.2.2.9 Reference 3D airframe and position of the

The DLR-F6 with the FX2B (F6) body fairing is used as the reference configuration¹⁹ (Fig. 3.13). The DLR-F6 is based on a medium range configuration with a cruise Mach number of 0.75 and a nominal lift coefficient(C_L) of 0.5, which is achieved at approximately 0.25 degrees of angle of attack. The FX2B fairing is an incorporation to the DLR-F6 airframe proposed in the Third NASA Drag Prediction

Workshop¹⁹, to improve the aerodynamic operation of the airframe. The airframe is scaled to the length of a reference medium-range single-aisle aircraft.

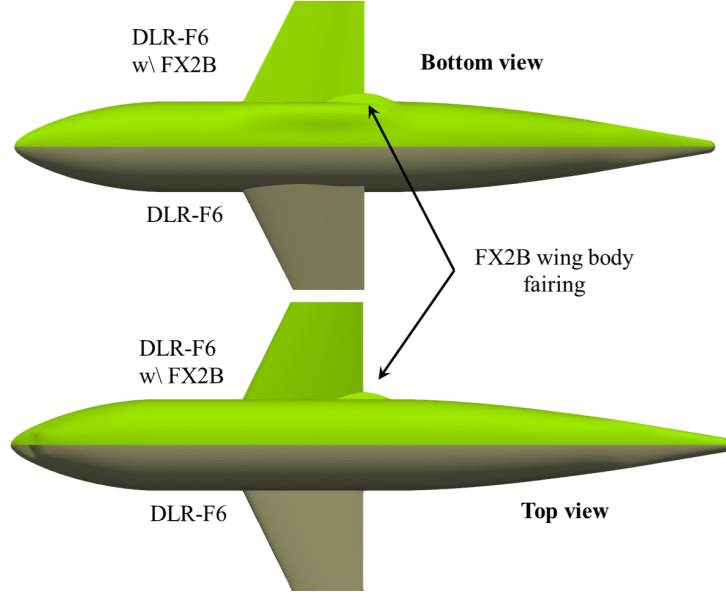


FIGURE 3.13: Comparison between the original DLR-F6¹⁸ and the DLR-F6 with the FX2B fairing¹⁹

To match the aft fuselage geometry with the rest of the airframe, a transversal cut of the geometry is defined at a distance $(l_{ac} \cdot f_{en})$ from the fuselage nose. Based on the central axis of the airframe (Fig. 3.14), the cross section of the fuselage is projected into cylindrical coordinates. The constraints $(\theta_{fus}$ and $\theta'_{fus})$ for the aft fuselage geometry (Fig. 3.5) are obtained with the axial derivative of the radial component (r) at a given azimuthal location (Φ) as defined in Fig. 3.14. A second order centred finite differences scheme is used to obtain the local derivatives. Additional definitions of the relative reference frames can be found in the Appendix A.

For the 3D configurations, the relative position of the propulsive system with respect to the aircraft in the vertical (XZ) plane (Fig. 3.14) is controlled with similar parameters to the 2D axisymmetric fuselage $(l_{ac}, f_{eng}, l_{tc})$. The location where the aft fuselage geometry modification starts is defined as a fraction of the fuselage total length (f_{eng}) . Then, the axial location of the propulsor is defined with the length of the aft fuselage curve relative to the end of the aircraft mid-body. The main addition to the 3D non-axisymmetric position control is the vertical displacement from the aircraft belly (h_{off}) . This value is controlled as a vertical offset and allows to control the ground clearance of the aircraft afterbody.

The relative orientation of the engine central axis to the aircraft central axis is controlled with the propulsor tilt angle, Ω_{engine} (Fig. 3.15). From the engine central

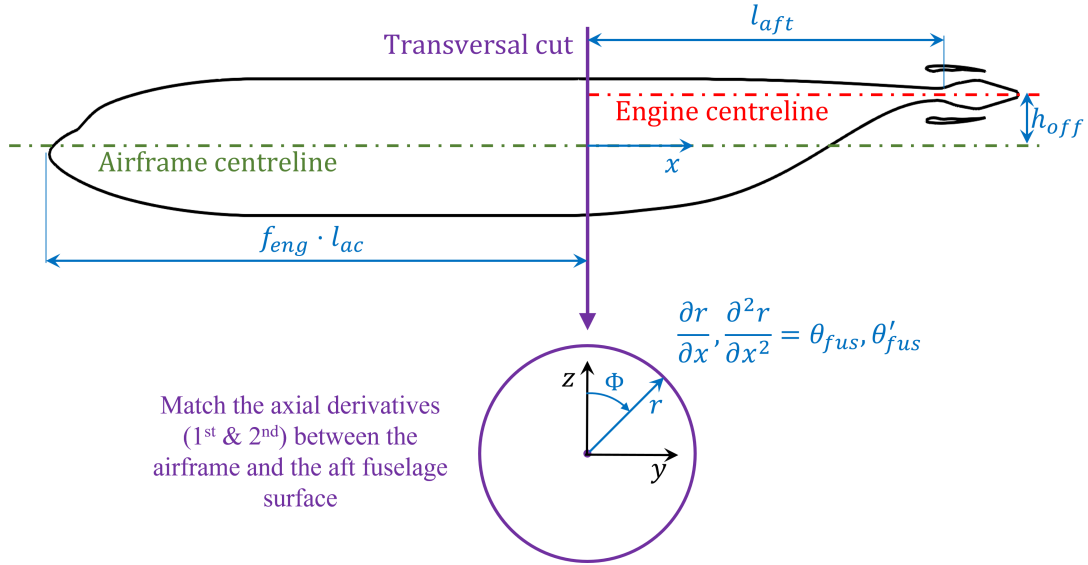


FIGURE 3.14: Definition of the propulsor location based on the 3D reference aircraft (DLR-F6 with FX2B¹⁹), and definition of the geometry matching between the airframe and the aft-fuselage surface

axis, the scarf angle (θ_{scarf}) is also defined to control the intake length azimuthally. The length of the intake is modified on the vertical plane (XZ). The position of the highlight at the top line and bottom line is displaced by the angular value defined by θ_{scarf} , with engine axis point at the side-line highlight axial location as center of rotation. Under this definition, for a positive θ_{scarf} the top-line length increases and the bottom line length decreases. The side-line length is constant and the intermediate lengths are defined based on the azimuthal iCST variation.

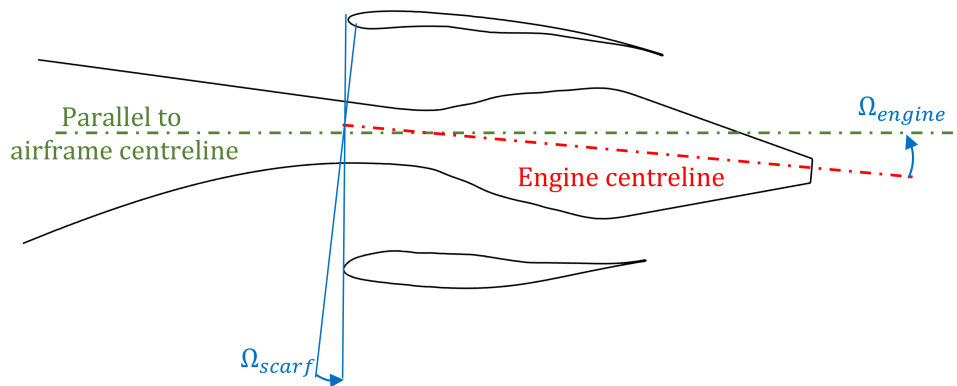


FIGURE 3.15: Sketch of the intake scarf and engine inclination (Not to scale)

Besides the orientation of the propulsor, an additional control is included to transform the sharp trailing edge of the 2D parametrisation into a more realistic blunt

trailing edge. The trailing edge of the nacelle is fixed while the position of the trailing edge of the exhaust is offset by a given value. The offset is moved perpendicular to the upper exhaust trailing edge angle, β_{noz} (Fig. 3.7).

3.3 Performance accounting

The increase on the aerodynamic coupling between the airframe and the propulsive system requires of an integral analysis of the aircraft and propulsor forces. At the same time, it is essential to evaluate the impact of the propulsive system at both system and aircraft level. This presents a challenge for the force book-keeping of the embedded propulsion systems.

While energy-based methods as the Power-Balance⁷⁹ and the Exergy Analysis¹⁶ methods have been presented as a potential solution to separate the force contribution in embedded propulsive systems, they require an increase of the computational cost when compared to the near-field momentum-based methods⁸⁰. This increase on the computational cost is due to the additional mesh resolution required for the far-field region of the domain. Another disadvantage on the application of these methods is the current absence of benchmark study cases to validate these energy-based methodologies. Therefore, it is necessary to reassess the conventional momentum-based approaches and adapt them for the requirements of the novel embedded propulsive systems. In the present work, a modified near-field method is proposed as the force book-keeping methodology. The force is normalised by the reference values to obtain metrics that define the aerodynamic performance of the propulsion integration at an aircraft and system level.

In addition to the force accounting methodology, established intake distortion⁴⁵ and exhaust performance accounting³⁶ methods are applied. The validity these approaches and the additional considerations for their implementation is also analysed in this section.

3.3.1 Modified Near-Field Method

Momentum based approaches have been commonly used to determine the force balance of the propulsive system and the aircraft^{15,35,94}. A Modified-Near Field Method is proposed (Fig. 3.16) with some changes from the original formulation of the AGARD 237¹⁴.

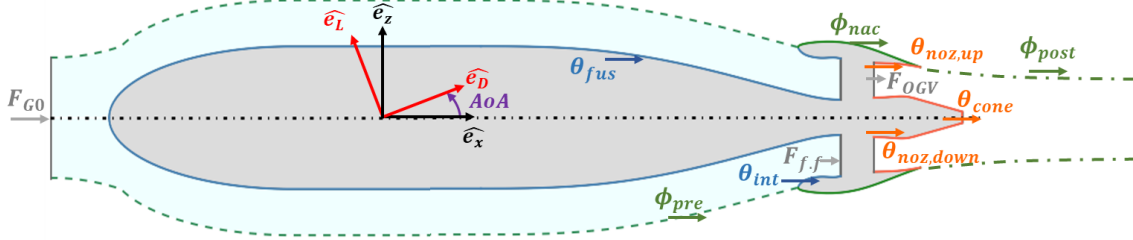


FIGURE 3.16: Modified Near-Field method domain and forces

In the Modified-Near Field Method, the forces are divided into three categories: wall forces, θ_i ; drag forces, ϕ ; and gauge stream forces, F . The wall forces, θ_i , are the surface forces that lie within the engine stream-tube (Fig. 3.16), i.e. thrust domain. These forces are obtained by the integration of the gauge pressure ($p - p_\infty$) and the axial contribution of the wall shear stress tensor ($\tau_{wall,x}$) over the surface of the aerodynamic body (Eq. 3.16). The drag forces are the forces that lie outside the engine stream-tube (Fig. 3.16), i.e. drag domain. The drag forces are formulated in the same form as the wall forces, but they are oriented towards the drag direction (Eq. 3.17). In addition, the drag forces can be divided into three main groups: the pre-entry forces, ϕ_{pre} ; the nacelle drag force, ϕ_{nac} ; and the post-exit terms, ϕ_{post} . Finally, the gauge stream forces are the result of the momentum integration at a given aerodynamic plane (Eq. 3.18).

$$\theta_i = \int [(p - p_\infty) + \tau_{wall,x}] dA \quad (3.16)$$

$$\phi_i = -\theta_i \quad (3.17)$$

$$F_i = \int [(p - p_\infty) + V_x (\rho \mathbf{V} \cdot \mathbf{n})] dA \quad (3.18)$$

The pre-entry force terms can be derived directly from a momentum balance to the inlet capture control volume (Eq. 3.19). The value of the upstream gauge force, F_{G0} , is computed analytically (Eq. 3.20) from the freestream velocity and the mass flow rate of the propulsive system ($\dot{m}$, V_∞). Traditionally, the separation of the force terms on the modified near-field method has been performed based on a thrust-drag force definition (T and D). However, on the accounting of the forces of the embedded propulsion system these definitions are obscured^{64,75}. The definition of a forward and backward axial force based on the aircraft coordinate system will be more accurate^{13,80}. Nevertheless, to maintain the nomenclature with the conventional application of the modified near-field method the thrust-drag split is maintained in this work. Since the aim of the accounting method is to evaluate

the net force contribution, this discrepancy on the nomenclature does not affect the final definition of the net force. Thus, the contribution of the forces is split in thrust T and drag D . These terms can be simplified to define the modified drag (Fig. 3.21), D^* , and thrust (Eq. 3.25), T^* , forces by removing the post-exit forces, ϕ_{post} , from the accounting. This is of special interest on complex cases where is complex to evaluate the aerodynamic interaction between the propulsor and the wake(e.g. three-dimensional embedded propulsive systems).

$$\phi_{pre} = -F_{G0} + F_{F,F} + \theta_{Intake} + \theta_{fus} \quad (3.19)$$

$$F_{G0} = \int_{\infty} V_x (\rho \mathbf{V} \cdot \mathbf{n}) dA = \dot{m} V_{\infty} \quad (3.20)$$

$$D^* = \phi_{pre} + \phi_{nac} \quad (3.21)$$

$$D = D^* + \phi_{post} = \phi_{pre} + \phi_{nac} + \phi_{post} \quad (3.22)$$

$$GPF^* = F_{OGV} - \theta_{nozzle,up} - \theta_{nozzle,down} - \theta_{cone} \quad (3.23)$$

$$GPF = F_{OGV} - \theta_{nozzle,up} - \theta_{nozzle,down} - \theta_{cone} + \phi_{post} \quad (3.24)$$

$$T^* = -F_{G0} + GPF^* \quad (3.25)$$

$$T = T^* + \phi_{post} \quad (3.26)$$

The reference frame to obtain the forces varies with the dimensionality of the problem (2D axisymmetric or 3D). In 2D axisymmetric cases, the thrust and drag forces are defined based on the central axis of the aft-mounted propulsive system ($\hat{\mathbf{e}}_x$ in Fig. 3.16). Obviously, no lift force is generated with this approach. In the 3D cases, the forces are obtained projected into the drag direction of the aerodynamic axis ($\hat{\mathbf{e}}_D$ in Fig. 3.16). This axis is defined by the relative orientation of the flow with respect to the engine central axis, based on the angle of attack (AoA). The lift force can also be obtained with this formulation. The importance of the angle of attack and the aircraft lift force is discussed in Section 3.3.6.

The defined modified thrust and drag forces are balanced to obtain the net force of the aircraft-propulsor combined system. This term is defined as the Net Vehicle Force (Eq. 3.27), NVF . The NVF represents the total force on the aircraft reference system. The comparison between different propulsion integration aerodynamic designs can be performed with this metric. However, since the propulsive system is embedded into the airframe, it is still required to separate the contribution of the aft-mounted propulsor to the overall aircraft force. Hence, it is necessary to define a metric to determine the impact of the new embedded propulsor and compare

different configurations.

$$NVF = D^* - T^* \quad (3.27)$$

3.3.2 Normalisation of the performance metrics

To compare the aerodynamic performance between different propulsor configurations, it is necessary to define a metric that describes the embedded propulsor contribution to the overall propulsive force. For this purpose, the method described by Valencia et al.¹³⁴ is applied. First, a reference airframe and power-plant is defined (Fig 3.17). The thrust requirement of the reference power plants (T_{ref}) at the selected operating conditions is used to scale the thrust forces. The forces over the airframe without propulsive system (drag forces) are applied as the Net-Vehicle Force of the reference airframe (NVF_{ref-af}). For 2D axisymmetric cases, a reference axisymmetric airframe is applied (Fig 3.17). In the 3D cases the reference airframe (DLR-F6¹⁹) is applied. After the references forces are defined, the forces obtained with the embedded propulsor are scaled to isolate the force contribution of the propulsor¹³⁴. The NVF (Eq. 3.27) is balanced with the Net Vehicle Force of the reference airframe (NVF_{ref-af}) to define the Relative Net-Thrust Force (Eq. 3.28), F_{RN} . Based on the assumption that any difference on the forces between the reference aircraft and the new system derives directly by the integration of the embedded propulsor, this metric defines the force contribution of the new propulsor.

$$F_{RN} = NVF - NVF_{ref-af} \quad (3.28)$$

The F_{RN} contribution to the overall propulsive force can be obtained by the normalisation of the F_{RN} by the reference thrust requirement, T_{ref} (Fig 3.17). The metric resulting from this normalisation is the Thrust Split (Eq. 3.29), TS . This metric is used to define the contribution of the integration of the new embedded system to the total thrust requirement of the vehicle. Hence, the aerodynamic performance of the new propulsive system at the full aircraft level is represented with the TS .

$$TS = -\frac{F_{RN}}{T_{ref}} \quad (3.29)$$

In addition to the thrust contribution of the system it is required to define a metric that can characterize the performance of the propulsive system. For this purpose, the Thrust Specific Power Consumption (Eq. 3.30), $TSPC$, is applied⁶⁴. This metric is defined as the ratio between the required power of the propulsor, W_{prop} ,

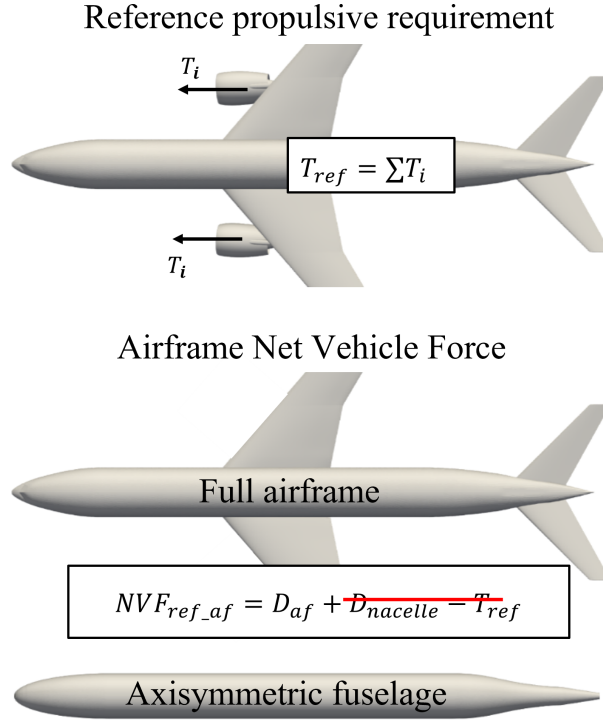


FIGURE 3.17: Schematic of the reference thrust and the reference airframe net-vehicle force for a sample configuration (NASA Common Research Model²⁰) for illustration purposes. Not to scale

and the Relative Net Thrust Force (F_{RN}). The W_{prop} is obtained through a work balance to the inlet and the outlet of the fan stage (Eq. 3.31). The $TSPC$ can be used to define a propulsive efficiency (Eq. 3.32), η_{prop} . This requires the definition of a reference velocity. This velocity can be the reference freestream value of the flow^{64,134}. However, this can produce propulsive efficiency values higher than unity, so the utility of the propulsive efficiency as a metric is reduced. While more detailed definitions of the reference velocity can be applied (e. g. the velocity at the propulsor highlight), this is still an arbitrary definition of the efficiency metric. Hence, to maintain the consistency across this work, the $TSPC$ is applied as the reference performance metric.

$$TSPC = -\frac{W_{prop}}{F_{RN}} \quad (3.30)$$

$$W_{prop} = \left| \iint_{OGV \text{ OUTLET}} \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} + p\mathbf{V} - \bar{\bar{\tau}}\mathbf{V} \right] \cdot d\mathbf{S} - \iint_{FAN \text{ FACE}} \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} + p\mathbf{V} - \bar{\bar{\tau}}\mathbf{V} \right] \cdot d\mathbf{S} \right| \quad (3.31)$$

$$\eta_{prop} = \frac{V_{\infty}}{TSPC} \quad (3.32)$$

3.3.3 Intake performance evaluation

The application of more embedded propulsion systems is expected to increase the flow distortion levels at the intake^{39–41}. This increase on the flow distortion levels could increase the blade loads and lead to losses on the turbomachinery efficiency^{42,43}. This could produce significant losses on the system aerodynamic performance and increase the power requirements. Hence, it is necessary to adequately quantify the intake performance for the design of embedded propulsors.

In this work, the intake performance is addressed based on three reference stations (freestream, fuselage and AIP) of the domain (Fig. 3.18). First, a plane defined upstream the aircraft where the metrics of interest are equal to the freestream values (Freestream station). Second, a reference plane at a defined station of the fuselage (Fuselage station) is selected to represent the boundary layer properties (Appendix B). This plane is selected at a position where the boundary layer shape is not affected by the transition at the fuselage forebody and the presence of the propulsor geometry in the fuselage afterbody. The Fuselage station is bounded radially by the fuselage surface and the shear layer of the engine stream-tube defined by the plane of streamlines that end at the nacelle highlight (Fig. 3.18). Third, a reference plane inside the propulsor intake is defined at a fixed distance (5 cm) of the fan face (AIP plane).

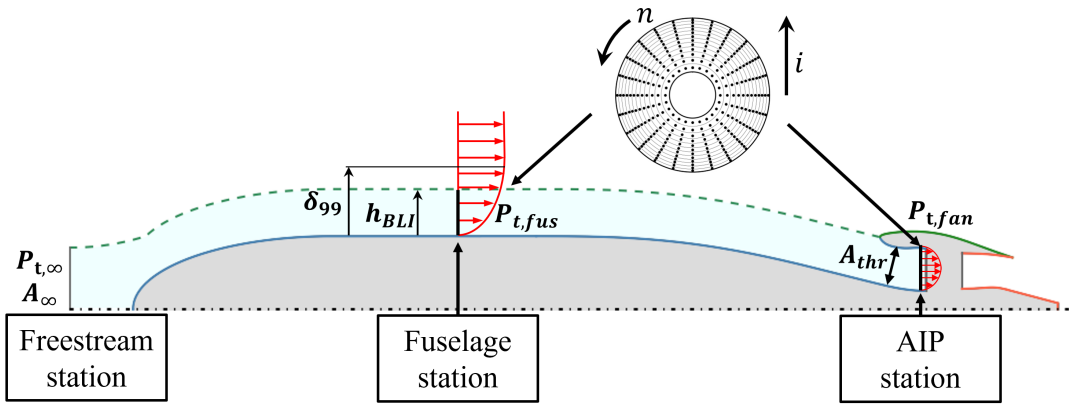


FIGURE 3.18: Intake performance reference measure planes and rake distribution

To sample the properties of interest of the flow (static and total pressure) at the reference stations, a method analogous to an experimental set up^{39,45} is applied

(Fig. 3.18). A total of 12 radial probes placed on equal area sectors are defined in each azimuthal station. The radial probes are split into 72 equally spaced azimuthal positions to sample the flow properties every five degree sector. For the 2D axisymmetric cases, the number of azimuthal divisions is considered infinite so only one set of radial probes is applied. The metrics of interest are obtained by an area-weighted average of the samples. The metrics averaged with this approach are presented in the text with an over-line (e.g. $\overline{P_{t,fan}}$).

Established methodologies are used to define the intake performance and distortion accounting⁴⁵. The area weighted total pressure at the fan face station (Fig. 3.18) is compared with the freestream total pressure value to derive the Intake Pressure Ratio (Eq. 3.33). The IPR can be compared with the fuselage station averaged total pressure to obtain the Ingested Pressure Ratio (Eq. 3.34), IPR_{Ing} , to represent the losses coming from the designed aerodynamic installation of the propulsive system. To quantify the distortion level in a 2D axisymmetric analysis the Radial Distortion Index is applied (Eq. 3.35), RDI . For the 3D configurations the Circumferential Distortion Index (Eq. 3.36), CDI ; and the sectional distortion coefficient in a sector of 60 degrees (Eq. 3.37) are also applied.

$$IPR = \frac{\overline{P_{t,fan}}}{P_{t,\infty}} \quad (3.33)$$

$$IPR_{Ingested} = \frac{\overline{P_{t,fan}}}{\overline{P_{t,fus}}} \quad (3.34)$$

$$RDI = \max \left(\frac{\overline{P_{t,fan}} - \overline{P_{t,i=0}}}{\overline{P_{t,fan}}}, \frac{\overline{P_{t,fan}} - \overline{P_{t,i=N}}}{\overline{P_{t,fan}}} \right) \quad (3.35)$$

$$CDI = \max_1^{12} \left(0.5 \left[\frac{\overline{P_{t,i}} - \min(P_{t,i})}{\overline{P_{t,fan}}} + \frac{\overline{P_{t,i+1}} - \min(P_{t,i+1})}{\overline{P_{t,fan}}} \right] \right) \quad (3.36)$$

$$DC_{60} = \frac{\overline{P_{t,fan}} - \overline{P_{t,60}}}{\overline{q_{t,fan}}} \quad (3.37)$$

In integrated systems under boundary layer ingestion, it is necessary to quantify the amount of boundary layer ingested. For this purpose, the Boundary Layer Ingestion Rate (Eq. 3.38), BLR , is defined as the fraction of the boundary layer thickness ingested at the reference fuselage station. This metric can be controlled for a given flight condition by the modification of the propulsive system required mass flow rate. A detailed discussion about the boundary layer evaluation can be found in the Appendix B.

Since the BLR cannot be controlled under variable flight operation conditions, the

MFCR (Eq. 3.39) is defined as the ratio between the throat area (A_{thr}) and the Freestream station area (A_∞), determined by the mass flow rate and the flight conditions ($\rho_\infty V_\infty$). The conventional definition of the *MFCR* applies the highlight area of a podded engine to illustrate the pre-entry capture duct shape without the inclusion of the intake design effect. However, in the embedded systems, the presence of the fuselage afterbody limits the definition of the highlight area. Thus, the conventional definition of the metric cannot be applied without a misrepresentation of the original physical meaning. In this work, the throat area is enforced as a geometric constraint for each design (Fig. 3.6). This provides a more consistent definition of the reference dimensions. Hence, it is used to define the non-dimensional mass flow requirements in this work (Eq. 3.39).

$$BLR = \frac{h_{BLI}}{\delta_{99}} \quad (3.38)$$

$$MFCR = \frac{A_\infty}{A_{thr}} = \frac{\dot{m}}{\rho_\infty V_\infty A_{thr}} \quad (3.39)$$

3.3.4 Exhaust performance evaluation

Non dimensional coefficients are applied for the definition of the exhaust performance based on the work of Goulos et al.³⁶. The coefficients are the discharge coefficient, C_D , and the velocity coefficients, C_V and C_V^* . The discharge coefficient (Eq. 3.40), is defined as the ratio between the actual mass flow at the exhaust duct and the ideal isentropic mass flow (Eq. 3.41) through the exhaust nozzle throat (A_{noz}). The velocity coefficients (Eq. 3.43-3.44) are defined as the ratio between the Gross Propulsive Force and the Ideal Propulsive Force (Eq. 3.45), *IPF*. The *IPF* is defined based on the ideal velocity at the exhaust nozzle throat (Eq. 3.46) and the actual mass flow through the exhaust.

$$C_d = \frac{\dot{m}_{actual}}{\left(\frac{\dot{m}}{A}\right)_{ideal} A_{noz}} \quad (3.40)$$

$$\left(\frac{\dot{m}}{A}\right)_{ideal} = P_t \left(\frac{1}{\lambda}\right) \sqrt{\frac{2\gamma}{(\gamma-1)RT_t} \left(1 - \left(\frac{1}{\lambda}\right)^{\frac{\gamma-1}{\gamma}}\right)} \quad (3.41)$$

$$\lambda = \frac{P_{t,ex}}{p_\infty} \quad (3.42)$$

$$C_V = \frac{GPF}{IPF} \quad (3.43)$$

$$C_V^* = \frac{GPF^*}{IPF} \quad (3.44)$$

$$IPF = \dot{m}_{actual} \cdot V_{ideal} \quad (3.45)$$

$$V_{ideal} = \sqrt{\frac{2\gamma RT_t}{(\gamma - 1)} \left(1 - \left(\frac{1}{\lambda} \right)^{\frac{\gamma-1}{\gamma}} \right)} \quad (3.46)$$

The reference metrics (Eq. 3.41-3.46) are obtained from the mass weighted average values of the total pressure and total temperature at the exhaust duct. The nozzle pressure ratio, λ , (Eq. 3.42) is defined as the ratio between the mass averaged total pressure at the exhaust ($P_{t,ex}$) and the ambient static pressure (p_∞). The reference ambient static pressure (p_∞) is defined by the operating altitude based on the International Standard Atmosphere model¹⁴⁵. In the isentropic mass flow (Eq. 3.41) definition, if the fan nozzle pressure ratio exceeds the critical value ($\lambda \geq \lambda_{crit} = ((\gamma + 1)/2)^{\frac{\gamma}{\gamma-1}}$) the value of λ is set to λ_{crit} . In the ideal velocity (Eq. 3.46) the nominal value of λ is always used.

3.3.5 Correction of the force contribution with mass flow imbalances

The CFD methodology could generate mass flow imbalances between the intake and the exhaust duct (Appendix B). This is a result of the changes on the effective static pressure in the exhaust due to the fuselage boundary layer and the nacelle geometry. These mass flow imbalances could lead to a misrepresentation of the propulsive contribution of the embedded system. As a solution to this problem, a mass flow matching methodology is proposed in Section 3.4.2.1. Nevertheless, some small differences between the mass flow at the fan face and exhaust duct are still observed (up to $\sim 0.5\%$). To mitigate these uncertainties, the gross propulsive force applied in the accounting of the NVF is corrected based on the velocity coefficients based on the approach proposed by Otter¹²². The NVF is corrected based on the modified gross-propulsive force (Eq. 3.47). After the C_V^* is obtained with the mass flow of the CFD solution at the exhaust, the modified gross propulsive force is obtained for the equivalent target mass flow prescribed at the fan ($\dot{m}_{tgt}$) to correct the force contribution. This approach was verified by Otter¹²² for separate flow exhausts to operate between -0.5% and 1.5% mass flow error in the bypass. For the current applications the mass flow at the exhaust is corrected by the one dimensional fan model or the low order fan model, so the error is always below this range.

$$GPF^* = C_V^* (\dot{m}_{tgt} V_{ideal}) \quad (3.47)$$

3.3.6 Three-dimensional effects in the performance accounting

The aerodynamic performance evaluation was proposed with the 2D axisymmetric case as reference, but the definition of the figures of merit (TS , $TSPC$, IPR , RDI , C_D , C_V ...) holds for 3D configurations. However, a notable exception is the ingested pressure recovery, IPR_{ing} (Eq. 3.34). Under non-homogeneous 3D flows over the fuselage (Fig. 3.19), the definition of the engine stream-tube is not as straightforward as in the 2D axisymmetric cases (Appendix B). While the streamlines of the engine stream-tube could be captured, the integration and definition of the streamtube cross sectional area is still complex. Thus, to simplify the problem and avoid the introduction of any arbitrary definition of the average total pressure at the Fuselage station (Fig. 3.18), this metric is excluded from the 3D studies and replaced by the conventional intake pressure recovery, IPR (Eq. 3.33). This does not imply a loss of information, since the IPR contains the losses at the fuselage. However, these pressure losses cannot be excluded to isolate the effects of the aerodynamic design of the propulsion integration.

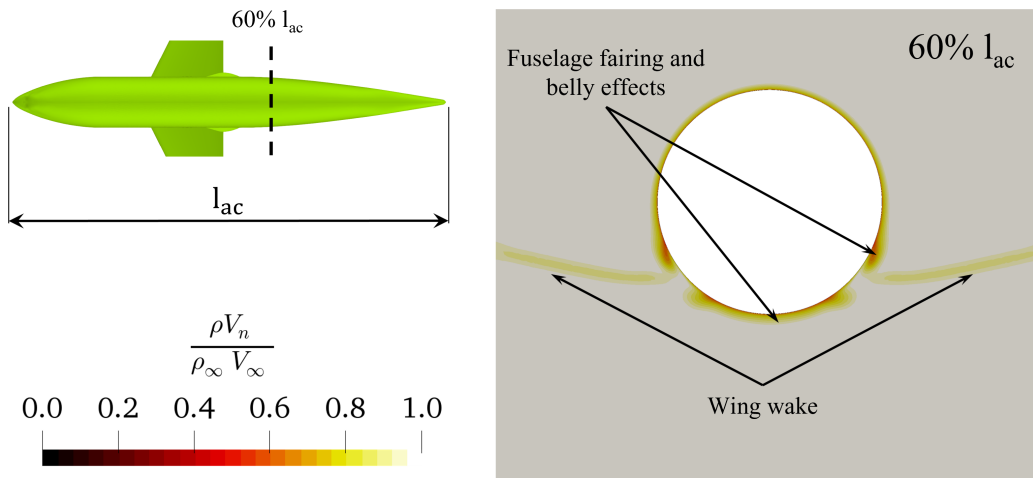


FIGURE 3.19: View of the non-dimensional mass flow at the Fuselage station of the DLR-F6¹⁹

An additional effect that must be accounted for the 3D configurations is the variation on the lift coefficient due to the propulsion integration effects. The reference aircraft¹⁹ has a target lift coefficient (C_L) equal to 0.5 at cruise conditions. To represent the effects of the propulsion integration on the overall aircraft aerodynamics, the reference C_L must be maintained. To match this value of the lift coefficient, two

angles of attack near to the value of the reference aircraft are simulated. The post-processing methodology is applied for each angle independently and then a linear interpolation is applied to match the reference C_L . The angles are selected with an interval of 0.5° to minimise the interpolation uncertainty.

3.3.7 Low order performance accounting

As a complement to the numerical evaluation of the aerodynamic performance, a low order force accounting methodology was included for the preliminary evaluation of the performance. In this approach, the forces are defined based on the drag of the fan cowl and the fuselage in isolation. These forces are defined based on the approaches defined by Raymer¹⁴⁶ and Torenbeek^{147,148} for preliminary drag estimation. In these approaches, the zero-lift drag coefficient, $C_{D,0}$, (Eq. 3.48) of the surfaces is computed based on the viscous coefficient and a form factor. For this work the form factor, the definitions of viscous coefficient, C_f , (Eq. 3.49), and the fuselage and cowl form factors, FF , (Eqs. 3.51-3.52) are obtained by the formulation defined by Raymer¹⁴⁶. The viscous coefficient (C_f) includes the changes on the viscous contribution over the surface with the variation of the Reynolds number, Re , and the Mach number, M (Eq. 3.50). The form factor accounts for the contribution of the form drag based on the fineness ratio of the fuselage and the cowl ($\zeta_{fus} = l_{ac}/(2r_{fus})$ and $\zeta_{cowl} = l_{nac}/(2r_{max})$).

$$C_{D,0} = C_f FF \frac{S_{wet}}{S_{ref}} \quad (3.48)$$

$$C_f = \frac{0.455}{(\log(Re))^{2.58} (1 + 0.144M_\infty^2)^{0.65}} \quad (3.49)$$

$$Re = \frac{\rho_\infty V_\infty l_{ref}}{\mu} \quad (3.50)$$

$$FF_{(fus)} = 0.9 + 0.0025\zeta_{fus} + \frac{5}{\zeta_{fus}^{1.5}} \quad (3.51)$$

$$FF_{(cowl)} = 1 + \frac{0.35}{\zeta_{cowl}} \quad (3.52)$$

The drag coefficient is normally scaled by a reference area, S_{ref} (e.g. the cross-sectional area). However, the viscous coefficient is based on the wetted area of the aerodynamic body, S_{wet} . Hence, to scale the drag values it is required to provide a representative wetted area estimation. The formulations defined by Torenbeek¹⁴⁷ for the fuselage wetted surface (Eq. 3.53) and the fan-cowl (Eq. 3.54) are applied for this purpose. The formulation of both terms have been adapted to match the

parametric geometry formulation proposed in the Section 3.2.2.

$$S_{wet,fus} = \pi 2r_{fus}l_{ac} \left(1.0 - \frac{2}{\zeta_{fus}}\right)^{2/3} \left(1 + \frac{1}{\zeta_{fus}^2}\right) \quad (3.53)$$

$$S_{wet,cowl} = 2l_{nac}r_{max} \left(2.0 + 0.35f_{max} + 0.8f_{max}\frac{r_{hi}}{r_{max}} + 1.15(1 - f_{max})\frac{r_{te}}{r_{max}}\right) \quad (3.54)$$

3.3.7.1 Net Vehicle force from momentum deficit ingested

The fuselage drag provides a reference value for a non-propelled fuselage in isolation. However, a method to account for the drag contribution of the embedded propulsion system is still required. Seitz et al²⁶ proposed to account for the thrust split of the aft-mounted Boundary Layer Ingestion propulsor based on the amount of momentum deficit ingested by the propulsor (Eq. 3.55). In this formulation, the effect of the propulsion integration and the aerodynamic coupling between the propulsor and the fuselage is not directly captured. Furthermore, the effect of the nacelle drag is not accounted. With this method, the ingested drag by the BLI system ($D_{fus,ing}$) is defined as a fraction of the fuselage drag in isolation (Eq. 3.56). The ingested fraction of the momentum flux deficit of the boundary layer is accounted with the factor f_β . This value is obtained from the boundary layer properties over the fuselage based on a CFD reference. The amount of momentum and mass flow rate ingested based on the of the boundary layer is estimated in the Appendix B. The effect of the axial location of the BLI propulsor is accounted by the relative position of the highlight plane of the propulsor ($\chi_{prop} = (f_{enf}l_{ac} + l_{aft})/l_{ac} - 2r_{fan}AR_{int}$). Finally, any additional effect of the propulsion integration is included by an integration factor (f_{int}). In the present work, the value of f_{int} is set to 0.9 based on the estimation of Seitz et al²⁶.

$$TS_{ideal} = \frac{T_{BLI}}{T_{ref} - D_{fus,ing}} \quad (3.55)$$

$$D_{fus,ing} = D_{fus}f_\beta\chi_{prop}f_{int} \quad (3.56)$$

The aforementioned formulation presents two clear limitations for the preliminary design of the aircraft and the embedded propulsion system. First, the nacelle contribution is completely neglected from the force accounting. Hence, any change on the propulsion integration design is not captured. Second, the effect of the aerodynamic coupling between the propulsor and the airframe is not directly accounted. Previous research such as Hall et al¹⁴⁹ and Sanders⁶⁴ demonstrated that the presence of the boundary layer ingestion propulsor modifies the shear stress distribution and static

pressure distributions over the surface in the regions near the propulsor based on the operating conditions. Hall et al¹⁴⁹ evaluated the effect of the shape factor¹⁵⁰ of the aspirated boundary layer. However, in a conceptual design with low order models this cannot be accounted. As a solution, Bijewitz et al¹⁵¹ proposed a calibration of the propulsion integration factor (f_{int}) based on correlations with CFD data. However, this reduces the applicability of the method as an alternative to the CFD, since higher fidelity data is also required.

In addition to these limitations, the formulation of the thrust split (Eq. 3.55) is not equivalent to the one defined by the Equation 3.29. Thus, to adapt the definition proposed by Seitz and Gologan²⁶ to the current methods and include the nacelle contributions, a modified methodology based on the momentum deficit ingested is proposed (Fig. 3.20). The proposed method accounts for the nacelle drag effects and provides an split between the thrust and the drag contributions of the propulsion integration. The ingested drag is applied to define the recovered part of the reference fuselage. Thus, the drag contributors on the preliminary analysis are the nacelle drag D_{nac} and the remaining non-ingested part of the fuselage drag (Eq. 3.57). This links the installation effects to the preliminary drag estimation by accounting the boundary layer ingestion contribution and the cowling design to the momentum deficit. A detailed comparison between the original approach defined by Seitz and Gologan²⁶ and the current approach is provided in the Appendix B.

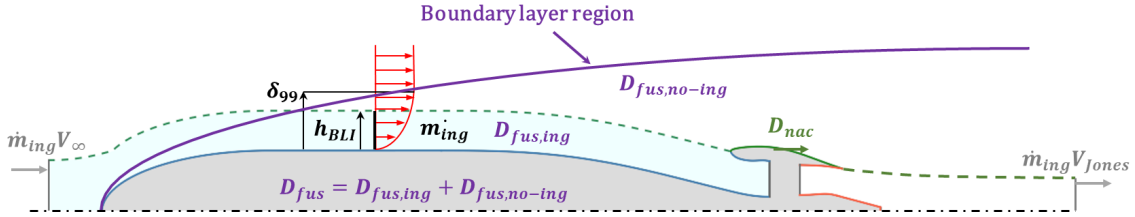


FIGURE 3.20: Schematic of the forces over the fuselage and the propulsor with the low order performance accounting based on the momentum deficit ingested

$$D = D_{nac} + D_{fus,no-ing} = D_{nac} + (D_{fus} - D_{fus,ing}) \quad (3.57)$$

To account for the thrust contribution of the BLI propulsor, Jones thrust definition is applied¹⁵². This approach assumes an isentropic expansion from the nozzle exit to the freestream pressure. The transfer of momentum and energy from the exhaust streamtube to the ambient is neglected on this approach. With this approach the velocity of the propulsor's jet on the freestream can be computed based on the exhaust exit velocity (V_{ex}), pressure and Mach number (Eq. 3.58). This approach

has some limitations for some operating conditions of the exhaust. Significant uncertainties are expected in cases where the exhaust boundary layer growth is more significant or where exhaust shock wave patterns are present¹⁵³. However, it is considered sufficient to account for the thrust contributions on this low order approach. With this approach the net uninstalled thrust, T_{BLI} , (Eq. 3.59) is computed as the difference between the gross propulsive force obtained from Jones definition and the gauge force of the pre-entry streamtube at infinity (Fig. 3.16).

$$V_{Jones} = V_{ex} \left(1 + \frac{2}{M_{ex}^2 (\gamma - 1)} \left(1 - \left(\frac{p_{\infty}}{p_{ex}} \right)^{\frac{\gamma-1}{\gamma}} \right) \right)^{0.5} \quad (3.58)$$

$$T_{BLI} = \dot{m} (V_{Jones} - V_{\infty}) \quad (3.59)$$

With T_{BLI} and D , the net vehicle force of the vehicle can be obtained (Eq. 3.60). Assuming that the reference drag of the aircraft is obtained from the zero-lift definition of the isolated fuselage (Eq. 3.48), the relative net force can be defined as the difference between the nacelle drag and the thrust sources (Eq. 3.61). These thrust sources are the uninstalled thrust and the amount of fuselage drag layer ingested by the propulsor, $D_{fus,ing}$ (Eq. 3.56).

$$NVF = D - T_{BLI} \quad (3.60)$$

$$F_{RN} = NVF - NVF_{ref} = NVF - D_{fus} = D_{nac} - D_{fus,ing} - \dot{m} (V_{Jones} - V_{\infty}) \quad (3.61)$$

The thrust split TS , and the $TSPC$ are obtained with the methodology defined in Section 3.3.2. The power requirements of the one-dimensional propulsor are obtained from the performance of the propulsor, defined by the fan pressure ratio (FPR) and the fan stage polytropic of isentropic efficiency (η_{poly} or η_{is}). The reference boundary layer properties are used to size the propulsor as defined in the Appendix B. Also, the exhaust efficiency is included by the assumption of preliminary C_V and C_D values. Similarly, a preliminary estimation of the IPR_{ing} is included in the sizing process.

3.3.7.2 Net Vehicle Force from modified near-field method

The aforementioned approach is based on the assumption that there are no aerodynamic coupling effects between the propulsor integration and the rate of boundary layer ingested. Hence, the drag is estimated directly from the propulsor cowl and the fuselage in isolation. The interaction between both systems is estimated by the amount of fuselage drag ingested, based on a reference boundary layer distribution.

This methodology presents some limitations and it may result on a higher discrepancy from a higher order evaluation method such as CFD. To reduce the uncertainty of this preliminary design method, an additional methodology based on the modified near-field momentum method¹⁴ (Section 3.3.1) is proposed (Fig. 3.21).

The drag forces is divided into three terms: ϕ_{pre} , ϕ_{nac} and ϕ_{post} (Eq. 3.22). The nacelle surface drag (ϕ_{nac}) is accounted as the zero-lift drag (viscous and form) of the axisymmetric nacelle in isolation^{146–148} (Eq. 3.62). To obtain the post exit force (ϕ_{post}), a momentum balance at the exhaust stream tube is applied. Previous work on nacelle aerodynamics¹⁵³ highlighted the complexity of accounting the forces on the exhaust cone (θ_{cone}) with low order methods. As a solution, Ramirez-Rubio et al¹⁵³ created a surrogate method based on CFD data to estimate the force in the external wall of separated exhaust configurations in conventional podded powerplants. However, to simplify the present approach, the forces on the cone (θ_{cone}) are neglected (Eq. 3.63). This assumption is introduced to resemble a conceptual design approach where no CFD or surrogate data is available. Finally, the pre-entry drag forces (ϕ_{pre}) are obtained from the application of a momentum balance on the engine entry stream tube (Eq. 3.64). The gauge forces, F_G , are evaluated at the Freestream station and at the highlight plane (Fig. 3.21). The static pressure and velocity at the highlight are obtained from the boundary layer properties of the reference fuselage. The surface forces of the fuselage are defined from the reference fuselage in isolation based on the relative position of the propulsor highlight (χ_{prop}). In this case, the integration factor (f_{int}) is applied to account for some of the changes on the shear and pressure forces by the presence of the propulsor. The value is considered constant for this preliminary analysis ($f_{int} = 0.9$).

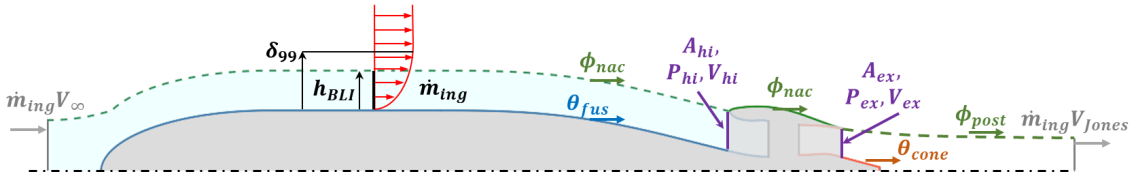


FIGURE 3.21: Schematic of the forces over the fuselage and the propulsor with the low order performance accounting based on the modified near-field method

$$\phi_{nac} = C_f F F_{(cowl)} S_{wet,cowl} 0.5 \rho_{\infty} V_{\infty}^2 \quad (3.62)$$

$$\phi_{post} = \dot{m} (V_{Jones} - V_{ex}) - A_{ex} p_{ex} \quad (3.63)$$

$$\phi_{pre} = \dot{m} (\bar{V}_{hi} - V_{\infty}) - A_{hi} p_{hi} - \theta_{fus} \quad (3.64)$$

$$\theta_{fus} = D_{fus} \chi_{prop} f_{int} \quad (3.65)$$

Finally, the net uninstalled thrust (T_{BLI}) is included from the definition in Equation 3.59. The remaining metrics (TS and $TSPC$) are obtained from the Net Vehicle Force definition (Eq. 3.60). A comparison between this and the other low order drag estimation methods is provided in the Appendix B.

3.4 Computational Fluid Dynamics methods

The main methods behind the CFD methodology are defined in this section. This includes the numerical solver, the different turbomachinery models, and the mesh methodology for each of the configurations.

3.4.1 Boundary conditions and numerical solver

A steady compressible viscous computational fluid dynamics approach is applied on the present works. Fravre-Averaged Navier-Stokes equations are applied with the $k-\omega$ SST¹⁵⁴ turbulence closure model. The cell-centred solver ANSYS Fluent is used¹⁵⁵. A second order upwind scheme is utilized for the space discretization. The gradient computations is based on a Green-Gauss cell-based method. The gas thermodynamic properties are defined applying the kinetic theory and a real gas model. The flow conditions with altitude are defined by the International Standard Atmosphere (ISA) model¹⁴⁵. The fluid viscosity is adjusted with altitude based on Sutherland's law. The governing equations of the CFD model and the reference frames applied are described in Appendix A. The summary of the validation cases for the current CFD approach can be found in Appendix D.

3.4.1.1 Boundary conditions for the 2D axisymmetric model

Pressure far-field boundary conditions are applied for the freestream boundaries of the domain based on the operation altitude and Mach number (Figs. 3.22-3.23). Adiabatic non-slip walls are used for the fuselage and housing components surfaces. A y^+ value of unity is applied for all the wall boundaries. The axis of rotation is defined on the x coordinate of the system (Figs. 3.22-3.23). The boundary conditions of the CFD approach are changed based on the turbomachinery model (Fig. 3.2):

- **One-dimensional fan approach:** In the one-dimensional fan approach the fan boundaries are defined by a target mass flow outlet at the fan face and a

total pressure inlet boundary condition on the OGV outlet (Fig. 3.22). The total pressure and total temperature at the OGV outlet are defined based on the conditions at the fan face and the turbomachinery requirements. The total pressure value is iterated to match the mass flow values at the OGV outlet and the fan inlet. The mass flow matching process is defined in Subsection 3.4.2.1.

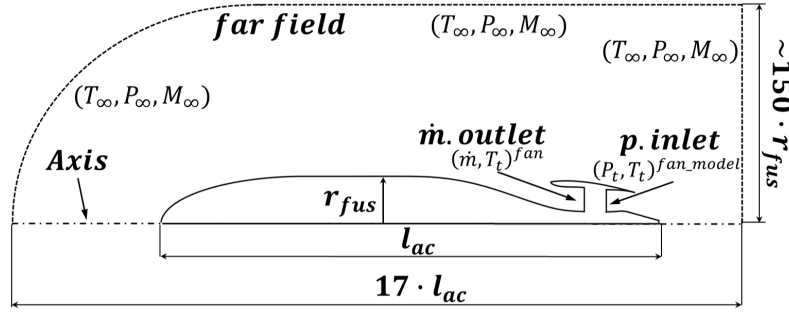


FIGURE 3.22: Boundary conditions for the 2D axisymmetric model with one dimensional fan

- **Throughflow turbomachinery approach:** In the throughflow turbomachinery approach there are no boundaries between the main domain and the fan duct. A continuous fluid domain is applied at the fan (Fig. 3.23). The turbomachinery is modelled through the application of source terms based on the local flow and turbomachinery geometry as described in Subsection 3.4.2.3. The tangential velocity component is considered constant across the azimuthal coordinate.

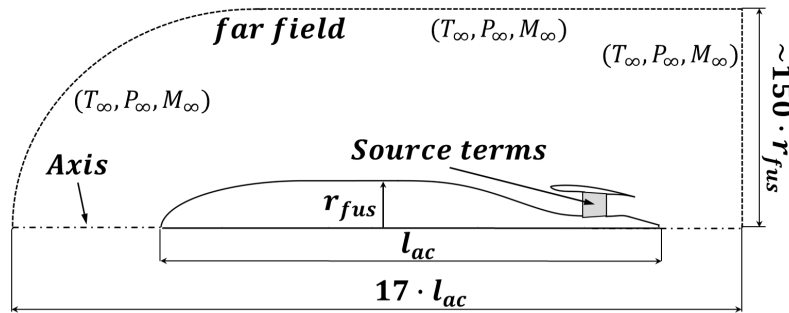


FIGURE 3.23: Boundary conditions for the 2D axisymmetric model with through-flow turbomachinery model

3.4.2 Turbomachinery models

The evaluation of the impact of the propulsion integration on the aerodynamic performance requires a representative model of the turbomachinery. On the present work, two approaches (Fig. 3.2) are presented: a semi-coupled one-dimensional fan model, and a fully coupled throughflow turbomachinery model. The semi-coupled model defines the turbomachinery effects based on a one-dimensional cycle with a fixed fan efficiency (isentropic or polytropic) and a variable fan pressure ratio (FPR). On the other hand, the coupled throughflow approach models the turbomachinery effects based on the local flow conditions, the fan geometry and the rotational speed. The methods behind both approaches are presented in this section.

3.4.2.1 One-dimensional fan model

On this approach, the effect of the turbomachinery on the flow is controlled by a fan pressure ratio, FPR , (Eq. 4.2) and the stage efficiency (Eqs. 3.67-3.68). A prescribed point on the turbomachinery cycle could be applied with these inputs. However, the operating conditions of the embedded propulsor can affect the exhaust flow requirements and the operating point of the cycle becomes complex to match. This can be easily understood by the observation of the flow conditions at different stations of the embedded propulsor (Fig. 3.25).

The mass flow rate prescribed at the fan face (Station 3) must agree with the mass flow at the exhaust duct (Stations 4 and 5). However, the mass flow at Station 4 depends on the dimensions of the propulsor, the local static pressure at the discharge plane (Station 5), and the total pressure and total temperature defined by the fan operation (Fig. 3.25). The system dimensions are defined based on the boundary layer at a selected station of the fuselage (Station 1). The aft fuselage geometry and the evolution of the boundary layer result in a total pressure drop (ΔP_t) from the Station 1 to the highlight plane of the propulsor (Station 2). This changes the total pressure at the intake for the target mass flow rate ($\dot{m}_{tgt}$) defined at the fan face boundary. The change on the total pressure at the fan face ($P_{t,3}$) modifies the total pressure and total temperature values at the exhaust inlet (Station 4). As defined in Equation 3.41, the mass flow rate in the exhaust duct is prescribed by these two magnitudes ($P_{t,4}$ and $T_{t,4}$), the area at the discharge plane (Station 5), and the effective fan nozzle pressure ratio (λ_{eff}). This presents two problems for the preliminary design of the exhaust duct. First, the aft-fuselage and intake geometry changes the ΔP_t between stations 1 and 3. This change affects the exhaust

operating conditions by the modification of the total pressure at the fan face. While a preliminary guess of this value can be obtained, some uncertainty on the mass flow rate at the exhaust duct is expected. Second, the effective static pressure at the discharge of the embedded propulsor is strongly affected by the boundary layer and the exhaust geometry⁴⁹. This varies the effective fan nozzle pressure ratio based on the fan cowl geometry and affects the effective value of the mass flow at the exhaust. These changes on the mass flow rate at the exhaust duct violate the continuity of the flow on the model. Hence, a correction of this value based on the fan performance is required.

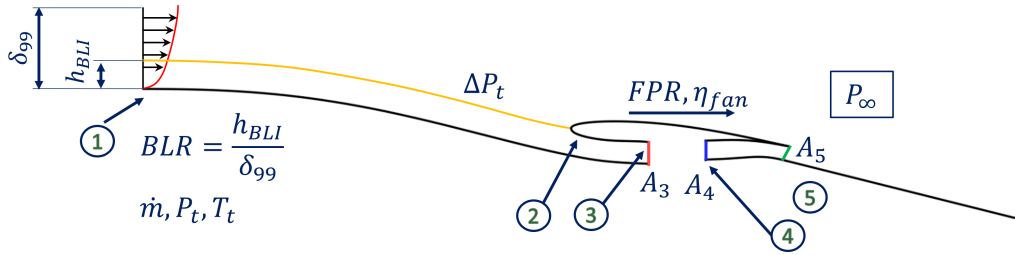


FIGURE 3.25: Sketch of the stations of the flow on the aft mounted boundary layer propulsor

Since the static pressure at the charging plane (Station 5) and the total pressure at the fan face (Station 3) are defined by the aerodynamic design and the operating conditions, the only degrees of freedom to match the mass flow rate between Stations 3 and 4 are the FPR (Eq. 4.2) or the propulsor efficiency (Eqs. 3.67-3.68). To simplify the model for the propulsor response, a variable FPR approach with a fixed efficiency was proposed (Fig. 3.26). The FPR correction is based on the capacity value at Station 4 (Eq. 3.69). The values of total pressure and temperature at station 4 are obtained with a mass average method as recommended by Cumpsty et al¹⁵⁶. The effective value of the mass flow rate at the exhaust ($\dot{m}_4$) is also obtained at Station 4. Then, the Q_4 value is changed to match the target mass flow rate value of the fan face $\dot{m}_3$ and an ideal capacity based on the FPR and the selected efficiency. Two approaches have been developed, one for a fixed isentropic efficiency (Eq. 3.70) and a second approach for a constant polytropic efficiency (Eq. 3.71). In both approaches, the new value of the FPR at a given iteration is obtained by a Newton-Raphson scheme from Equations 3.70-3.71 until equilibrium is reached (prediction error below 1×10^{-6}). With the new FPR guess, the CFD solver approach is run for a fixed number of iterations to converge the mass flow at the exhaust. Then a new guess of FPR is proposed. The FPR value is iterated until the mass flow imbalance (Eq. 3.72) is below 0.05%.

$$FPR = \frac{P_{t,4}}{P_{t,3}} \quad (3.66)$$

$$\eta_{is} = \frac{FPR^{\frac{\gamma-1}{\gamma}} - 1}{\frac{T_{t,4}}{T_{t,3}}} \quad (3.67)$$

$$\eta_{poly} = \frac{\gamma - 1}{\gamma} \frac{\ln\left(\frac{P_{t,4}}{P_{t,3}}\right)}{\ln\left(\frac{T_{t,4}}{T_{t,3}}\right)} \quad (3.68)$$

$$Q_4 = \frac{\dot{m}\sqrt{T_t}}{P_t A} \Big|_{Station\ 4} \quad (3.69)$$

$$Q_4 = \frac{\dot{m}_3\sqrt{T_t}}{P_t A} \Big|_{Station\ 4} = \frac{\dot{m}_3 \sqrt{T_{t,3} \left(1 + \frac{FPR^{\frac{\gamma-1}{\gamma}} - 1}{\eta_{is}}\right)}}{P_{t,3} FPR A_4} \quad (3.70)$$

$$Q_4 = \frac{\dot{m}_3\sqrt{T_t}}{P_t A} \Big|_{Station\ 4} = \frac{\dot{m}_3 \sqrt{T_{t,3} FPR^{\frac{\gamma-1}{\eta_{poly}\gamma}}}}{P_{t,3} FPR A_4} \quad (3.71)$$

$$\widehat{m_{imb}} = \frac{\dot{m}_3 - \dot{m}_4}{\dot{m}_3} \times 100 \quad (3.72)$$

To maintain numerical stability during the mass flow rate iteration, the mass flow correction is activated on three steps (Fig. 3.27). First, the CFD model is converged for a fixed preliminary total pressure and temperature values at the exhaust inlet. Then, the initial FPR value and the efficiency are used to define the new total pressure and temperature values at the exhaust inlet based on the mass weighted average of these metrics at the fan face. After that, the FPR value is evaluated again every few CFD iterations until the mass flow imbalance tolerance is achieved.

3.4.2.2 Radial profiles at the stage exit

In addition to the fixed value one-dimensional fan model, a methodology to define radial profiles at the fan stage exit (Station 4) for the total temperature and total pressure values is applied based on the work of Goulos et al.⁹⁹. A parabolic definition of the radial distributions of total temperature and total pressure is used. With this approach, different load distributions and intensities can be applied to model the system response (Fig. 3.28). With the radial profiles the sensitivities of the exhaust system to the fan stage outlet conditions can be evaluated.

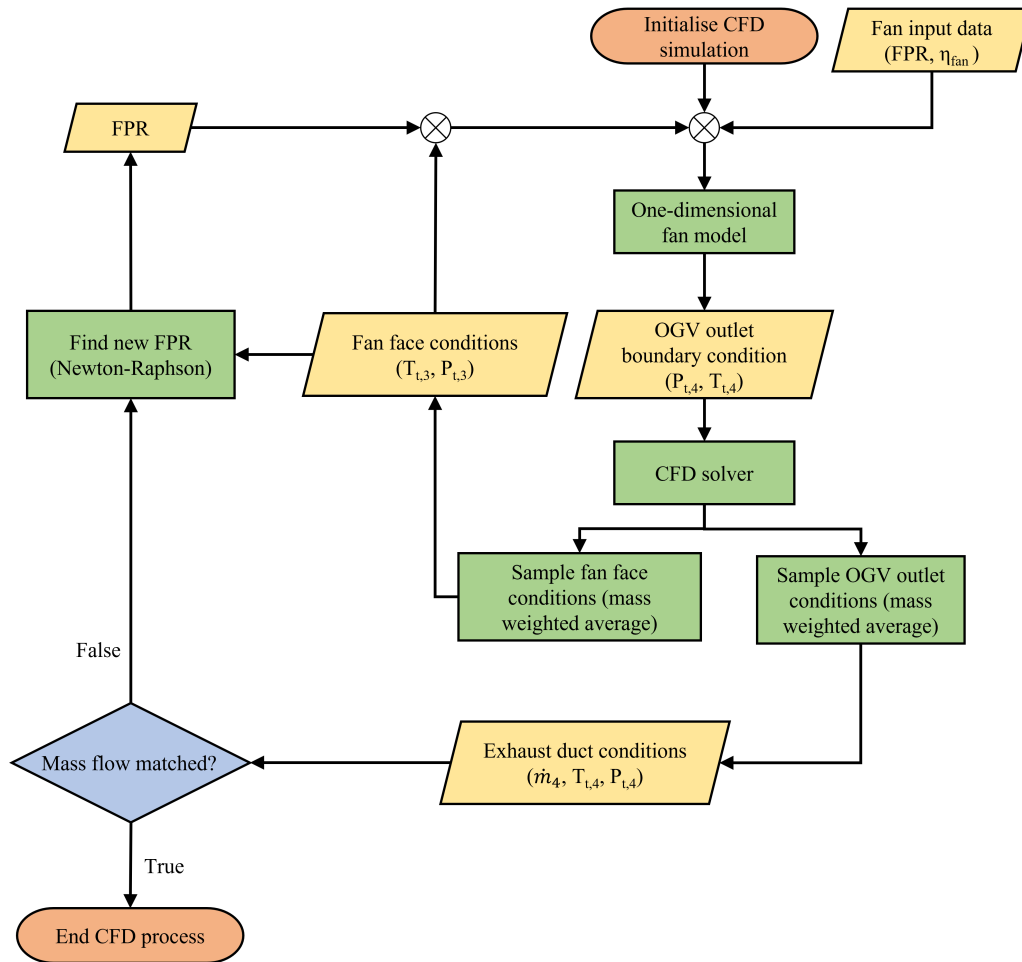


FIGURE 3.26: Workflow for the mass flow rate matching model

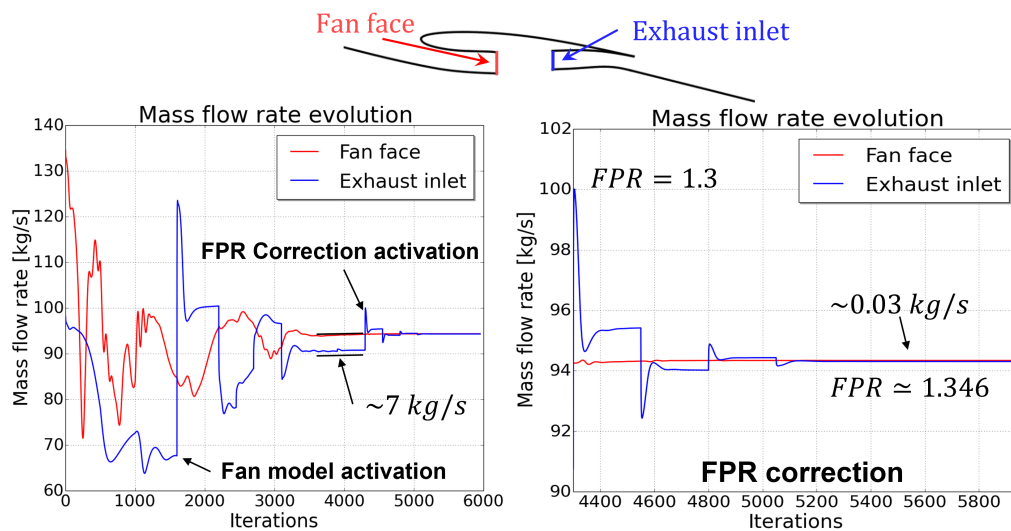


FIGURE 3.27: Verification of the mass flow matching condition at the fan

The profiles are controlled by two main parameters (Eqs. 3.73-3.74): the ratio of the maximum value ($P_{t,max}$) to the area averaged value ($\bar{P}_t$), and the radial location of the maximum total pressure ($r_{max\ P_t}$). The radial location of the maximum pressure is normalised by the inlet boundary upper and lower radial constraints to define the parameter h (Eq. 3.74). Then, the parabolic profile (Eq. 3.75) is defined by the amplitude, a ; the position of the maximum, h ; and the maximum value, $P_{t,max}$. The amplitude, a , is derived analytically based on the space constraints and the maximum value (Eq. 3.76). An analogous definition of the total pressure profile can be applied for the total temperature value and the swirl angle. The mass flow correction of the one-dimensional fan model is also applied for the radial profiles.

$$\bar{P}_t = \frac{2\pi \int_{r_{min}}^{r_{max}} P_t(r) dr}{\pi (r_{max}^2 - r_{min}^2)} \quad (3.73)$$

$$h = \frac{r_{max\ P_t} - r_{min}}{r_{max} - r_{min}} \quad (3.74)$$

$$P_t = a (r - r_{max\ P_t})^2 + P_{t,max} \quad (3.75)$$

$$a = \frac{\bar{P}_t - P_{t,max}}{\left(\frac{r_{max}^2 + r_{min}^2}{2} + r_{max\ P_t}^2 - \frac{4}{3} \frac{r_{max}^2 + r_{min}^2 + r_{max\ P_t} r_{min}}{r_{max} - r_{min}} r_{max\ P_t} \right)} \quad (3.76)$$

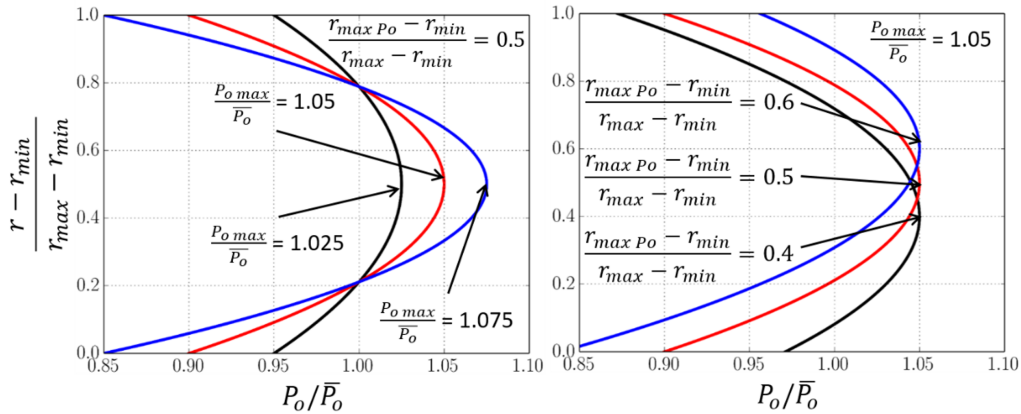


FIGURE 3.28: Example of the radial profiles distribution for the OGV outlet boundary

3.4.2.3 Hall-Thollet Model

An inviscid Body Force Model (BFM) based on the discrete turbomachinery element was originally proposed by Hall¹¹¹. The formulation was modified for viscous flows

with an explicit formulation of the turbomachinery losses and compressibility effects by Thollet¹⁷. The source terms are formulated in a flow relative reference frame with two main components: the normal body force, f_n , and the parallel body force, f_p (Fig. 3.29). The normal force is responsible of the generation of the main turning of the flow, and the parallel force is responsible of the generation of the losses of the turbomachinery. These components are projected to the absolute frame of reference and added to the main flow equations (Eqs. 3.77-3.79). The metal blockage effect (Fig. 3.29) is modelled with the blockage factor (Eq. 3.80).

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = -\frac{1}{b} (\rho \mathbf{V} \cdot \nabla b) \quad (3.77)$$

$$\frac{\partial \rho \mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V} - \nabla \cdot \bar{\bar{\tau}} + \nabla p = \rho \mathbf{f} - \frac{1}{b} (\rho \mathbf{V} \cdot \nabla b) \mathbf{V} \quad (3.78)$$

$$\frac{\partial \rho e_t}{\partial t} + \nabla \cdot (\rho h_t \mathbf{V} - \bar{\bar{\tau}} \cdot \mathbf{V}) = \rho \Omega f_\theta - \frac{1}{b} (\rho h_t \mathbf{V} \cdot \nabla b) \quad (3.79)$$

$$b = N_{blades} \frac{\theta_{ps} - \theta_{ss}}{2\pi} \quad (3.80)$$

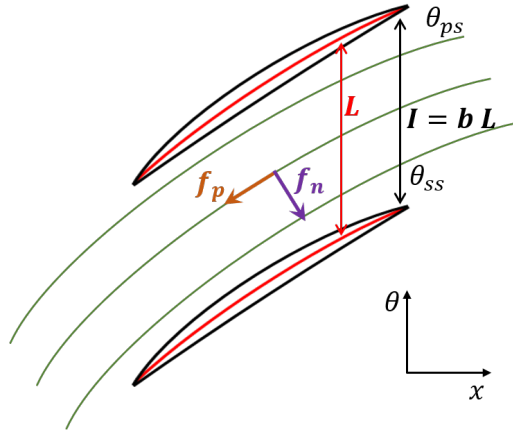


FIGURE 3.29: Definition of the body force components and the blockage factor

Hall's BFM¹¹¹ includes a normal force definition for an inviscid and incompressible flow formulation (Eq. 3.81). This model provides the theoretical lift coefficient of an isolated flat plate for the blade and it is consistent with the behavior in the infinite solidity limit. This formulation does not require calibration, and it is only dependent on the geometric definition of the turbomachinery element defined by the normal of the blade or vane camber, $\mathbf{n}$. The formulation of the normal force depends on the local pitch, s (Eq. 3.82); the local deviation angle, δ (Eq. 3.83);

and the local relative velocity magnitude, W (Eq. 3.84).

$$f_n = 0.5 \frac{W^2 2\pi \delta}{s |n_\theta|} \quad (3.81)$$

$$s = \frac{2\pi r}{N_{blades}} \quad (3.82)$$

$$\delta = \arcsin \left(\frac{\mathbf{W} \cdot \mathbf{n}}{|\mathbf{W}|} \right) \quad (3.83)$$

$$\mathbf{W} = \mathbf{V} - \Omega r \cdot \mathbf{e}_\theta \quad (3.84)$$

Thollet¹⁷ proposed a compressible and viscous approach of Hall's BFM by the addition of a blockage and compressibility correction, as well as a parallel force definition to model the blade losses. The compressibility correction is based on the Prandtl-Glauert correction for the incompressible regime and an Ackeret formulation for the supersonic regime. The compressibility effect is included to the normal (Eq. 3.85) and parallel force (Eq. 3.86) components by a compressibility correction factor, K_{Mach} (Eq. 3.87). To model the losses Thollet defines two main loss sources. The first source models the losses with a simple friction coefficient correlation for a turbulent flow flat plate (Eq. 3.88). The second is a calibration term to match the maximum efficiency point location by the selection of a reference point for the calibration deviation angle.

To define the body forces orientation, the plane containing the local relative velocity vector and the local blade normal is first defined. Then the normal force component must be orthogonal to the normal of this plane and to the parallel force component. Since the parallel force component is defined to be oriented against the relative velocity (Eq. 3.89), the normal force must be orthogonal to both the normal to the plane and the relative velocity vector (Eq. 3.90).

$$f_n = \frac{K_{Mach} 0.5 W^2 2\pi \delta}{sb |n_\theta|} \quad (3.85)$$

$$f_p = \frac{0.5 W^2}{sb |n_\theta|} \left(2C_f + 2\pi K_{Mach} (\delta - \delta^{\eta_{cal}})^2 \right) \quad (3.86)$$

$$K_{Mach} = \begin{cases} \min \left(\frac{1}{\sqrt{1-M_r^2}}, 3 \right), & \text{if } M_r < 1 \\ \min \left(\frac{1}{2\pi\sqrt{M_r^2-1}}, 3 \right), & \text{if } M_r > 1 \end{cases} \quad (3.87)$$

$$C_f = 0.0592 Re_x^{-0.2} \quad (3.88)$$

$$\frac{\mathbf{f}_p}{|\mathbf{f}_p|} = \frac{\mathbf{W}}{|\mathbf{W}|} \quad (3.89)$$

$$\frac{\mathbf{f}_n}{|\mathbf{f}_n|} = \mathbf{W} \times (\mathbf{W} \times \mathbf{n}) \quad (3.90)$$

3.4.2.4 Hall-Thollet model implementation

The Hall-Thollet model was implemented (Fig. 3.30) with the User Defined Functions (UDF) framework of ANSYS Fluent¹⁵⁷. The UDF framework allows the addition of source terms to the flow equations and to carry the necessary intermediate calculations. The model requires the introduction of the geometry ($\mathbf{n}$), metal blockage terms (b and ∇b), calibration deviation angle values ($\delta^{\eta_{cal}}$), and the fan current operating conditions (Ω). The methodology applied to obtain these values can be found in the Appendix E. These inputs are stored within the memory of the solver for each cell center of the prescribed domain with User Defined Memory tools. To smooth the gradient of the values for different mesh resolutions a bilinear interpolation of the input values at each cell center location is used.

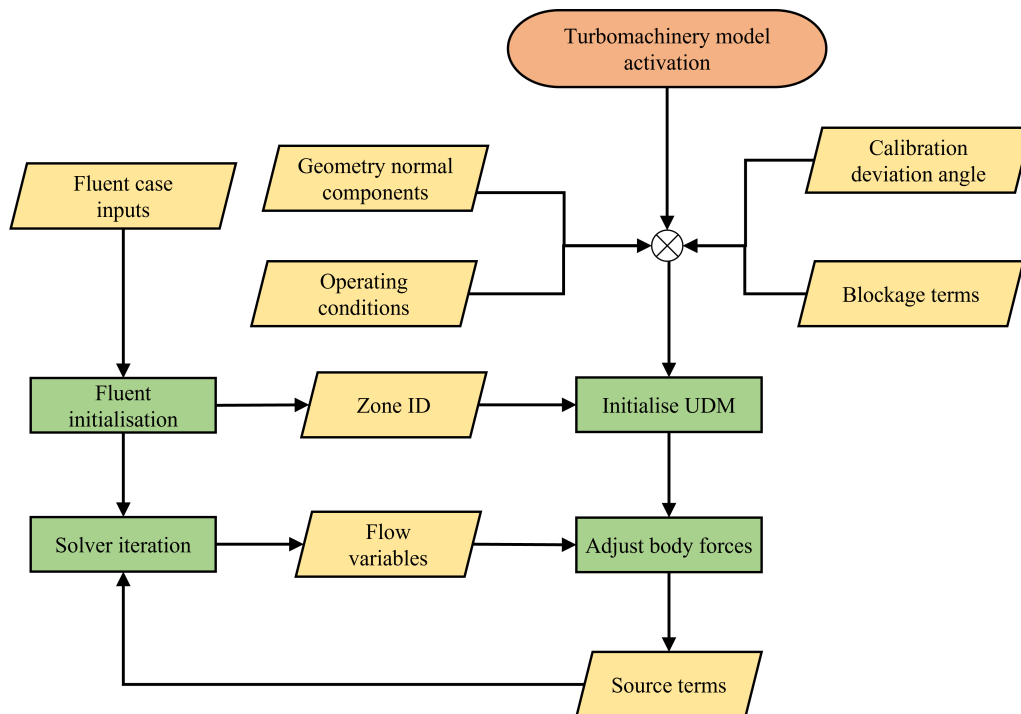


FIGURE 3.30: Hall-Thollet BFM workflow on the User Defined Function framework of ANSYS Fluent

Two numerical approaches were generated: a 2D-axisymmetric with swirl¹⁵⁵ approach and a 3D implementation. The 2D approach models the flow in a 2D domain

and assumes no variation on the out of plane tangential or swirl component based on a reference rotation axis. The 3D approach models the full 3D flow around the turbomachinery domain. A detailed validation of this implementation for a reference configuration can be found in Chapter 6.

3.4.3 Mesh methodology

This work evaluates the aerodynamic impact of the propulsion integration design in different levels of detail (Fig. 3.2). Thus, different mesh methodologies are applied across the different chapters to capture the main aerodynamic characteristics of the embedded propulsion system. The main mesh methods used in the present work are:

- Structured multi-block mesh for the 2D axisymmetric configurations with semi-coupled one-dimensional propulsor
- Structured multi-block mesh for the 2D axisymmetric configurations with coupled through-flow propulsor
- Hybrid mesh for the 3D non-axisymmetric configuration with through-flow propulsor

3.4.3.1 Structured mesh for the 2D axisymmetric configurations with semi-coupled one-dimensional propulsor

In this mesh approach, a C-shape domain (Fig. 3.31) is applied with the axisymmetric airframe and propulsor located in the center. The turbomachinery stage of the propulsor is not included in the flow domain. A multi-block structured mesh is used in this case. A y^+ value of one is used for all the wall boundaries. The maximum growth ratio of the cells normal to a surface is set to 1.2. The domain extends 8 aircraft lengths (l_{ac}) vertically and 17 aircraft lengths axially. A grid sensitivity study based on the methodology proposed by Celik et al.¹⁵⁸ was adopted. Three progressive mesh refinements were applied with 2.79×10^5 , 5.09×10^5 , and 9.28×10^5 elements. The intermediate mesh level (5.09×10^5) was selected.

A detailed definition of the uncertainty accounting for this mesh configuration can be found in Appendix C.2.2. The uncertainty includes the iterative convergence uncertainty and the mesh uncertainty. For the selected mesh resolution, an uncertainty

of $\pm 1.84 \times 10^{-4}$ units of the TS , and ± 2.3 m/s for the $TSPC$ was obtained. On the intake performance, the error obtained was of $\pm 5 \times 10^{-4}$ units and $\pm 7.7 \times 10^{-5}$ for the RDI and IPR_{ing} , respectively. Similarly, for the exhaust performance the cumulative error is estimated to be $\pm 3.1 \times 10^{-5}$ units, $\pm 4.31 \times 10^{-5}$ units, and $\pm 1.4 \times 10^{-5}$ units for the C_V , C_V^* , and the C_D , respectively.

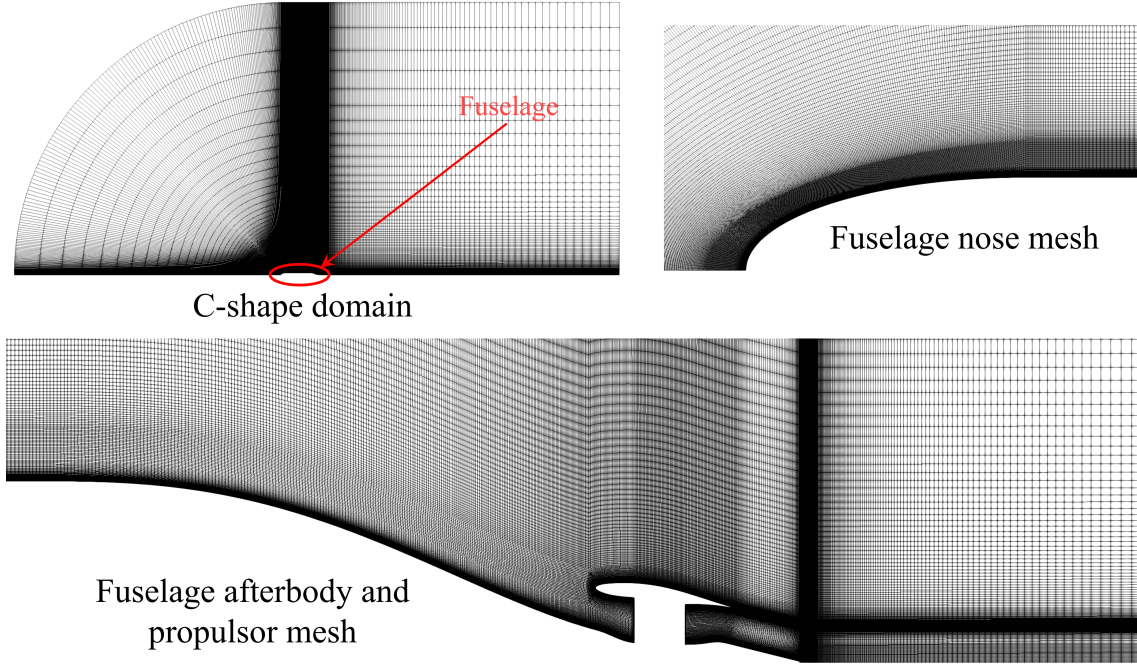


FIGURE 3.31: Mesh domain and details for the 2D axisymmetric configuration with semi-coupled one dimensional propulsor

3.4.3.2 Structured mesh for the 2D axisymmetric configurations with coupled through-flow propulsor

In this approach, a C-shape domain (Fig. 3.31) is applied with the airframe and propulsor located in the center. The turbomachinery domain is also included. A multi-block structured mesh is applied. A y^+ value of one is used for all the wall boundaries. The maximum growth ratio of the cells normal to a surface is set to 1.2. The mesh on the turbomachinery section maintains the radial element continuity from the intake to the exhaust. The blocking on the turbomachinery section is adjusted to match the blade and vane domain for the low order turbomachinery model (Fig. 3.32). The domain extends 8 aircraft lengths (l_{ac}) vertically and 17 aircraft lengths axially. A grid sensitivity study based on the methodology proposed by Celik et al.¹⁵⁸ was adopted. Three progressive mesh refinements were applied

with 3.91×10^5 , 7.08×10^5 , and 1.286×10^5 elements. The intermediate mesh level (7.08×10^5) was selected.

A detailed definition of the uncertainty accounting for this mesh configuration can be found in Appendix C.2.3. The uncertainty includes the iterative convergence uncertainty and the mesh uncertainty. For the selected mesh resolution, an uncertainty of $\pm 1.925 \times 10^{-5}$ units of the TS , and ± 1.95 m/s for the $TSPC$ was obtained. On the intake performance, the error obtained was of $\pm 9.98 \times 10^{-5}$ units and $\pm 1.956 \times 10^{-4}$ for the RDI and IPR_{ing} , respectively. Similarly, for the exhaust performance the cumulative error is estimated to be $\pm 4.03 \times 10^{-5}$ units, $\pm 1.53 \times 10^{-5}$ units, and $\pm 5.05 \times 10^{-5}$ units for the C_V , C_V^* , and the C_D , respectively.

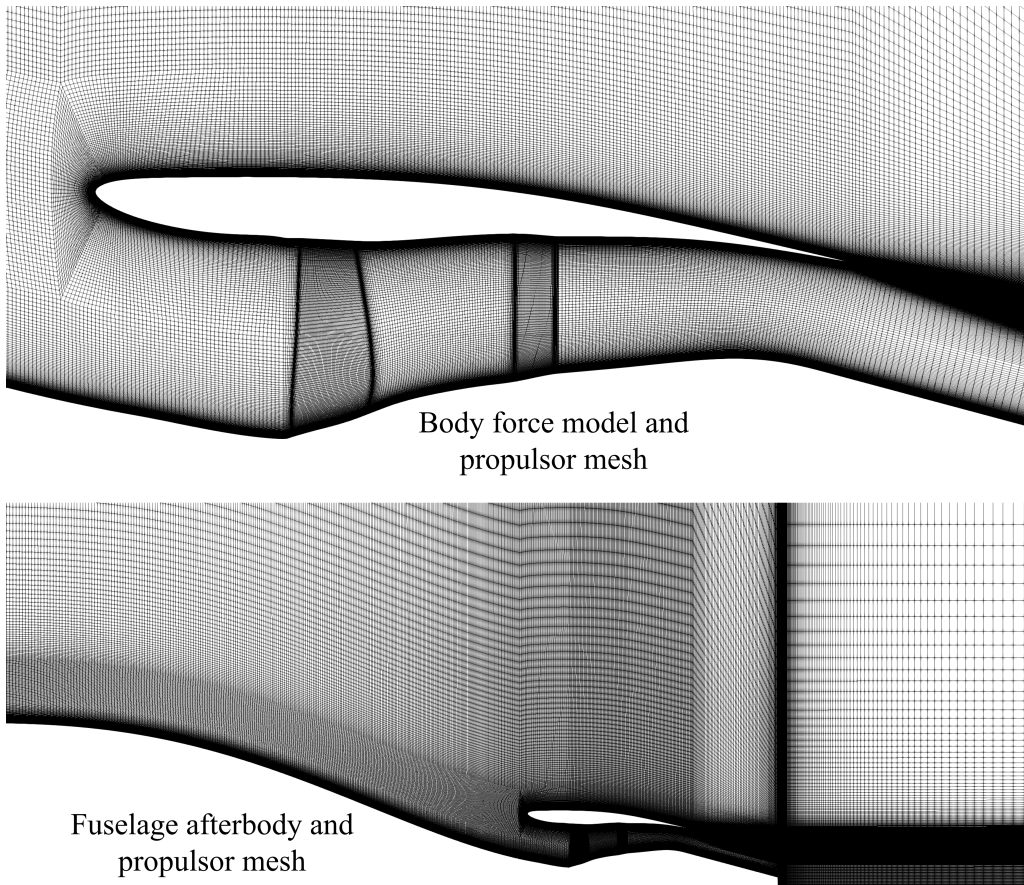


FIGURE 3.32: Mesh details for the 2D axisymmetric configuration with coupled throughflow low order turbomachinery model

3.4.3.3 Hybrid mesh for the 3D non-axisymmetric configuration with through-flow propulsor

An spherical domain with the 3D airframe and propulsor placed in the center is applied for this approach. A hybrid mesh approach is used, with an unstructured tetrahedral mesh over the airframe and the propulsor housing and a structured hexahedral mesh in the turbomachinery domain (Fig. 3.33). This mesh methodology was selected due to the complexity of the airframe and embedded propulsor geometry. These geometries limit the application of structured multi-block approaches, due to the increase on human time and effort to define the geometry. The structured multi-block approaches are also more complex to generalise for multiple designs with similar future. Under this context, the application of a hybrid unstructured mesh with tetrahedral elements enables an adequate representation of the surface boundaries of the mesh with limited human effort, but with a higher number of elements in comparison with structured multi-block meshes. On the other hand, the geometry within the turbomachinery domain is simple enough to apply a structured mesh approach, reducing the number of elements locally and maintaining an adequate orientation of the cells towards the flow gradients.

The presence of the aft-mounted annular propulsor on the aircraft breaks the vertical (XZ) symmetry of the problem. Hence, for the CFD evaluation of this configuration the full aircraft must be modelled (port and starboard sides). However, the unstructured mesh methodology presents two potential problems to model the full aircraft. First, the application of the unstructured mesh around the full aircraft does not generate identical discretisation between the starboard and port sides of the vehicle. This could lead to the emergence of non-physical lateral forces on the CFD solution. Second, the coupled low order turbomachinery model benefits of structured grids to control the local definition of the cells where the source terms are applied. Hence, a hybrid mesh methodology in two parallel stages was proposed to avoid any of this problems (Fig. 3.34).

To obtain equal grid spacing between both sides of the unstructured aircraft the mesh is generated with a symmetry plane. Then the mesh is mirrored and the boundaries and domain are merged at solver level (Fig. 3.35). This approach has the additional advantage of reducing the mesh generation time and the required computational resources to half. The turbomachinery domain is created as a full annulus structured mesh on an independent process. Both mesh approaches cannot be merged at solver level because of the discrepancy between the element shape and position. As a solution, a flow interface is defined between both meshes at

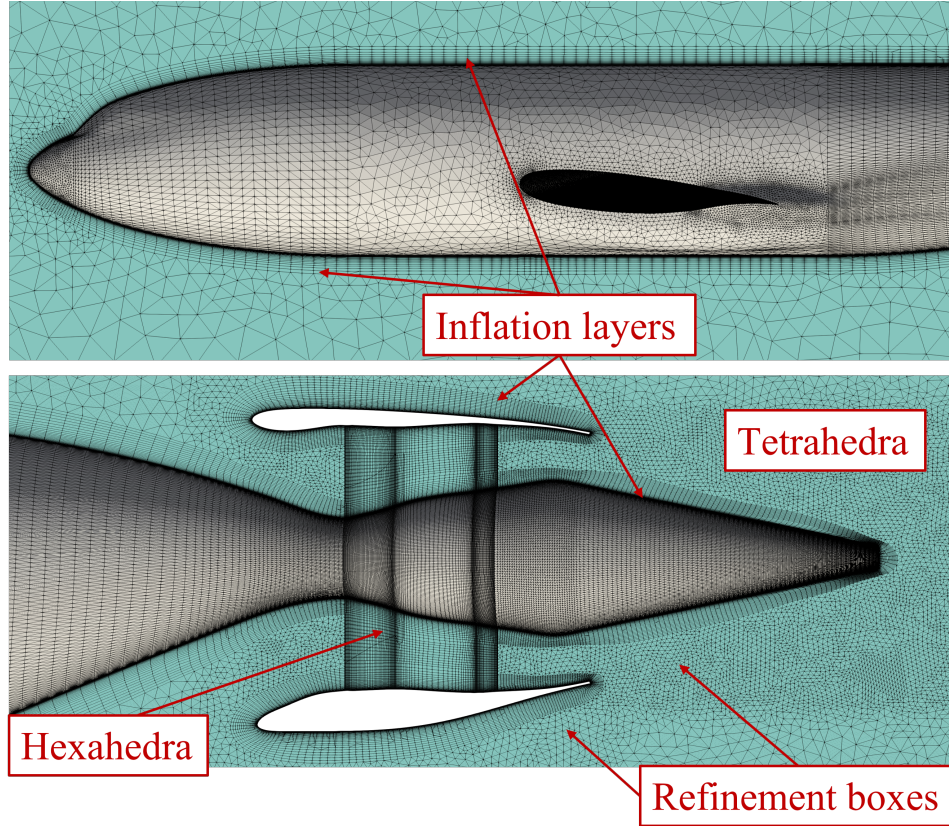


FIGURE 3.33: Detailed view of the main regions of the 3D aircraft mesh

the turbomachinery flow entry and exit. This approach interpolates the flow fields between boundaries. The mesh generation was adapted to minimize the interference between boundaries and reduce the interpolation error.

The half aircraft mesh is included in a hemispherical domain with a radius of a 100 times the reference aircraft chord. To control the mesh resolution of the unstructured mesh, density boxes are defined in the regions of interest of the flow (Fig. 3.33). Inflation layers are applied to improve the capture the boundary layer by the mesh (Fig. 3.33). Standard wall functions are applied to reduce the mesh density and reduce the computational cost. The y^+ value is between 50 and 100 to allow the standard wall functions to operate. A grid sensitivity study based on the methodology proposed by Celik et al.¹⁵⁸ was applied. Three progressive mesh refinements were applied with 50.11×10^6 , 88.5×10^6 , and 168.2×10^6 elements. The intermediate mesh level (88.5×10^6) was selected.

A detailed definition of the uncertainty accounting for this mesh configuration can be found in Appendix C.3.2. The uncertainty includes the iterative convergence

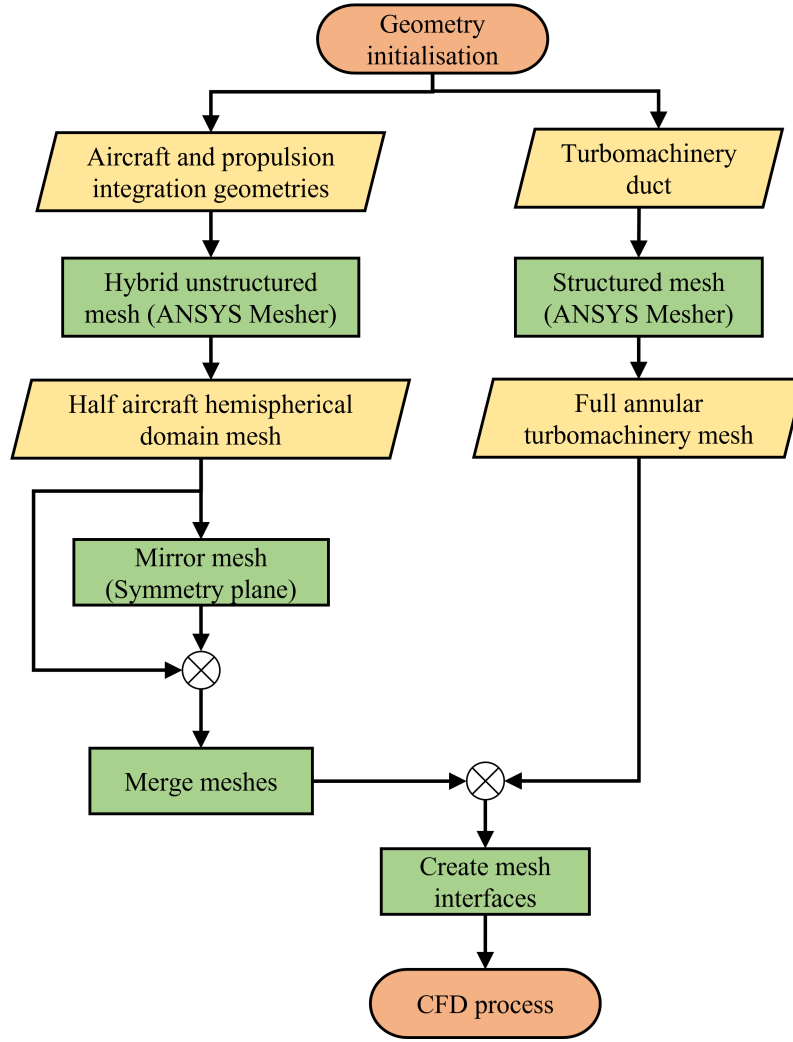


FIGURE 3.34: Methodology for the 3D hybrid mesh methodology

uncertainty and the mesh uncertainty. For the selected mesh resolution, an uncertainty of $\pm 3.51 \times 10^{-3}$ units of the TS , and ± 5.37 m/s for the $TSPC$ was obtained. On the intake performance, the error obtained was of $\pm 3.5 \times 10^{-3}$ units of IPR , $\pm 1.03 \times 10^{-3}$ units of RDI , $\pm 1.0269 \times 10^{-3}$ units of CDI , and $\pm 9.745 \times 10^{-3}$ units of DC_{60} . Similarly, for the exhaust performance the cumulative error is estimated to be $\pm 1.58 \times 10^{-3}$ units, and $\pm 3.49 \times 10^{-4}$ units for the C_V^* , and the C_D , respectively.

3.5 Multi-objective optimisation

The design and CFD methodology is compatible with a multi-objective optimisation strategy based on the parametric definition of the geometry. Evolutionary optimisation algorithms (EA) and particle swarm optimisation algorithms (PSO) were tested

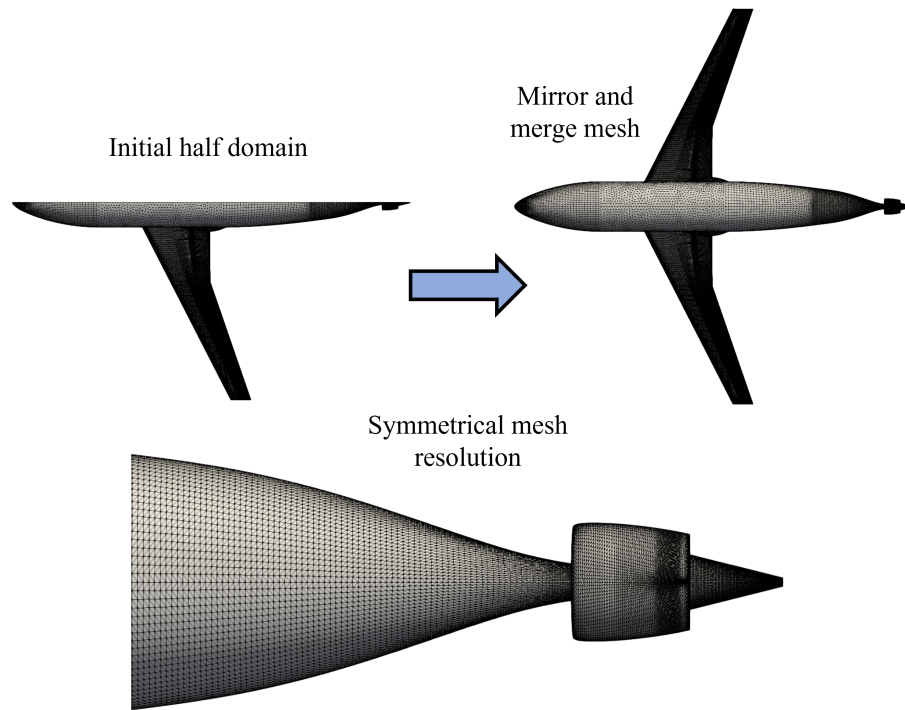


FIGURE 3.35: Sample of mirrored mesh control

for the optimisation of the embedded propulsive systems. The optimisation method used in the present work is based in two different sources. First, the Routine for Intake and Nacelle Genetic Optimisation (RINGO) developed by Robinson^{22,159} based on the Non-dominated Sorting Genetic Algorithm (NSGA-II)¹⁶⁰. Second, the open source optimisation library Platypus¹⁶¹ in python is applied with the Indicator-Based Evolutionary Algorithm (IBEA)¹⁶², and the Reyes-Coello Multi-Objective Particle Swarm Optimisation method¹⁶³ (OMOPSO). Both of the applied algorithm sources can be applied independently of the method to obtain the objective functions. Hence, besides the CFD-in-the-loop methodology, surrogate and low order models of the performance could be used to guide the optimisation question.

3.5.1 Framework

The optimisation process is different for the evolutionary algorithms such as NSGA-II and IBEA (Fig. 3.36), and the Particle Swarm algorithms as OMOPSO (Fig. 3.37). Nevertheless, the initialisation strategy and performance evaluation is the same for both types of algorithm. The process starts with a seed design space exploration and it is followed by a sequence of different generations with a fixed number of individuals. This sequence continues until the convergence condition is achieved. Each

generation proposes a number of individuals with particular attributes or degrees of freedom (DoF). In this work, the degrees of freedom are the individual parameters required to define the geometry of the propulsion integration. At the end of each generation, the objective functions (OF) for each individuals are evaluated. Then, the algorithm proposes the next generation of individuals based on these objective functions.

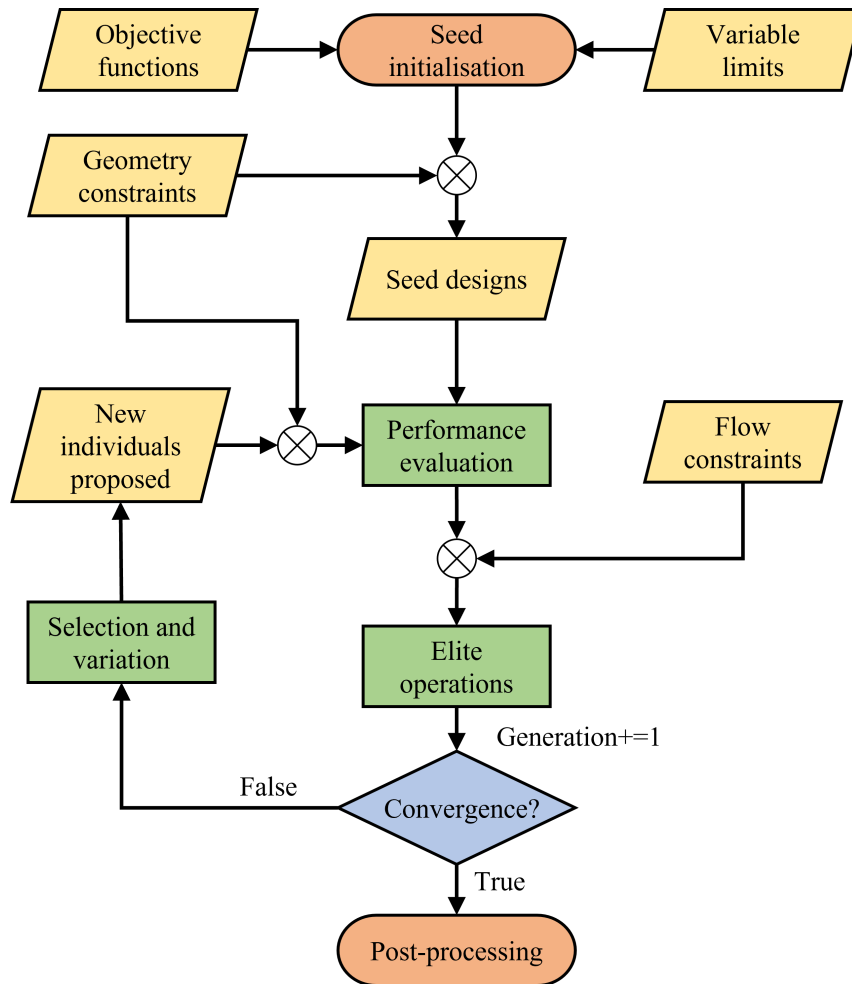


FIGURE 3.36: Work flow for the multi-objective optimisation process for the evolutionary algorithms

At the beginning of the optimisation process, the seed is initialised with an initial value for each degree of freedom (geometry parameters) for each design. This initialisation requires the definition of a target value of individuals for the initial seed, as well as the number of degrees of freedom and their limits. At this stage, geometry filters are applied to eliminate the designs with undesirable geometry features from the optimisation process. These filters are determined on predefined geometric constraints. To summarise, the geometry filters can be grouped into three types:

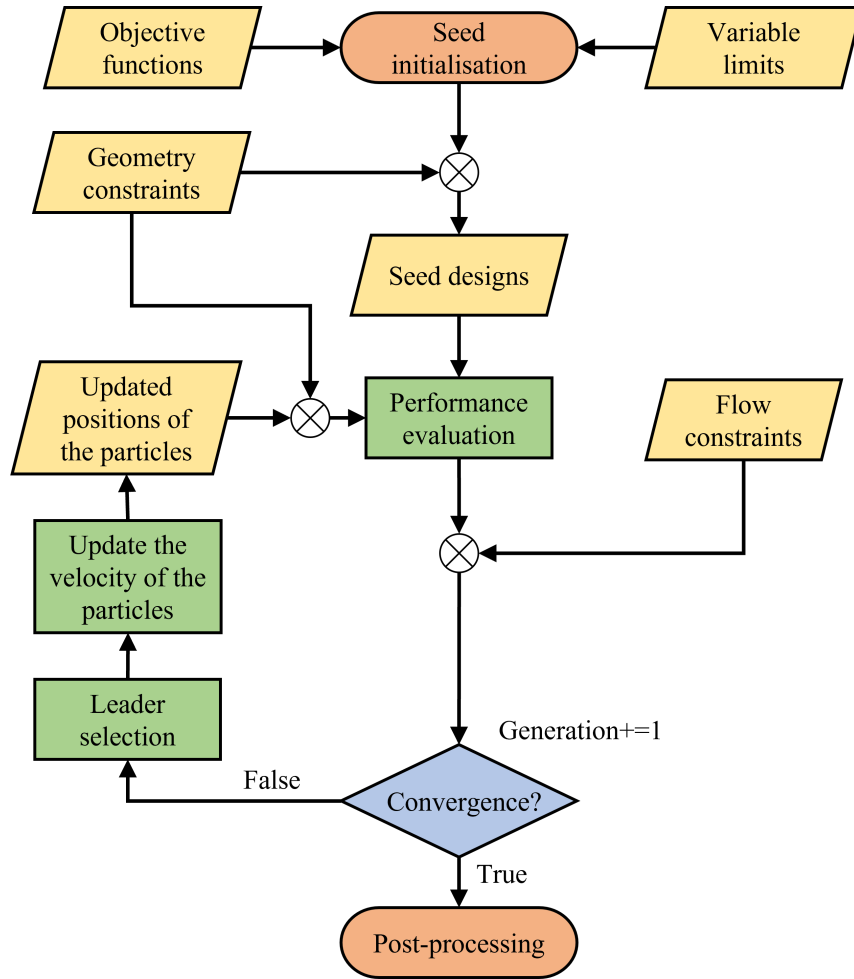


FIGURE 3.37: Work flow for the multi-objective optimisation process for the particle swarm algorithm

- **Length constraints:** These constraints fix the maximum or minimum length of a part of the system. E.g. the maximum length of the aircraft is constrained for all the optimizations based on the reference aircraft dimensions
- **Surface inflections:** These constraints control the change on the curvature polarity. This can force specified surfaces to be continuously concave ($\xi''(\varphi) < 0$) or continuously convex ($\xi''(\varphi) > 0$) by the control of the number of allowed inflection points ($\xi''(\varphi) = 0$)
- **Controlled offset filters:** These filters control the distance between two surfaces to avoid possible intersections between the different iCST curves of the housing components. Euclidean distance at a fixed axial location is applied ($\Delta\xi = \xi_{iCST-1} - \xi_{iCST-2} > Offset$). E.g. after a fixed distance from the trailing edge, the minimum distance between the fan cowl and the outer exhaust geometry is constrained to keep the design under manufacturing tolerance

The specific geometry filters for each optimisation problem are defined in the different chapters of this work. The application of the geometry constraints reduces the number of available geometries whose performance can be evaluated. To ensure that the target number of individuals on the initialisation seed is obtained, the seed sizing is performed in two stages. First, a sample with a fixed number of individuals is proposed. This initial seed is based on a Latin Hypercube Sampling (LHS)¹⁶⁴ to evenly populate the design space. The selected geometry filters are then applied to this initial sample. Finally, the number of successfully generated geometries is then used to obtain the new seed dimensions (Eq. 3.91).

$$N_{seed} = N_{seed-required} \frac{N_{initial-sample}}{N_{valid-geometries}} \quad (3.91)$$

The new proposed seed dimension is produced again with a LHS methodology. The performance of the designs is then evaluated based on the CFD results. At this stage a classification of the designs based on a set of aerodynamic constraints is applied. The constraints are evaluated from the CFD results. The individuals of the generation are classified into viable and non-viable designs. The non-viable designs are excluded from the optimisation before the algorithm proposes a new set of designs for the next generation. The aerodynamic constraints applied on the present work are the following:

- **No-separation of the flow:** This condition is imposed in all the housing components (Fig. 3.3) of the design. The flow must remain attached for the design to be considered. Separation is assumed when negative values of the axial wall shear stress are present ($\tau_{ws,x} < 0$). Under some conditions with lower *MFCR* or *BLR* values, the location of the stagnation point moves towards the intake duct. This produces some spillage loss with a local reverse flow direction around the fan cowl tip and a change on the $\tau_{ws,x}$ polarity. Nevertheless, this does not imply flow separation. Thus, the region of the nacelle highlight is excluded from the separation analysis to avoid any misclassification of the designs
- **Maximum fan face Mach number.** This condition sets a maximum tolerable fan face Mach number based on current blade design requirements. The condition is evaluated at the fan face boundary. A mass weighted average of the Mach number¹⁵⁶ is applied
- **Maximum Mach number at the intake duct.** Shock waves could appear at the intake duct based on the overacceleration of the flow at the intake lip.

This will have a negative impact on the aerodynamic performance of the system and could lead to flow separation at the intake. Thus, the maximum value of the Mach number at the intake is constrained to filter any over-acceleration at the intake during the optimisation process

Once the aerodynamic performance of the designs is evaluated, the optimisation algorithm is applied to propose the new designs. The new designs are proposed by the update of the particle velocities and position in OMOPSO¹⁶³ (Fig. 3.37), or the evolutionary selection and genetic variation in NSGA-II¹⁶⁰ and IBEA¹⁶² (Fig. 3.36). After the initial seed, the process is repeated until the convergence condition or the maximum number of generations is achieved.

3.5.2 Convergence evaluation

In a multi-objective optimisation problem a single optimum point cannot be defined. Instead, a set of optima can be identified over a n -dimensional Pareto front or Pareto-optimal set³⁰, where the n dimensions are the number of objective functions. The Pareto front is formed by the set of non-dominated designs on the direction of the objective functions¹⁶⁵. As in single-objective optimisation problems, Local Pareto-optimal sets ($\bar{Y}'$) can be found around local optima¹⁶⁵. The Global Pareto-optimal set ($\bar{Y}$) constitutes the set of the overall non-dominated results of the problem¹⁶⁵.

The identification of the Global Pareto-optimal set is the main objective of the MOO process. However, the design of experiments and the selection of the algorithm have an impact in the achievement of this goal. An assessment on the performance of the algorithms of the present work based on benchmark problems can be found in Appendix F. In this work, the final Pareto-optimal set of the algorithm is interpreted to be the Global after the convergence criteria is achieved. The convergence is evaluated through the Hypervolume Index²¹, HV (Eq. 3.92). This metric quantifies the evolution of the dimensions of the Local Pareto-optimal sets across the generations of individuals (Fig. 3.38). The convergence criteria was set based on the evolution of the HV in the last five generations. An minimum offset of 0.5% of the final HV value is imposed to achieve convergence.

$$\overline{HV} = volume \left(\bigcup_{i=1}^{|\bar{Y}'|} v_i \right) \quad (3.92)$$

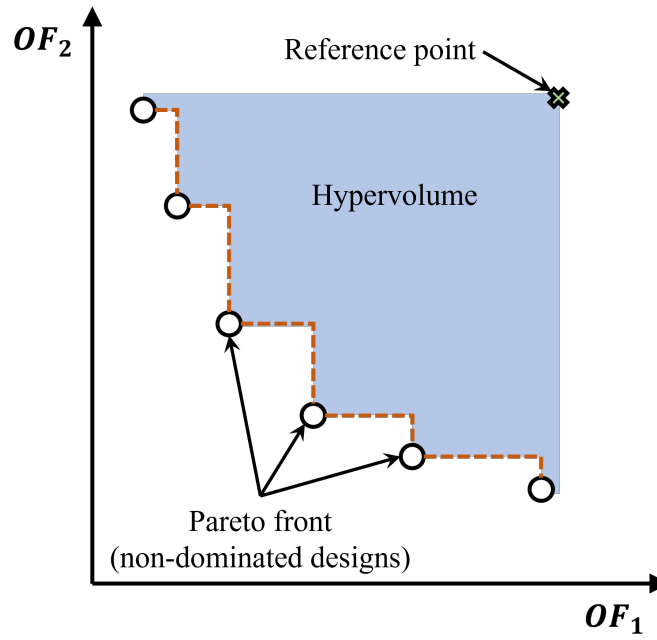


FIGURE 3.38: Schematic example of a two-dimensional hypervolume, based on Deb²¹ and Robinson²²

3.5.3 Configuration of the optimisation algorithms

3.5.3.1 NSGA-II

The Non-dominated Sorted Genetic Algorithm II (NSGA-II) is an evolutionary algorithm based on elite-preservation and diversity-preservation¹⁶⁰. In the algorithm the individuals of a generation are sorted by an elitism rank and the crowding distance between each other. The elitism rank is used to maintain the best individuals of each generation, while the crowding distance is applied to maintain the diversity of the population to avoid the stagnation of the solution in local Pareto fronts. The subsequent generations are defined based on the information of the ranked individuals. A mutation technique is required to merge the characteristics or genes of the selected designs. A crossover technique is also necessary to randomly introduce some diversity on the mutation. The RINGO implementation of NSGA-II^{22,159} applies a Gaussian Mutation operator and a blended crossover technique (BLX- α). Based on previous work on nacelle aerodynamics optimisation²², the standard deviation of the Gaussian mutation (σ) is set to 0.001 and α is set to 0.5.

3.5.3.2 IBEA

Indicator-Based Evolutionary Algorithm (IBEA)¹⁶² adapts the search based on a target function (Indicator Metric) of the performance of the optimisation. In this formulation, the Hypervolume Index is applied as the Indicator Metric. The fitness value of each individual is calculated based on the Indicator Metric. The best individuals are selected based on a ranking of the fitness value. The next generation of individuals is generated by the mutation and crossover of pairs of the selected individuals. In this implementation, A Simulated Binary Crossover approach is applied with a uniform mutation operator.

3.5.3.3 OMOPSO

The Multi-Objective Particle Swarm algorithm (OMOPSO) defined by Reyes-Sierra and Coello-Coello¹⁶³ is used. As all PSO algorithms, the method is based on semi-autonomous particles (individual designs) that communicate with each other and update their position in the design space. This is later used to formulate an updated velocity of the particle in a favourable direction for the optimisation. This algorithm used bounded random values for the inertia weight and the constraints that define the individual particle velocity. A Pareto Dominance and a crowding distance factor based on binary tournament¹⁶⁰ are applied for the leader selection. The maximum set of leaders is set to the size of the swarm. A combination of three different mutations is applied to the swarm to update the velocity and the position of the particles: no-mutation, uniform mutation, non-uniform mutation.

CHAPTER 4

Analysis of the propulsive requirements

In this chapter, the potential power savings derived from the aft mounted boundary layer ingestion (BLI) annular propulsor are evaluated. In this configuration, the power required by the aft-mounted propulsor can be either provided by an independent system (i.e. gas turbine or electric batteries) or be partially or fully extracted from the wing-mounted engines. In any of these cases, the presence of the BLI propulsor reduces the thrust requirement of the wing-mounted power-plants, which can result in power savings for the overall aircraft. In this work, the power savings are accounted based on the reduction of the fuel consumption of the wing-mounted power-plants due to the presence of the BLI propulsor. An exploration of the requirements for this approach is provided in Section 4.1. The methodology to obtain the cycle operating requirements for the wing-mounted conventional power plants is defined based on the cruise and take-off propulsive requirements of the aircraft (Section 4.2). Then, the methodology is applied to define a baseline wing-mounted engine configuration for the reference single-aisle medium-sized aircraft (Section 4.3). From this baseline engine configuration, the effect of the aerodynamic performance of the BLI propulsor is accounted for different types of power configurations for the aft mounted propulsor (Section 4.4). Finally, the main conclusions from this approach are summarised in Section 4.5.

4.1 Background

The aim of the present work is to define the methods for the aerodynamic design of the propulsion integration of novel embedded systems. This will develop into a detailed analysis of the aerodynamics and model requirements of such embedded

systems in future chapters. However, before the aerodynamic requirements for the design of the propulsion integration are evaluated, it is necessary to identify the relation between the aerodynamic performance of the aft-mounted propulsor and the potential power (or fuel) savings of the aircraft. Thus, an analysis of the potential benefits on the vehicle power consumption is performed in this chapter. The analysis is focused on the case study of an aft-mounted boundary layer ingestion annular propulsor. The aft mounted propulsor is assumed to be part of a distributed propulsion system with two other wing-mounted conventional power plants.

The definition of the power savings obtained with the embedded systems is a line of research by itself and the focus of many dedicated works^{1,44,89,149,151}. Thus, the selection of the most adequate approach to define the power benefits derived from the aft mounted propulsor requires further discussion. For example, in a very simplified approach, the power reduction of the system could be analysed only based on the changes of propulsive efficiency of the aft-mounted propulsor. However, this presents some limitations. The main limitation is that this approach would not include the effect of the propulsive power reduction on the power requirements for the conventional wing-mounted power-plants. Also, as it is observed before (Chapter 3.3.2), the determination of the propulsive efficiency is somehow abstract when applied to boundary layer ingestion systems.

Another approach could be the application of energy-based accounting methods such as the Power Balance⁷⁹ or the Exergy Analysis^{16,81} to account for the power benefits of the vehicle based on gains in kinetic energy. However, this would still require information about the performance of any additional propulsion system on the aircraft, such as the wing mounted engines. Besides, as it is discussed in Chapter 2.2, the application of these methods will have an associated increase in the computational cost.

Under these conditions, it becomes evident that the changes in the aircraft thrust requirements derived by the boundary layer ingestion propulsor need to be transformed into changes in the operation of the wing mounted power-plants. The inclusion of these wing mounted power plants on a high fidelity aerodynamic analysis such as CFD could be a computationally expensive process and could divert the attention from the design of the aft-mounted propulsor. A possible solution for this is to apply lower fidelity approaches (e.g. drag correlations) to estimate the drag reduction¹⁵³. Thus, to model the wing mounted engines, it is advisable to apply an approach that is initially decoupled from the aerodynamics. Then, the aerodynamic aerodynamic

performance of the aft-mounted propulsor must be linked to the decoupled approach to determine the benefits on power consumption of the full aircraft.

Works such as Welstead and Felder¹, and Seitz and Gologan⁴⁴ proposed a decoupled approach to quantify the benefit of the aft mounted propulsor on the overall power consumption of the aircraft. However, these methods are not defined to account for changes in the aerodynamic performance (i.e. thrust split and propulsive efficiency). One approach to combine the aerodynamic performance with the power savings is proposed by Sanders⁶⁴. Sanders accounted for the changes on power consumption by the application of a reference power-plant cycle for the non-BLI propulsors on the aircraft. On this approach, any benefit on the thrust contribution of the system is mapped to the change into the power requirements for the power-plant. With this approach, the power savings could be coupled with the aerodynamic performance of the aft-mounted propulsor. Nevertheless, the application of a fixed reference cycle does not provide an accurate representation of the power plant design process. During the conceptual design process, when propulsive power requirements are withdrawn from the wing-mounted engines the dimensions and cycle requirements of the engines are recalculated accordingly. Then, these benefits can be compared with the reference configuration to understand the actual benefits on the system power consumption. At the same time, the power consumption of the aft-mounted propulsor should be included to evaluate the savings derived from the increase on the aerodynamic performance of the overall system.

Hence, an improvement of the methodology defined by Sanders⁶⁴ is proposed in this chapter to account for the changes in the cycle design for the wing mounted engines. A baseline configuration is first defined for the wing-mounted engines of the reference aircraft. This configuration is optimised based on the current technology levels on gas turbines to produce a representative reference. This requires the definition of a cycle optimisation methodology (Section 4.2). To couple the aerodynamic performance of the aft-mounted propulsor with the changes in the power requirements, the cycle optimisation methodology is applied again based on the reduction of the thrust requirement for the wing mounted engine and the required extracted power from the engine. This approach can be also extended to different types of power configurations for the BLI propulsor.

4.1.1 Power distributions for the aft mounted boundary layer ingestion propulsor

For the case study of this work (i.e. an aft-mounted boundary layer ingestion propulsor on a medium-range single-aisle aircraft configuration), two representative reference power distribution scenarios are contemplated (Fig. 4.1):

- **Independent power source for the aft-mounted propulsor.** This case accounts for the possibility of including a secondary generation source to power the additional propulsor. The type of power plant and its performance is not addressed, but assumed as a black box that provides power based on the propulsor aerodynamic requirements. The wing mounted engines are scaled based on the change on the thrust required by the application of the BLI system
- **Aft-mounted propulsion power extracted from the wing-engines.** This case accounts for a fully mechanical distribution of the BLI propulsor power requirement. The wing-mounted engines are scaled based on the changes on the thrust requirement of the BLI propulsor and the power required at a given system efficiency

Both cases represent two different extreme scenarios of the power distribution for these configurations. On the current configurations for aft-mounted annular boundary layer propulsors proposed in the literature (STARC-ABL¹¹ and CENTRELINE¹²) hybrid-electric propulsion is also introduced to improve the overall efficiency of the system. The estimation of the level of hybridisation is a complex subject out of the scope of this project, and the dedicated focus of a range of other studies^{4,60,166,167}. Hence, the two extreme baseline cases are applied to determine the limits of both types of operation. The actual benefits on power consumption derived from the mix of BLI and hybrid-electric propulsion are expected to lie in between both scenarios.

To evaluate the power savings derived from the integration of the BLI propulsor, it is required to define the thrust requirement for each of the engines (wing mounted and aft-mounted). In Chapter 3.3.2, the Thrust Split (Eq. 3.29) was introduced as the ratio between the net force variation provided by the integration of the aft-mounted propulsor, F_{RN} , and the total thrust required, T_{ref} . In this chapter this Thrust Split definition is maintained to estimate the power savings. However, since this is a preliminary analysis, the F_{RN} is interpreted as the thrust provided by the

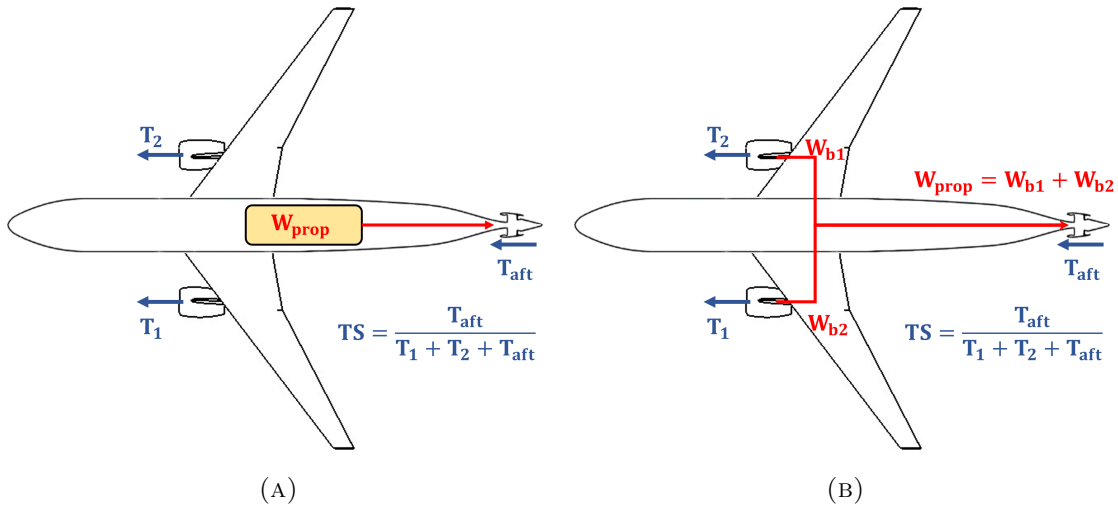


FIGURE 4.1: Schematic of the aft mounted boundary layer powering possibilities: (A) Independent power source case, and (B) power extracted from the wing mounted propulsors

aft-mounted propulsor (T_{aft}). In this way, the reference thrust T_{ref} , is the result of the addition of the thrust provided by the two wing mounted engines (T_1 and T_2), and the aft-mounted propulsor (Eq. 4.1). In this way, by fixing T_{ref} to a nominal value, and providing a value of TS the thrust requirement for the wing mounted engines can be obtained.

$$TS = \frac{-F_{RN}}{T_{ref}} = \frac{T_{aft}}{T_1 + T_2 + T_{aft}} \quad (4.1)$$

4.2 Cycle sizing methodology

To match the cycle performance with the thrust and power requirements in mid-cruise and take-off conditions, the software tool TURBOMATCH^{168,169} is applied. This tool allows a modular definition of the engine cycle components for an iterative matching of the cycle operation. While a manual design of the cycle requirements could be performed, an automated design approach for the configurations has to be introduced to carry out a systematic comparison of the different cycles. Goulos et al³⁶ and Stankowski¹⁷⁰ proposed an automatic routine to match the engine cycle requirements based on the propulsor dimensions and thrust class (AUTOMATCH). The fan isentropic efficiency (η_{fan}), the combustor outlet temperature (COT), and the overall pressure ratio (OPR) are used to define a given technology level. For that technology level, the cycle operation is optimised to match the thrust requirement

and minimise the specific fuel consumption (SFC) by varying the fan pressure ratio (FPR) and the engine bypass ratio (BPR). This allowed the exploration of the different technology levels to identify the most adequate operation conditions for the selected architecture. However, the selection of the desired operating point still requires further assumptions about the technology limitations of the selected configurations. Besides, the AUTOMATCH methodology is designed to estimate cycles based on existing configurations and do not allow the change on the system dimensions. This is required for the current approach, since the actual dimensions of the power-plant cannot be predicted beforehand. For these reasons, the routine is modified to allow the variation of the radial dimensions of the fan. An inbuilt capability of TURBOMATCH is applied for this purpose. Within the solver, for a given FPR and BPR the mass flow requirement is automatically calculated to provide the thrust requirement of the system based on the cycle performance. The mass flow rate is later used to determine the fan radius dimensions to match a fixed Mach number at the fan face in cruise ($M=0.55$). In addition, the power out-take from the low pressure turbine is included to model the cases with power extraction. This modification is included in both cruise and take-off conditions.

The optimisation process of the engine operation for a given technology level (η_{fan} , COT, OPR) is based on a three separate optimisation steps (Fig. 4.2). First, the technology level, the thrust requirements and the power extraction requirements are input into TURBOMATCH for a defined engine architecture and stage efficiencies. Then, the most adequate values of BPR and FPR are explored through a random seed between selected boundaries. The seed is applied as the initial sample of a single objective optimisation using NSGA-II based on the SFC minimisation. The genetic algorithm optimisation is applied until a fixed number of designs (1000) is achieved. This optimisation is used as an initial exploration of the design space to identify the most suitable FPR and BPR values to start a gradient based optimisation of the SFC using the L-BFGS-B algorithm¹⁷¹. This algorithm is run until a tolerance below 1×10^{-12} is achieved. The final design emerging of this optimisation is interpreted as the global optimum design for cruise conditions at that given technology level. The cruise condition cycle is then input in TURBOMATCH with an initial value of the take-off COT. Then the L-BFGS-B algorithm is applied to match the thrust requirement under take-off conditions with the COT as the active variable. After this process, for a given technology level this approach provides the optimum FPR and BPR for minimum SFC at cruise conditions and the required COT to match the take off requirements.

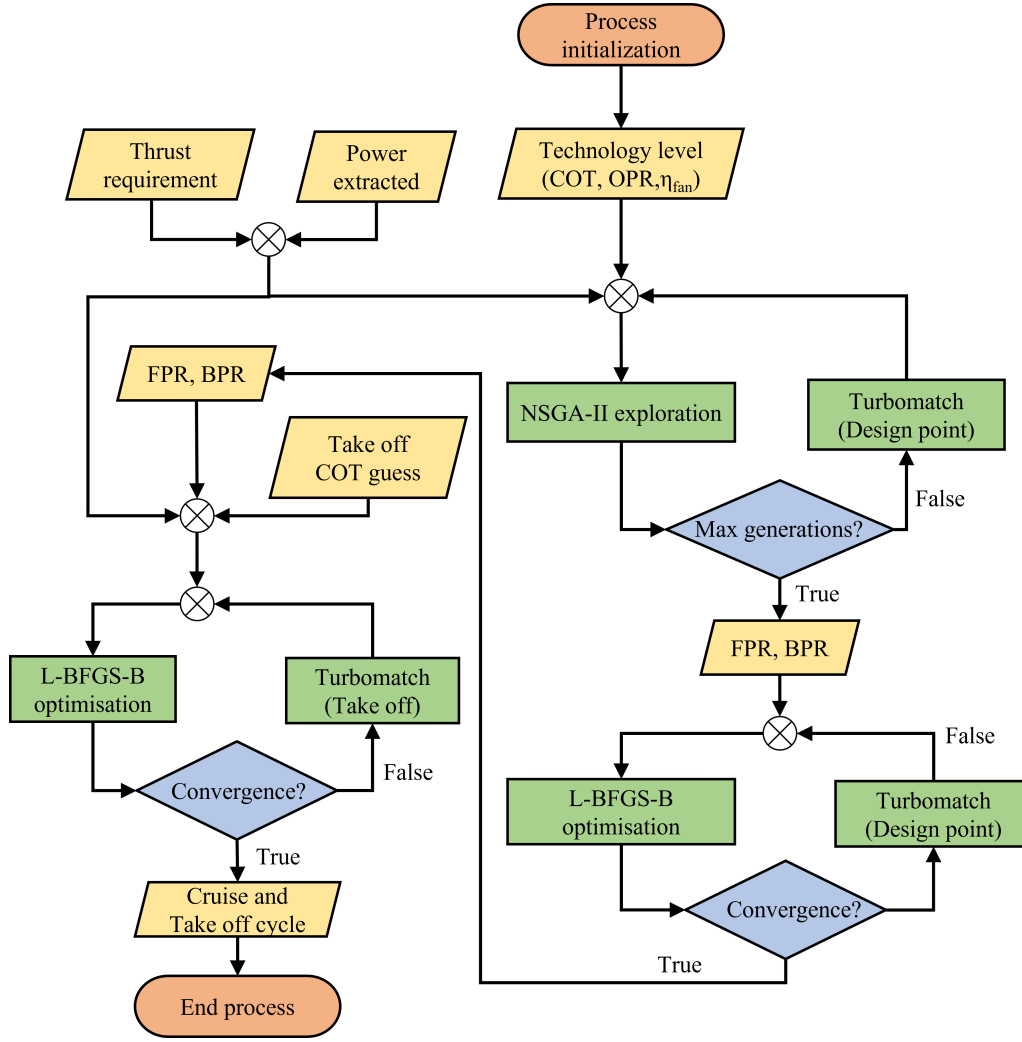


FIGURE 4.2: Workflow for the cycle matching and selection methodology

The adapted methodology is used to map the design space for the different technology levels (Fig. 4.3a). As it is expected the SFC in cruise reduces with the increase of the COT and the OPR¹⁷² (Fig. 4.3a). However, some criteria are still required to define the optimal viable technology level for each thrust-power combination. Hence, the decrease on the SFC must be limited based on the current or future technology limitations. For this work, two critical temperatures under take-off conditions are applied as the technology limits (Fig. 4.3). The two reference temperatures are the Compressor Discharge Temperature (CDT) and the Combustor Outlet Temperature (COT). The maximum take-off COT is limited to 1850K and the maximum CDT is set to 950K¹⁷³. Under these conditions, the optimum design within the technology requirements is found to be always at the intersection between the two temperature limit lines. Once this point is found, the gradient based optimisation is run again for the optimum technology level found for the given restrictions.

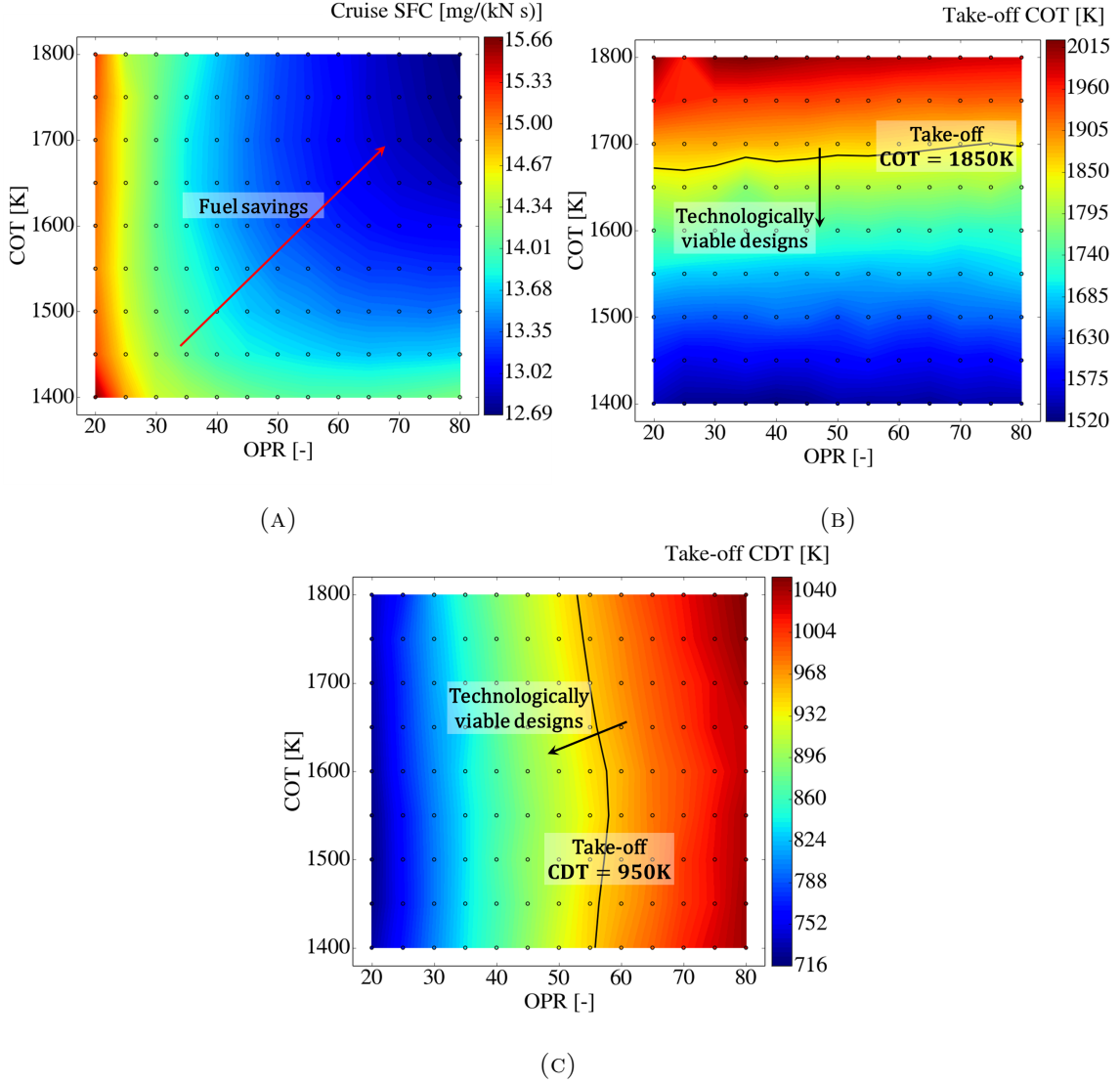


FIGURE 4.3: Sample operation maps for optimised cycle design at different technology levels (η_{fan} constant): (A) Optimal Specific Fuel Consumption in cruise conditions, (B) required Combustor Outlet Temperature in take-off conditions, and (C) required Compressor Discharge Temperature in take-off conditions

The aim of this thesis is to evaluate the impact of the design of the embedded propulsor integration in the aerodynamic performance. The evaluation of the power savings derived from the BLI propulsor is just a necessary stage to provide context about the potential benefits of the propulsive system. Hence, the aim of this chapter is not to provide definitive numbers of the actual power savings, but to generate the necessary background for the rest of this thesis. Under this context, in the present study, some important assumptions are taken. All transmission losses (mechanical and power) are neglected when the power is supplied to the BLI propulsor. The variation of the nacelle drag with the change of the wing-mounted propulsor dimensions

is neglected to simplify the engine sizing approach. For a more complete evaluation of the thrust requirement of the wing-mounted engines the change on the installation drag should be considered. In addition, the effects of the system weight are also neglected. This includes the weight for the wing-mounted power-plants and the aft-mounted propulsor. This also includes the changes on the structural weight in the wings and the rear of the fuselage. For the final design of the novel configuration, the structural impact of the change of the propulsor dimensions should be included. However, the weight estimation is out of the scope of the present work. Similarly to the weight, the effects of the change on the propulsive efficiency are not accounted in the full aircraft range and flight operation. Also, for any power extracted from the wing-mounted engine, the transmission or conversion losses are neglected.

4.3 Baseline wing-mounted power plant

To evaluate the potential power savings of the distributed BLI propulsion, it is necessary to define a baseline configuration. This baseline is optimised with the same methodology as the distributed propulsion cases (Section 4.2). This allows a systematic comparison between the optimal designs for each power-thrust pair requirement. To size all the configurations, sample mid-cruise conditions are applied based on the operation of the BLI propulsor (Tab. 4.1). In this work, two cases are contemplated. The first case (Case A) corresponds to the reference medium-size single-aisle aircraft in mid cruise with a Mach number of 0.78. The second case (Case B) is based on a reference 3D airframe (DLR-F6¹⁹) found on the open literature. This airframe is designed for a cruise operation of Mach 0.75.

In both cases, the take-off operation is selected for the reference thrust requirement¹⁷⁴ at sea level in a hot day (40 °C). The international standard atmosphere (ISA) model¹⁴⁵ is used to define the temperature and static pressure at each condition. The increase in the reference temperature on take-off is determined as ΔT from the ISA value. A constant intake pressure recovery (IPR) is assumed in both cases.

For the baseline engine and all the subsequent designs, a three-spool configuration is applied (Fig. 4.4). The pressure ratio at the fan tip (bypass duct) is set to the nominal value of the optimisation. The root pressure ratio of the fan is corrected using a constant ratio between the tip and the root (Eq. 4.2). The split of the OPR between the lower pressure stages and the high pressure compressor is kept constant during the optimisation process (Eq. 4.3). The ratio between the stages is

TABLE 4.1: Operation condition for the engine performance optimisation

Operating conditions						
Case	Point	Mach #	Alt. (ft)	IPR (%)	ΔT (K)	Thrust (kN)
A	Mid-cruise	0.78	32,000	99.75	0.0	22
	Take-off	0.0	0.0	99.75	25	100
B	Mid-cruise	0.75	32,000	99.75	0.0	22
	Take-off	0.0	0.0	99.75	25	100

selected from the suggested ideal pressure distribution for a three spool configuration suggested by Epstein¹⁷⁵. A fixed hub-to-tip ratio of 0.265 is assumed for all the configurations.

$$\frac{FPR_{tip} - 1}{FPR_{root} - 1} = 1.23 \quad (4.2)$$

$$\frac{FPR_{root} PR_{IPC}}{PR_{HPC}} = 1.5 \quad (4.3)$$

Fixed isentropic efficiencies are assumed for the compression and expansion stages (Tab. 4.2). A constant low power bleed is applied to the high pressure turbine (HPT) to simulate the electric power requirements for the cabin (Fig. 4.4). A constant ratio of mass flow is bled from the IPC to supply the aircraft cabin. Also, a constant mass flow ratio is bled from the high pressure compressor (HPC) to cool the HPT blades and vanes. Also, despite the continuous progress towards more environmentally efficient fuels such as hydrogen⁵⁵, to maintain consistency with the current configurations kerosene is assumed to be the operating fuel. Nevertheless, this does not change the overall power requirements of the engine in isolation. Since the weight and tank sizing is excluded from this analysis, the change on the power requirements are only transmitted on the value of SFC. This metric could be converted to any alternative fuel with a simple adjustment of the specific calorific power.

TABLE 4.2: Assumed isentropic efficiencies for the turbine and compressor stages of the wing-mounted three-spool turbofan configuration

Summary of isentropic efficiencies					
Compression			Turbine		
Fan	IPC	HPC	HPT	IPT	LPT
0.94	0.92	0.91	0.9	0.92	0.93

With the aforementioned assumptions, the optimisation routine is applied to a range of technology levels. The fan isentropic efficiency is fixed to the nominal value (Tab. 4.2). The cruise OPR is increased from 20 to 80 in steps of 5. At the same time

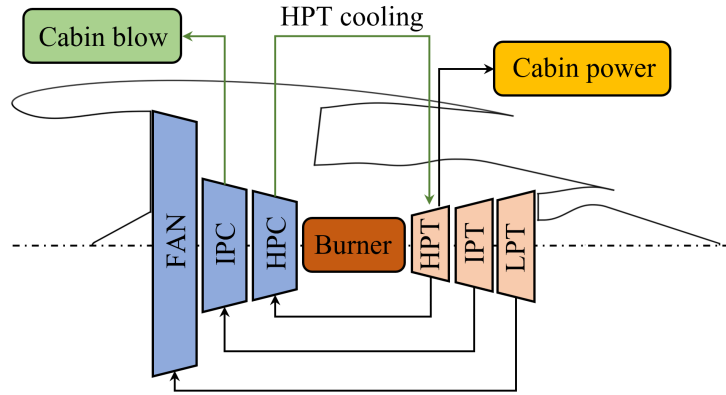


FIGURE 4.4: Schematic of the turbomachinery model for the wing podded propulsor in the configuration of reference (not to scale)

the cruise COT is increased for each OPR between 1400K and 1800K in intervals of 50K. The take-off limit temperatures are applied to find the optimum cruise COT and OPR configuration within the selected technology limits (Figs. 4.5-4.6). For both cases (Case A and Case B), the optimum baseline design resulted in an ultra-high bypass ratio configuration (Tab. 4.3) with a fan pressure ratio of the order of 1.44. For the target Mach number at the fan face during cruise conditions a fan diameter of approximately 1.945 meters (76.55 inches) is obtained for the Case A, and a diameter of approximately 1.98 meters (77.92 inches) is obtained for the Case B. These configurations are aligned with the predicted future aero-engine configurations found in the literature^{117,118,125,173}. Thus, the configurations obtained from the analysis are considered representative of optimal power-plants for a future medium range aircraft configuration without the boundary layer ingestion system. This configuration will be used as the baseline to compare the potential power savings obtained by distributing the thrust to the aft mounted boundary layer propulsor.

TABLE 4.3: Optimum operating conditions for the baseline wing-mounted engine at cruise

Baseline wing-mounted turbofan optimum cruise operation								
	Radius [m]	httr	BPR	FPR	OPR	COT [K]	SFC [mg/(kN s)]	Sp. Thrust [m/s]
Case A	0.9725	0.265	14.59	1.441	54.75	1686.2	13.138	97.67
Case B	0.9896	0.265	14.86	1.446	56.07	1678.45	13.179	94.97

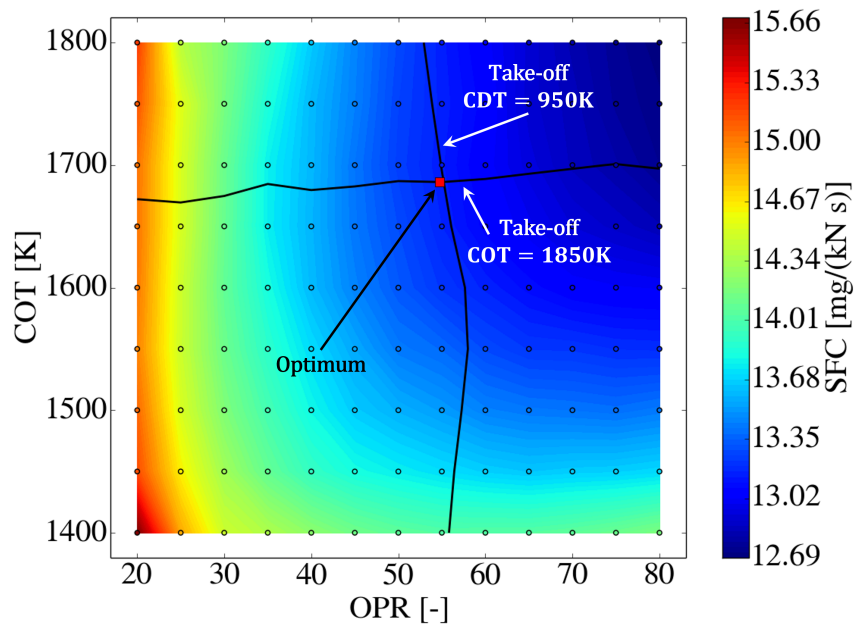


FIGURE 4.5: Selected baseline configuration based on the technology levels at take-off for the Case A (Tab 4.1)

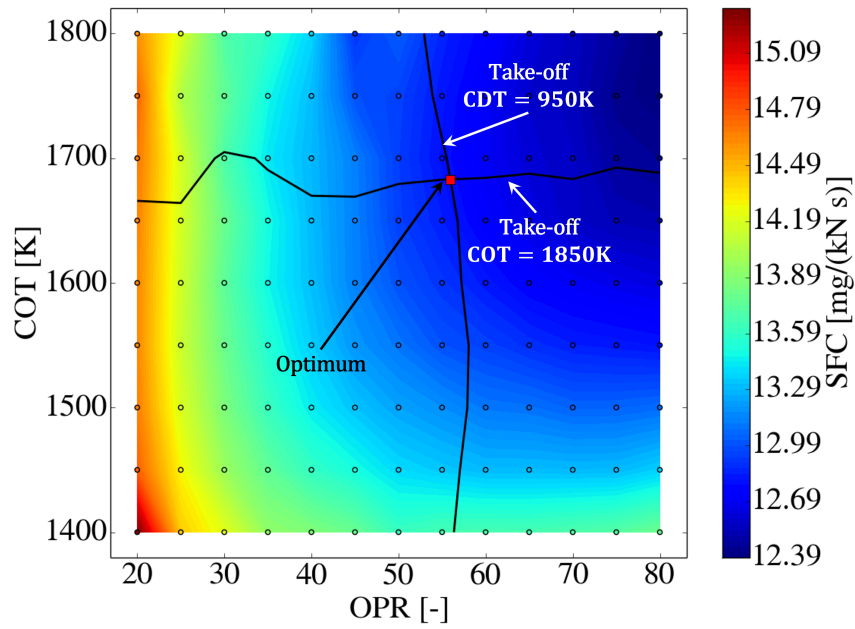


FIGURE 4.6: Selected baseline configuration based on the technology levels at take-off for the Case B (Tab 4.1)

4.4 Power savings with boundary layer

As discussed in Section 4.1, the most direct approach to quantify the power savings from the integration of the aft-mounted BLI propulsor is to account for all the power

requirements of the distributed propulsion system and compare them with the power requirement of the baseline configuration with wing-mounted engines. A Power Saving Coefficient, PSC , is defined for this purpose (Eq. 4.4). The power required for the cruise operation for each power configurations (Section 4.1.1) is applied to obtain the requirements of the aft-mounted propulsor. To obtain the power of the wing mounted engines, the Specific Fuel Consumption (SFC) is multiplied by the specific calorific value of the fuel (HV) and the operating thrust requirement (Eq. 4.6). For the aft-mounted annular propulsor, the power is extracted from the $TSPC$ and TS requirements at a given point (Eq. 4.5). When the power of the baseline ($W_{baseline}$) or the power of the podded engines (W_{podded}) is used, this metric includes the added power of the two wing mounted engines.

$$PSC(\%) = \frac{W_{baseline} - W_{new}}{W_{baseline}} \times 100 = \frac{W_{baseline} - (W_{prop} + W_{podded})}{W_{baseline}} \times 100 \quad (4.4)$$

$$W_{prop} = TSPC \cdot T_{ref} \cdot TS \quad (4.5)$$

$$W_{podded} = SFC \cdot HV \cdot T_{ref} \cdot (1 - TS) \quad (4.6)$$

With this approach the aerodynamic analysis of the aft mounted BLI propulsor can be coupled with the performance of the wing mounted engines. This method also accounts for changes on the engine performance for the theoretical BLI propulsor by the introduction of the $TSPC$ into the analysis. Thus, a reference map can be created to determine the power consumption benefits between different combinations of $TSPC$ and TS for the reference power configurations. This methodology is applied for the two cases defined in Section 4.1.1 to provide a reference for the possible power savings obtained with the aft mounted propulsor. Based on the current assumptions, the PSC values should be interpreted as a reference of the potential power savings and not as a definitive value for the power savings of the system. Nevertheless, this is sufficient for the current work, where the principal focus is on the study of the aerodynamics of the propulsion integration of the embedded propulsion system.

4.4.1 Independent power source

The first case study is the independent power source for the aft-mounted propulsor (Fig. 4.1a). In this case the source of the power required by the BLI propulsor is removed from the cycle analysis. However, the reduction in thrust requirement

based on the input TS is included to resize the cycle. For this purpose, the three-spool gas turbine configuration defined for the baseline engines is applied (Fig. 4.4). The ratios between the compressor stages and the efficiency of the components is maintained as defined in Section 4.3.

To provide a guideline of the possible power savings derived from the aft-mounted boundary layer ingestion propulsor, the PSC is mapped for different combinations of TS and $TSPC$. The thrust split is increased progressively in intervals of 0.25 between 0.125 and 0.325. The $TSPC$ is evaluated in a range of 175 to 325 m/s in intervals of 25 m/s. The same value of thrust split is applied for the take off and cruise conditions. Each thrust combination is evaluated based on the approach described in Section 4.2. In this configuration, the propulsive requirements of the wing-mounted power-plants are evaluated once for each TS value. This gives a fixed value of W_{padded} (Eq. 4.6) for each TS point. Then, the $TSPC$ is applied to provide an analytical value for W_{prop} (Eq. 4.5) and obtain the variation of the PSC .

For cases A and B (Tab. 4.1), the PSC increases monotonically with the TS (Figs. 4.7-4.8). This is a direct consequence of diverting more thrust to the aft-mounted propulsor. The wing-mounted power-plant under these circumstances reduces its size and power input. This leads to a notable power reduction for the wing mounted engines. For example, in case A ($M = 0.78$), the increase of the thrust split at a constant $TSPC$ (240 m/s) from 0.20 to 0.21 results in an increment on the PSC from 10.9% to 11.5%.

The PSC also increases with the reduction of the $TSPC$ (Figs. 4.7-4.8). For example, the reduction of the $TSPC$ from 260 to 240 m/s at a fixed TS (0.20), the power saving coefficient increases from 10.2% to 10.91%. The $TSPC$ is reduced by the increase on the thrust contribution of the BLI propulsor or the reduction of the power required for the aft-mounted power-plant (Eq. 4.5). This means that the $TSPC$ is inversely correlated to the propulsive efficiency of the embedded propulsor. Hence, at a fixed power input for the aft-mounted propulsor, the $TSPC$ can only decrease with the increase of the relative net thrust force, F_{RN} (Eq. 3.28), or simply the increase on the TS (Eq. 3.29).

To give further context about the power savings of the BLI propulsor, operation lines at fixed fan pressure ratios, boundary layer ingestion rates and fan polytropic efficiencies are introduced (Figs. 4.7-4.8). These lines are defined based on the sizing considerations described in Appendix B. These are based on an ideal fan stage model to determine the valid operating points for a fixed power requirement for the aft-mounted power-plant.

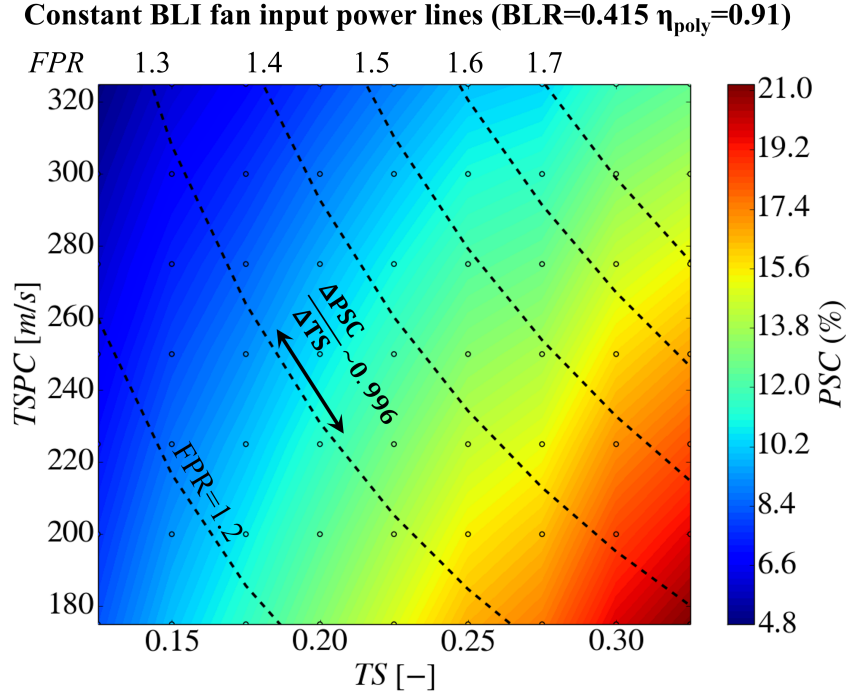


FIGURE 4.7: Power Saving Coefficient for the different TSFC and TS values on the Case A ($M_{cruise} = 0.78$) with an independent power source. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.415 and polytropic efficiency of 0.91

For the present work, a fan pressure ratio of 1.3 and a polytropic efficiency of 0.91 is assumed for the sizing of the aft-mounted BLI propulsor (Appendix B). After the sizing of the boundary layer ingestion propulsor, the thrust contribution of the BLI propulsor is expected to be around $TS = 0.2$ for the Case A ($M = 0.78$), and $TS = 0.225$ for the case B ($M = 0.75$). At this point, the expected PSC from this preliminary assessment is 11.23% and 13.62% for the cases A ($M = 0.78$) and B ($M = 0.75$), respectively. If the aerodynamic optimisation process is assumed to increase the thrust contribution at a fixed power input level, a exchange rate between the PSC and the TS of almost one is expected (Figs. 4.7-4.8). Specifically, if the TS increases 0.01 units, the PSC increases in 0.996% in the Case A ($M = 0.78$) and 0.997% in the Case B ($M = 0.75$). Thus, for this configuration (Fig. 4.1a), any gain on the thrust contribution of the aft-mounted propulsor can be traced back to a similar increase on power saving or fuel burn reduction.

This indicates that there is a higher interest in increasing the thrust contribution of the aft-mounted propulsive system to reduce the overall power requirements of the aircraft. For a fixed power input, if the turbomachinery design of the aft-mounted propulsor is assumed constant, the only strategy to increase the thrust contribution

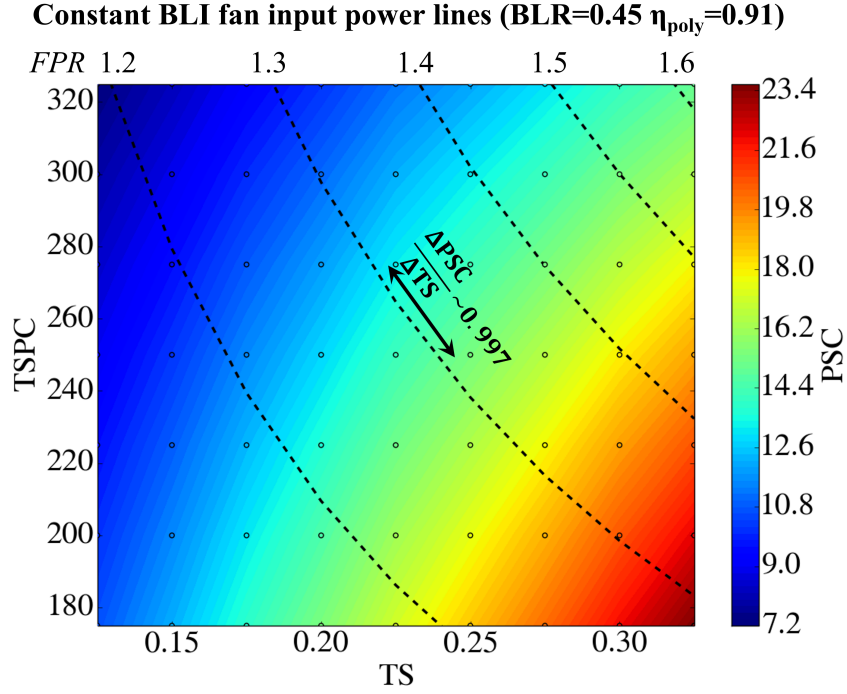


FIGURE 4.8: Power Saving Coefficient for the different TSPC and TS values on the Case B ($M_{cruise} = 0.75$) with an independent power source. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.45 and polytropic efficiency of 0.91

of the aft-mounted propulsor is to improve the aerodynamic design of the propulsion integration (i.e. aft-fuselage, intake, nacelle and exhaust design). Hence, it is on the designer's interest to understand the impact of the aerodynamic design of the propulsion integration in the thrust contribution of the BLI propulsor.

In addition, further benefits could be obtained by the reduction of the dimensions of the wing-power plant with the thrust split. The increase on TS requires the reduction of the trust requirement of the wing-power plant. With the automatic cycle optimisation methodology of the Section 4.2, this was achieved with the reduction of the mass flow rate required while maintaining similar bypass ratios ($\pm 3\%$) and fan pressure ratios ($\pm 1.5\%$) to the baseline configurations (Tab. 4.3). This results in a monotonic reduction of the fan radius for the wing-mounted propulsor (Fig. 4.9). The small variations of bypass ratio ($\pm 3\%$) mean that the reduction of the fan radius scales down both, the core and the bypass regions of the power plant. This could potentially increase the power savings by the reduction of the weight and the drag of the wing mounted engines. Nevertheless, the analysis of the contribution of both factors is considered out of the scope of this project.

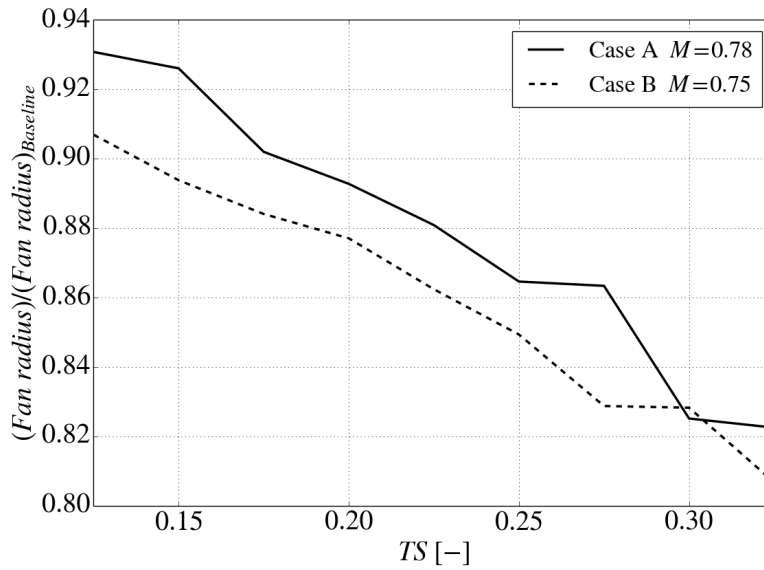


FIGURE 4.9: Change on the dimensions of the wing-mounted propulsor with the increase on TS , for the configurations with the independent power source for the BLI propulsor

The benefits in the PSC obtained by this analysis are in line with the values found in the open literature for similar analyses. For example, Welstead and Felder¹ identified a potential saving of approximately 15% in fuel reduction for the STARC-ABL configuration at the start of cruise condition with a FPR of 1.25 and isentropic efficiency of 95.7% for the BLI propulsor. This analysis was performed without any aerodynamic input and evaluated top of climb and take off conditions. Also, this study contemplated higher TS values (~ 0.46) with a similar value of BLR ($\sim 45\%$). This work also included weight considerations and a hybrid-electric power distribution for the aft-mounted propulsor. However, the analysis of any of these factors is considered out of the scope of this thesis.

In a similar study, Seitz and Gologan⁴⁴ estimated values of PSC of approximately 10%. However, this study did not include the take-off conditions and it was focused on the sizing aspect of the configuration (weight and range). It is understood that the weight of the system will limit the maximum PSC achievable. However, this thesis is focused on the aerodynamic design and analysis of the propulsion integration of the embedded propulsive systems. Future work may include a detailed weight evaluation for the vehicle. This will require a detailed definition of the power source of the aft-mounted propulsor to determine the impact of the energy storage (electric, hybrid or fuel) in the power savings.

More detailed studies have reduced the predicted power saving coefficient predictions. For example, Gray et al¹¹ estimated a 3% power saving for the STARC-ABL configuration. Seitz et al¹⁷⁶ estimated the power reduction of the CENTRELINE concept between 6-9%. These studies also incorporated the aerodynamic analysis of the aft-mounted propulsor using a CFD approach. This will be addressed in later chapters of this thesis.

4.4.2 Power extracted from the wing mounted engines

The second case study is the configuration where the power for the BLI propulsor is completely sourced from the wing mounted power plants (Fig. 4.1b). In this configuration, all the mechanical power required for the aft-mounted propulsor is derived from the wing-mounted engines. A three-spool configuration is selected for the wing-mounted engines based on the assumptions described in the Section 4.2). The power for the rear mounted engine is extracted from the low pressure turbine (Fig. 4.10). The transmission losses are neglected.

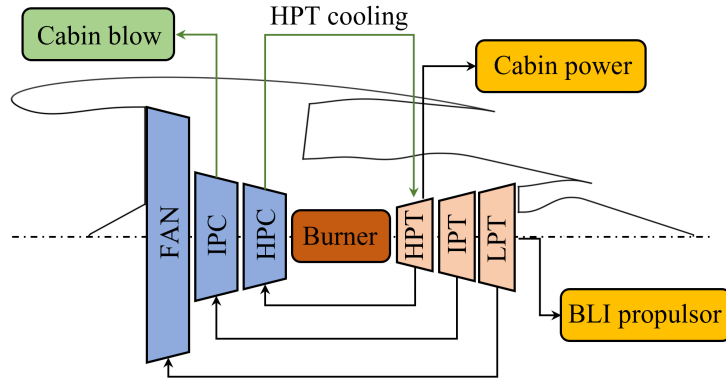


FIGURE 4.10: Schematic of the turbomachinery model for the wing podded propulsor in the configuration with power extracted from the turbine (not to scale)

To provide a guideline of the possible power savings derived from the aft-mounted boundary layer ingestion propulsor, the PSC is mapped for different combinations of TS and $TSPC$. The thrust split is increased progressively in intervals of 0.25 between 0.125 and 0.325. The $TSPC$ is evaluated in a range of 175 to 275 m/s in intervals of 25 m/s. Higher values of $TSPC$ (above 300 m/s) introduced convergence problems in the methodology (Section 4.2), so they are not included in the analysis. In this configuration, the W_{prop} is included into the W_{podded} values and into the engine optimisation methodology. The addition of the power supply requirement to

the engine results in a more complex design space for the optimisation methodology. This affects the smoothness of the PSC distributions (Figs. 4.11-4.12). However, the maps are considered sufficient to provide a reference of the power savings for this configuration based on the current assumptions.

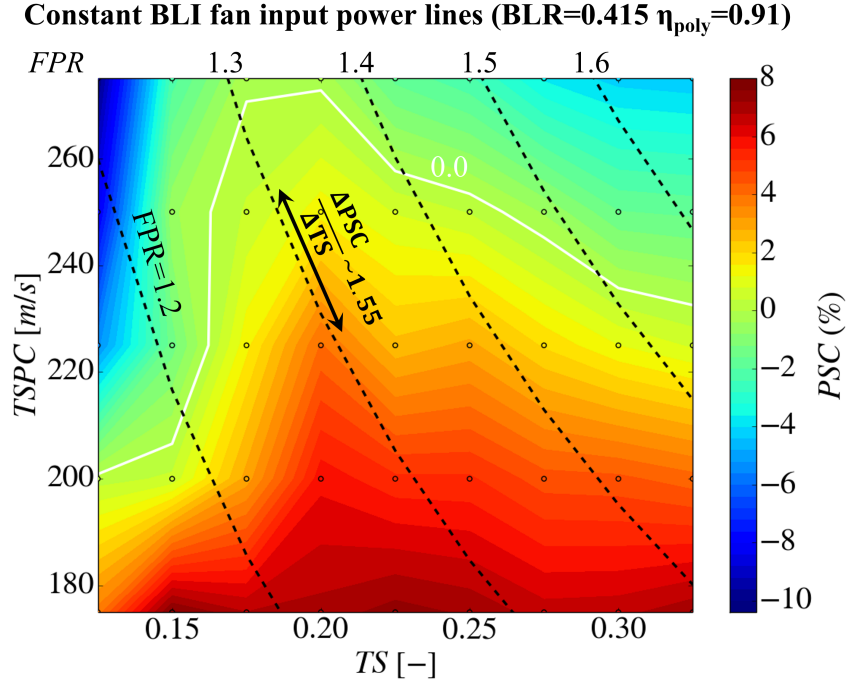


FIGURE 4.11: Power Saving Coefficient for the different TSPC and TS values on the Case A ($M_{cruise} = 0.78$) with the power extracted from the wing-mounted power-plants. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.415 and polytropic efficiency of 0.91

The same value of thrust split is applied in the take off and cruise conditions. To determine the power requirement for the aft-mounted propulsor in take-off conditions, it is necessary to assume a level of thrust in the stage. The mass flow rate is derived from the area of the fan face for a fixed Mach number of 0.55 based on the sizing considerations in Appendix B. The fan pressure ratio is then iterated to match the TS input. Jones velocity (Eq. 3.58) is applied to obtain the ideal thrust at sea level for the aft-mounted propulsor. A fixed polytropic efficiency of 0.91 is assumed to derive the power input of the stage.

For the Cases A and B (Tab. 4.1), at a fixed $TSPC$ the increase on the TS does not provide significant benefits on the PSC for values of TS higher than 0.15 (Figs. 4.11-4.12). The increase on the PSC is mainly driven by the $TSPC$. This means that when the power of the BLI propulsor depends on the wing-mounted power plants, there is no benefit in solely increasing the thrust of the aft-mounted propulsor. The

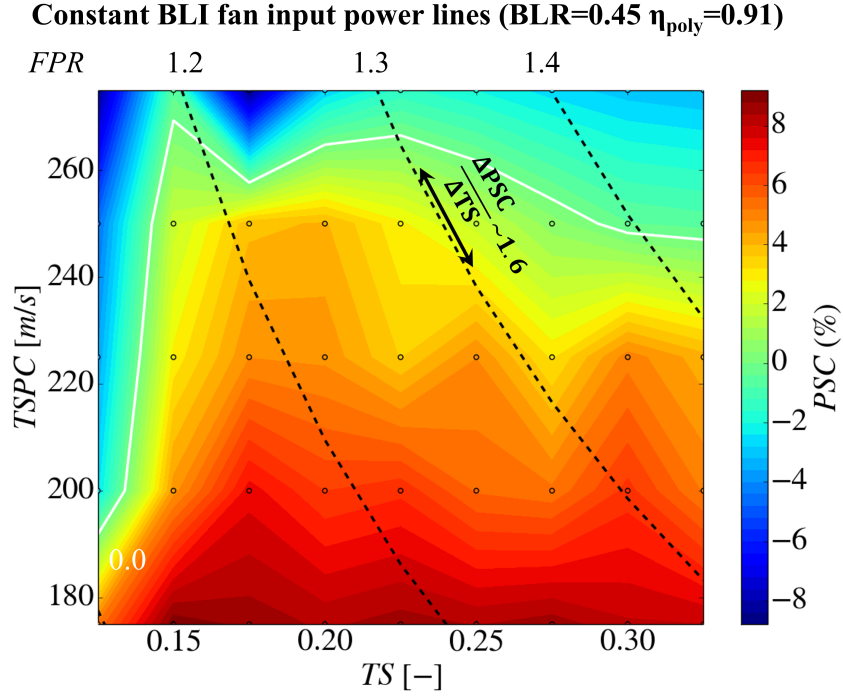


FIGURE 4.12: Power Saving Coefficient for the different TSPC and TS values on the Case A ($M_{cruise} = 0.75$) with the power extracted from the wing-mounted power-plants. Dashed lines corresponding to BLI fan power input isolines for a BLR of 0.45 and polytropic efficiency of 0.91

increase on TS must be associated to an increase on the propulsive efficiency (i.e. reduction of $TSPC$) of the aft-mounted propulsor. To achieve the reduction of $TSPC$ with the increase on TS , the power input of the aft-mounted has to remain as constant as possible.

In comparison with the configuration with the independent power source (Fig. 4.1a), the predicted power savings are reduced substantially. For example, at the preliminary design point of the Case A ($TS = 0.2$, $TSPC = 230 m/s$) the PSC decreases from 11.23% to 3.5%. At the same time, the sensitivity of the PSC to changes on TS at fixed power increases (Figs. 4.11-4.12), with a approximately a 1.6% increase in PSC per 0.01 units of TS based on the preliminary design point. This means that there is a higher potential benefit on the increase of TS when the power source of the aft-mounted propulsor depends on the wing-mounted power plants. However, higher reductions in the PSC are expected if the TS is reduced. For higher values of $TSPC$, this can result in penalties in the power consumption (Figs. 4.11-4.12).

Similar to the independent power case (Fig. 4.9), the radial dimensions of the wing-mounted power plants decrease with the rise on the TS of the aft-mounted propulsor

(Fig. 4.13c). In this case, the radius of the fan also increases with the $TSPC$. This is due to the increase on the power requirements on the wing-mounted power plant with the increase of TS and $TSPC$. Under these conditions, the turbines of the power-plant (Fig. 4.10) demand additional power to supply the aft-mounted propulsor. This requires the increase on the OPR and the COT. This causes the increase of the fan pressure ratio (Fig. 4.13b) for a given thrust requirement. This rise on the fan pressure ratio increases the thrust provided by the bypass of the power-plant with a lower bypass ratio (Fig. 4.13a). Thus, the fan radius of the wing-mounted engines decrease, but the relative size of the core with respect to the bypass must increase. This would result in additional benefits on the power consumption with the reduction of the nacelle drag, but the size of the core of the power-plant may not allow further weight reductions. Future work should include a weight model to determine the impact of the dimensions of the propulsor.

4.5 Summary

In this chapter, an approach was presented to account the power savings obtained with the integration of the aft-mounted propulsor. This method is based on the reduction of the thrust requirement on the wing-mounted power-plants due to the distributed propulsion. The methodology has been based on existing tools^{168,169} to determine the cycle requirements of the wing-mounted power plants.

For this purpose, two baseline power-plant configurations have been defined based on an optimisation methodology for two cruise Mach numbers ($M = 0.78$ and $M = 0.75$) at a constant altitude (32,000 ft). The benefits on the power consumption are defined with respect to these baseline configurations. The definition of the optimal configuration was based on representative technology limits under take-off and cruise conditions.

Two configurations for the power supply of the embedded propulsor were evaluated. First, a configuration where the power of the BLI propulsor is sourced independently of the wing-mounted power-plants. Second, a configuration where all the power of the BLI propulsor is mechanically derived from the wing mounted power-plants. These two cases represent two limit configurations to understand the design space.

The net value of the power savings is strongly dependent on the power source for the embedded propulsor. From the two power configurations for the aft-mounted propulsor, power savings between 3% and 11% were predicted based on the baseline

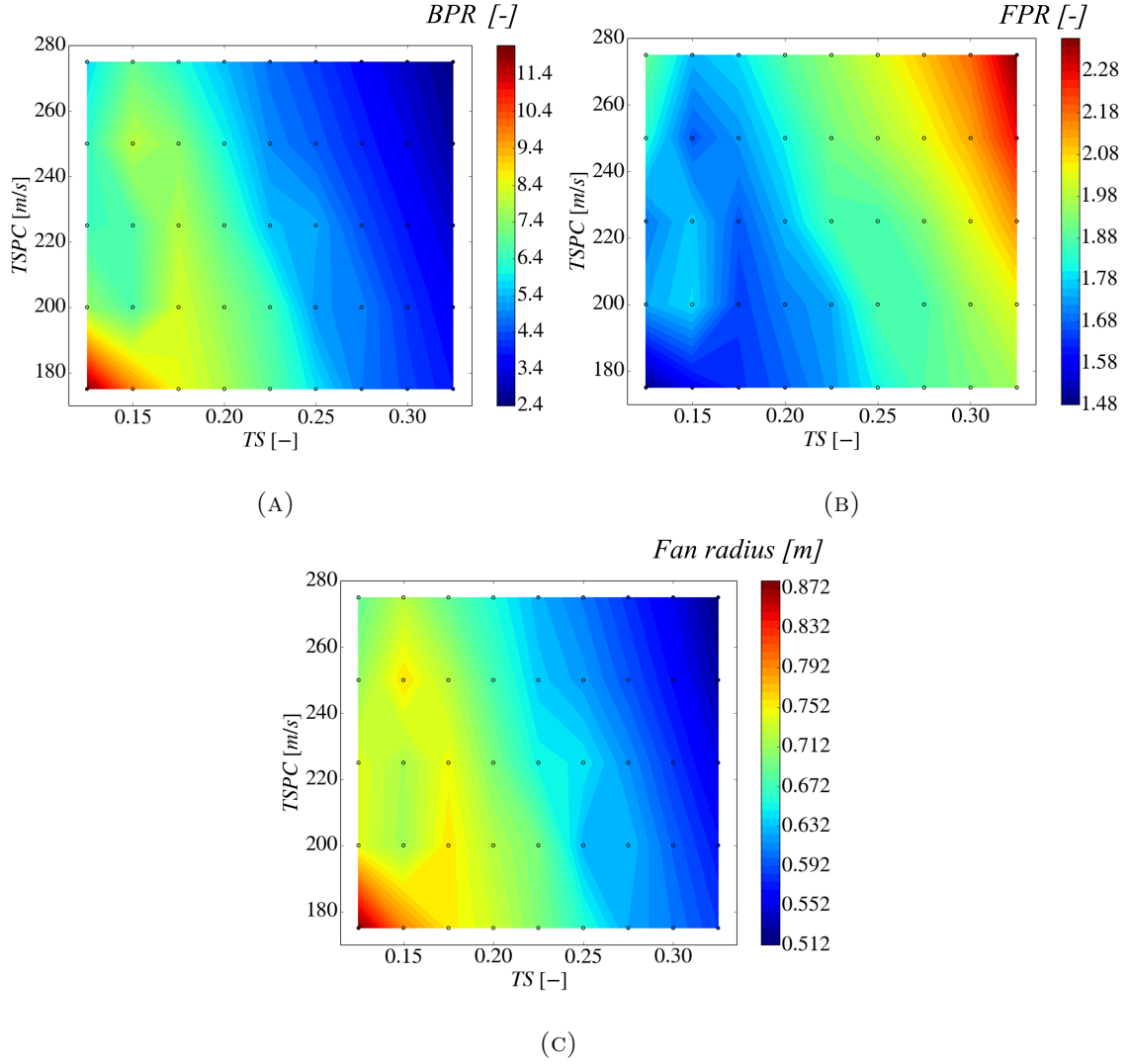


FIGURE 4.13: Changes on the wing-mounted power-plant requirements with the $TSPC$ and the TS for the Case A in cruise conditions ($M = 0.78$): (A) Bypass ratio, (B) Fan Pressure Ratio, and (C) Fan radius

power-plant. These values are aligned with results in the open literature^{1,11,12}, where benefits in power consumption between 3% and 15% are reported.

The power savings obtained by the integration of the aft-mounted propulsor are strongly dependent on the thrust contribution of the embedded propulsive system (measured by the TS), and the propulsive efficiency (evaluated with the $TSPC$). For a fixed power input, the increase in power savings is achieved by the increase in the propulsive force contribution of the aft-mounted propulsor. If the fan stage design is also fixed, the increase on TS is only achievable by the aerodynamic design of the propulsion integration of the embedded system. Thus, the aerodynamic analysis of the propulsion integration of embedded systems can be evaluated based on the

relative increase in thrust split. Based on a preliminary estimation for a nominal TS of 0.2, the increase of 0.01 in TS leads to an approximate increase in power savings of between 1% and 1.6% with respect to an equivalent aircraft without the embedded propulsor.

It is acknowledged that these estimates do not include the changes on the power savings due to the weight and drag reduction with the fan radius. Future work should include a comprehensive analysis of the weight requirements for the distributed BLI propulsive systems. However, this will require higher insight of the source of the power supply of the aft-mounted propulsor (i.e. electric, turbo-electric, mechanical ...).

CHAPTER 5

Axisymmetric propulsion integration aerodynamics with semi-coupled turbomachinery

This chapter reports on the studies carried out using the 2D axisymmetric methodology with a semi-coupled model for the turbomachinery. In this approach, the turbomachinery effects are modelled by an aspirated inlet with a fan stage defined by the cycle requirements. The first part of this chapter is dedicated to the definition of the case study and the operating conditions (Section 5.1). In the second section, the requirements and methods for the optimisation of the aerodynamic performance of the embedded system are evaluated (Section 5.2). The third part of the chapters presents a practical application of the optimisation methodology to evaluate the effect of the aft-mounted propulsor radial and axial location (Section 8.3). In the following sections, the robustness of the designs is evaluated for off-design operating conditions (Section 5.4). Finally, the closing remarks of the present chapter are summarised in Section 5.5.

5.1 Case study

As a case study, an aft-mounted annular boundary layer ingestion (BLI) propulsor is considered. The propulsion integration is based on a medium-range single-aisle aircraft configuration. The reference mid-cruise operating conditions (Tab. 5.1) are defined based on representative aircraft data of the same type (A320-200¹⁴⁰). The amount of boundary layer ingested was evaluated during the sizing of the propulsive system based on the boundary layer properties (Appendix B). A 2D axisymmetric model of the embedded propulsor (Chapters 3.4.1.1-3.4.3.1) is used with a semi-coupled one-dimensional fan model (Chapter 3.4.2.1). The fan nominal conditions

are based on a representative low thrust contribution BLI propulsor⁹⁶. However, the value of the *FPR* will be slightly modified from the nominal condition to match the exhaust mass flow requirement (Chapter 3.4.2.1). The BLI fan efficiency was defined with the polytropic value to provide a more representative variation of the isentropic efficiency with the changes on *FPR* in the absence of nominal cycle data.

TABLE 5.1: Sample mid-cruise operating conditions.

Cycle parameter	Value	Unit
<i>FPR</i>	1.3	-
Altitude	32,000	ft
M_∞	0.78	-
<i>BLR</i>	0.415	-
η_{poly}	0.91	-

In this chapter, the analysis is focused on the aerodynamic performance of the propulsion integration. A methodology for the evaluation of the performance of the propulsion integration design was proposed in the Chapter 3.3. In Chapter 4, the thrust contribution and its propulsive efficiency were linked to the power savings of the full vehicle (aircraft, aft-mounted propulsor and wing mounted power plants). Nevertheless, on the evaluation of the aerodynamic performance of the embedded propulsor, the intake and exhaust performance should be also included. This requires a multi-objective design approach to evaluate the performance of the system.

5.2 Optimisation of the aft-fuselage, intake and fan cowl geometries of an embedded propulsor

The aerodynamic design of the propulsion integration for embedded propulsion systems requires a simultaneous evaluation of different housing components. This is due to the increase on the aerodynamic coupling between the different components. Hence, to account for the effect of the design of the housing components, a multi-objective approach is required.

As observed in Chapter 2.1.5, the distortion transfer through the turbomachinery stage will affect the exhaust design requirements. In the present chapter, the distortion transfer is not accounted for directly by the one-dimensional fan model (Fig. 5.1). Thus, it is not of direct interest to focus the design effort on this particular housing component at this stage. Nevertheless, the effects of the aft-fuselage, intake and fan cowl design on the aerodynamic performance can still be evaluated. The rest

of this chapter will be focused on these housing components and the aerodynamic coupling between them. The influence of the exhaust inlet conditions is also evaluated in Section 5.4.2, but the design requirements of the exhaust are not evaluated. The exhaust design requirements and the impact of the design on the performance will be studied in more detail in Chapter 7.

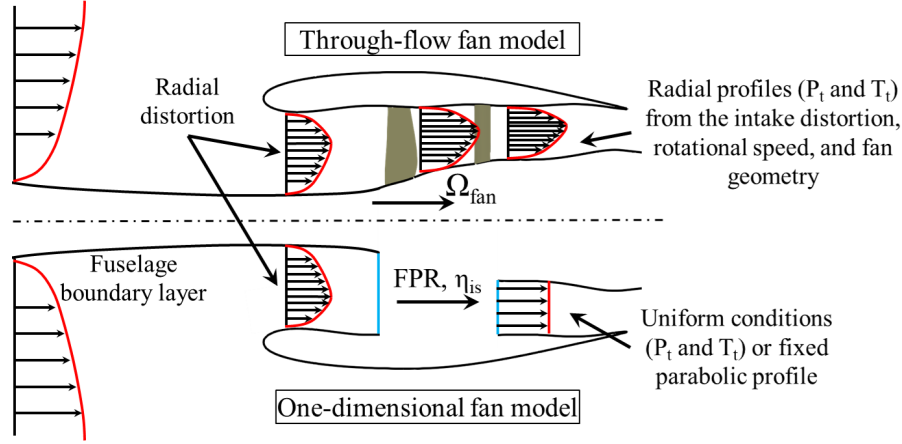


FIGURE 5.1: Schematic of the differences between the one-dimensional fan and the through-flow fan model. Effects on the definition of the exhaust inlet conditions.
Not to scale

For the evaluation of the aerodynamic performance in the 2D axisymmetric configurations, four candidate objective functions are found (TS , $TSPC$, RDI and IPR_{ing}). The aerodynamic performance of the combined propulsion-aircraft system is evaluated with the TS , while the propulsive performance of the aft-mounted power plant is evaluated with the $TSPC$. The intake performance is accounted for based on the total pressure losses due to the aft-fuselage and intake design with the ingested pressure recovery (IPR_{ing}), and the radial intake distortion defined by the radial distortion index (RDI).

These four candidate objective functions are to study the change on the aerodynamic performance due to changes on the propulsion integration design. For some types of studies (e.g. manual design activities or design space explorations) the use of multiple objective functions is not a limitation. However, for multi-objective optimisation (MOO) problems this could require to increase the population per generation and the number of generations to achieve convergence to the global Pareto front. This will obviously incur a notable increase of the computational cost and will limit the applicability of these studies. Thus, it is of interest to explore if the number of objective functions can be reduced to diminish the computational requirements. This will be evaluated during the definition of the optimisation problem.

5.2.1 Definition of the optimisation problem

To evaluate the requirements and limitations of the design space, an initial design space exploration for the parameters of the aft-fuselage, intake and nacelle was proposed (13 degrees of freedom as in Figure 5.2). The aft mounted propulsor location was fixed at radially ($r_{hub}/r_{fus} = 0.1$) and axially ($l_{aft}/l_{ps} = 0.85$). The fuselage geometry is fixed to match the reference fuselage from the nose to the 75% of its length (Chapter 3.2.2.2). This is approximately the length of the pressurised cargo vessel of the baseline aircraft¹⁴⁰. The fan dimensions were fixed based on cruise conditions (Tab. 5.1) according to the sizing methodology defined in Appendix B. This results on a hub to tip ratio of 0.29. A sample fixed convergent exhaust geometry was applied for all the designs. The trailing edge position of the fan-cowl is fixed, but the boat-tail angle (β_{nac}) was allowed to change (Fig. 5.2).

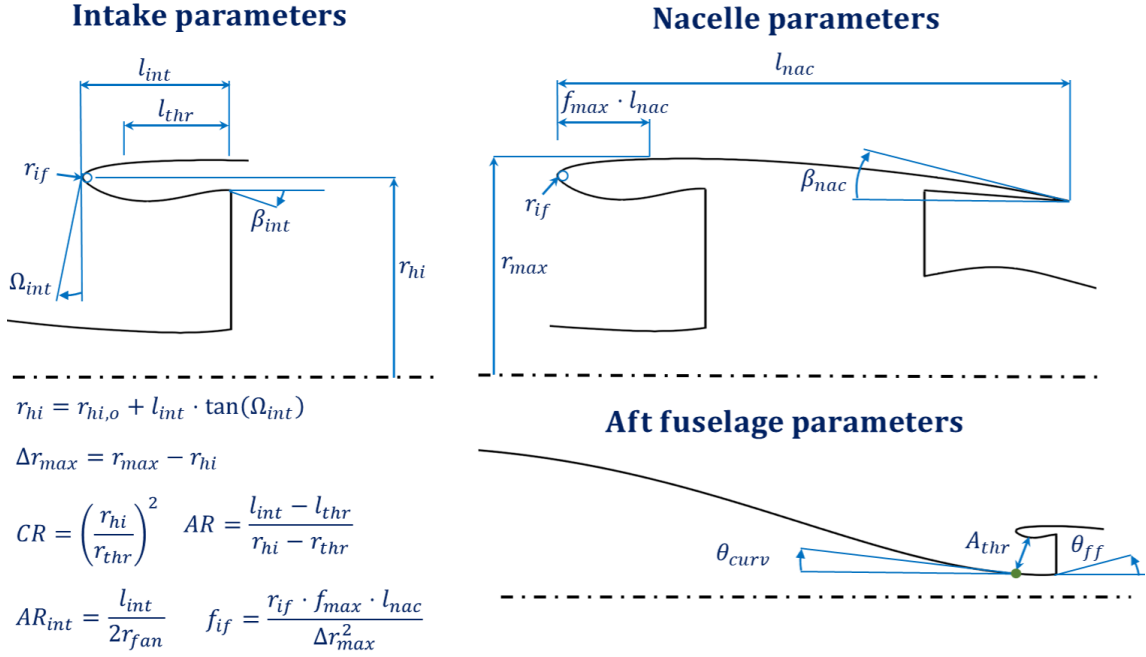


FIGURE 5.2: Definition of the degrees of freedom for the optimisation aft-fuselage, intake and nacelle geometries. Axial and radial location of the propulsor fixed

A Latin Hypercube Sampling¹⁶⁴ was used to evenly populate the design space. The surface inflation constraints were applied for the intake and the fan cowl (Chapter 3.5) to filter undesired geometries. Aerodynamic filters were also applied to the designs. Maximum Mach number values at the fan face ($M \leq 0.6$ and the intake duct ($M \leq 0.85$) are imposed (Chapter 3.5). Also, the cases were discarded if the flow was not attached in any of the housing components.

After the filtering process (Chapter 3.5), 220 valid designs were obtained. The correlation and monotonicity between the candidate objective functions are compared to understand the dependency between the different objective functions (Fig. 5.3). To address the correlation the Pearson index was applied while the Spearman index was used to evaluate the monotonicity of the sample distributions. A very strong negative correlation and monotonicity was found between the $TSPC$ and the TS , with Pearson and Spearman indexes of -0.91 and -0.991 , respectively. This is a direct effect of the relation between both metrics (Eq. 5.1). The $TSPC$ and TS are inversely correlated by a factor that depends on the required propulsor power W_{prop} . This value varies according to the flow conditions at the propulsor with the correction of the FPR to match the target mass flow requirement (Chapter 3.4.2.1).

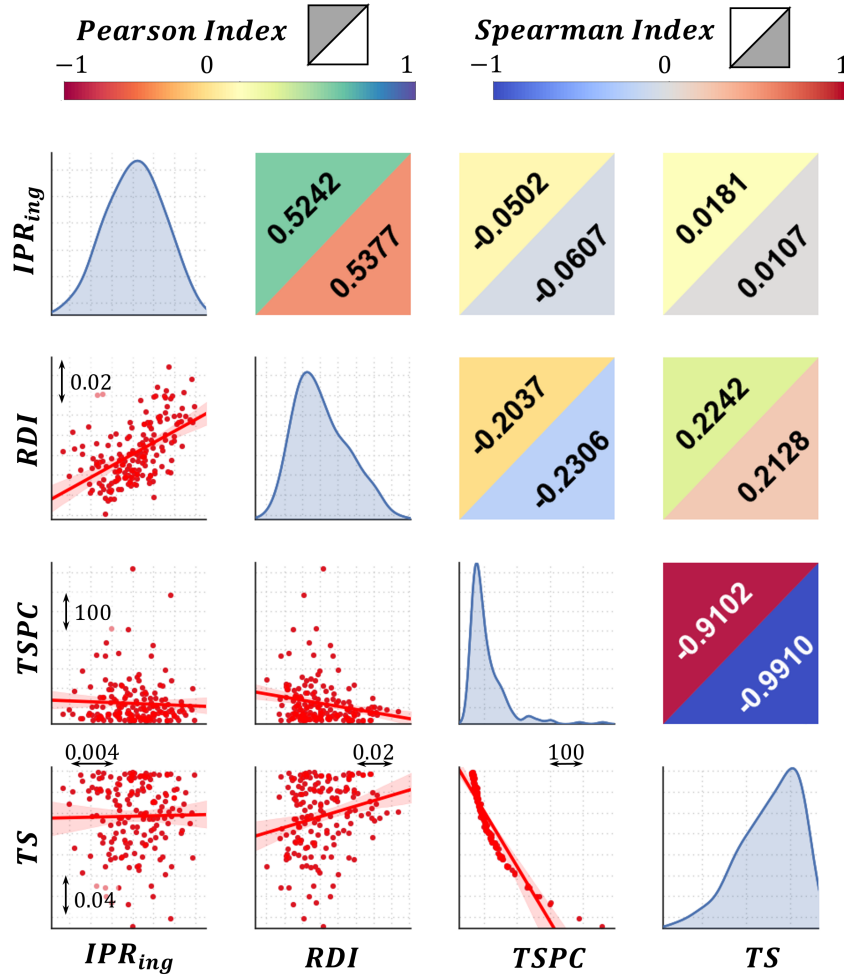


FIGURE 5.3: Comparison of the candidate objective functions for the optimisation of the aft-fuselage, intake and fan cowl aerodynamic design ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

$$TSPC = \frac{W_{prop}}{T_{ref}} \frac{1}{TS} \quad (5.1)$$

On the other side, the intake performance parameters (IPR_{ing} and RDI) have a moderate positive correlation and monotonicity (Fig. 5.3), with Pearson and Spearman indices higher than 0.5. These values are insufficient to determine the dependency between both functions. Furthermore, the polarity of the indices is positive. This would mean that any improvement in the ingested pressure recovery will show a penalty on the radial distortion index if the correlation values were higher. Thus, these two metrics must be considered in an optimisation problem to fully characterise the intake performance. Finally, very weak correlation and monotonicity are found between the intake performance metrics (RDI and IPR_{ing}) and the system performance metrics ($TSPC$ and TS). This does not enable a further reduction of the metrics required for the aerodynamic optimisation of the aft-fuselage, intake and fan cowl design of the embedded propulsive system. Thus, the problem must contain at least three objective functions (Eq. 5.2). For this work, the Thrust Split (TS) was selected to represent the aerodynamic performance of the overall system. Hence, the problem consist of two metrics to maximise (IPR_{ing} and TS) and one metric to minimise (RDI).

$$\text{Objective functions} : \begin{cases} \text{maximise } (TS) \\ \text{maximise } (IPR_{ing}) \\ \text{minimise } (RDI) \end{cases} \quad (5.2)$$

5.2.2 Multi-objective optimisation of the aerodynamic performance

The requirements for the different optimisation algorithms were evaluated in Appendix F. The Indicator-Based Evolutionary algorithm (IBEA) was found to be the most suitable for a constrained multi-objective optimisation problem with a similar number of degrees of freedom. Thus, this algorithm is applied in the present analysis. The initial design space exploration was used as the seed of the optimisation problem. The optimisation was run for 85 additional generations with 65 individuals each (5 designs per degree of freedom)²². The convergence criteria defined in the Chapter 3.5.2 was applied.

The non-dominated designs form a three-dimensional Pareto surface (Fig. 5.4) for the three objective functions. The Thrust Split increases approximately 0.005 units

from the non-dominated design with the highest value to the non-dominated design with lowest TS . This equates to an approximate variation of the power savings of 0.5%-0.8%, based on the preliminary power saving prediction of 3-11%. At the same time, the RDI changes in approximately 0.05 units from the worst to the best performing non-dominated designs. This is a significant variation. To put this in context, in a clean undistorted flow the RDI value is approximately 0, and increases with the sectoral and radial distortion to moderate values at $RDI \sim 0.02 - 0.04$ and high values at $RDI \sim 0.04 - 0.1$ ¹⁷⁷. Also, the maximum IPR_{ing} has a significant benefit (0.014) compared with the minimum IPR_{ing} non-dominated design. The impact of the total pressure losses from the design point varies with the fan model. As a guideline, a change in IPR of 0.01 from the design point can result in variations of the order of 1% in fuel burn¹⁷⁸. Nevertheless, this is based on the design point for a fixed fan design and cycle in a conventional under-wing podded configuration. In this conceptual design stage, lower values of IPR_{ing} would require to resize the fan to match the fan capacity requirements.

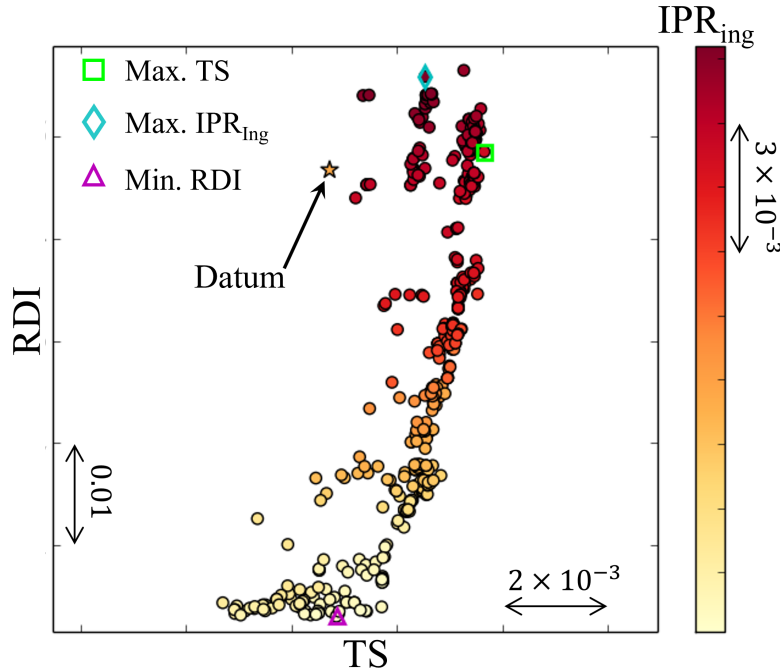


FIGURE 5.4: Non-dominated designs from the optimisation of the aft-fuselage, intake and fan cowl aerodynamic design ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

In addition, the improvement of the TS and IPR_{ing} in the Pareto surface has a negative effect on the predicted radial distortion (Fig. 5.4). This has some important implications for the design space. In this particular configuration, a compromise may

be required between the operating distortion levels at the intake and the improvement of the aft-mounted propulsor thrust contribution. Nevertheless, the minimum *RDI* design is not in the lowest *TS* performance (Fig. 5.4). The improvement margin between the minimum *RDI* and the maximum *TS* non-dominated designs is 0.003 *TS* units (0.3%-0.45% changes in power savings). Also, some non-dominated designs with similar *RDI* values and higher *TS* values can be observed on the Pareto surface. Hence, some design margin is still available to compromise between both metrics. On the other hand, the maximum *IPR_{ing}* and the minimum *RDI* configurations are almost placed on opposite sides of the Pareto surface with a penalty of 0.011 in *IPR_{ing}* for the minimum *RDI* design. Furthermore, there is almost a linear increase of the *IPR_{ing}* and the *RDI* for this particular configuration, as was noted on the preliminary design space exploration (Fig. 5.3). Thus, a significant compromise between both metrics may be required to balance the total pressure losses due to the propulsion integration and the distortion levels. A potential solution for this could be a resizing of the propulsor to compensate the capacity loss at the fan face, but this may have a negative effect on the overall drag.

The non-dominated designs on the Pareto surface have provided some perspective about the optimal design space for the propulsion integration of this particular geometry. However, some context is still required to illustrate the need of detailed aerodynamic design and multi-objective optimisation. For this purpose, a Datum design with the same propulsor dimensions and location (axial and radial) was included (Fig. 5.4). The Datum design (Fig. 5.5) was defined with a compact nacelle, intake and exhaust design (Tab. 5.2), with the aim to increase the thrust contribution of the system with respect to a Preliminary design defined based on general knowledge of propulsion integration design for conventional wing mounted engines (Appendix B.7). Some notable benefits can be observed by the selection of some non-dominated designs in comparison with the Datum. The maximum *TS* non-dominated design provides an improvement from the Datum of 0.003 in *TS* and 0.006 in *IPR_{ing}*, with an almost negligible penalty of 0.0017 units in the *RDI*. Similarly, the maximum *IPR_{ing}* non-dominated design has benefits of 0.002 and 0.008 in the *TS* and *IPR_{ing}*, respectively. However, a more significant penalty on the radial distortion values can be appreciated an increase of the *RDI* on 0.01 units. On the other hand, the maximum *RDI* non-dominated design does not provide notable changes on the *TS* value, with a negligible 0.00013 benefit (i.e. approximately 0.001% of power savings). The *IPR_{ing}* is severely penalised with a loss of 0.006 from the Datum design. However, an important reduction of the radial distortion could be achieved with a benefit of 0.044 units on the *RDI*.

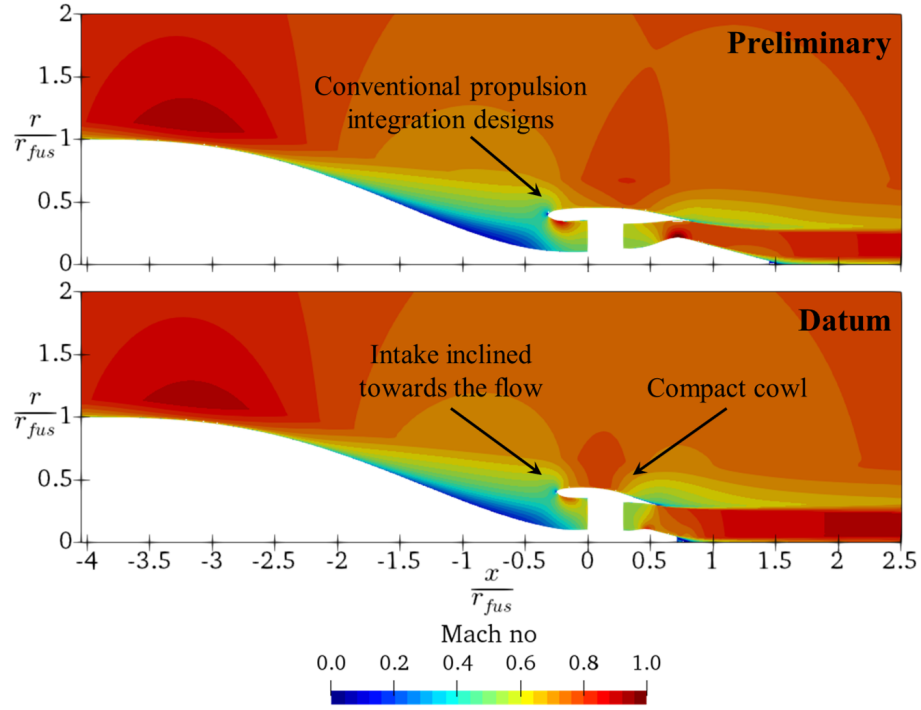


FIGURE 5.5: Mach number contours for the Datum and Preliminary designs used as reference ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

TABLE 5.2: Main dimensions of the cowl for the reference designs and the down-selected non-dominated designs of the multi-objective optimisation

Main cowl dimensions for the reference designs					
Metric	Preliminary	Datum	max TS	max IPR_{ing}	min RDI
$\frac{r_{hi}}{r_{fan}}$	1.08	1.14	1.18	1.148	1.195
$\frac{r_{te}}{r_{hi}}$	0.9	0.74	0.71	0.728	0.7
Ω_{int}	0.0°	11.5°	19.8°	19.9°	14.0°
$r_{max} - r_{hi}$ [m]	0.15	0.067	0.069	0.057	0.087
f_{max}	0.35	0.25	0.226	0.234	0.252
AR_{int}	0.45	0.35	0.24	0.201	0.2
$\frac{l_{nac}}{r_{fan}}$	3.26	2.28	2.024	1.98	1.97

It can be concluded that the design space for the aerodynamic design of the propulsion integration of this configuration of embedded propulsor requires special attention. The multi-objective optimisation, when compared with more conventional manual design approaches, can be used to identify key improvements on the aerodynamic performance based on changes in the aerodynamic design. The multi-objective optimisation strategy has also shown that there is a conflict between some groups of objective functions, such as the IPR_{ing} and the RDI . For this reason, the potential benefits in the aerodynamic performance are conditioned by the compromise

between the objective functions. To understand the origin of these benefits, it is of interest to analyse the aerodynamics of the different selected configurations and the impact of the design parameters on the performance. Both analyses are provided later in this Chapter (Sections 5.2.2.2-5.2.2.3).

5.2.2.1 Importance of the aerodynamic analysis

Prior to the detailed study of the different designs, it is of interest to further contextualise the importance of the aerodynamic design and evaluation in the preliminary design stage. Low order models based on drag correlations have been extensively applied to evaluate the impact of the propulsion integration in the aerodynamic performance^{1,26,153,179}. However, to the author's knowledge, the validity of these models in embedded propulsion systems has not been adequately compared with higher fidelity approaches such as CFD. This could lead to misleading conclusions when these models are applied for embedded systems. In this section, the CFD approach is compared with two low order preliminary design methods for embedded propulsion systems (Chapter 3.3.7). These are the Momentum Flux Deficit method (MFD) based on the definition of Seitz et al²⁶, and the preliminary accounting with a Modified Near Field Method (MNFM).

A Datum design for the MOO methodology was introduced in the previous section (Fig. 5.5). However, it was considered of interest to introduce a representative preliminary design (Preliminary) based on conventional propulsion integration requirements (Tab. 5.2). This preliminary design will be used to further illustrate the impact in the performance of the aerodynamic design of the propulsion integration. In comparison with the Datum design, a less compact fan-cowl is used with larger intake and exhaust ducts (Fig. 5.5). Also, the intake inclination is maintained horizontal ($\Omega_{int} = 0$) as it would be on a conventional wing mounted power-plant. These designs represent two different regions of the design space and can be used to illustrate the differences between the low order models and the CFD.

When the CFD and the low order performance prediction methods are compared (Fig. 5.6), it is observed that the low order performance accounting methods fail to capture the net values and changes the TS and TS_{PC} for the different designs. The MFD method over-predicts the TS in a 40-50% more with respect to the CFD value. The MNFM approach also over-predicts the TS by approximately a 20% of the CFD value. These differences on the thrust contribution affect the TS_{PC} prediction directly, since the predicted power input of the low order approaches

remains constant and the changes are only dependent on the increase in the thrust performance. The combination of the discrepancies on TS and $TSPC$ result in notable changes on the Power Saving Coefficient, PSC , (Fig. 5.7) independently on the power configuration selected for the aircraft (Chapter 4.1.1). For example, in the configuration with the independent power source, the MFD predicts a PSC 10-12% higher than the CFD. This is almost double than the predicted power savings for this configuration. Similarly, the MNFM approach predicts a PSC 5-7% higher than the CFD evaluations. Thus, even improved preliminary prediction methods such as the MNFM are insufficient to provide an accurate prediction of the net propulsive contribution of the embedded propulsor. The low order methods are unsuitable for the prediction of the aerodynamic performance of embedded propulsion systems.

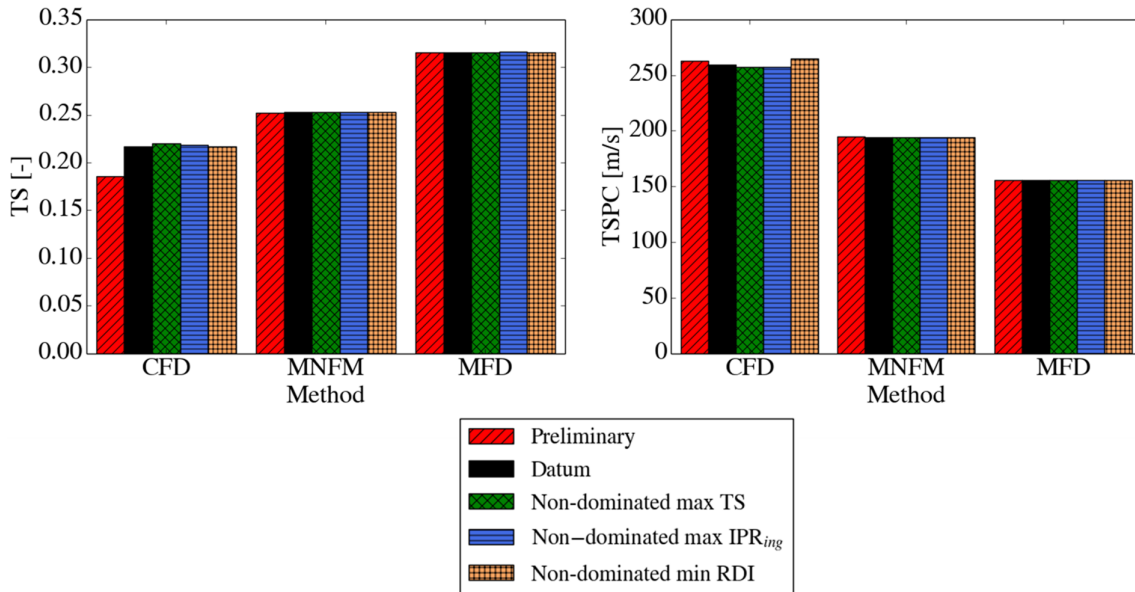


FIGURE 5.6: Comparison of the TS and $TSPC$ values predicted between the CFD and the low order performance prediction approaches for three configurations: Preliminary design, Datum design and Optimum TS values ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

In addition, the low order accounting methods fail to predict significant changes on the performance due to the changes in the propulsion integration design. This becomes evident when the performance prediction of the Preliminary design is compared with the Datum and the selected non-dominated designs (Fig. 5.6). The 2D axisymmetric CFD analysis predicts a substantial improvement in the aerodynamic performance when a detailed aerodynamic design is applied. An increase of 0.031 in the TS (0.1% -1.7% in PSC) is observed between the Preliminary and Datum designs, with further improvements on the non-dominated designs (max TS , max IPR_{ing} and min_{RDI}) up to a 0.003 increase in TS from the Datum (0.16%-0.33%

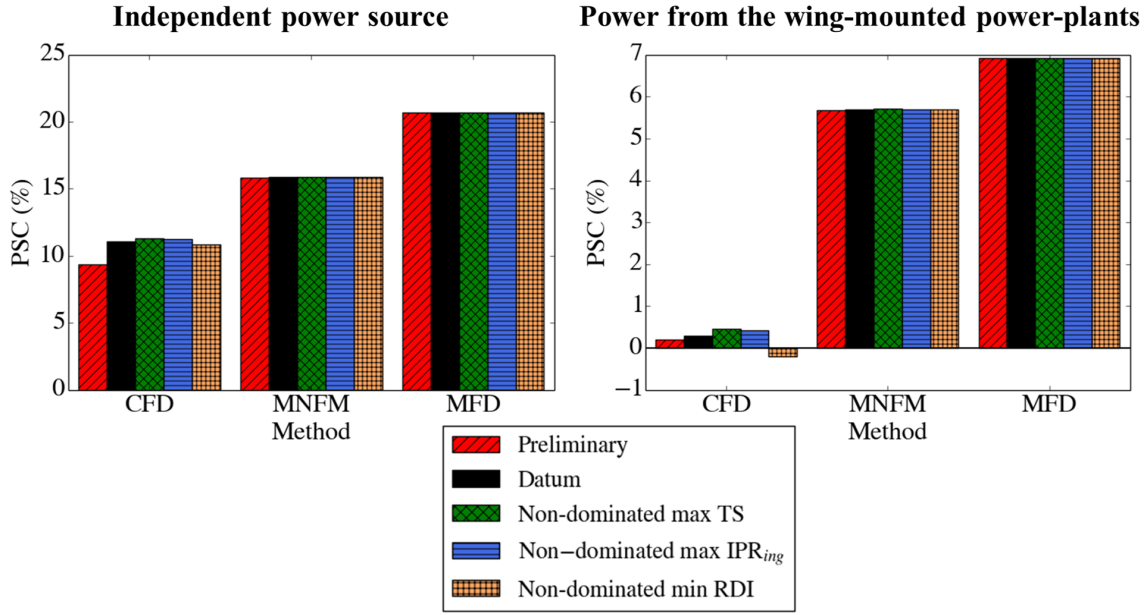


FIGURE 5.7: Comparison of the PSC predicted between the CFD and the low order performance prediction approaches for three configurations: Preliminary design, Datum design and Optimum TS values ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

in PSC). However, these changes accounted by the CFD approach are not translated into the low order performance prediction methods. With these methods, only small variations of TS between the designs are accounted for by the changes in the cowl geometry in both models. These variations (ΔTS) are of the order of 4.5×10^{-4} between the Preliminary design and the more detailed designs and lower than 1.0×10^{-4} between the detailed designs. This also misrepresents the changes on PSC from the different designs (Fig. 5.7). Thus, the low order approaches are also unsuitable to perform comparative analyses between different aerodynamic designs of the propulsion integration.

The differences observed between the low order approaches and the CFD can be attributed to two main factors. First, for the low order performance prediction methods (MNFM and MFD) the relative weight of the form and viscous drag of the fan cowl in the performance is much lower than the contribution of the fuselage. Thus, important changes in the cowl design are not fully accounted. Second, the impact of the aerodynamic coupling between the airframe (i.e. axisymmetric fuselage) and the embedded propulsor integration is not fully captured. Some of these effects were included by a correction factor, f_{int} , in the models (Chapter 3.3.7). However, this cannot capture the real impact of the changes of the design on the performance. A more detailed analysis of these integration effects is included in Section 5.2.2.2.

A potential solution to account for these effects was proposed by Bijewitz et al^{151,179}. In these works, it was proposed the creation of a distribution of f_{int} based on higher fidelity data, such as CFD. This could be used to calibrate the prediction for some of the effects of the propulsion integration. However, for more detailed aerodynamic designs this would require a high initial computational effort, reducing the advantages of the low order preliminary performance prediction methods. Furthermore, the integration effects are also expected to be dependent on the propulsor and the external operation conditions. Thus, there are not significant advantages on the application of such reduced order methods in comparison to a higher fidelity methods such as CFD.

5.2.2.2 Aerodynamics of the selected designs

In previous sections, notable benefits derived from the application of a detailed aerodynamic design have been reported. In this section, an analysis of the aerodynamics of such designs is provided to identify some of the main characteristics of the propulsion integration of the aft-mounted BLI propulsor. To approach this analysis, the Mach number distributions over the Preliminary design, the Datum design and the selected non-dominated designs are compared (Fig. 5.8).

The Preliminary design consists on a simplified cowl without any modifications on the cowl orientation (Tab. 5.2). The Preliminary design also has a smooth geometry transition from the cylindrical fuselage to the propulsor hub (Fig. 5.8). The flow accelerates locally on the transition from the cylindrical fuselage due to the immediate expansion at the origin of the aft fuselage curve. After the local acceleration, the flow continuously diffuses until the proximity of the highlight plane (Fig. 5.9). The intake section of the aft fuselage curve has a constant horizontal slope from the highlight to the fan face plane. A local acceleration is found in the intake of the Preliminary design around the throat due to the intake lip aspect ratio (AR). This could eventually lead to the emergence of shock waves for higher mass flow capture ratios. Hence, on a more detailed design this feature should be corrected to reduce the intake load. After the throat, the flow diffuses smoothly towards the fan face (Fig. 5.10a). The nacelle of the Preliminary design has a smooth curvature variation from the leading to the trailing edge. This reduces the flow diffusion in the nacelle afterbody (Fig. 5.10b). This has two direct effects. First, there is a reduction of the forward force (thrust direction) in comparison with the other designs, since the nacelle afterbody design is not used to reduce the pressure drag over the cowl. Second, the effective discharge pressure in the exhaust is higher relative to the other designs.

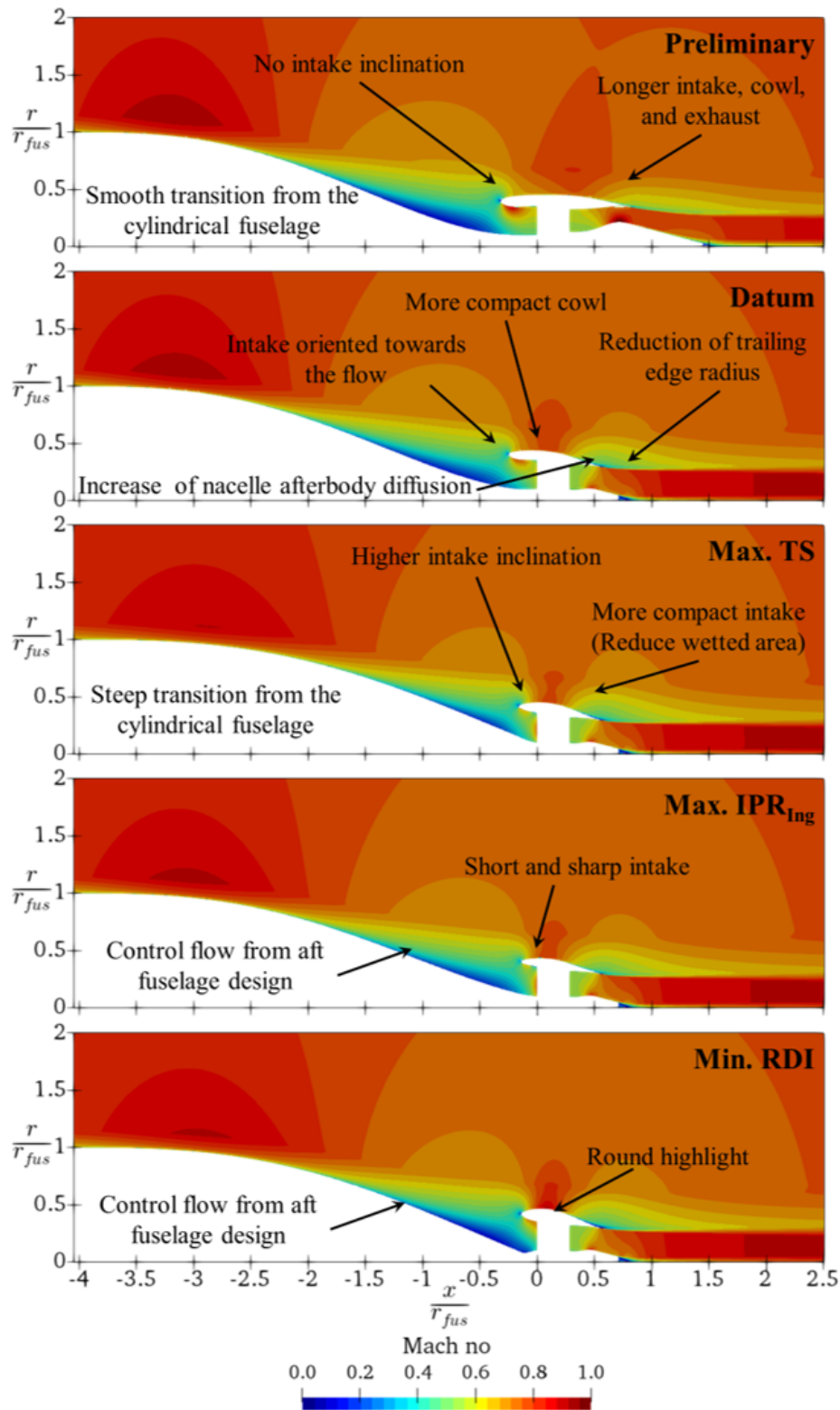
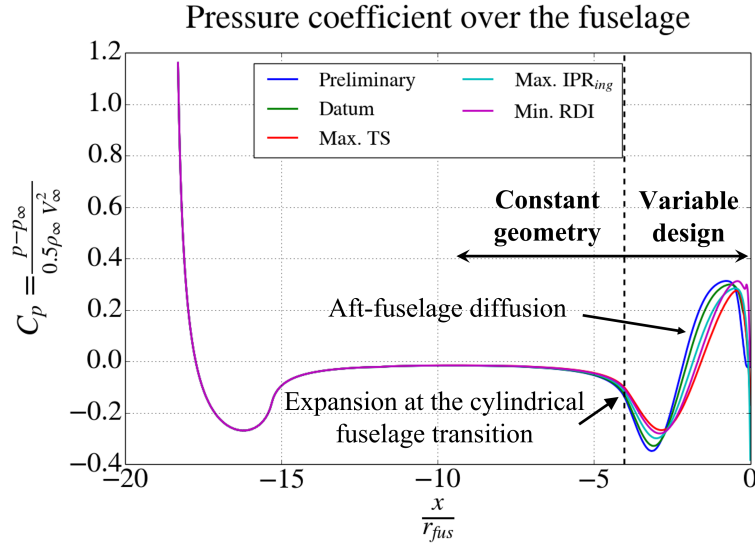
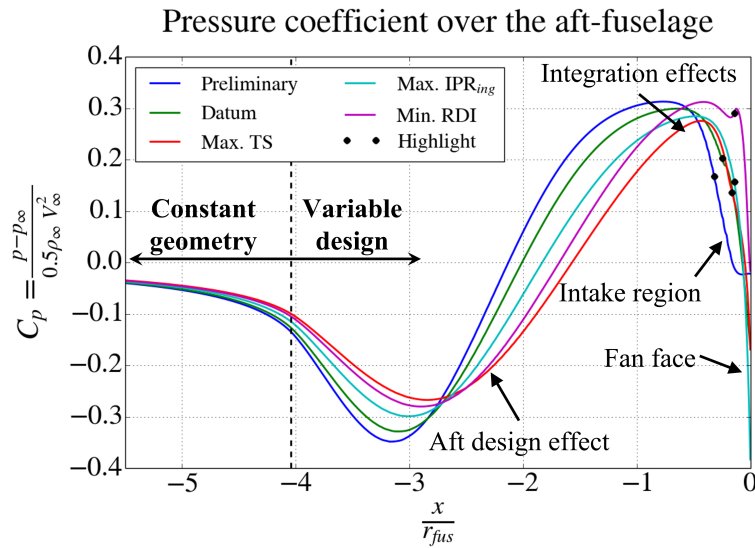


FIGURE 5.8: Mach number contours for the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

This has an effect on the exhaust performance and the amount of mass flow that the exhaust delivers for a given fan pressure ratio while the exhaust remain unchoked (Chapter 3.4.2.1).



(A) Pressure coefficient over the fuselage from the nose to the fan face



(B) Detail of pressure coefficient over the aft-fuselage curve, including the intake duct

FIGURE 5.9: Pressure coefficient distributions over the fuselage for the the base-line designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

The Datum design is a bespoke manual design to increase the propulsive contribution of the aft mounted embedded propulsor from the Preliminary design. A slightly steeper geometry transition from the cylindrical fuselage (Fig. 5.8) is included in combination with the increase of the intake inclination compared with the Preliminary design (Tab. 5.2). This aft fuselage and intake design reduces slightly the acceleration of the flow on the cylindrical fuselage transition and reduces the flow diffusion in the aft fuselage in comparison with the preliminary design (Fig. 5.9). However, the increase of the intake inclination and the slope over the aft fuselage enables some contribution of the aft fuselage pressure to the forward force of the propulsive system. A more compact cowl design is introduced in the Datum design with a lower intake and exhaust length in comparison with the Preliminary design (30% length reduction). The nacelle trailing edge radial position is also reduced to facilitate the fan cowl orientation towards the flow coming from the aft fuselage geometry (Tab. 5.2). This also increases the flow diffusion in the nacelle afterbody (Fig. 5.10b). This reduces the viscous drag contribution in the nacelle, due to the reduction of the local velocity in the nacelle afterbody (Fig. 5.11a). Also, the effective static pressure at the exhaust plane increases. As a consequence of this, the required fan pressure ratio to match the intake mass flow requirement increases. This has a positive effect on the thrust contribution of the system. However, additional power is required for the fan. The Datum design already provides a significant improvement on the thrust contribution of the aft mounted system ($\Delta TS \sim 0.031$) in comparison with the Preliminary design (Fig. 5.6). In addition, some additional benefits in the TS and the intake performance (IPR_{ing} and RDI) were obtained by the multi-objective optimisation process (Fig. 5.4). The top-performing non-dominated designs of the Pareto surface (Fig. 5.4) share some of the main aerodynamic characteristics of the Datum, but with some aerodynamic changes that improve the aerodynamic performance for each objective function.

In comparison with the Datum design, the maximum Thrust Split design (Fig. 5.4) has a greater increase of the intake inclination and a further reduction of the cowl and intake length (Tab. 5.2). The reduction of the cowl length reduces the overall wetted area with a further reduction of the viscous drag. At the same time, the increase in intake inclination increases the flow diffusion on the nacelle afterbody (Fig. 5.10b). This increases the forward force contribution of the cowl and the operating fan pressure ratio.

The non-dominated maximum TS design also has a more steep geometry transition from the cylindrical fuselage, with a higher overall slope on the aft-fuselage curve (Fig. 5.8). This reduces the acceleration of the first expansion in the aft fuselage

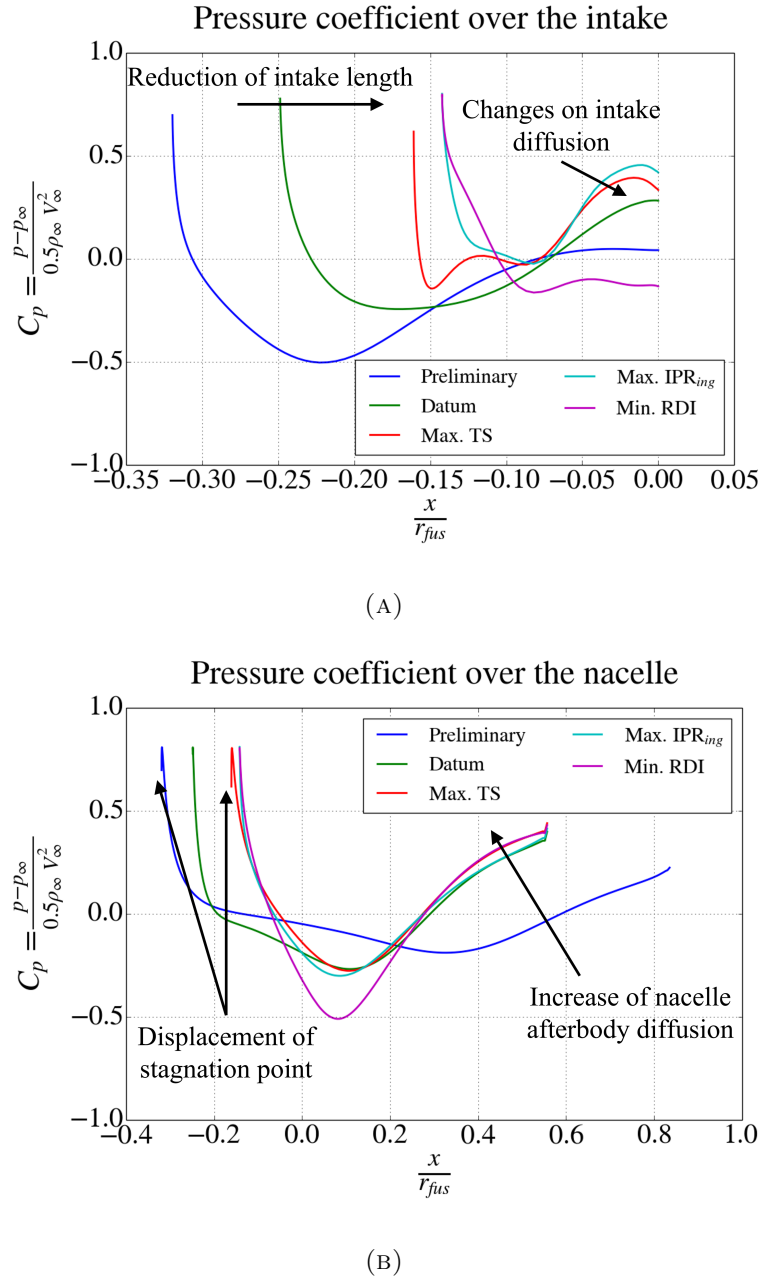
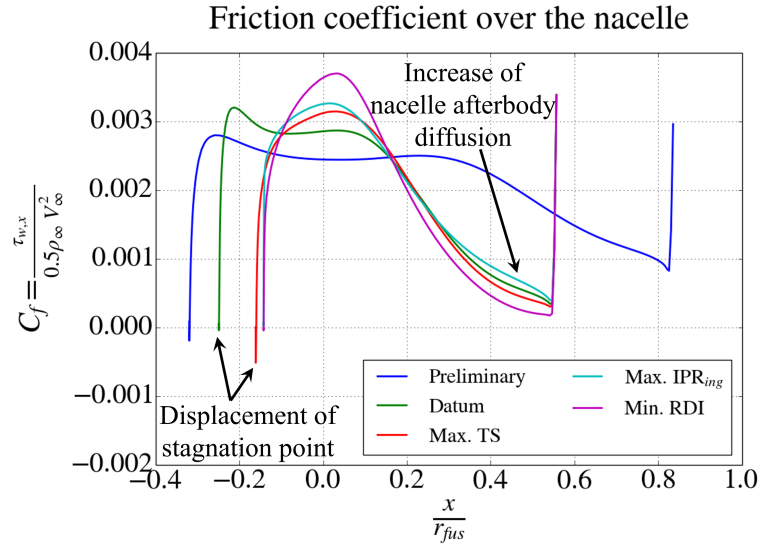
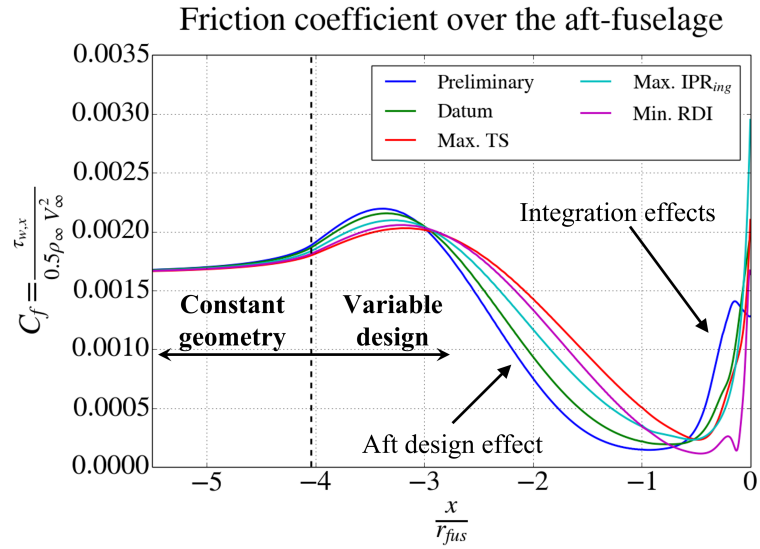


FIGURE 5.10: Pressure coefficient distributions over the intake (A) and the fan cowl (B) for the the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

curve and the overall diffusion in the aft fuselage geometry (Fig. 5.9b). The peak value of the friction coefficient (Fig. 5.11b) on the aft fuselage is reduced in comparison with the datum design, but the overall C_f value over the aft fuselage length increases. Hence, the steep inclination of the fuselage afterbody curve increases the viscous contribution of the drag. However, the increase on the pressure over the



(A)



(B)

FIGURE 5.11: Friction coefficient distributions over the fan cowl (A) and the aft-fuselage (B) for the the baseline designs and the top performing designs of the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$)

fuselage afterbody increases the forward force contribution of the propulsion integration due to the orientation of the surface. This is due to the increase of the axial component of the surface normal with the change on aft-fuselage geometry. The increase on the pressure force compensates partially the increase on the viscous drag.

The non-dominated maximum IPR_{ing} design has a shorter intake (Tab. 5.2) with a sharp leading edge (Fig. 5.8). This in combination with a less steep geometry transition from the cylindrical fuselage geometry allows the propulsive system to accommodate the flow at the fan face while the total pressure losses are reduced. The intake inclination (Ω_{int}) is corrected by the optimisation process to be oriented towards the flow. However, the presence of the sharp intake leading edge could have a negative impact on the performance under off design conditions where the stagnation point moves away from the highlight. If the $MFCR$ increases, fast accelerations of the flow could appear on the intake with a penalty on the intake performance. On the other hand, if the $MFCR$ decreases, the acceleration of the flow over the highlight could result in a significant increase of the spillage drag over the nacelle. Thus, a more detailed analysis of the performance under off-design conditions is required. This will be addressed later in this chapter (Section 5.4).

The non-dominated design with the maximum RDI also has a short intake (Tab. 5.2). However, the highlight curvature is higher than on the maximum IPR_{ing} non-dominated design. The aft fuselage also has a change in the slope within the intake duct (Fig. 5.8) combined with a steeper geometry transition from the cylindrical fuselage with a higher curve slope. With this aft fuselage curve design, the flow diffuses for longer than any of the other designs while the expansion at the origin of the aft fuselage is mitigated (Fig. 5.9b). In combination with an intake shape with low curvature rates from the throat to the fan face, the flow is compressed within the intake duct (Fig. 5.8) to reduce the radial variation of the total pressure within the duct. However, this results in a notable penalty on the IPR_{ing} values and the TS (Fig. 5.12).

The analysis of the different designs has identified some of the conflicts on the design space between the different objective functions. As well as some important shared characteristics. More compact cowl designs ($l_{nac}/r_{fan} \sim 2$ and $AR_{int} \sim 0.2$) with an increased intake inclination ($\Omega_{int} \sim 14 - 20$) could provide a significant benefit in TS . The fuselage aft design has a significant effect on the intake performance and the thrust contribution of the system. This affects the local flow at the intake and fan cowl, but also changes the distribution of the forces over the aft fuselage surface. The change on the forces is a combination of the aft fuselage design and the pressure gradient created by the propulsor (Fig. 5.10b). These effects were not accounted by lower order methods and require a higher order aerodynamic analysis such as CFD to be included in the system performance.

5.2.2.3 Influence of the design parameters on the aerodynamic performance

Prior to this point, the design and aerodynamic characteristics of the embedded propulsive system have been evaluated based on some reference designs. However, the information that can be obtained from a reduced number of designs is limited, and it cannot be used to fully derive design guidelines for the propulsion integration design of embedded propulsive systems. For this reason, it is important to extend the amount of data used for this purpose. Fortunately, the MOO results in a densely populated Pareto surface of 458 designs (Fig. 5.4). The non-dominated designs of the multi-objective optimisation can be used to identify the limits of the design space and the impact of the different design parameters on the aerodynamic performance.

When the non-dominated dataset of the Pareto surface is evaluated based on the prescribed optimisation limits (Figs. 5.12-5.13), some significant sensitivities to specific design parameters can be observed. On the aft fuselage control side, low slope values at the fan face (θ_{ff}) are required for all the designs of the non-dominated dataset (Fig. 5.12). Similarly, the throat area (A_{thr}) tends to go to the maximum limits of the design with a very small variation for any of the designs of the data set. A higher variation of the control of the aft fuselage slope (θ_{curv}) at the throat plane is observed. Some benefits on TS are found with the increase of θ_{curv} , while no clear relation between this parameter and the intake performance metrics (IPR_{ing} and RDI) can be identified (Fig. 5.13).

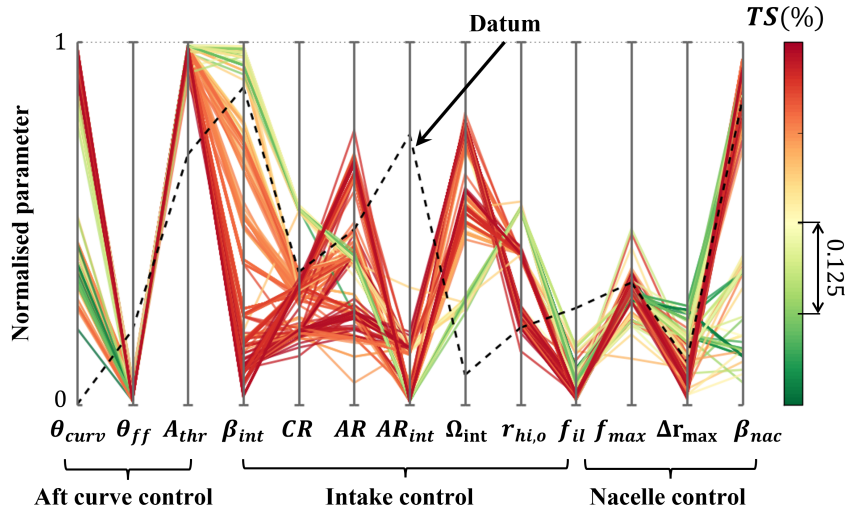


FIGURE 5.12: Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$) coloured by the Thrust Split value. Datum design as black dashed line. Values normalised with the optimisation limits

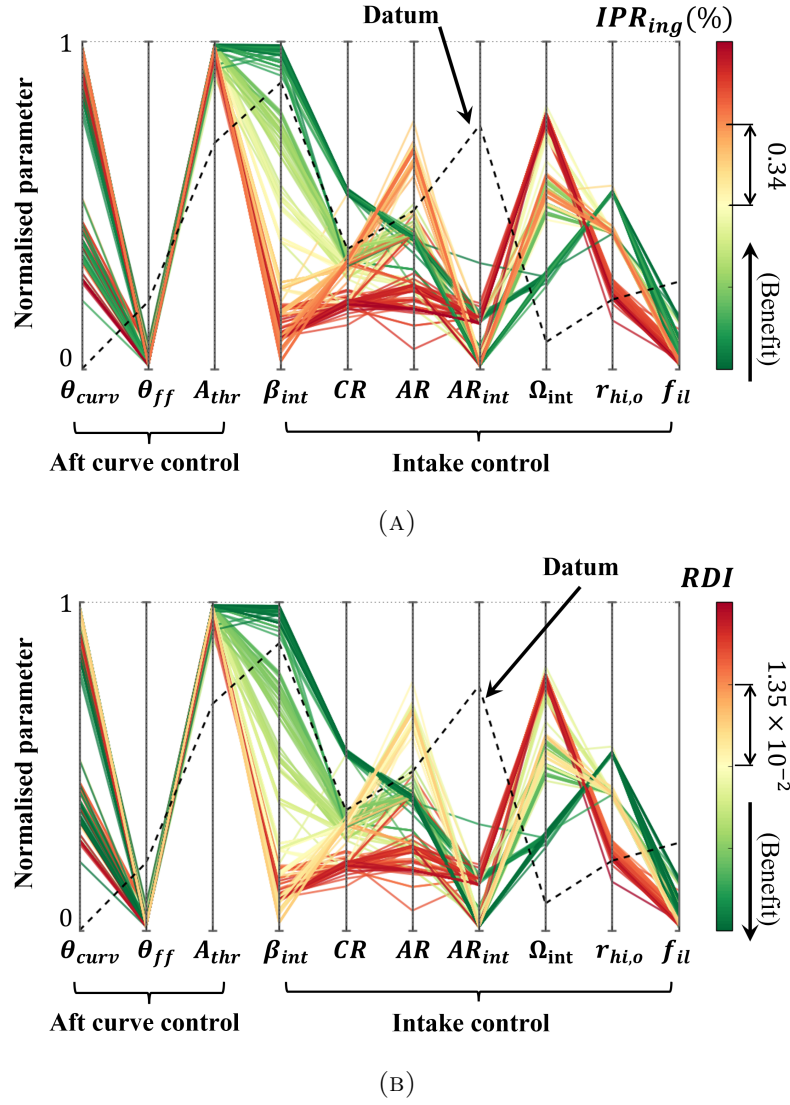


FIGURE 5.13: Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$) coloured by the intake performance: IPR_{ing} (A), and RDI (B). Datum design as black dashed line. Values normalised with the optimisation limits

The intake geometry control parameters present a higher variation in the Pareto front than the aft-fuselage curve parameters (Fig. 5.13). Some trends are identified between specific parameters and the objective functions. For the intake performance, the parameters that increase on the IPR_{ing} penalise the RDI . Lower contraction ratios (CR) and aspect ratios (AR) seem to improve the IPR_{ing} . This in combination with lower highlight curvature values (f_{il}), means that a thinner intake with a sharper leading edge is preferred to improve the pressure recovery at the intake. However, this penalises the intake radial distortion with a reduction of the flow control with the intake geometry. Higher values of the intake inclination (Ω_{int}) can have

a positive effect on the IPR_{ing} in combination with other parameters. Nevertheless, this could again have an immediate penalty on the RDI . The increase on the Ω_{int} could also have a positive impact on the TS , but the values are dispersed across a wider range (Fig. 5.12).

The nacelle geometry control parameters present a clearer trend with the TS (Fig. 5.12). Similar to the IPR_{ing} , lower highlight curvature radius (r_{if}) are preferred. This reduction of the nacelle radius with a lower maximum radius displacement from the highlight (Δr_{max}) indicate that flatter forebodies with sharp leading edges can provide some benefits on the TS . The best values of TS are found for the higher values boat-tail angle (β_{nac}). This in combination with the higher intake inclinations (Ω_{int}) could imply that a higher angle is required for higher positions of the maximum radius to enable the afterbody diffusion and maximise the thrust performance.

The parallel plot projection of the non-dominated designs has enabled the visual evaluation of some trends between designs. However, the dense amount of data requires a more focused analysis of the impact of each parameter on the aerodynamic performance. To identify the impact of the design parameters on the objective functions, a classification of the different non-dominated designs into performance groups is defined. To simplify the classification, the non-dominated designs were divided into four groups of equal size split by the quartiles for each of the three objective functions (TS , IPR_{ing} and RDI). The split of the groups is performed from the low to high values. Thus, the maximum values of each objective function are in group four and the minimum in the group one. In addition, for each group of each objective function, the space covered by each design parameter is explored by a box plot (Fig. 5.14). The box plot applies a categorisation of the value of the parameters also split by quartiles. To simplify the exploration and to focus on the regions of interest of the data, the outliers for each box plot were removed. To classify the outliers, the interquartile range (IQR) criterion is used. The IQR is defined as the range covered between the first (Q1) and the third quartile (Q3). Any data that lies out of the range between $Q1-1.5IQR$ and $Q3+1.5IQR$ is considered an outlier.

For the objective functions, the data groups of interest are mainly the fourth group for the variables to maximise (TS and IPR_{ing}) and the first quartile for the variable to minimise (RDI). The immediate consecutive groups (3 for TS and IPR_{ing} and 2 for RDI) can also be of interest to explore configurations that compromise between the different objective functions. For example, higher values aft fuselage slope control

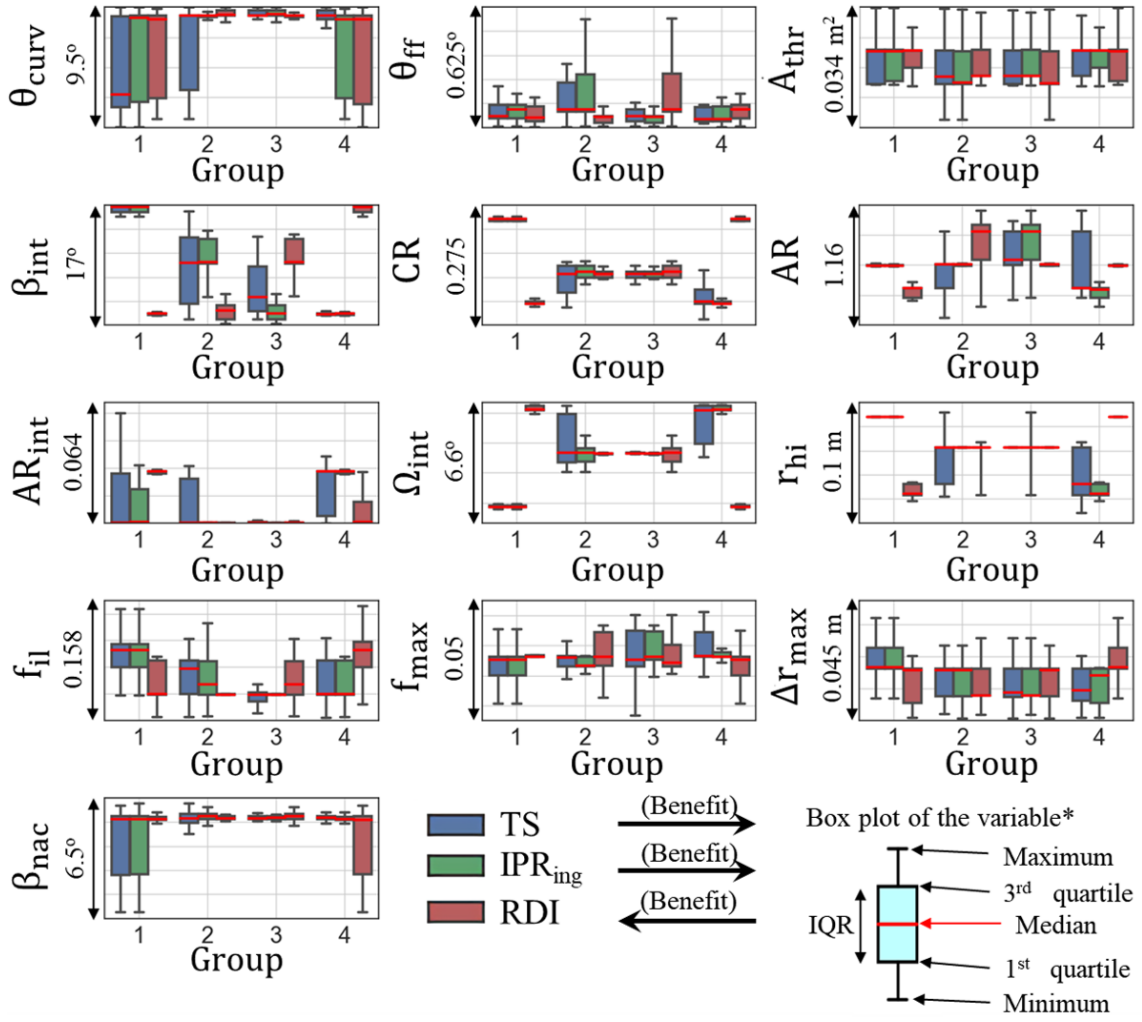


FIGURE 5.14: Box plots for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.1$ and $l_{aft}/l_{ps} = 0.85$) organised by the quartiles of each objective function (TS , IPR_{ing} and RDI)

(θ_{curv}) is required to maximise the TS with all data of group four centred inside a reduced range, while the best IPR_{ing} and RDI values are not completely centred in the same region of values, with approximately half of the samples out of this group. This means that some configurations with high TS could have adequate intake performance values. However, for the second best performing groups (i.e. 3 for TS and IPR_{ing} , and 2 for RDI), the range of θ_{curv} is concentrated around the same range with high slope angles required. Thus, a compromise could be achieved between the three objective functions while maintaining higher aft fuselage slopes.

For the other aft fuselage control variables (A_{thr} and θ_{ff}), as it was appreciated before, the designs of interest are located on a small range of the tested design space for all the variables (Fig. 5.14). For the intake design variables some other interesting

trends can be observed. Higher values of intake inclination (Ω_{int}) in combination with low angles of the intake slope at the fan face (β_{int}) can improve the intake performance. This is due to the correction of the orientation of the intake towards the flow from the aft fuselage and the orientation of the intake surface to minimise the intake total pressure losses. The TS has a slightly lower sensitivity to Ω_{int} , but this parameter will condition the design of the fan cowl by displacing the position of the highlight. Shorter intakes are preferred, with a very low variation of the AR_{int} independently on the data set. This helps to reduce the wetted area of the cowl to maximise TS , but also allows higher IPR_{ing} values and in some cases to reduce the RDI . The cowl design is conditioned by the highlight radial position and axial position. Overall, the position and value of the maximum radius determined by f_{max} and Δr_{max} are also placed within a reduced range from the overall data range. This was noted to indicate the preference for flatter and shorter nacelle forebodies (Fig. 5.12). Hence, higher values of the boat-tail angle (β_{nac}) are required to maximise the forward force contribution of the cowl and the flow diffusion at the trailing edge (Fig. 5.10b).

To summarise, some design guidelines can be derived from this study at this particular propulsor location in cruise conditions:

- For this given location of the propulsor ($r_{hub}/r_{fus} = 0.1$, $l_{aft}/l_{ps} = 0.85$), higher slopes of the aft fuselage can have a positive impact on the TS . However, this could have a negative impact in the intake operation and design. Short intakes can be used to reduce the fan cowl length and to improve the TS of the system while maintaining adequate intake performance
- The intake inclination (Ω_{int}) is a critical parameter to control the intake performance (RDI and IPR_{ing}). High intake inclination values are required to orient the nacelle towards the flow direction defined by the aft-fuselage slope. The intake inclination also defines the design requirements of the fan cowl
- Sharp leading edges with low curvature radius (r_{if}) can be used to improve the IPR_{ing} . However, higher values of the curvature radius at the highlight are required to reduce the radial distortion at the intake
- Short nacelle forebodies with low curvature rates can be used to improve the TS values. This has to be in combination with higher boat-tail angles to increase the flow diffusion in the nacelle afterbody

5.3 Effect of the system location

In the previous section, the requirements for the multi-objective optimisation for the aerodynamic performance of embedded propulsion systems were defined. The benefits of the optimisation methodology were evaluated for an specific configuration at a given propulsor axial and radial location. However, for closely integrated propulsion systems such as the aft mounted BLI configurations, the radial and axial position of the propulsor is expected to have a notable impact on the system performance. For the aft mounted annular propulsor, this location also defines the aft fuselage geometry requirements. As observed, this will define the intake performance and the thrust contribution of the system. Thus, it is necessary to understand the impact of the propulsor position on the aerodynamic performance.

In this section, the MOO methodology is applied to identify the sensitivities of the propulsion integration design to the propulsor location. The application of MOO enables the identification of the limits and characteristics of the design space. In addition, the influence of the propulsor location on the aerodynamics is evaluated. Finally, a study of the influence of the design parameters on the performance based on the location is provided.

5.3.1 Definition of the study

To evaluate the impact of the system location, a Design Space Exploration (DSE) for different radial and axial hub positions was carried out (Fig. 5.15). To define the points of interest, the geometry of the single-aisle medium sized aircraft of reference is used. The reference fuselage applied for the performance evaluation is based on the sideline of this aircraft. For this study, the side closing angle, δ_{side} , and the fuselage upsweep angle, δ_{ups} , are used to guide the propulsor locations of the system (Fig. 5.15). The δ_{side} is defined as the angle between the end of the cylindrical section and the end of the reference fuselage from the top view. The δ_{ups} is defined as the angle between the end of the cylindrical section of the aircraft's belly and the end of the reference fuselage from the side view. To keep the similarity with the reference axisymmetric fuselage, the geometry is kept constant from the nose to a 75% of the reference aircraft length¹⁴⁰. For each location, the reference fuselage length is used as a constraint for the maximum dimensions of the aft mounted propulsor integration design. Eight locations were defined for the propulsor location DSE. Three axial positions were defined for the propulsor hub at 50%, 70% and 85%

of the reference aft fuselage length (l_{ps}). The two reference angles (δ_{ups} and δ_{side}) and an intermediate angles were used to define the radial position of the propulsor at the given axial locations (Fig. 5.15).

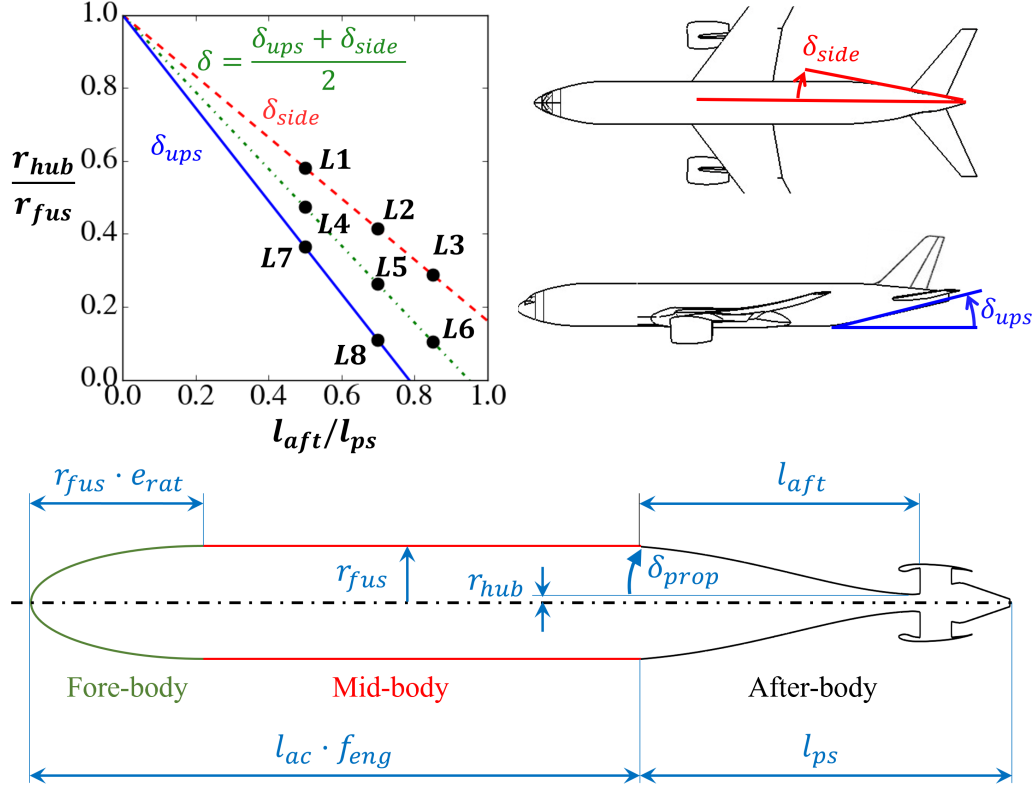


FIGURE 5.15: Definition of the different axial and radial locations of the propulsor for the study of the propulsor location effects. Definition of the reference angles based on a sample aircraft, not to scale

The reference areas of the propulsion system (A_{fan} , A_{noz} , $A_{OGV,out}$) were kept constant based on the required dimensions of the propulsion system (Appendix B). This maintains a constant mass flow rate for the propulsor and a target Mach number at the fan face and exhaust inlet. The eight selected points of the location DSE cover a representative range of fan design possibilities for the given fan area requirements. Configurations with high hub-to-tip ratios ($httr$) of the order of 0.8 with short blade spans are covered by the locations L1, L2 and L4 (Fig. 5.15). Configurations with $httr \sim 0.3$ and greater blade spans are represented by the locations L6 and L8. The fan dimensions of the first group are on the region of low pressure compressor blades, while the fans of the second group are closer to a conventional wing-mounted fan configuration. The axial length of the propulsor blade rows was kept constant for all the locations. The exhaust length and areas are kept constant between configurations, but the cone length is extended to match the overall system length requirements. With the exception of location L8, all the cones extend up to

the length of the reference fuselage, where they are cropped. Location L8 aft cone is shorter due to the space limitations. The location L6 corresponds to the location used in the case study in Section 5.1.

To provide a systematic comparison of the different locations of the study (Fig. 5.15), a multi-objective optimisation (MOO) of the aerodynamic performance is performed for all the cases. The MOO methodology defined in Section 5.2 is used for the eight locations of the DSE. In the seed initialisation for the 13 geometric degrees of freedom, an LHS of 260 individuals is used for each location. A maximum of 85 generations of 65 individuals each was set for each optimisation. The convergence criterion defined in Chapter 3.5 is applied for all the optimisations of the test. Aerodynamic and geometric filters were applied as defined in the Section 5.2.

Prior to the optimisation of the different locations, the correlation and monotonicity between the different objective functions was evaluated to understand if it is required to modify the optimisation problem (Fig. 5.16). The $TSPC$ and the TS have a strongly negative monotonicity for all the propulsor axial and radial locations (Fig. 5.16), with values of the Spearman index close to -1 . The Pearson index value for the $TSPC$ and the TS varies with the propulsor location. However, when the logarithms of the TS and the $TSPC$ are compared, the Pearson index rises to values between -0.97 and -1 . This is a consequence of the inverse correlation between both metrics (Eq. 5.1). The only metric that affects this correlation is W_{prop} . This metric varies with the changes of FPR to match the required mass flow at the exhaust. However, the effect of the W_{prop} in the $TSPC$ is small in comparison with the effect of the TS . Thus, only one of the metrics could be used to reduce the complexity of the optimisation problem. For the present work, only TS is selected as an objective function.

Compared with the consistent correlation and monotonicity observed between $TSPC$ and TS , a greater variation on the value of the Pearson and Spearman indices is observed between the RDI and the IPR_{Ing} (Fig. 5.16). The lower propulsor radial locations (L6 and L8) have a positive moderate correlation and monotonicity and the higher propulsor radial locations have a moderate negative correlation. A very weak correlation is observed for the intermediate propulsor radial values (L2, L3, L5, L7). A moderate negative correlation and monotonicity is observed at higher radial locations (L1 and L4). The change on the metric correlation shows a strong dependence of the system location in the inflow conditions and the intake aerodynamic behaviour.

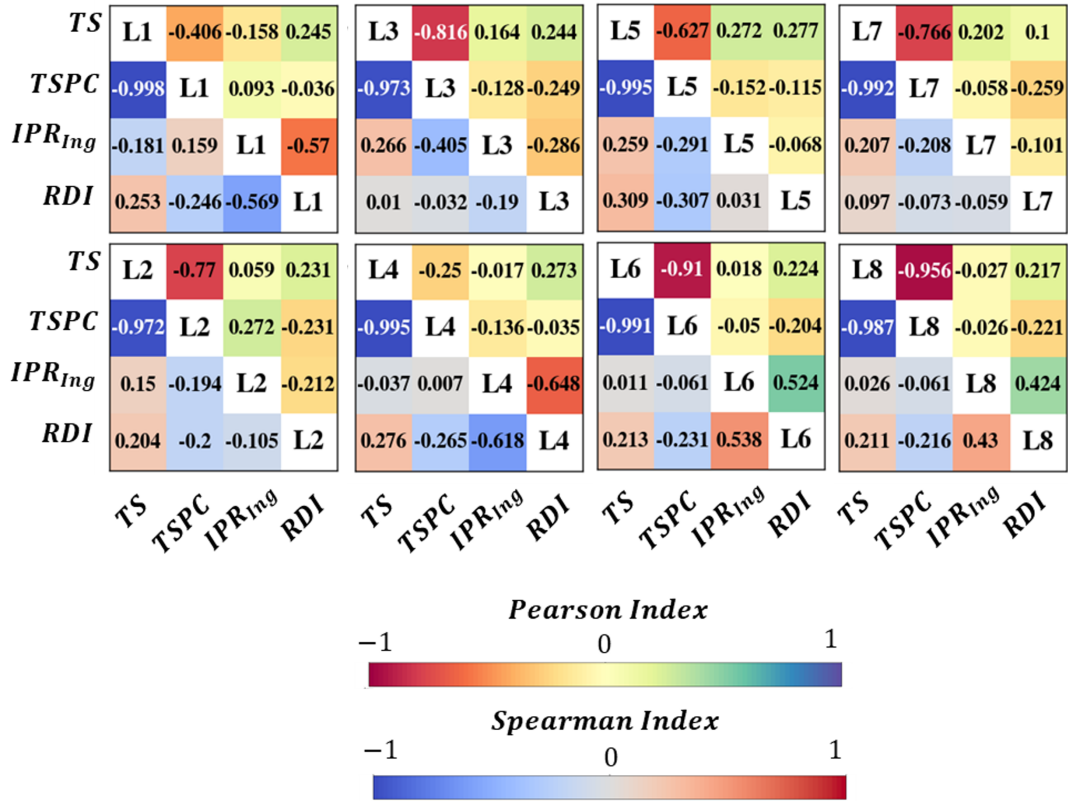


FIGURE 5.16: Candidate objective functions correlation for the 8 locations of the Design Space Exploration (Fig. 5.15). The Pearson Index is evaluated on the top section (upper right) of the correlation matrix of each location. The Spearman index is evaluated on the bottom section (lower left) of the correlation matrix

Thus, it is advisable to evaluate the correlation between objective functions prior to each MOO problem set-up to determine design requirements for the embedded propulsion system. The variation on the behaviour of both metrics with the location does not allow to reduce the number of objective functions in all cases. Hence, to keep a consistent methodology between the optimisations, both metrics are used as objective functions. Then, the optimization problem consists of three objective functions (Eq. 5.2), with two functions to maximize (TS and IPR_{ing}) and one to minimize (RDI).

5.3.2 Effect of the location of the propulsion system on the performance

The non-dominated designs on the Pareto surface for each of the locations of the DSE study are evaluated to determine the maximum possible benefits of each radial

and axial location (Fig. 5.15). The design space of each of the locations leads to different Pareto surface distributions (Fig. 5.17). In some locations the Pareto surface is more sparsely populated. The MOO convergence criterion (Chapter 3.5) was satisfied for all the locations.

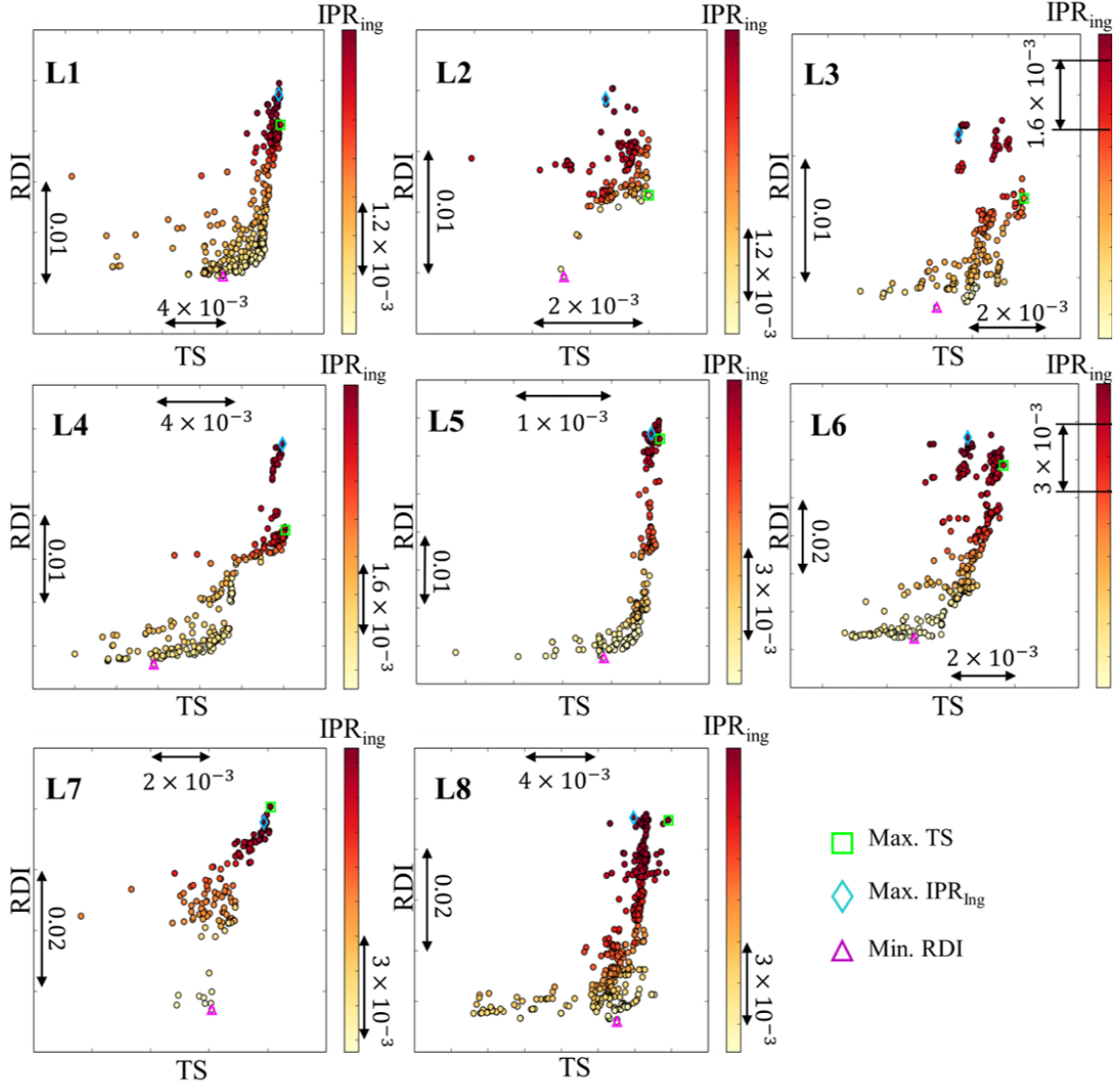


FIGURE 5.17: Pareto surfaces for the eight cases of the location design space exploration

The propulsor location affects the range covered by the non-dominated designs for each of the three objective functions (Fig. 5.18). This not only affects the optimal values for each of the objective functions, but the flexibility of the design space to obtain designs that compromise between the different metrics. For example, in the configurations placed on the upswep angle, δ_{ups} , (L7 and L8) the maximum IPR_{ing} and TS are concentrated in a very small range (Fig. 5.18). This shows that there

is very little improvement in either metric by changing the design. However, at the same time some gains in RDI can be obtained by selecting non-dominated designs that have a lower IPR_{ing} or TS benefits. On the other side of the design space are the non-dominated designs of the configurations positioned on the closing side angle, δ_{side} , such as L1, L2 and L3. In these configurations, the non-dominated cover a higher range of the IPR_{ing} and the TS , while there is less margin for changes on the RDI . Hence, it can be determined that the optimal design space for the propulsion integration aerodynamic design of the embedded system is highly sensitive to the propulsor axial and radial location. This will affect both the maximum performance obtained for each location and the flexibility on the design to compromise between the selected objective functions.

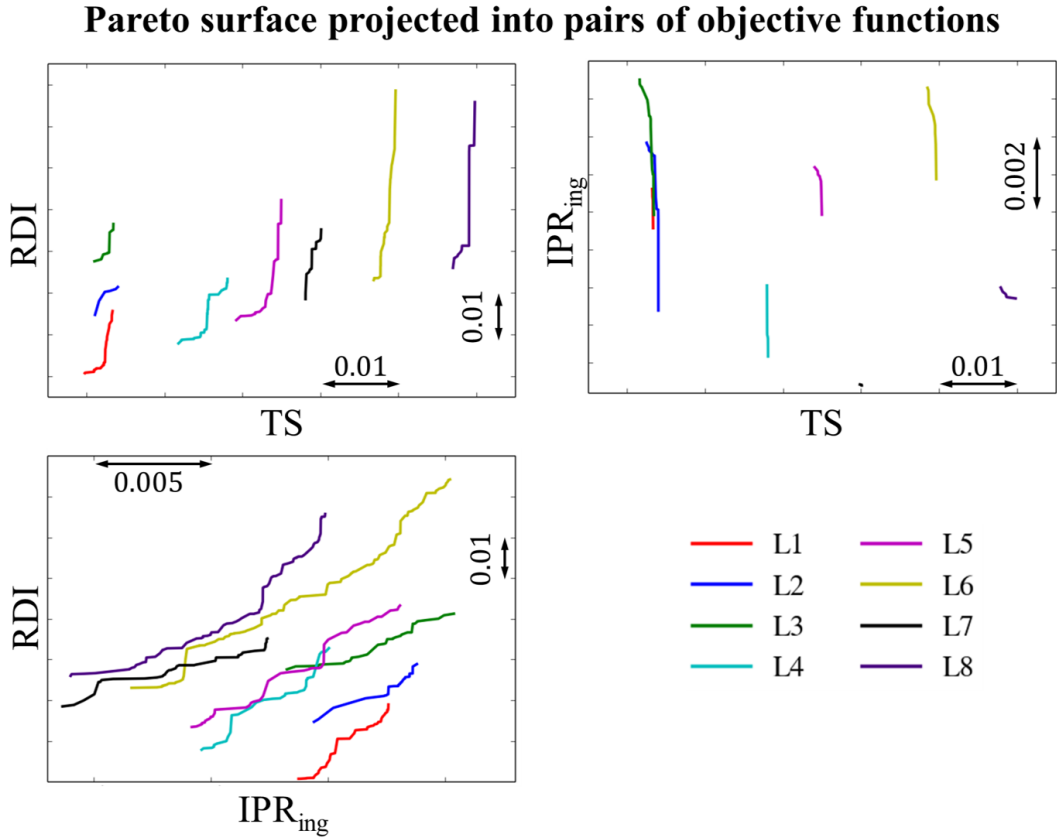


FIGURE 5.18: Pareto surfaces projected into pairs of objective functions for the eight cases of the location design space exploration

From the Pareto surfaces of the selected locations it can be observed that the peak value of each objective function is highly dependent on the position of the propulsive system (Fig. 5.17). To determine the limits of the design space for each location of the propulsor, the best performing non-dominated designs for each objective function were selected. With these non-dominated designs, the optimal performance was

mapped to the axial and radial location to identify the impact of the propulsor position on the aerodynamic performance (Figs. 5.19-5.21).

The maximum ingested pressure recovery, IPR_{ing} , increases with the axial and radial position of the propulsor (Fig. 5.19). From location L5 ($x/r_{fus} = 0.26$, $l_{aft}/l_{ps} = 0.7$), the increase of the radial hub location to L2 ($x/r_{fus} = 0.41$) can provide benefits of up to 0.0008 on the IPR_{ing} . If the axial location is increased, for example from L5 to L3 ($x/r_{fus} = 0.28$, $l_{aft}/l_{ps} = 0.85$) the maximum value of ingested pressure recovery can be improved in 0.0024. At the same time, the benefits derived from the increase of the axial location of the propulsor can outweigh the penalties from the reduction of the radial location. For example, from location L5 ($r_{hub}/r_{fus} = 0.26$, $l_{aft}/l_{ps} = 0.7$) to L6 ($r_{hub}/r_{fus} = 0.1$, $l_{aft}/l_{ps} = 0.85$) an improvement of 0.0022 on the IPR_{ing} is obtained.

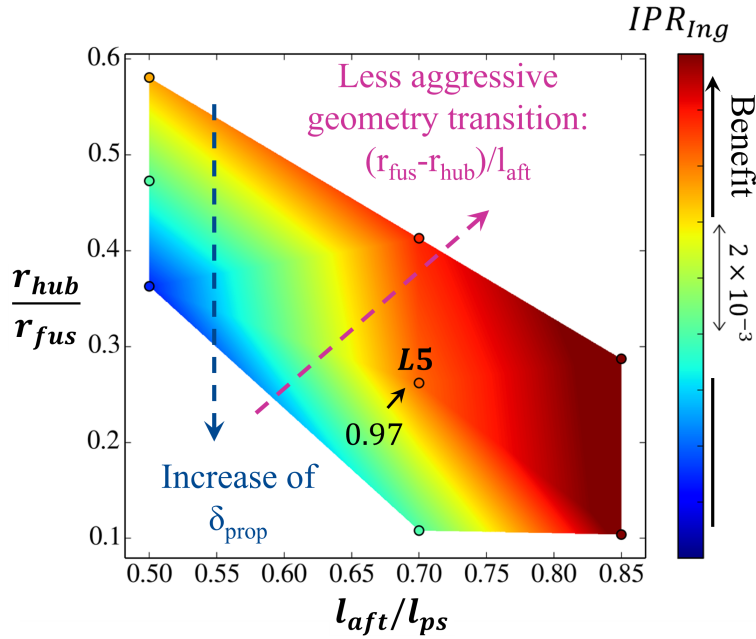


FIGURE 5.19: Propulsor radial and axial location effects evaluated from the best performing ingested pressure recovery non-dominated designs of each optimization

The increase on maximum TS is driven mainly by the reduction of the radial position of the propulsor, but it is also increased with the reduction of the axial position (Fig. 5.20). For example, from location L5 ($r_{hub}/r_{fus} = 0.26$, $l_{aft}/l_{ps} = 0.7$) to location L8 ($r_{hub}/r_{fus} = 0.106$, $l_{aft}/l_{ps} = 0.7$) a maximum increase of 0.015 on the Thrust Split is obtained. Also, when the radial location increases from L5 ($r_{hub}/r_{fus} = 0.26$, $l_{aft}/l_{ps} = 0.7$) to L2 ($r_{hub}/r_{fus} = 0.42$, $l_{aft}/l_{ps} = 0.7$), the maximum achievable TS is penalised by 0.021. At the same time, additional benefits on the maximum

TS can be achieved with less aft locations. For example, the maximum TS rises approximately a 0.01 from location L6 ($r_{hub}/r_{fus} = 0.103$, $l_{aft}/l_{ps} = 0.85$) to L8 ($r_{hub}/r_{fus} = 0.106$, $l_{aft}/l_{ps} = 0.7$).

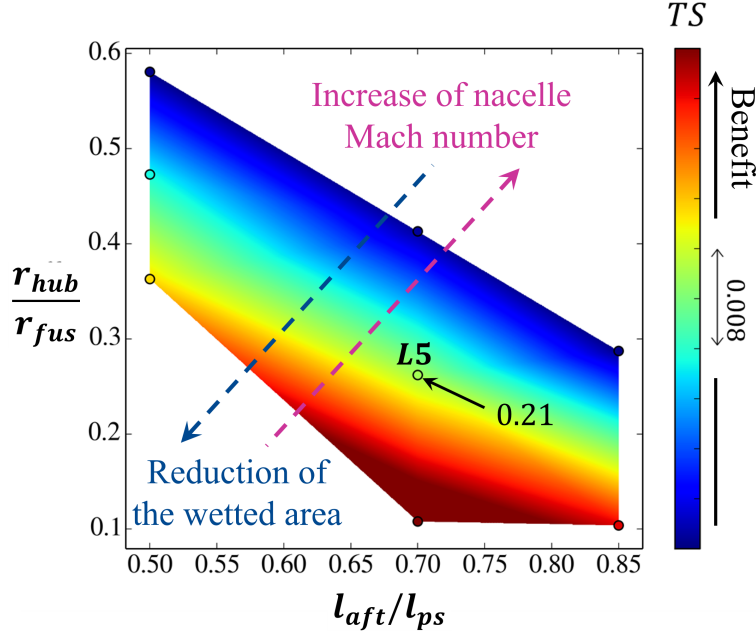


FIGURE 5.20: Propulsor radial and axial location effects evaluated from the best performing Thrust Split non-dominated designs of each optimization

It is clear that the propulsor radial and axial location can have a notable impact on the maximum TS achievable. A maximum difference of 0.04 on the TS values is observed between the locations L1 ($r_{hub}/r_{fus} = 0.58$, $l_{aft}/l_{ps} = 0.5$) and location L8 ($r_{hub}/r_{fus} = 0.106$, $l_{aft}/l_{ps} = 0.7$). This can lead to significant variations on the power savings derived from the aft mounted propulsor (Chapter 4). This translates to approximately 4-6% variations on the PSC value. Considering that the expected benefits for this configuration are approximately 3-11%, it is concluded that the change on the propulsor axial and radial location has a major impact on the maximum power savings. Under this context, the benefits in PSC obtained with a detailed MOO of the propulsion integration design (i.e. $\sim 0.3\%$ as in Section 5.2) have a second order effect in comparison with the impact of the bulk geometry of the propulsion parameters (i.e. system length, and axial and radial position).

The minimum achievable radial distortion (RDI) depends on the radial and axial location of the propulsor (Fig. 5.21). The increase on radial location of the propulsor can mitigate the radial distortion for a given axial position. For example, the minimum RDI decreases approximately from 0.07 to 0.06 with the increase on the radial location from L7 ($r_{hub}/r_{fus} = 0.36$, $l_{aft}/l_{ps} = 0.5$) to L4 ($r_{hub}/r_{fus} = 0.47$,

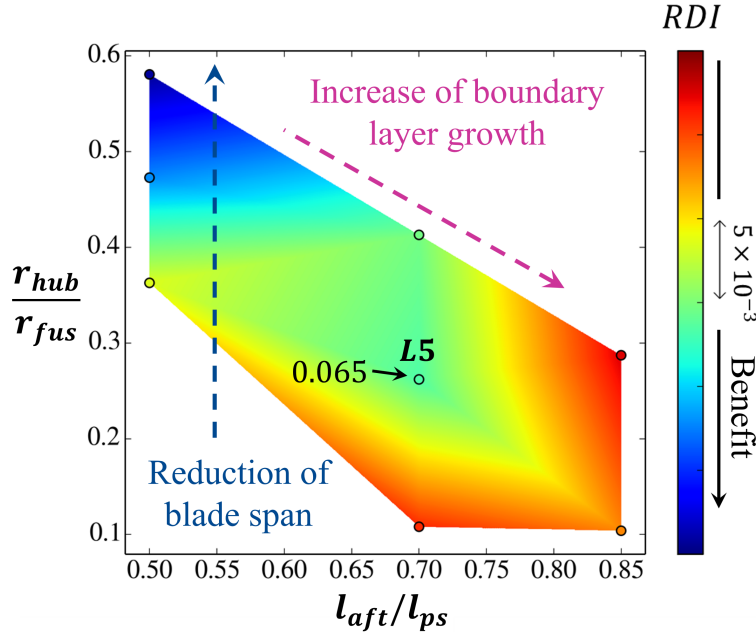


FIGURE 5.21: Propulsor radial and axial location effects evaluated from the best performing Radial Distortion Index non-dominated designs of each optimization

$l_{aft}/l_{ps} = 0.5$), and reduces to 0.05 for L1 ($r_{hub}/r_{fus} = 0.58$, $l_{aft}/l_{ps} = 0.5$). For the current DSE, the increase on the axial location of the propulsor is combined with a reduction of the radial location. This increases the radial distortion achieved in aft most locations. However, at the same time some reduction of the RDI is observed for intermediate axial locations such as L5 ($r_{hub}/r_{fus} = 0.26$, $l_{aft}/l_{ps} = 0.7$) compared with an aft most locations at a similar radial position such as location L3 ($r_{hub}/r_{fus} = 0.28$, $l_{aft}/l_{ps} = 0.85$). Nevertheless, the radial distortion level remains high for all locations¹⁷⁷, with a minimum of $RDI \sim 0.05$ in location L1.

Prior to any detailed aerodynamic analysis of the effects of the propulsor location, it is necessary to understand the interaction between the different objective functions when the location of the propulsive system is modified. Aft most locations provided the best possible IPR_{ing} values, while the TS improved mostly with the more forward and lower radial positions (e.g. L8), having the radial location the highest impact on the TS . Some compromise could be achieved between both metrics to maintain an adequate performance, with aft most locations at lower radial positions such as L6 ($r_{hub}/r_{fus} = 0.103$, $l_{aft}/l_{ps} = 0.85$). This shows that there is still some design flexibility on the propulsor location to balance the different requirements of the propulsion integration. However, at the same time the minimum RDI improves with the increase of the radial location and the reduction of the axial location. This is in direct contradiction to the requirements of the other two objective functions.

Hence, the maximum levels of intake radial distortion condition the benefits obtained in the other objective functions (TS and IPR_{ing}) with the change of the propulsor position.

The main purpose of the aft mounted boundary layer ingestion propulsor is the reduction of the power requirements of the aircraft. This is achieved mostly by the increase of the thrust contribution of the embedded propulsor (Chapter 4). At the same time, the intake distortion and total pressure values will determine the turbomachinery design, performance, and potential lifespan. Hence, the designer has to prioritise designs with higher TS values, but at the same time maintain reasonable levels of intake performance. Thus, there is an interest in the advance of turbomachinery technology to design distortion tolerant fans that can endure the distortion conditions that higher TS configurations require⁷⁸.

5.3.3 Effect of the location of the propulsion system on the aerodynamics

Before this section, the limits of the design space were explored for different axial and radial location of the propulsive system (Fig. 5.15). Lower r_{hub}/r_{fus} were found to increase on the thrust contribution of the propulsive system (i.e. higher maximum TS values) while they could penalise the maximum IPR_{ing} and RDI . The penalties in the maximum IPR_{ing} could be mitigated with the increase on the l_{aft}/l_{ps} , but this will produce some penalties on the TS and RDI . In this section, the aerodynamics of such locations are explored to understand their effect on the performance and design requirements.

To illustrate the effects of the change of r_{hub}/r_{fus} and l_{aft}/l_{ps} on the aerodynamics of the propulsion integration, the first three designs (L4, L5 and L6) placed at an equal propulsor angle (Eq. 3.9), δ_{prop} , are selected (Fig. 5.15). For each of these locations, the Mach number distributions of the non-dominated designs with the best performance for the three objective functions (TS , IPR_{ing} and RDI) are analysed (Figs. 5.22-5.24).

The non-dominated designs with higher radial locations such as L4 are exposed to higher Mach numbers over the fan cowl than the aft most locations, such as L5 and L6 (Fig. 5.22). This increases the overall drag force over the cowl, with changes on the viscous and form drag contributions of the cowl. At the same time, the increase on the r_{hub}/r_{fus} results in a higher wetted area of the propulsion integration housing

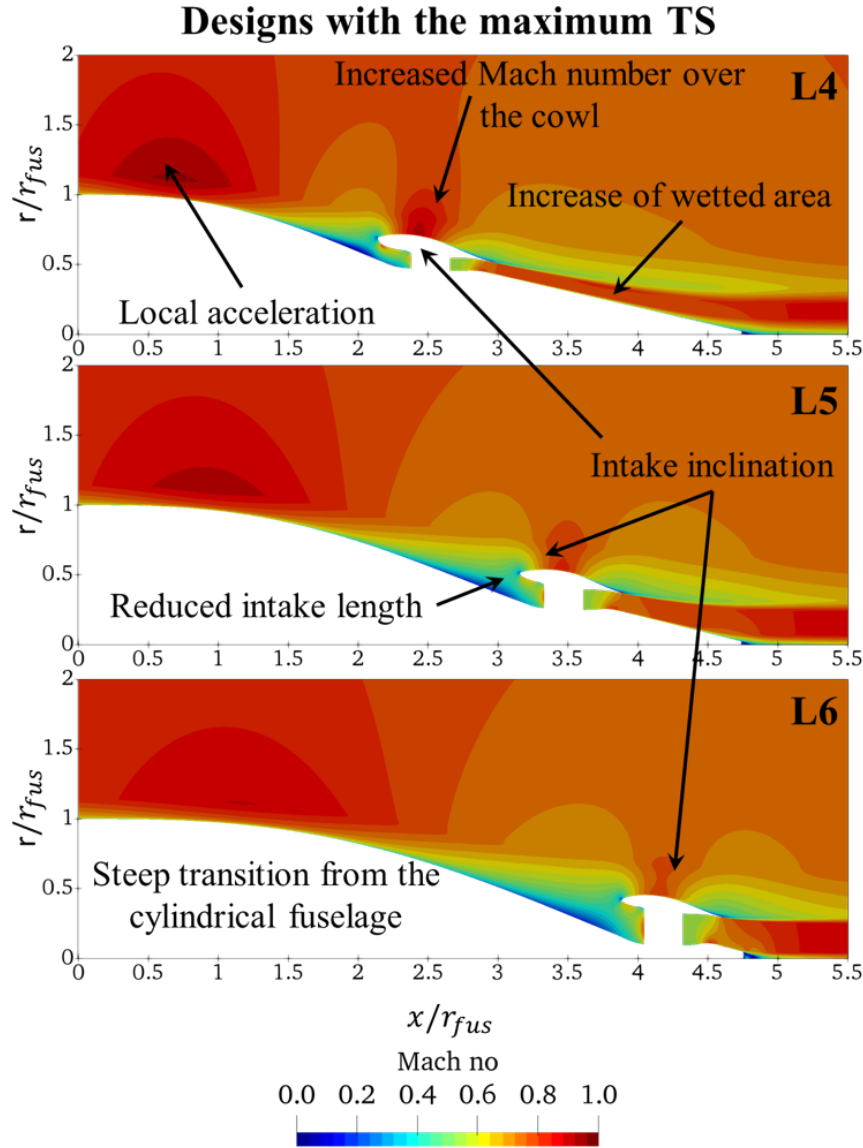


FIGURE 5.22: Mach number contours for the best TS non dominated designs of the locations L4, L5, and L6

components. This adds a further penalty on the drag force that leads to lower achievable TS values. For higher radial locations such as L4, forward positions of the propulsor (i.e. lower l_{aft}/l_{ps}) could apply the increase of the area to increase the pressure force contribution on the exhaust cone and compensate the rise on viscous drag. However, lower radial locations such as L6 still provide a more significant improvement on the maximum TS than higher radial locations, with approximately a 0.022 benefit on the maximum TS for L6 compared with L4 (i.e. 2.2%-3.5% in PSC).

The reduction of the total pressure loss (i.e. increase in IPR_{ing}) with the changes

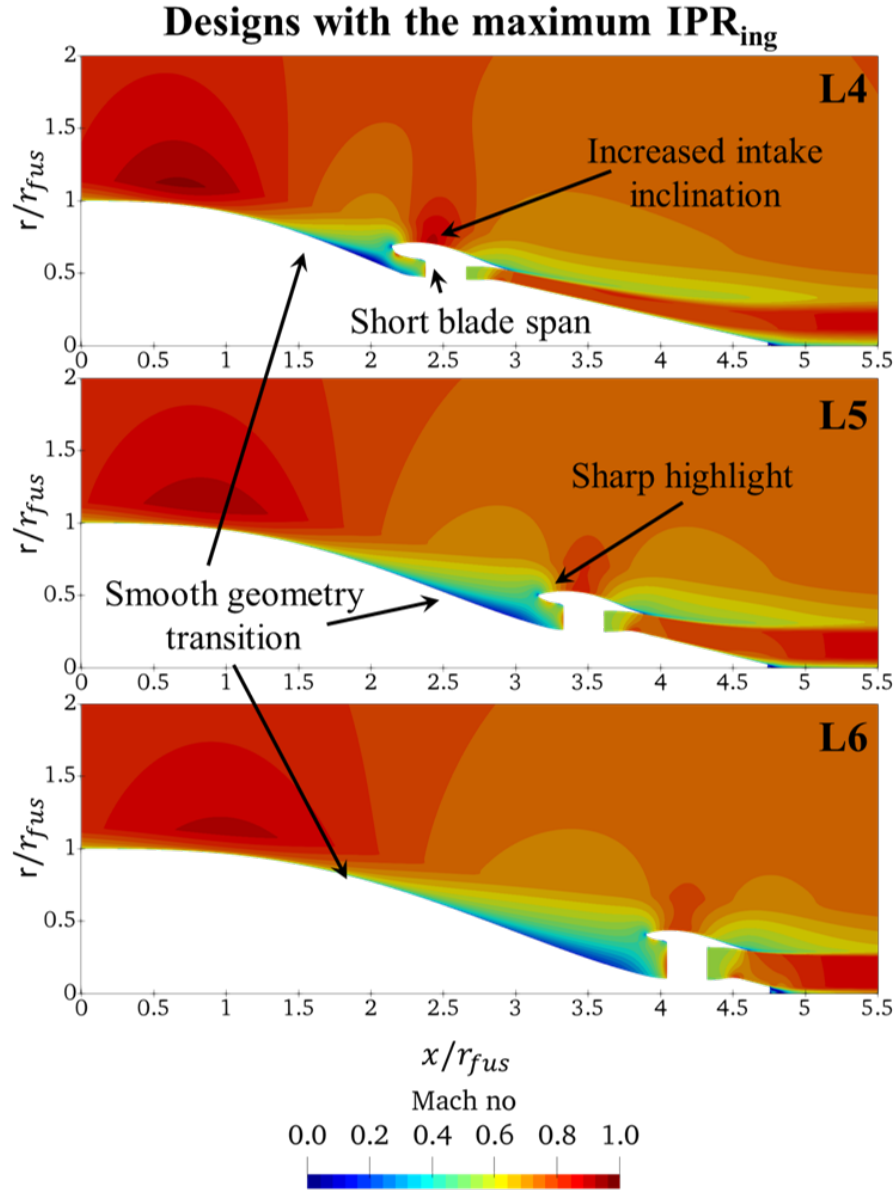


FIGURE 5.23: Mach number contours for the best IPR_{ing} non dominated designs of the locations L4, L5, and L6

on the propulsor location (Fig. 5.19) can be attributed to the change of design requirements on the aft-fuselage geometry (Fig. 5.23). More aft propulsor locations such as L6 extend the length of the aft fuselage curve and allow a smoother transfer of the flow to the intake propulsor, with a less aggressive pressure gradient at the aft-fuselage. At the same time, the increase on the radial position reduces the offset between the cylindrical fuselage radius and the propulsor hub with a less aggressive geometry transition ($(r_{fus} - r_{hub})/l_{aft}$) than the cases with a lower hub location. Thus, it is understood that while some balance is maintained between the axial and

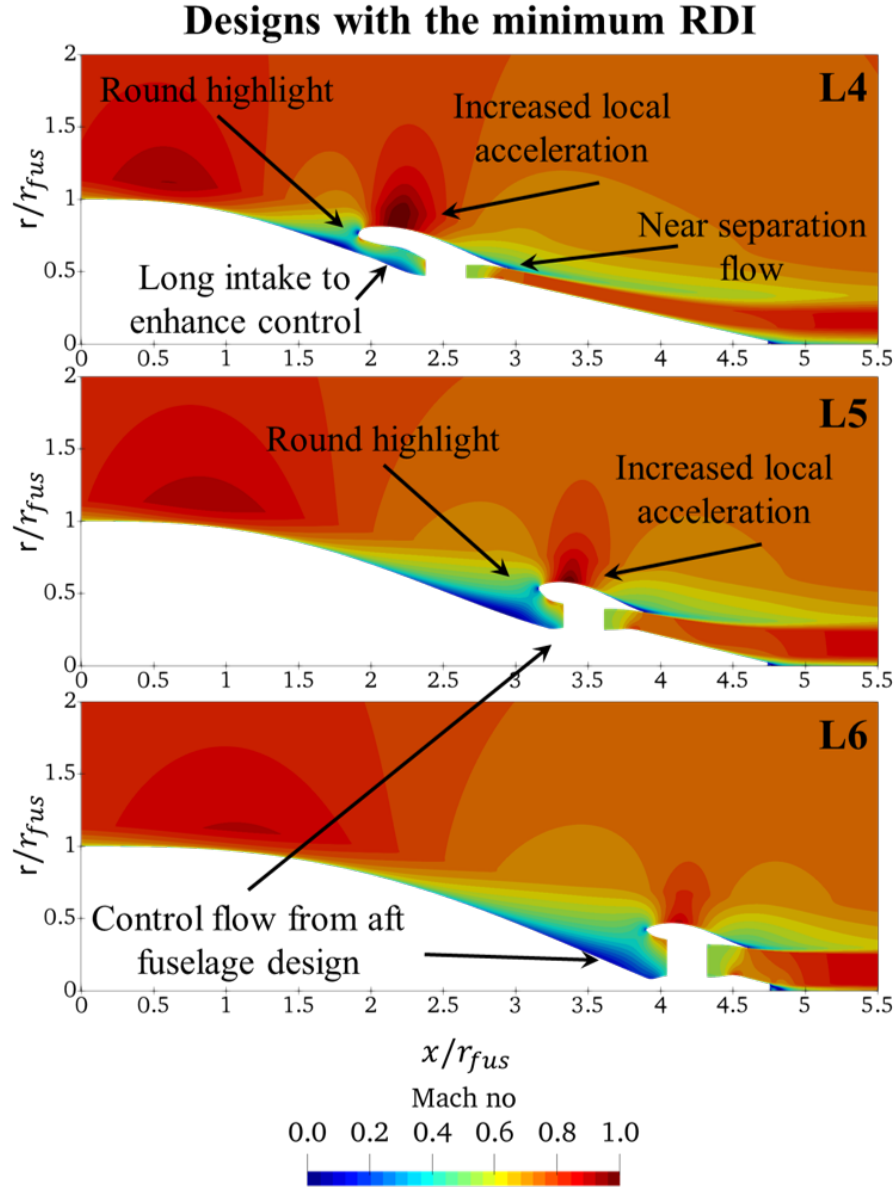


FIGURE 5.24: Mach number contours for the best RDI non dominated designs of the locations L4, L5, and L6

the radial position, higher values of IPR_{ing} can be obtained.

Under these conditions, the design of the aft-fuselage surface seems to be the main variable to control in order to maintain an adequate value of the IPR_{ing} at a given propulsor location. The maximum TS designs favour a steeper geometry geometry at the aft fuselage. This is believed to increase the forward force contribution of the aft-fuselage with the geometry orientation and the change on the diffusion pattern over the curve (Section 5.2.2.2). Compared with the maximum TS non-dominated designs (Fig. 5.22) a smoother geometry transition between the cylindrical fuselage

and the fan face is observed in the maximum IPR_{ing} non-dominated designs (Fig. 5.23). This favours the reduction of the total pressure losses at the aft-fuselage surface and increases the control over the boundary layer profile at the propulsor intake.

For the non-dominated designs with minimum RDI at the selected locations (L4, L5 and L6), the radial location is observed to have a notable effect on the design requirements and aerodynamics (Fig. 5.24). At high radial locations and foremost positions of the propulsor such as L4, there is a limited space for the aft fuselage geometry to modify the boundary layer flow from the cylindrical section of the fuselage to the intake. Under these conditions, the best option to reduce the intake distortion by the optimisation algorithm is to favour longer intakes to control the flow within the intake duct. On the other side, the aft most locations with lower r_{hub}/r_{fus} such as L5 and L6 have more space to accommodate the flow in the aft fuselage geometry before the intake duct. Thus, the optimiser can favour shorter intakes. This is important for the drag reduction of the system, since the longer intakes at higher radial locations result in a significant increase of the wetted area of the fan cowl. However, the intake distortion is expected to be significantly higher for location L6 in comparison with L4, with a penalty of approximately 0.014 units of RDI .

As it was noted in the previous section, it is of the interest of the designer to prioritise the TS to achieve higher reductions in the power required by the vehicle. This is better achieved in locations lower r_{hub}/r_{fus} and l_{aft}/l_{ps} . Thus, it is advisable to focus on this region of the design space (Figs. 5.25-5.26). The maximum global TS value was obtained for location L8, which is aligned with the location L5 radially and with the location L6 axially. These two can provide some improvements on the intake performance with increases on the maximum IPR_{ing} in both locations, and some reductions in the minimum RDI for location L5.

If the design effort is focused in such locations (L5, L6, and L8), the aerodynamic operation and design requirements are fairly similar. The design requirements of the fan cowl are dictated by the aft fuselage geometry, mainly by the flow incidence. The aft fuselage geometry can be used to accommodate the flow at the intake to reduce the total pressure losses (Fig. 5.26) or to obtain an additional forward force contribution of the aft fuselage body (Fig. 5.25). Under any of these conditions, the intake must be oriented towards the flow defined by the aft fuselage. Then, the cowl geometry needs to be adjusted to reduce the local Mach number and to balance the nacelle afterbody diffusion to increase the thrust contribution of the system (Section

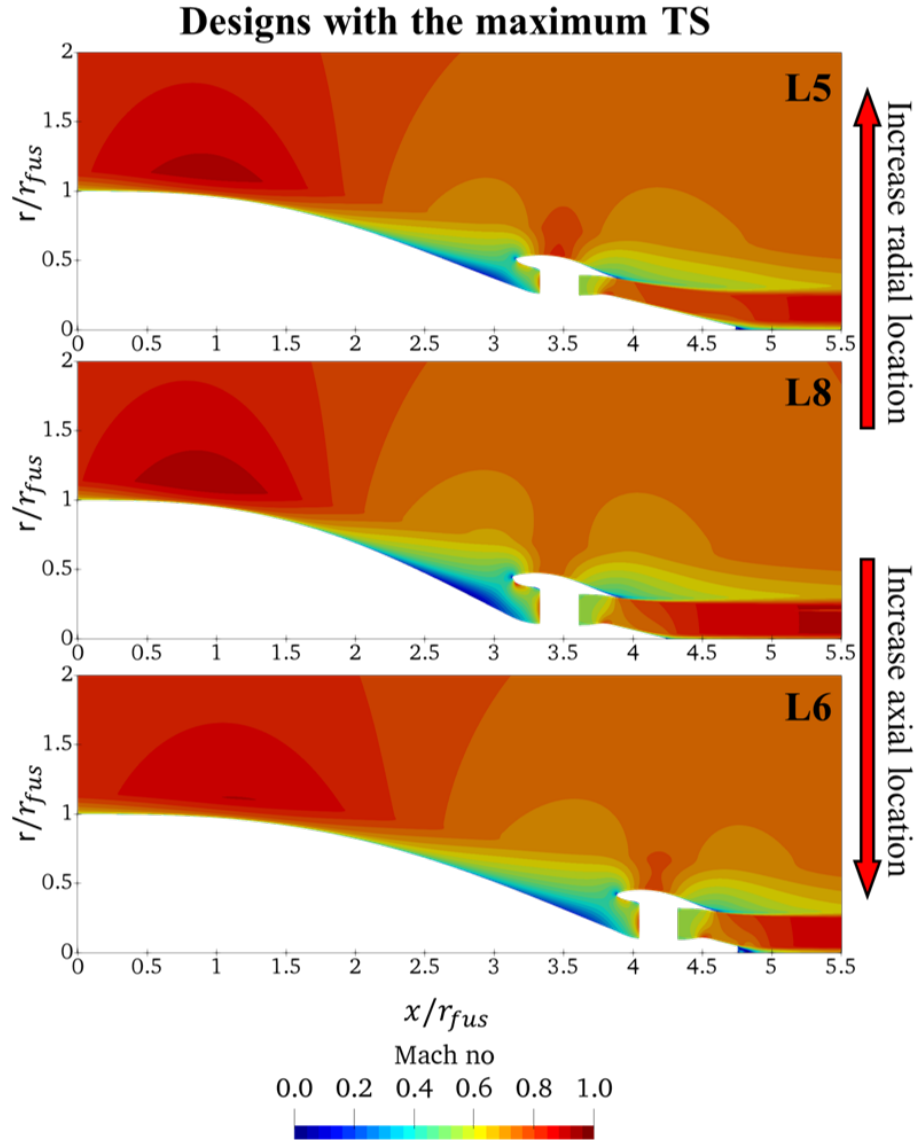


FIGURE 5.25: Effect of the change on radial and axial position: Mach number contours for the best TS non dominated designs of the locations L5, L8, and L6

5.2.2.2). Also, thinner and more compact intakes and cowls can be used to further increase the thrust contribution of the embedded propulsor for locations in which the aft fuselage geometry has enough space to transition from the cylindrical fuselage to the propulsor hub, for example L5 and L6 (Fig. 5.25). In the locations where the smooth geometry transition at the aft fuselage is not possible (e.g. L4 and L8), slightly longer intakes with higher curvature radii at the highlight are required to better adapt the flow conditions at the intake.

The reduction of the cowl length and the intake thickness can provide some advantages on the design operating conditions. However, for off design conditions the

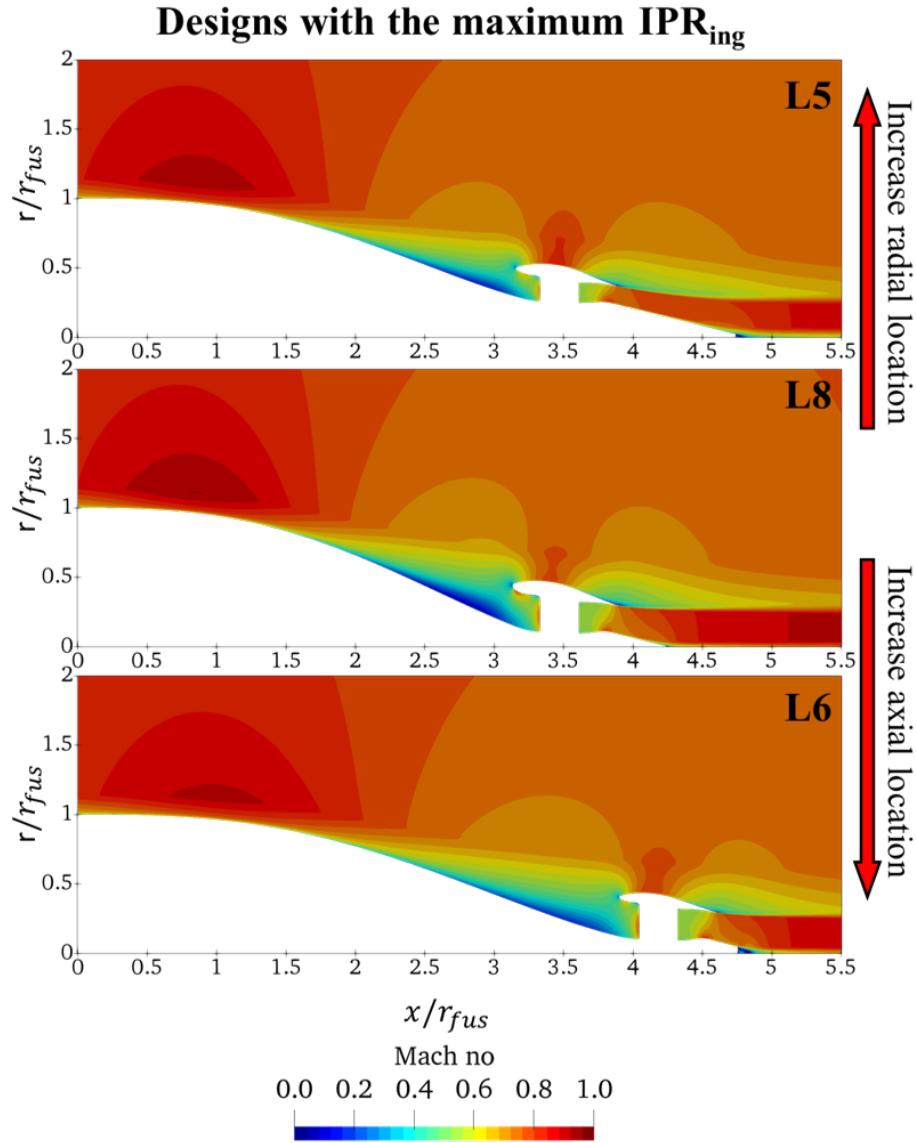


FIGURE 5.26: Effect of the change on radial and axial position: Mach number contours for the best IPR_{ing} non dominated designs of the locations L5, L8, and L6

lower radius of curvature could lead to penalties on the nacelle and intake performance. For example, it could be expected that the increase of the mass flow capture ratio ($MFCR$) leads to a local acceleration of the flow in the intake. This could eventually result in a shock wave that could lead to flow separation. On the other side, if the $MFCR$ was to be reduced the thinner outline of the highlight could generate an acceleration of the flow over the cowl and result in higher spillage drag or even flow separation on the nacelle afterbody. Thus, it is of interest to perform some level of robustness assessment to the off-design operation for the geometries optimised for cruise conditions. This will be evaluated in more detail in Section 5.4.

5.3.4 Effect of the location of the propulsion system on the design parameters

The optimal design space for each propulsor location of the DSE (Fig. 5.15) and the main aerodynamic requirements for a selected group of designs were evaluated in the previous sections. This has provided a better overview of the operation of the embedded propulsor and the aerodynamic requirements for the propulsion integration design. However, the design requirements for the bulk of the non-dominated designs for each MOO of the test have not been analysed yet. In this section, an analysis of the design characteristics for all the different propulsor locations is provided. This analysis is based on the parametric definition of the geometry of the propulsor.

The non-dominated designs for the individual MOOs at each propulsor position were observed to cover a wide range of the design space of the three objective functions of each optimisation (Figs. 5.17-5.18). The evaluation of the effect of the parameters on all the objective functions at each location simultaneously can lead to a convoluted analysis of the design space. Thus, it is of interest to compartmentalise the designs of interest for each objective function. For each objective function (TS , IPR_{ing} and RDI), the non-dominated designs of each MOO have been divided into four equally populated groups based on the objective function value in ascending order. To focus exclusively on the designs of interest, only the groups with the best performance for each objective function (i.e. maximum TS and IPR_{in} and minimum RDI) are showcased (Figs. 5.27-5.33).

To enable a clearer interpretation of the data, the parameters have been normalised to a frame relative to the propulsor location. The aft fuselage and intake angular parameters (θ_{fus} , θ_{ff} , β_{int} , Ω_{int}) are normalised with the propulsor angle, δ_{prop} (Eq. 3.9). This reflects the relative change on the angular magnitudes based on the relative angular offset between the fan face and the cylindrical fuselage. The boat-tail angle (β_{nac}) is normalised with the local intake inclination Ω_{int} to normalise the aftbody geometry based on the nacelle forebody orientation. The intake and aft fuselage areas (A_{thr}) and dimensions $r_{hi,o}$ are normalised based on the fan face area (A_{ff}) and tip radius (r_{fan}), respectively. The radial offset between the highlight and the maximum radius of the cowl (Δr_{max}) is normalised with the nacelle forebody length ($f_{max} \cdot l_{nac}$) to represent an aspect ratio. Meanwhile, the non-dimensional control parameters (CR , AR , AR_{int} , f_{il} and f_{max}) are maintained in their original reference frame.

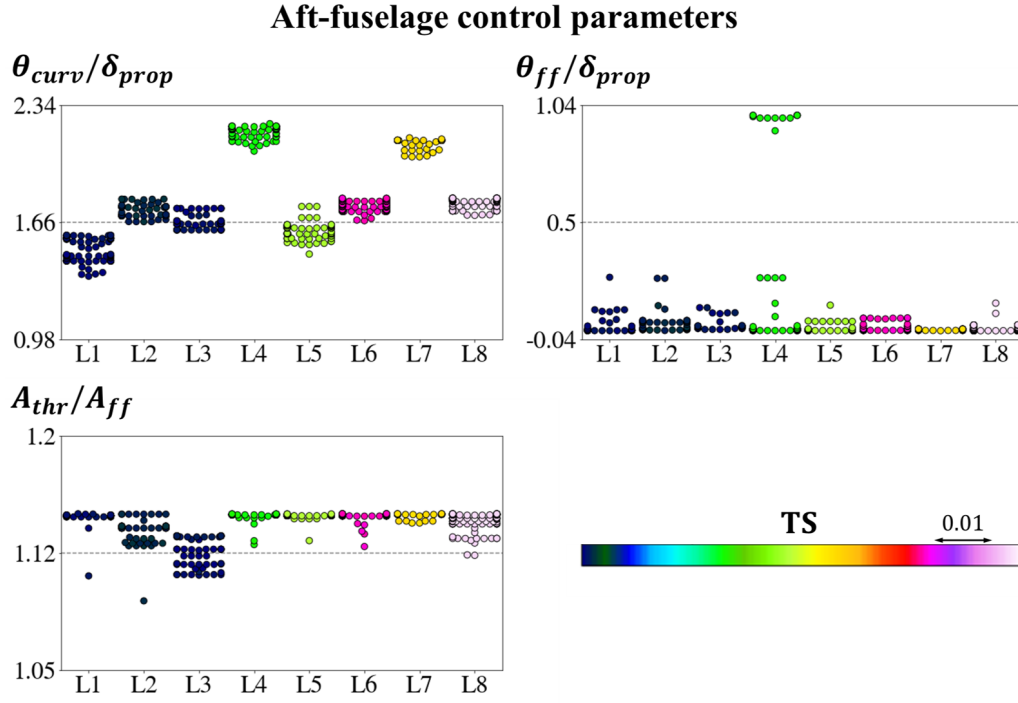


FIGURE 5.27: Aft fuselage control parameters of the non-dominated designs for the 25% highest values of TS for each of the locations of the DSE (Fig. 5.15)

When the designs with the highest TS are evaluated, a notable homogeneity is observed on the TS values for each location (Fig. 5.27-5.29). This means that the TS values are mainly controlled by the axial and radial location of the propulsor, and the detail design parameters of the propulsion integration have a second order effect. With that idea in mind, some information on how to increase the thrust contribution of the propulsion system can be still obtained. However, this will not have such a notable effect on the power savings as the bulk design parameters.

To increase the TS values at a given location, higher fuselage slope angles (θ_{curv}) are preferred (Fig. 5.27). A dependence on the location is observed for particular cases such as L4 and L7, where the relative aft fuselage slope ($\theta_{fus}/\delta_{prop}$) is higher than the other groups. These positions correspond to the more aggressive geometric transitions from the cylindrical fuselage to the fan face, where higher slopes on the fuselage are required due to space limitations. The TS has a significant sensitivity to changes on A_{thr} and θ_{ff} , since all the non-dominated designs of the studied locations are bound to specific regions of the design space, with the exception of location L4 where the space constraints may favour higher values (Fig. 5.27). The intake inclination, Ω_{int} , is another fundamental parameter to define the propulsion integration design space. In general, high relative intake inclination ($\Omega_{int}/\delta_{prop}$)

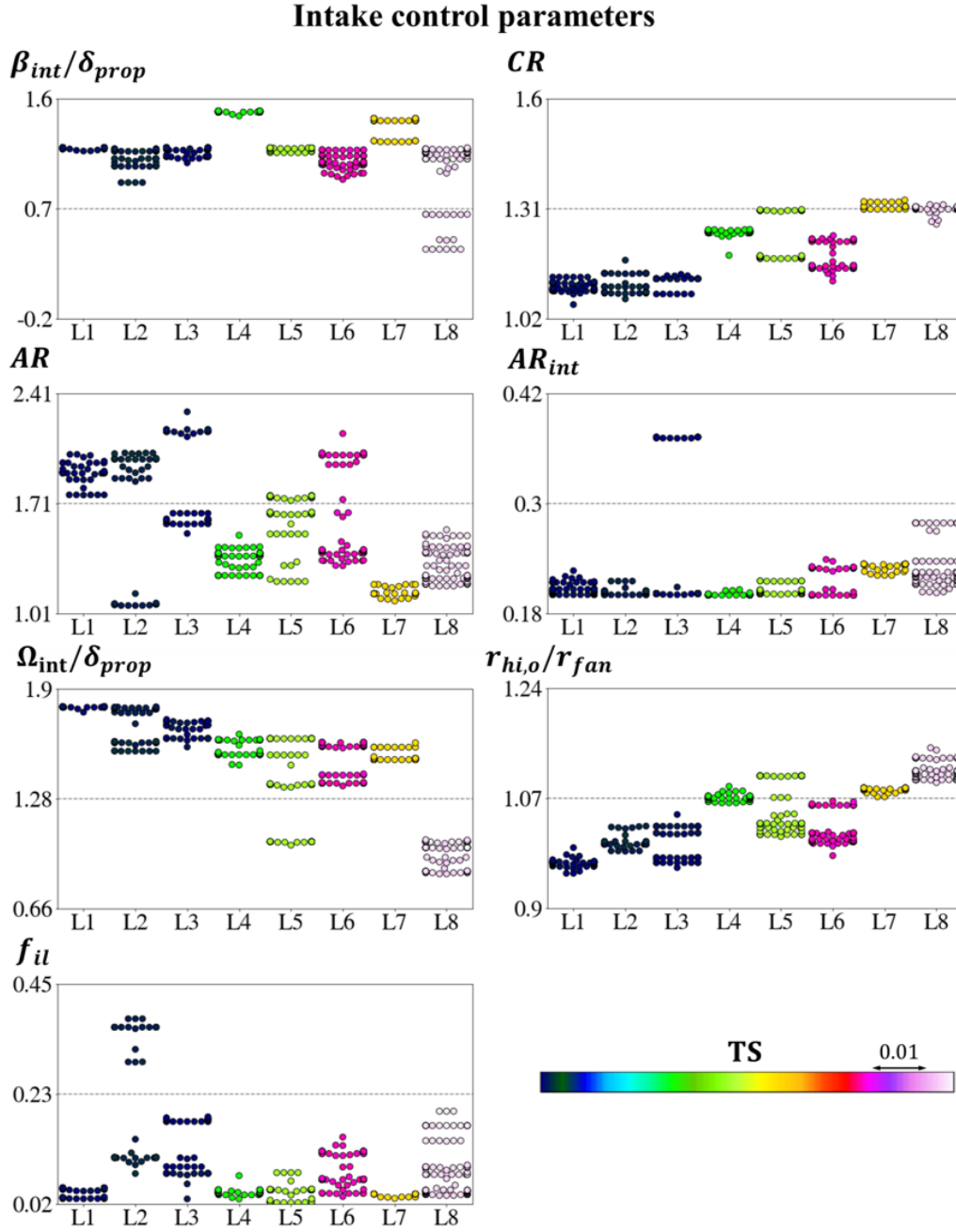


FIGURE 5.28: Intake control parameters of the non-dominated designs for the 25% highest values of TS for each of the locations of the DSE (Fig. 5.15)

values are preferred (Figs. 5.28), with the exception of location L8. For the non-dominated designs with the higher TS , the value of Ω_{int} on the different locations are grouped into clear bands or regions of the design space. These are different groups of intake inclination that define the design requirements for the other parameters. Thus, the intake and fan cowl design is conditioned by this parameter. At the same time, this parameter is influenced by the aft fuselage design and the axial and radial

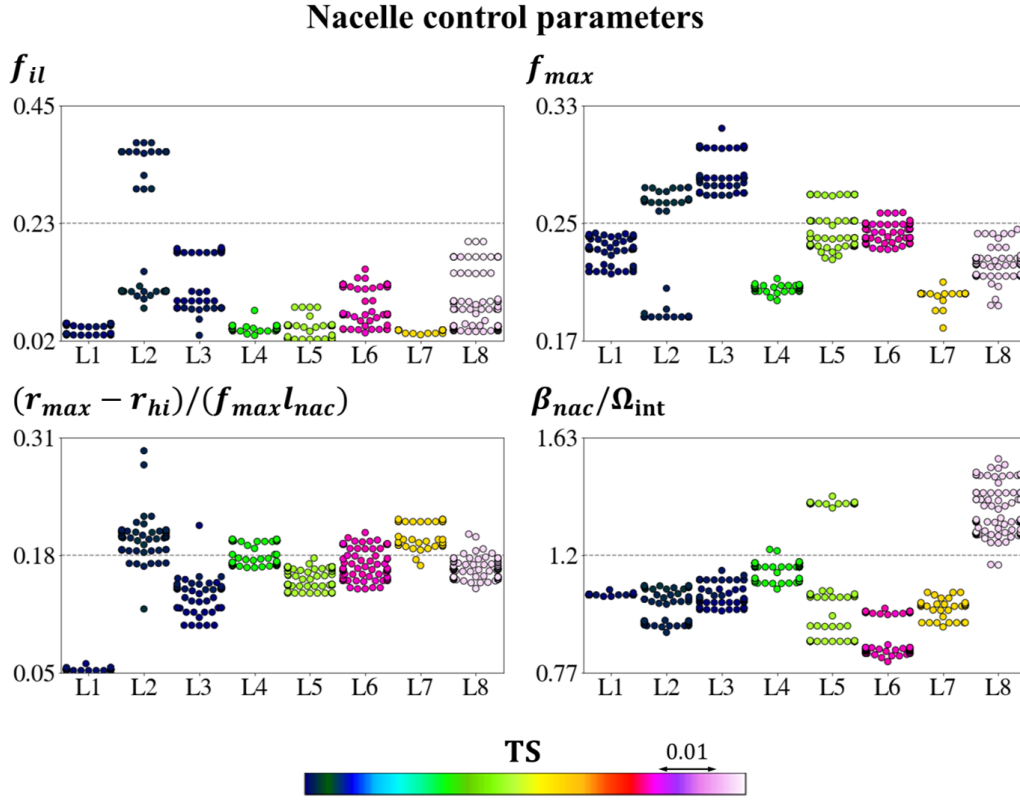


FIGURE 5.29: Aft fuselage control parameters of the non-dominated designs for the 25% highest values of TS for each of the locations of the DSE (Fig. 5.15)

position of the propulsion system.

In the designs with the highest TS , the fan cowl design also has some common characteristics across the different propulsor locations (Fig. 5.29). In general, forward axial locations of the maximum radius of the nacelle (f_{max}) are preferred. Also, the relative offset between the maximum radius of the highlight is placed on the same range for all locations, but location L1. This means that the forebody is required to be within some specific values of fineness ratio (i.e. length over height) to reduce the drag contribution with the reduction of wetted area and the cowl Mach number. On the location L1, even lower Δr_{max} are required. This could be due to the increase on the radial position of the propulsor. Under these conditions, even lower maximum nacelle radius are convenient to minimise the force penalties on the cowl. The boat-tail angle (β_{nac}) value is then set by the space requirements defined by the length of the nacelle afterbody. High boat-tail angles are preferred to increase the thrust contribution of the nacelle afterbody and to modify the exhaust operation to increase the thrust contribution of the system.

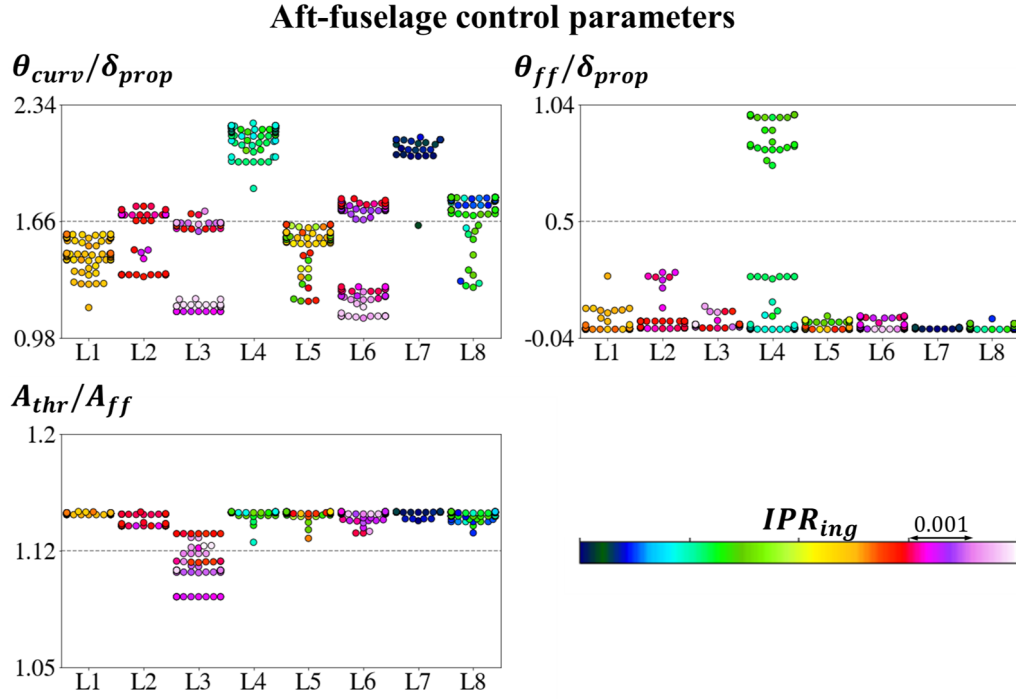


FIGURE 5.30: Aft fuselage control parameters of the non-dominated designs for the 25% highest values of IPR_{ing} for each of the locations of the DSE (Fig. 5.15)

The IPR_{ing} and the RDI have a higher sensitivity to changes on the design parameters than the TS , with higher variations of the metrics value with the parameters at a given propulsor location. Nevertheless, a higher influence of the location on the intake performance is still observed (Figs. 5.30-5.33). In contrast to the non-dominated designs with the higher TS , in the non-dominated designs with the higher IPR_{ing} (Fig. 5.30) and minimum RDI (Fig. 5.32), θ_{curv} has a higher variation for the different locations. At the same time, within the designs at a given location, there are some improvements on the IPR_{ing} with the reduction of the relative fuselage slope. This is clearer for designs with higher l_{aft}/l_{ps} such as L3 and L6, but can be observed for some designs at intermediate axial locations such as L5 and L6. Similarly to the designs with higher TS , lower θ_{ff} values are preferred with the exception of location L4. For the non-dominated designs with the minimum RDI (Fig. 5.32), a higher variety of θ_{curv} is observed for the locations with intermediate and low r/r_{fus} values such as L3, L5, L6 or L8. The θ_{ff} is restricted to lower values than the groups with the highest TS and IPR_{ing} .

Additionally, higher values of $\Omega_{int}/\delta_{prop}$ could improve the IPR_{ing} performance in some specific locations such as L3, L5, L6 and L8. The increase of Ω_{int} aligns the intake with the incident flow. This can reduce the possible total pressure loss

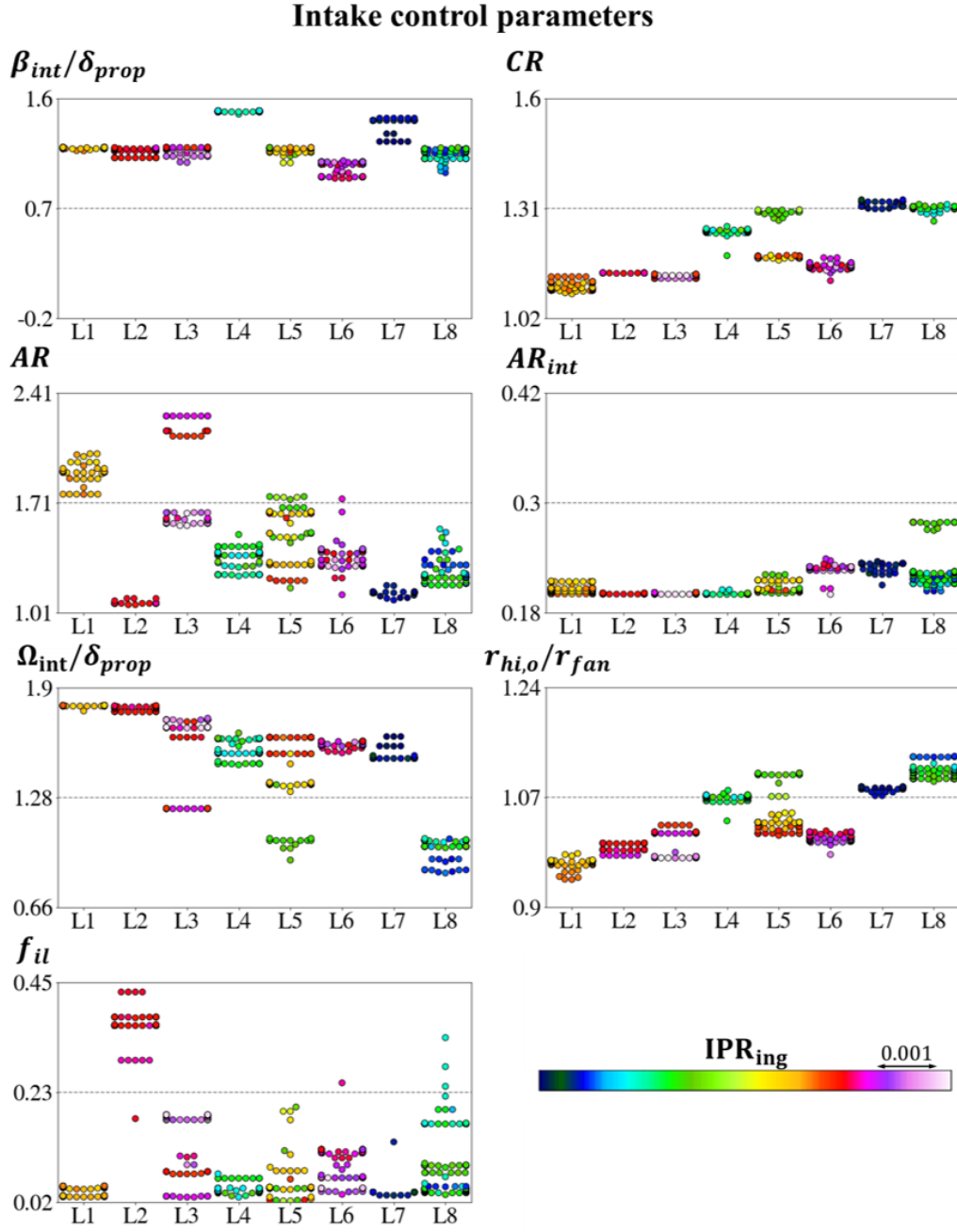


FIGURE 5.31: Intake control parameters of the non-dominated designs for the 25% highest values of IPR_{ing} for each of the locations of the DSE (Fig. 5.15)

derived from the displacement of the stagnation point. However, the designs with the lowest RDI favour lower intake inclinations and some locations as L5, L6, L7 and L8 (Fig. 5.33). These locations also have lower intake aspect ratios (AR_{int}) and higher highlight base radial position ($r_{hi,o}$). This could mean that the intake highlight still requires a higher radial position, but there is a lower need to adjust the highlight curvature with the Ω_{int} . In addition, the lower Ω_{int} in these locations

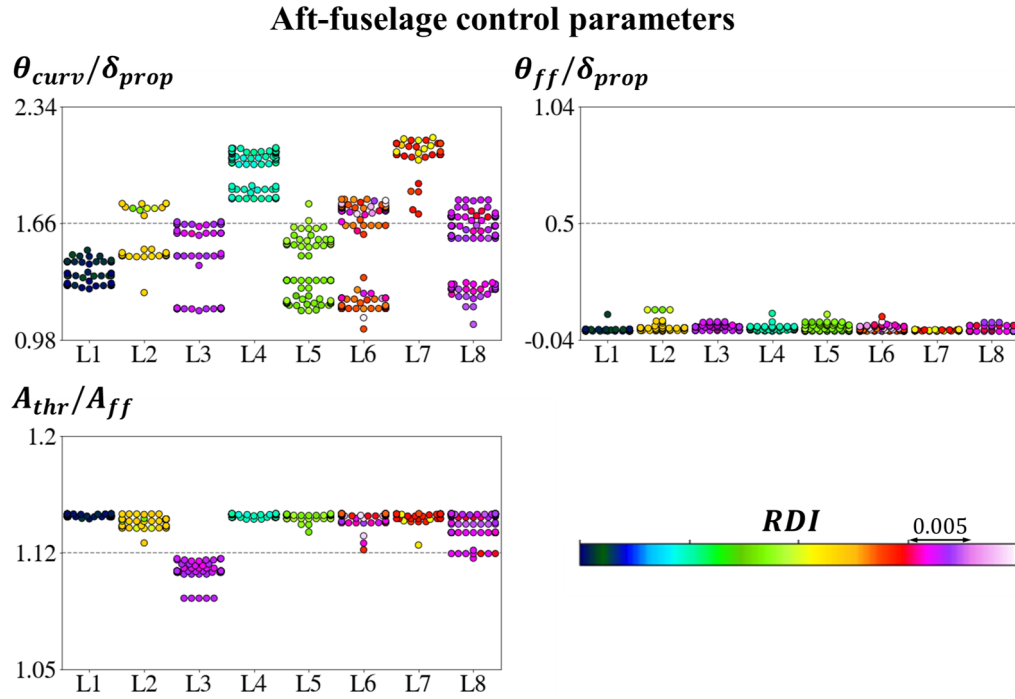


FIGURE 5.32: Aft fuselage control parameters of the non-dominated designs for the 25% lowest values of RDI for each of the locations of the DSE (Fig. 5.15)

to obtain designs with lower RDI appears to require higher intake contraction ratios (CR) and highlight radius of curvature (i.e. higher f_{il}) than the designs with higher TS and RDI . This means that thicker intakes are preferred to control the radial distortion, while thinner intakes can reduce the total pressure losses and increase the propulsive contribution of the system.

5.4 Robustness assessment

In the previous sections of this chapter, a design methodology for the aerodynamic design of the propulsion integration of the aft mounted BLI propulsor was used. In this section, the robustness of this design methodology is evaluated. In the first part of this section, the sensitivity of the intake performance to changes in the operation conditions is evaluated for some downselected non-dominated designs. In the second part, the effect in the aerodynamic performance of the flow conditions at the fan stage outlet is studied to understand the possible effects of the intake distortion transfer through the fan stage.

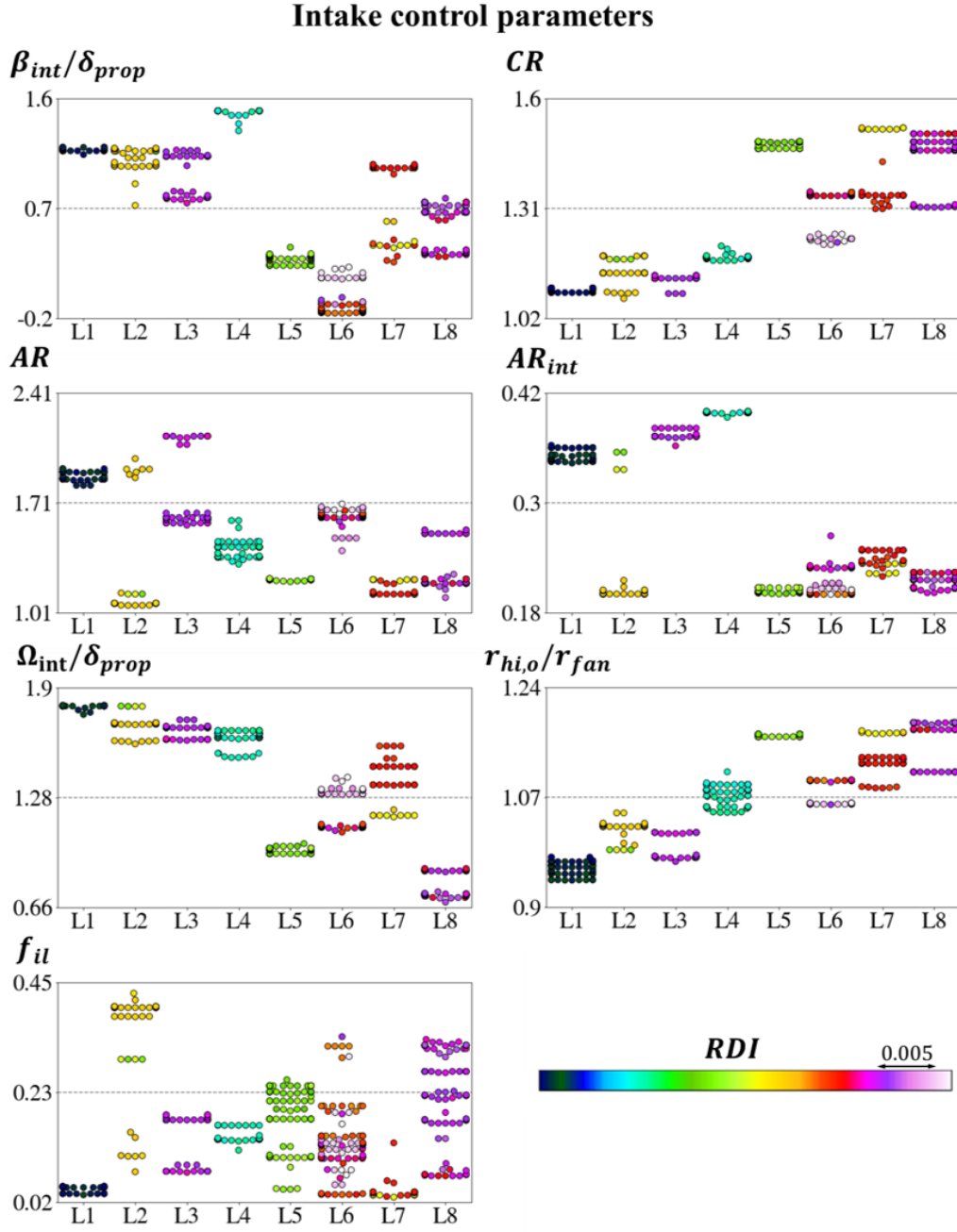


FIGURE 5.33: Intake control parameters of the non-dominated designs for the 25% lowest values of RDI for each of the locations of the DSE (Fig. 5.15)

5.4.1 Intake performance sensitivity to off-design conditions

As observed in the previous sections, the multi-objective optimisation strategy focused on cruise conditions sometimes leads to non-dominated designs with short intakes ($AR_{int} \sim 0.2$) and low curvature radii in the highlight ($f_{il} \sim 0.025$). These design characteristics improved the aerodynamic performance at the design point.

However, these particular geometric characteristics could have a negative impact on the aerodynamic performance for off-design conditions. For example, a lower curvature radius at the nacelle highlight could result in losses on the total pressure for higher BLR values due to the displacement of the stagnation points towards the fan cowl (Fig. 5.34). Under these conditions the flow moves towards the intake through a higher curvature gradient that could result in an over-acceleration of the flow at the intake, and in the worst scenario a shock wave emerging that could lead to flow separation. Similarly, if the BLR was reduced, the stagnation point would move towards the intake, increasing the spillage drag over the fan cowl (Fig. 5.34).

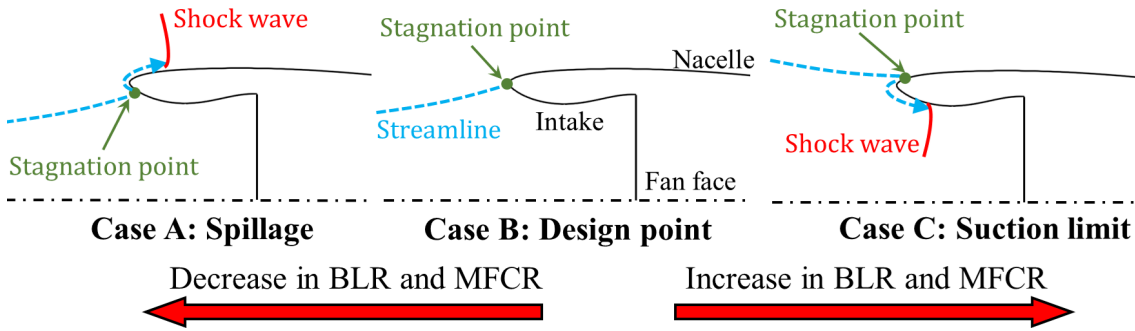


FIGURE 5.34: Schematic of the effects of the variation of the BLR in the intake and nacelle operation

To study the robustness of the non-dominated designs of the selected locations, an exploration of the Mach number and BLR values was performed at constant altitude. Four representative non-dominated designs are selected. These are the designs with the maximum TS and IPR_{ing} of the locations L6 and L8. The freestream Mach number was evaluated in six equally spaced points between $M = 0.65$ and $M = 0.9$ for each BLR value. The BLR was evaluated in eight equally spaced points between $BLR = 0.36$ and $BLR = 0.46$ for each Mach number value. The boundary layer thickness at the fuselage reference plane was evaluated for each Mach number to define the mass flow requirements for the target BLR . While the variation of the BLR in the proposed study appears to be narrow, the change in the distribution of mass flow rate and momentum flux (Appendix B.5) across the boundary layer means that small variations of the BLR can result in notable changes on the mass flow rate and momentum ingested by the BLI propulsor (Fig. 5.35). The intake performance metrics (IPR_{ing} and RDI) are studied to determine the robustness of the intake and aft-fuselage design.

For the selected non-dominated designs, IPR_{ing} is relatively insensitive to changes in BLR for a fixed freestream Mach number (Fig. 5.36). The changes on the

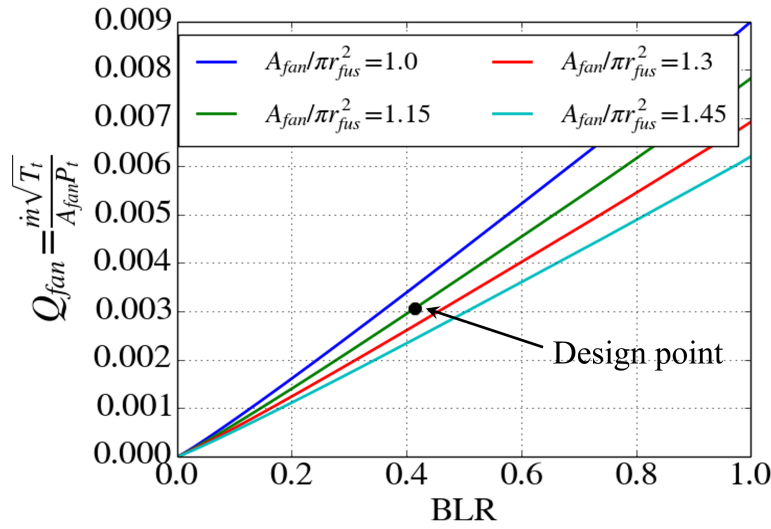
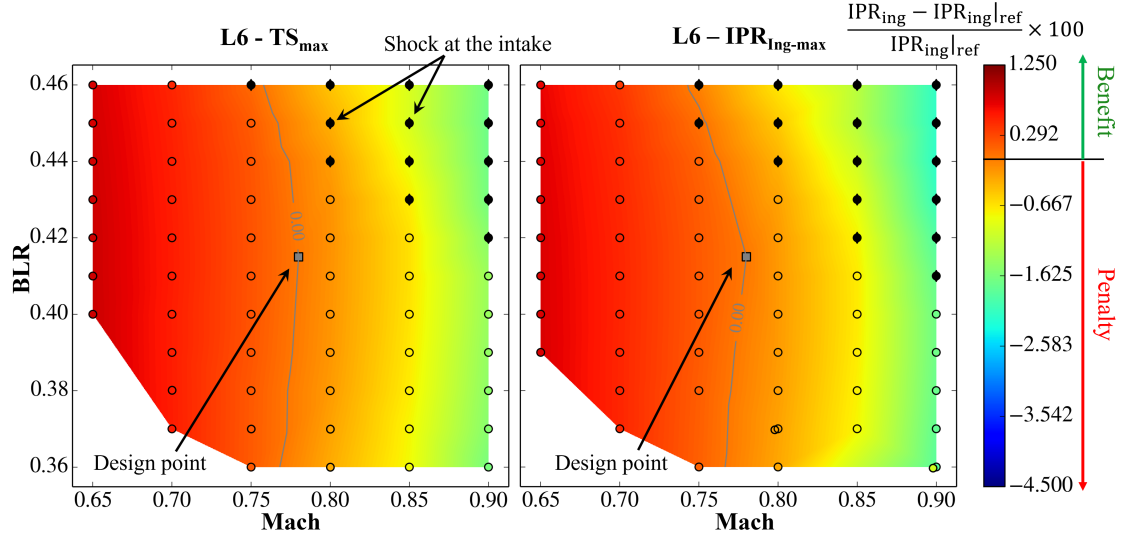


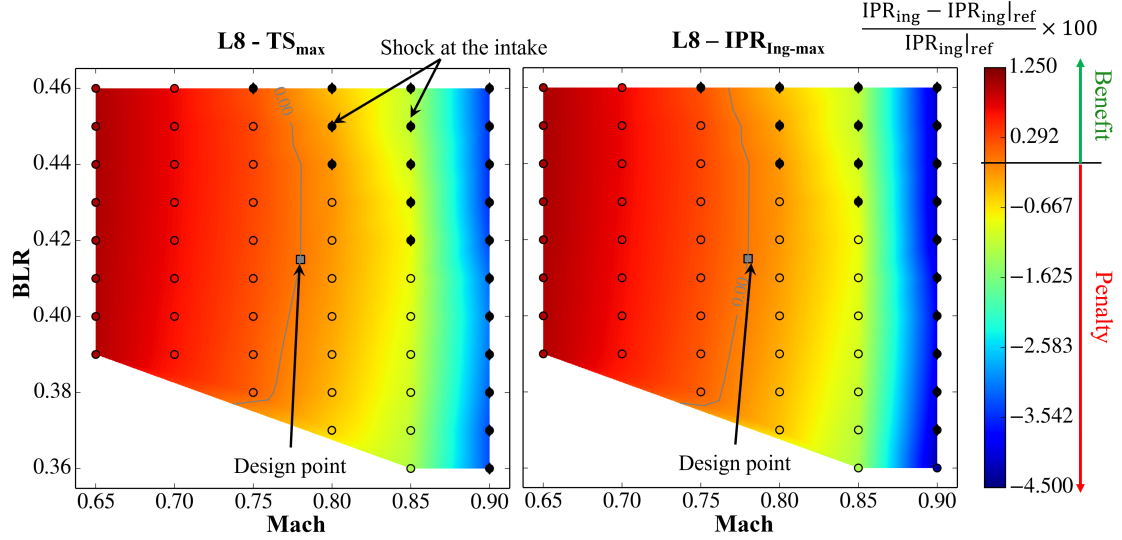
FIGURE 5.35: Effect of the BLR on the fan face capacity for the reference axisymmetric fuselage in mid-cruise conditions

performance become significant when the Mach number increases from the mid-cruise value ($M=0.78$). Small variations on the pressure recovery ($\sim 0.1 - 0.5\%$) at a fixed freestream Mach number are observed with the variation of the BLR from the design point ($BLR = 0.415$). This indicates that for a given flight Mach number the displacement of the stagnation point by changes in the operation mass flow does not have a major effect on the total pressure loss. Thus, the design allows some flexibility in the amount of boundary layer that can be ingested for a given flight condition. This allows the system to modify the throttle requirements for the different stages of cruise operation. However, the margin to increase the BLR at flight Mach numbers higher than 0.75 is limited by the over-acceleration of the flow at the intake (Fig. 5.36). This is ultimately limited by the generation of shocks in the intake duct. The increase on the mass flow rate moves the location of the stagnation point towards the fan-cowl. The free-shock operation range increase for the aft-most location (L6). Nevertheless, the operation of the intake remains relatively robust with increases in the Mach number within normal operating ranges of an aircraft in cruise ($\Delta M \pm 0.025$).

The sensitivity of IPR_{ing} to BLR (Fig. 5.36b) is smaller for the aft most location L6 ($l_{aft}/l_{ps} = 0.85$) compared with the location L8 ($l_{aft}/l_{ps} = 0.7$). This decrease on the IPR_{ing} sensitivity could be caused by the lower propulsor inclination δ_{prop} and the lower highlight curvature radius (Figs. 5.25-5.26) of the aft most location. The lower δ_{prop} increases the space for the aft fuselage curve to accommodate the



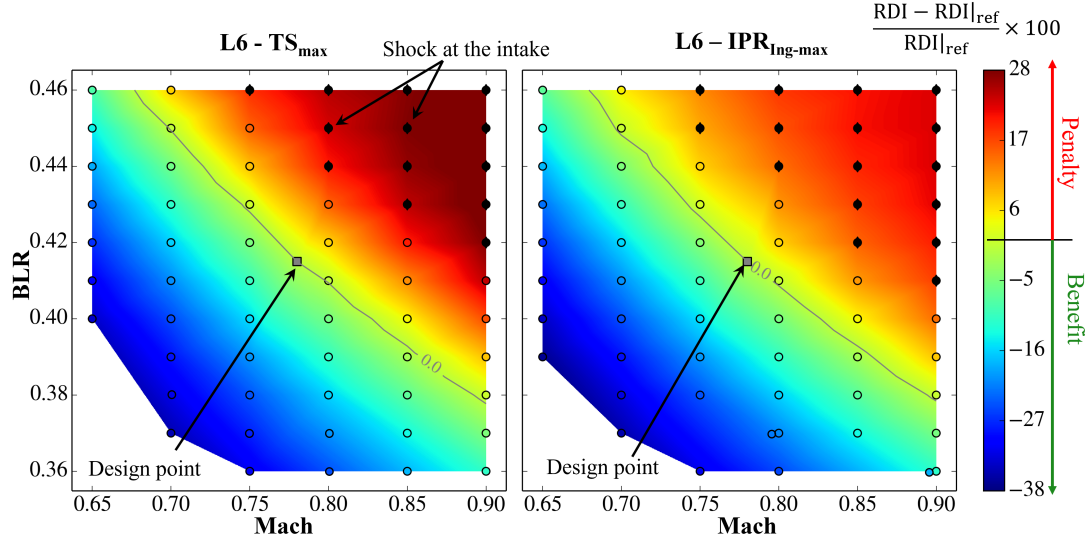
(A) Sensitivity of the IPR_{ing} to changes on the operation conditions for the maximum TS and IPR_{ing} non-dominated designs of the location L6



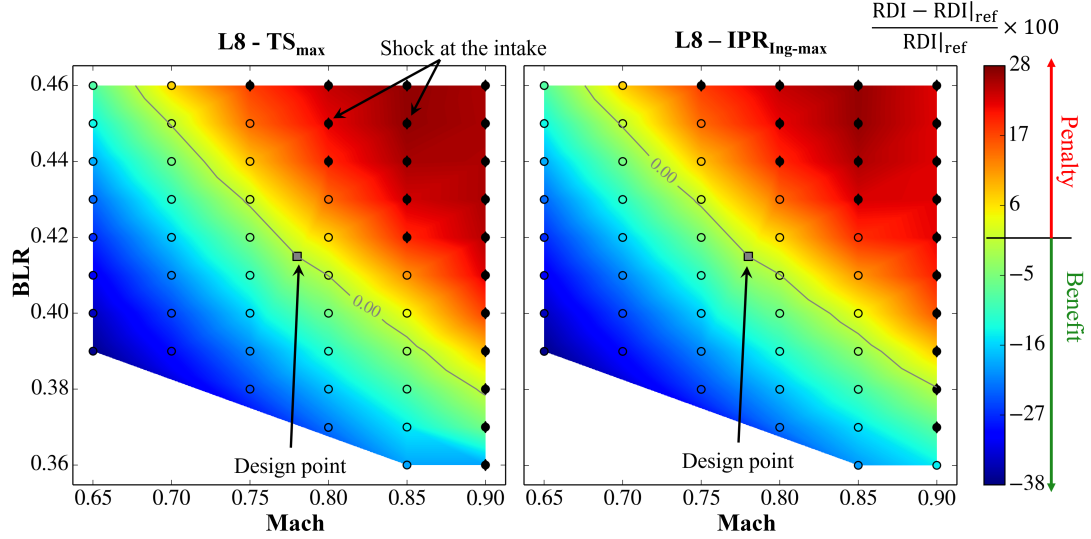
(B) Sensitivity of the IPR_{ing} to changes on the operation conditions for the maximum TS and IPR_{ing} non-dominated designs of the location L8

FIGURE 5.36: Maps of sensitivity of the IPR_{ing} to the operation conditions for the maximum TS and IPR_{ing} designs of the locations L6 and L8

fuselage flow to the fan face and reduces the viscous losses. The increase in the intake thickness (CR) and highlight curvature radius (f_{il}) for the location L8 requires of a higher acceleration of the flow into the intake and results in higher viscous losses in the intake duct. However, potential benefits could arise from the increase of the highlight curvature with the increase on the nacelle separation margin in off-design conditions⁹⁴. Future work focused on off-design conditions will be required to understand this effect.



(A) Sensitivity of the RDI to changes on the operation conditions for the maximum TS and IPR_{ing} non-dominated designs of the location L6



(B) Sensitivity of the RDI to changes on the operation conditions for the maximum TS and IPR_{ing} non-dominated designs of the location L8

FIGURE 5.37: Maps of sensitivity of the RDI to the operation conditions for the maximum TS and IPR_{ing} designs of the locations L6 and L8

For the selected non-dominated designs, the sensitivity of IPR_{ing} to the changes in Mach number at fixed BLR is higher than the sensitivity to BLR changes at fixed Mach. Penalties in the IPR_{ing} of between 0.5% and 1% are observed with an increase on the flight Mach number from 0.75 to 0.8. At the same time, potential benefits on the pressure recovery are observed with the reduction of the freestream Mach number. The changes on the IPR_{ing} with the Mach number are driven by the change of momentum and mass flow rate on the boundary layer. These changes

on the fuselage boundary layer for a given BLR modify the mass flow requirement with the Mach number. The modifications on the propulsor mass flow rate displace the stagnation point to the fan cowl side increasing the losses at the intake with the Mach number.

The combined effect of the BLR and the Mach number variation is most notable on the radial distortion (Fig. 5.37). The sensitivity of the RDI to the operating conditions is similar for the locations L6 and L8 with strong gradients on the performance. A slightly lower sensitivity to the operational changes is observed for the maximum IPR_{ing} non-dominated design of the location L6. Possible benefits on the radial distortion could be obtained by the reduction of the Mach number and the BLR . The strong sensitivity to the operation conditions on the radial distortion limits the operation on off-design conditions of the non-dominated designs. The increase on the throttle conditions and cruise Mach number could lead to a significant rise on the radial distortion associated with losses on the system performance^{42,43}. The variation on the radial distortion with the operation point is driven by the variation of the mass flow rate at the intake (Fig. 5.37) with lower BLR and freestream Mach numbers. The lower Mach number increases the boundary layer thickness and reduces the freestream velocity. So, for lower Mach numbers higher BLR values represent lower mass flow rates at the intake.

Overall, the non-dominated designs can maintain some robustness with the operation near the design point. This provides some flexibility during the operation at different stages of cruise. The RDI has a higher sensitivity to the changes in operating conditions than the IPR_{ing} . Under these conditions, the robustness of the aft-mounted propulsor design will be determined by the fan design and its tolerance to distortion. However, it is worth to mention that in the case that a "more robust" intake design was introduced based on off-design criteria (e.g. maximum incidence at take-off or cross-wind conditions), the impact on the TS and power savings is expected to be small in comparison with the effect of the bulk design parameters (i.e. main dimensions and propulsor location). Nevertheless, these studies have been carried with an axisymmetric design with no angle of attack. To properly understand the robustness of the intake and aft-fuselage conditions, a three-dimensional analysis of the effects of the flow incidence will be required. This could not be addressed in this work and it is recommended for future activities.

5.4.2 Sensitivity to fan outlet conditions

The design and optimisation of the propulsion integration design was carried assuming uniform flow conditions at the fan stage outlet. This is a simplification to reduce the dependency on the fan modelling for the semi-coupled one-dimensional fan model. However, this neglects the impact of the fan work distributions on the design requirements. In a more representative modelling of the aft-mounted propulsor power plant, the changes on radial distortion at the intake would propagate to the exhaust based on the added work of the fan stage. Also, the mass flow distribution across the fan and the isentropic efficiency of the turbomachinery will be modified according to the intake conditions. For these reasons, to reduce the uncertainty, it was considered appropriate to neglect the role of the exhaust design on the multi-objective optimisation process.

Despite all that, is it still of interest to understand how the aerodynamic performance of the propulsion integration design would be affected under different fan work distributions. Thus, a simplified test is proposed based in the modification of the total pressure and total temperature radial distributions at the exhaust inlet for one downselected design. This design is the configuration with maximum TS of location L6. To define the radial variation of the magnitudes of interest (P_t and T_t), parabolic profiles are applied (Chapter 3.4.2.1). The profiles are controlled by two parameters. First, the rate between the maximum value and the area weighted average of the magnitude ($P_{t,max}/\overline{P_t}$ for the total pressure). Second, the relative radial position of the maximum total pressure value (Eq. 3.74), h , normalised with the vane span.

The test consist on the variation of h in 7 progressive intervals from 0.2 to 0.8, and the variation of the total pressure to average ratio was set between 1.01 and 1.08 in 8 equally spaced intervals. The location of the maximum total temperature was set equal to the same on total pressure. Based on the work of Goulos et al⁹⁹, a constant ratio $\frac{P_{t,max}/\overline{P_t}}{T_{t,max}/T_t} = 0.995$ was applied. The FPR of the system was allowed to change to match the mass flow at the exhaust with the mass flow at the inlet (Chapter 3.4.2.1). The polytropic efficiency was set to a constant value of 0.91. The TS , the $TSPC$ and the exhaust performance metrics (C_V and C_D) are evaluated compared with the reference values with uniform conditions at the exhaust (Figs. 5.38-5.39).

The application of more representative exhaust inlet profiles has a significant effect on the aerodynamic performance (Fig. 5.38). The thrust contribution of the aft-mounted propulsor is observed to increase with h and $P_{t,max}/\overline{P_t}$, with a maximum

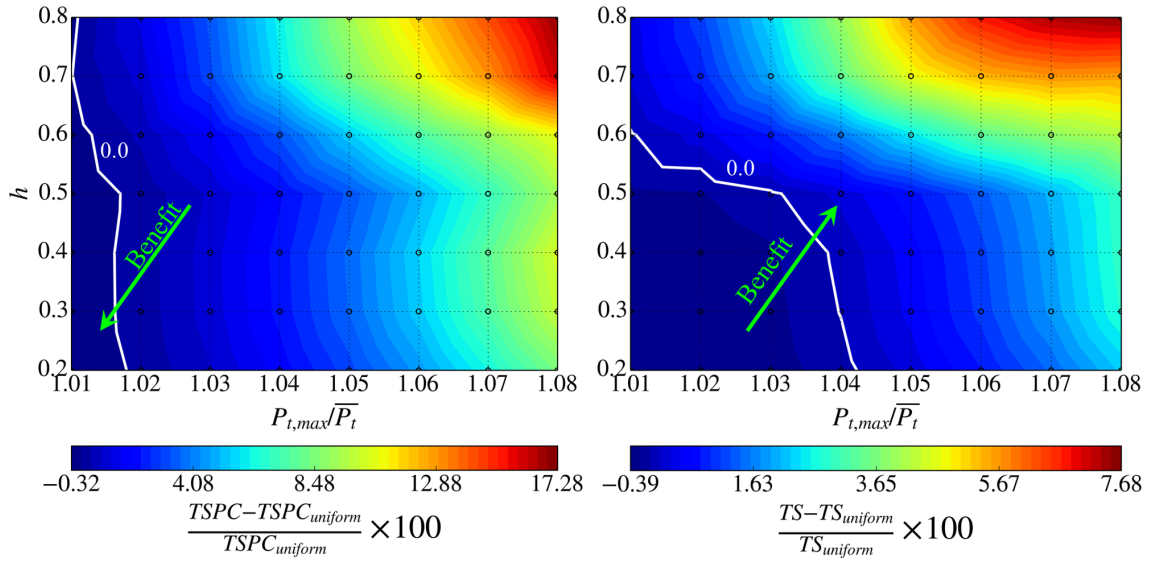


FIGURE 5.38: Effect of the exhaust inlet conditions on the aerodynamic performance for the design with the maximum TS of the location L6

increase of 7.68% in TS based on the case with uniform boundary conditions. At the same time, a reduction in TS from the uniform conditions reference is observed with values of $P_{t,max}/\bar{P}_t \leq 1.04$ and mid and hub-loaded profiles (i.e. $h \leq 0.5$).

The increase on TS comes at the expense on the propulsive efficiency with an increase on the $TSPC$ for all the cases with $P_{t,max}/\bar{P}_t \geq 1.02$. This means that the system requires to increase the FPR to match the mass flow requirements at the exhaust at higher $P_{t,max}/\bar{P}_t$ and h . Since the polytropic efficiency of the turbomachinery stage is fixed, the increase on FPR results in the rise on total power required by the propulsive system. The $TSPC$ reflects the propulsive efficiency of the system.

The increase on TS with h and $P_{t,max}/\bar{P}_t$ is driven by the rise of the GPF . This rise is due the increase on FPR and the reduction on viscous losses in the exhaust walls⁹⁹. The reduction of the viscous losses in the exhaust walls compared with the reference with uniform exhaust conditions is a result of the changes in the radial mass flow distributions (Fig. 5.40). The variable distributions of P_t and T_t at the exhaust result on regions with lower mass flow and velocities near the walls. This is more significant for tip loaded configurations (i.e. $h \geq 0.6$), since the lower walls of the exhaust and the cone have a higher wetted surface than the upper walls of the exhaust.

When the exhaust efficiency metrics are evaluated (Fig. 5.39), it is observed how the changes in total pressure and total temperature modify the GPF and the discharge

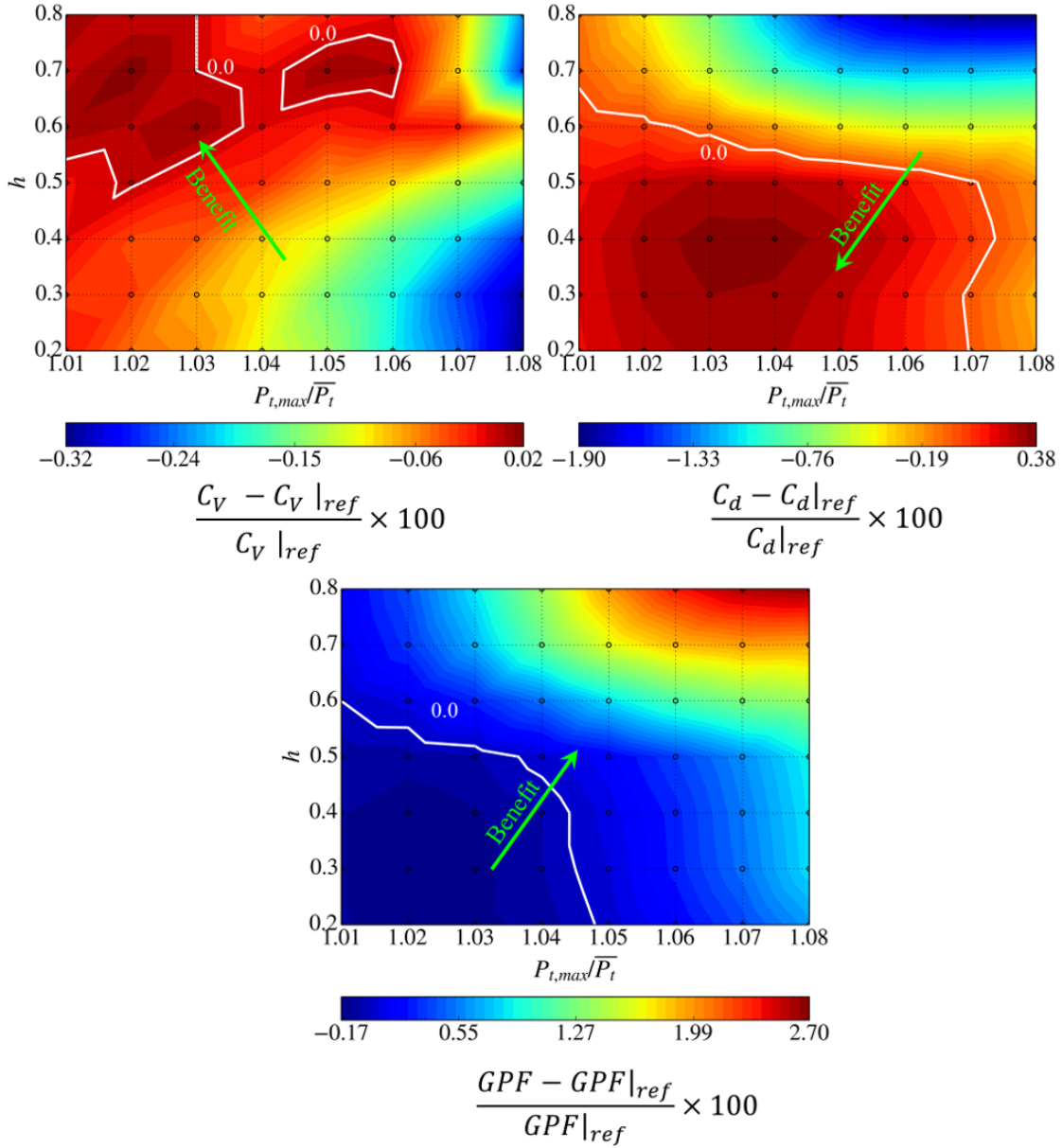


FIGURE 5.39: Effect of the fan outlet conditions for the design with the maximum TS of the location L6. Variation from the reference uniform conditions at the fan stage outlet for the C_V , C_D , and GPF

and velocity coefficients (C_d and C_V). The C_V increases for tip loaded configuration ($h \geq 0.6$) at and with the reduction of $P_{t,max}/\bar{P}_t$. In this region, there are some gains on GPF , but the increase in C_V is limited by the change on the FPR for the profiles. The increase in FPR leads to an increase on the average total pressure and temperature in the exhaust. This affects the ideal propulsive force (Eq. 3.45) by changing the ideal discharge velocity for a given mass flow rate. At the same time, the exhaust suppression (i.e. the lost in discharge coefficient) increases with

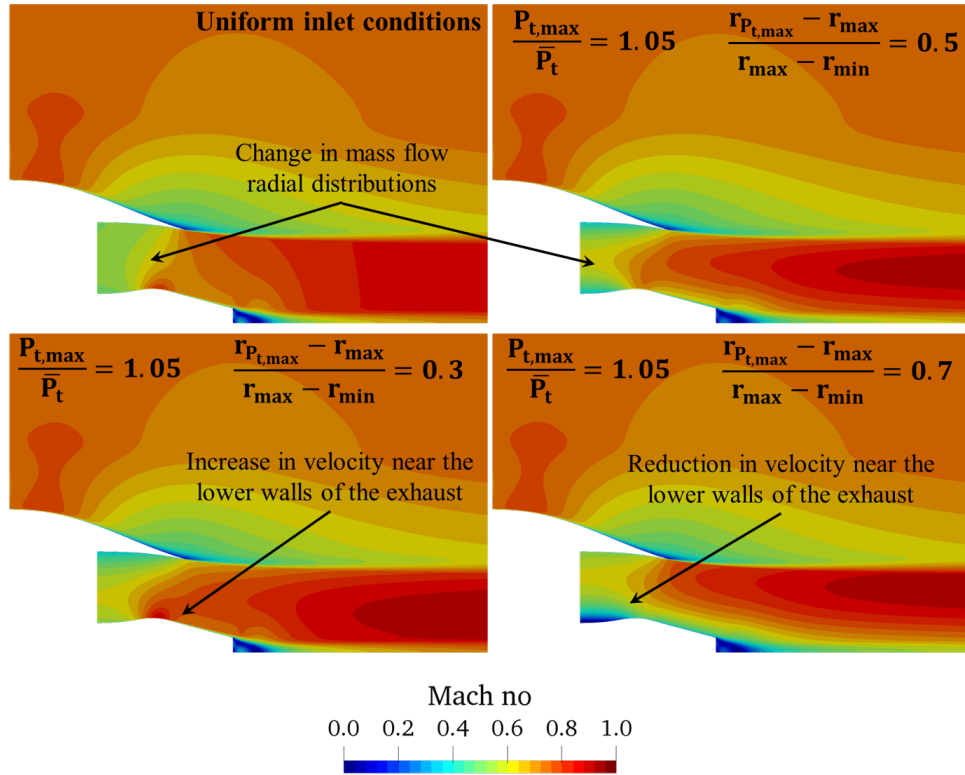


FIGURE 5.40: Effect of the fan outlet conditions for the design with the maximum TS of the location L6. Mach number contours at the exhaust for the baseline Uniform conditions case and three different radial distributions ($h = 0.3-0.5-0.7$) at a fixed load ($P_{t,max}/\bar{P}_t = 1.05$)

$P_{t,max}/\bar{P}_t$ and h . This is again a consequence of the change on the average total pressure and temperature at the exhaust duct. In this case, since the exhaust is not choked (fan nozzle pressure ratio lower than critical value), the discharge coefficient is more sensitive to these changes on exhaust operation. Thus, it is clear that for these embedded configurations with low fan nozzle pressure ratios, the exhaust inlet conditions can have a notable effect on the exhaust aerodynamic performance.

The increase in TS for some exhaust inlet conditions was observed to come at the expense of the propulsive efficiency of the system with an increase on the $TSPC$ (Fig. 5.38). As it was observed in Chapter 4, the interaction between both metrics will limit the power savings achievable by the integration of the aft-mounted BLI propulsor. How the exchange between TS and $TSPC$ affects the power savings is highly dependent on the power source selected for the embedded propulsor (Fig. 5.41). If the aft-mounted propulsor is powered by a source independent of the wing mounted power plants (Chapter 4.1.1), the power reduction of the vehicle is mainly determined by the TS , with a less significant influence of the $TSPC$. Under

these conditions, the tip loaded total pressure and total temperature distributions ($h \geq 0.6$) with higher $P_{t,max}/\bar{P}_t$ can increase the power saving coefficient (PSC) up to 0.26% compared with the baseline with uniform exhaust conditions (Fig. 5.41). For other h and $P_{t,max}/\bar{P}_t$ combinations, a reduction in the power savings is expected. If all the power of the aft mounted propulsor was obtained from the wing mounted power plants (Chapter 4.1.1), the impact of the $TSPC$ in the PSC increases. From the reference with uniform conditions at the exhaust inlet, no combination of h and $P_{t,max}/\bar{P}_t$ can increase the power savings of the system. Furthermore, the PSC is penalised by at least 0.2% in the best case scenario. Also, for $P_{t,max}/\bar{P}_t$ higher than 1.02, any power savings start to erode due to the increase in $TSPC$.

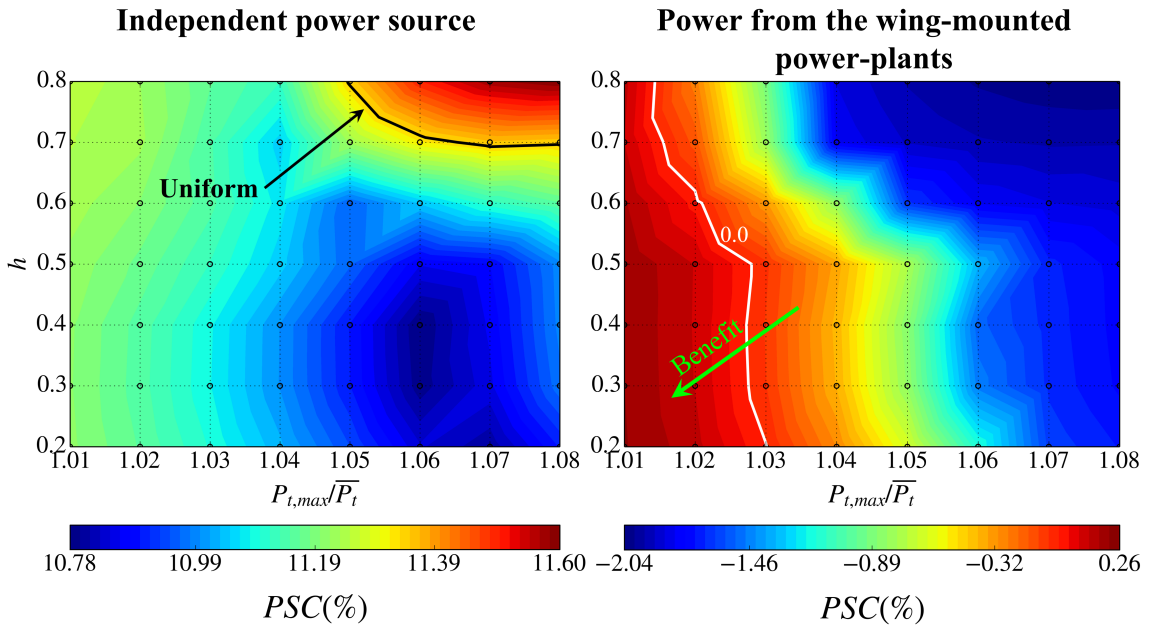


FIGURE 5.41: Effect of the fan outlet conditions for the design with the maximum TS of the location L6. Change on the power saving coefficient (PSC) for two different propulsor power sources

The two presented power sources for the aft mounted propulsor (Chapter 4.1.1) represent two extreme cases. For these embedded systems, the current concepts are focused on hybrid-electric configurations^{1,11,12}. For these configurations, the power savings are expected to lie between both of the proposed limits. Nevertheless, these cases can be used to highlight the impact of the exhaust design and operating conditions in the power savings of the system. An overestimation of the power savings is to be expected when a non-representative exhaust model is applied. For some limit cases, this could even lead to penalties on the system performance. To the author's knowledge, the design of the exhaust system for the embedded propulsion

systems has not been addressed in the open literature. For this reason, a more detailed study about the impact of the exhaust design on the performance will be carried in Chapter 7.

In addition, it has been observed that the exhaust inlet conditions define the operation of the exhaust and its design requirements. In a more representative model of the embedded propulsion system, these exhaust conditions will be determined by the transfer of the flow distortion through the fan. The transfer of this distortion is also expected to affect the operation of the fan and reduce the isentropic efficiency of the system, based on the current state of the art^{43,77}. This could be addressed with new generations of distortion tolerant fans⁷⁸. Nevertheless, independently on the selected fan configuration it is clear that the fan outlet conditions have a significant effect on the system performance. Thus, for a detailed analysis of the exhaust design requirements a more representative fan model is required. For this reason, in Chapter 6 the validation of a low order throughflow method is performed. This model will enable a more detailed exhaust analysis based on the fan operating conditions due to the flow distortion at the intake (Chapter 7).

5.5 Summary

Across this chapter, different studies with a 2D axisymmetric CFD approach have been performed. These models used a semi-coupled one-dimensional fan model based on a target fan efficiency and mass flow rate. As a case study an aft-mounted annular BLI propulsor is used. The CFD approach was compared with low order models based on thrust and drag correlations. These models were found to be unsuitable to predict the aerodynamic performance of the embedded propulsion system. Differences between 20-50% in the thrust split were obtained with the different low order models based on the CFD approach. This translates to an approximately 5-12% change in the power savings, while the expected benefit for this configuration is around 3-11% in total.

A multi-objective optimisation (MOO) methodology was used for the aerodynamic design of the aft-fuselage, intake and nacelle at a fixed radial and axial position of the propulsor. A benefit of approximately 0.003 in TS from a Datum manual design was obtained. This translates to 0.3-0.45% in power savings, based on the preliminary prediction for total power savings between 3-11%. The MOO approach was used to determine the impact of the axial and radial position of the propulsor. More forward axial placements with lower radial positions of the propulsor are preferred.

A maximum difference in TS of 0.04 units was observed between locations. This translates to approximately 4-6% changes on power savings. Under this context, it becomes clear that the axial and radial location of the propulsor, with the bulk dimensions of the propulsion integration, are the major contributors to the changes on the power savings due to the aerodynamic design of the propulsion integration. Thus, the detailed optimisation of the propulsion integration at a fixed axial and radial position has a second order impact ($TS \sim 0.003$) in comparison to the bulk geometry effects.

The MOO strategy was based only on cruise conditions. This produced some non-dominated designs that could result on penalties in the intake performance under off-design conditions. A robustness assessment of some down-selected design was carried out. The designs maintained acceptable robustness for limited variations on the operating Mach number and mass flow, with high penalties with extreme changes in the performance. For future work, it was determined that a study of the effects of flow incidence with three-dimensional configurations will be required to determine the performance of the propulsion integration design in off-design conditions.

A study of the exhaust inflow conditions in the aerodynamic performance was carried out. Notable changes on the power savings were observed with the change in the total pressure and total temperature distributions at the exhaust inlet. These changes will condition the aerodynamic design requirements for the exhaust system. Thus, a more representative fan model is required to characterise the exhaust operating conditions and determine the exhaust design requirements.

CHAPTER 6

Low order turbomachinery model for propulsion integration aerodynamics

The embedded propulsive systems of the novel aircraft configurations are expected to operate under strong levels of sector and radial flow distortion at the intake^{39,42,78}. This increase on the distortion at the intake has been reported to have a significant effect on the propulsor performance by multiple works as Gunn et al.⁴², Defoe et al.⁴³, and Hall et al.⁹¹. The fan also plays an important role on the distortion suppression and propagation. Cao et al.⁹² found that on intakes under high-incidence the fan can have a suppressive effect on the distortion and increase the separation free operation. Hence, a transfer model is required to define the effects of the turbomachinery on the aircraft aerodynamic performance to achieve a fully coupled aerodynamic analysis.

It is not always possible to apply a high fidelity model that includes the turbomachinery geometry under close-coupled flow aerodynamic configurations because of computational cost limitations. To avoid this problem, multiple low order models can be found in the literature to characterize the turbomachinery response in cases where some loss in the accuracy is permitted in favour of a reduction of the computational cost. However, not all of them are suitable to characterise the turbomachinery effects under strong flow distortion.

After the literature research, the Hall-Thollet body force model (BFM)¹⁷ was identified as a suitable candidate to reduce the computational cost of the flow simulation. This model was implemented in the numerical framework of ANSYS Fluent¹⁵⁵ (Fig. 3.30). The implementation of this model in the compressible viscous CFD approach of the present work required an additional validation process for a reference geometry. The configuration selected for this purpose was the NASA Rotor 4 (R4)

fan configuration^{23,24}. In this chapter, a summary of the validation process can be found. The Hall-Thollet model was validated against the stage performance defined by the experimental results²³. The fan outlet radial profiles for the total pressure, isentropic efficiency, and swirl angle were also compared with the experiments. Additionally, as a higher fidelity reference for the body force model, a mixing-plane CFD approach was also validated and compared with the experimental reference.

6.1 NASA Rotor 4

The implementation of the Hall-Thollet model on the ANSYS Fluent¹⁵⁵ framework is validated using the NASA Rotor 4 (R4) geometry²³. The R4 is a scaled single fan stage representative of a medium pressure ratio, high bypass ratio turbofan (Tab. 6.1). The measure stations are defined based on the experimental reference^{23,24} (Fig. 6.1). Four rotational speeds have been applied for the validation of the stage performance based on the maximum rotational speed value (50%,70%,87.5%, and 100%). Experimental values of the radial distributions of the pressure ratio, isentropic efficiency, and swirl angle were available for the flow between the fan and the OGV at 87.5% and 100% of the maximum fan rotational speed²⁴.

TABLE 6.1: NASA Rotor 4 design characteristics²³

Rotor 4 design characteristics	
N_{blades}	22
N_{vanes}	54
Fan tip diameter	0.65 m
httr	0.3
Corrected RPM (100%)	12,657
Stage PR	1.47
Corrected design mass flow rate	45.6 kg/s
Tip relative Mach number	1.26

6.2 Mixing Plane Model

As a complement to the BFM calculations and to provide a comparison with a higher fidelity approach; a steady, viscous compressible solution of a single blade and vane passage of the NASA R4 stage was carried using a Mixing Plane approach

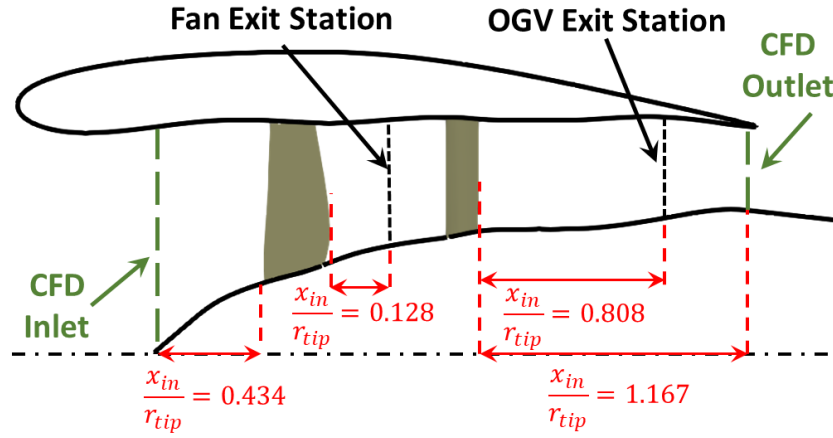


FIGURE 6.1: NASA Rotor 4 geometry²³ with the relative locations of the measurement rakes

(Fig. 6.2). The blade and the vane domains are generated independently, and their interaction is modelled with a Mixing Plane Model with an mixed-out averaging¹⁵⁵. The mesh was generated using TurboGrid¹⁸⁰ for a target y^+ of unity. The $k-\omega$ SST model is applied as the turbulence model. A tip gap of 0.15% of the fan blade radius is used. A total pressure inlet boundary condition was imposed at the domain inlet to match the reference wind tunnel²³ boundary conditions. Rotational periodic boundary conditions were applied for the blade and the vane side domains. Target inlet capacity for each fan rotation speed is modelled by changing the static pressure value at the vane outlet. Non-conducting, non-slip boundary conditions were applied for the domain wall surfaces. The OGV domain hub, shroud and vane, and the blade shroud walls were modelled as static elements. A relative rotation matching the fan rotation speed value is applied for the blade walls and the blade domain hub boundary. A detailed grid sensitivity study based on the methodology proposed by Celik et al¹⁵⁸ can be found in Appendix C.4 for the mixing plane CFD approach.

6.3 Reference body force model implementation

The BFM implementation was validated for 3D and 2D axisymmetric flows. For the 2D axisymmetric a meridional cut of the intake, stage and exhaust duct was applied. The off-plane contribution of the body forces were modelled by an axisymmetric azimuthal velocity component¹⁵⁵. For the 3D implementation an azimuthal section equivalent to the extend one blade passage (16.36°) was applied. For both approaches

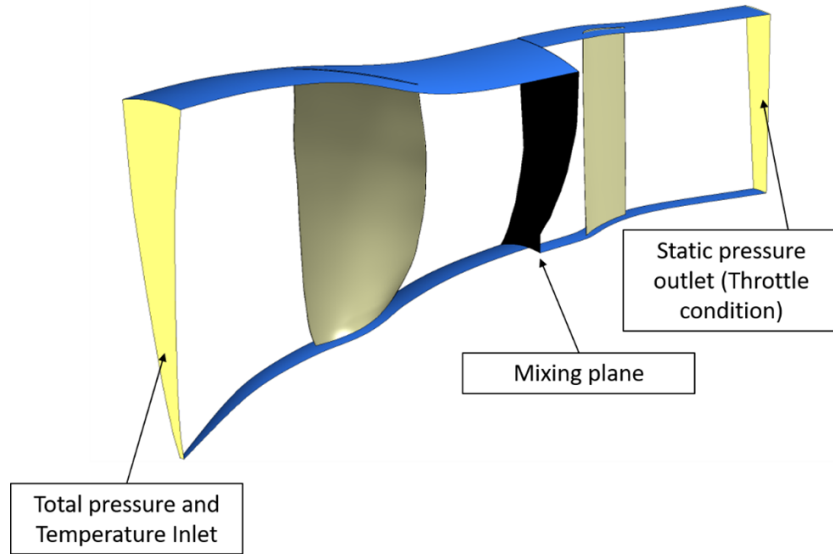


FIGURE 6.2: Sample view of the NASA Rotor 4²³ domain for the mixing plane CFD approach

the computational domain is extended from a point near the spinner tip to a plane near the exhaust throat (Fig. 6.1). For the present work, only clean inlet flows were available for the validation.

A steady viscous compressible flow CFD approach is used with the $k-\omega$ SST turbulence model. The hub and the shroud are defined as no-slip, non-conductive walls with a y^+ value near unity. The inlet boundary condition is set as a total pressure inlet matching the wind tunnel²³ conditions. A static pressure outlet is applied to model the outlet of the domain (Fig. 6.1). The target fan flow capacity the static pressure at the outlet was modified. Periodic rotational boundary conditions are used for the lateral faces of the domain on the sector 3D approach. To model the effect of the fan blade tip gap, the source terms are set to zero on that region. A detailed grid sensitivity study based on the methodology proposed by Celik et al¹⁵⁸ can be found in Appendix C.4 for the 2D and 3D body force model test cases. The selected mesh resolutions of the 3D and 2D axisymmetric models have approximately a 40% and a 0.11% of the computational cost of the mixing plane approach, respectively.

6.4 Rotor 4 validation

6.4.1 Stage performance

To validate the BFM implementation within the Fluent framework the fan stage pressure ratio, FPR , and the total-to-total isentropic efficiency, η_{is} , were obtained for four fan rotational speeds ($\Omega/\Omega_{max} = 50 - 70 - 87.5 - 100\%$) at different inlet capacity, Q , values. The 2D-axisymmetric and 3D sector BFM approaches are compared against the R4 experimental results²³, and the Mixing Plane Model. The results are sampled using an area-weighted-average at the designated measure locations – Fan Exit Rake and OGV Exit Rake (Fig. 6.1) – following the reference experimental results. As defined in Hughes et al.²³, total temperature values are taken at the Fan Exit Rake and no total temperature losses are assumed at the OGV. Total pressure values are sampled at the OGV Exit Rake. The mass flow is computed at the numerical domain inlet boundary and is normalized as the capacity, Q , with the reference inlet conditions and the fan face area. Each of the two BFM approaches has been calibrated for each of their four operation lines. The calibration approach is defined in the Appendix E.2.

An adequate agreement with the stage pressure ratio and isentropic efficiency results is observed for all the approaches. The discrepancy on the stage pressure ratio between the numerical approaches and the experiments increases when the operation point moves towards the surge limit (Fig. 6.3). The maximum discrepancy on the stage operation pressure ratio is observed for the $\Omega/\Omega_{max} = 100\%$ operation line, with an average difference with the experimental results of 1.65% for the 3D Sector BFM, a 1.2% for the Mixing Plane Model, and a 1.11% for the 2D-axisymmetric BFM. A larger discrepancy of the isentropic efficiency (Fig. 6.4) is observed for the BFM 3D Sector and the Mixing Plane approaches compared with the 2D axisymmetric approach for higher rotational speeds ($\Omega/\Omega_{max} = 87.5 - 100\%$). The Mixing Plane model has a 0.72% difference with the experimental results in the near peak efficiency point at $\Omega/\Omega_{max} = 87.5\%$, and of 0.97% at $\Omega/\Omega_{max} = 100\%$. In the near peak efficiency point, the difference of the 3D sector BFM approach with the experimental results is 0.63% at $\Omega/\Omega_{max} = 87.5\%$, and 1.44% at $\Omega/\Omega_{max} = 100\%$.

The rise in discrepancy of the stage performance from the experiments with the increase of the fan rotational speed is due to the increase of the Mach number, with transonic effects from the $\Omega/\Omega_{max} = 87.5\%$ operation line upwards²³. The Mach number rise initiates different loss mechanism on the blade performance that are

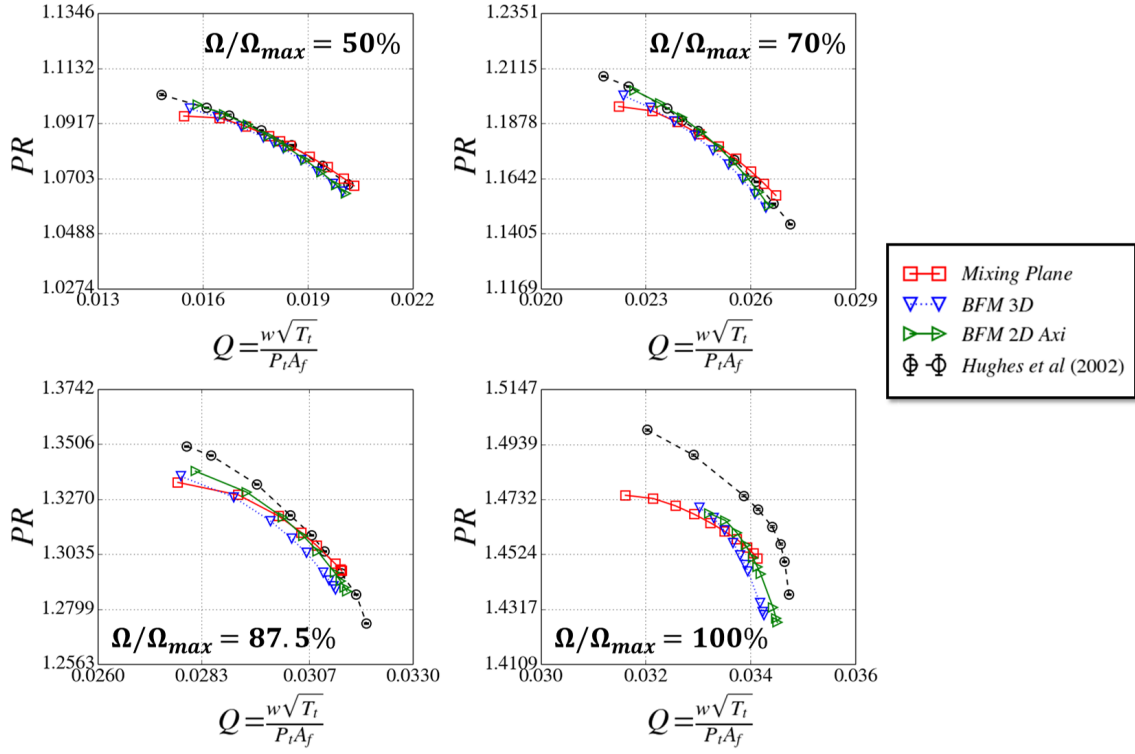


FIGURE 6.3: Stage Pressure Ratio (PR) of the NASA-R4 at 50-70-87.5-100% max rpm operation line. Experimental data from Hughes et al. ²³

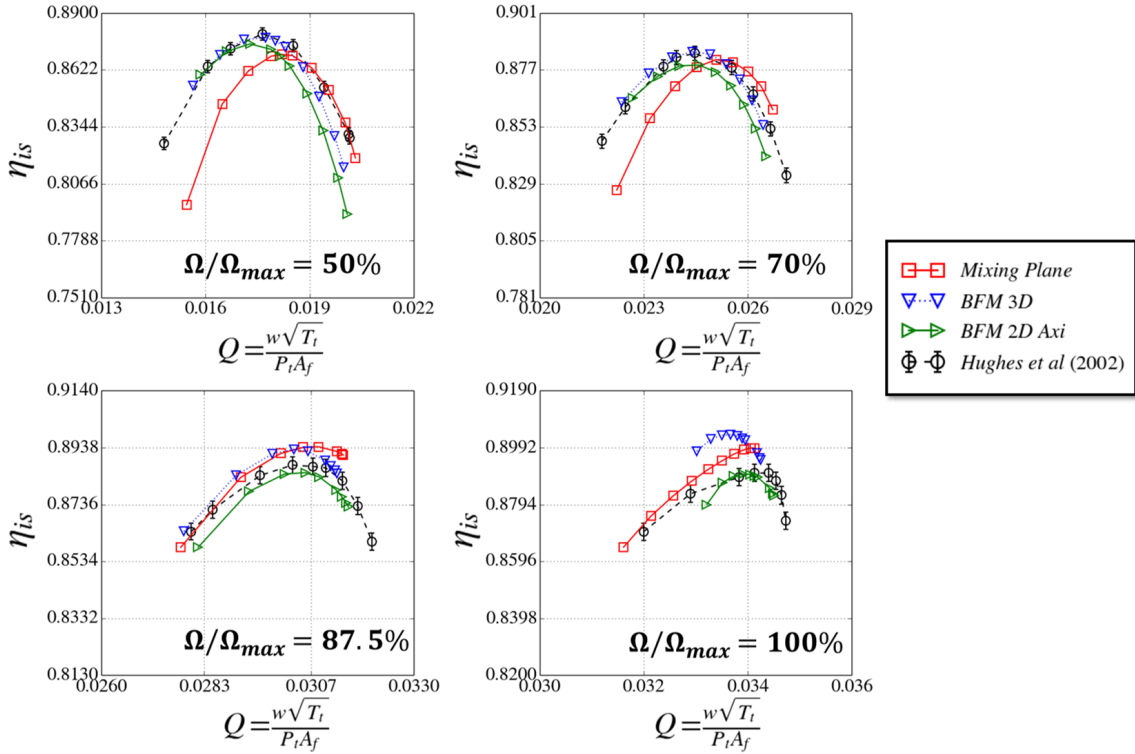


FIGURE 6.4: Stage isentropic efficiency (η_{is}) of the NASA-R4 at 50-70-87.5-100% max rpm operation line. Experimental data from Hughes et al. ²³

not fully represented on the Hall-Thollet model. On the Mixing Plane Model, the Mach number increase leads to the increase of losses not fully captured by the mesh definition and the emergence of transonic effects and non-steady mechanisms. The 2D-axisymmetric BFM approach allows a better calibration on these higher speeds regime due to the absence of modelling 3D viscous loss mechanism on the flow and near walls, closer to the ideal non-azimuthal variation of the flow defined by the Hall-Thollet model.

Overall, the agreement of the Hall-Thollet model at different operation lines with the experimental results is considered sufficient to model preliminary aero-engine installation aerodynamics and provides a reasonable representation of the stage performance when compared with a higher order approach such as the Mixing Plane model.

6.4.2 Radial distributions

The radial distributions of the pressure ratio, total-to-total isentropic efficiency, and swirl angle were measured at the Fan Exit Rake station (Fig. 6.1) at the nominal operation points for $\Omega/\Omega_{max}=87.5\%$ and 100% ²⁴. At these conditions, the fan is operating at $Q=0.02845$ and $Q=0.033$, respectively. Each radial probe is circumferentially averaged using an area-weighted approach. A total of 25 equal area distributed probes are used for the sample.

The radial distributions of PR_{fan} for the Mixing Plane model pressure ratio radial distribution has a good level of agreement with the experiments (Fig. 6.5), with a difference of 0.04-0.92% at $\Omega/\Omega_{max} = 87.5\%$ and 0.53-1.67% for $\Omega/\Omega_{max} = 100\%$. The 3D sector BFM pressure ratio radial distribution has a higher discrepancy with the experiments, with a peak difference of 0.23-1.4% for $\Omega/\Omega_{max} = 87.5\%$. A better match with the experiments is found at $\Omega/\Omega_{max} = 100\%$ for the 3D sector BFM compared with the Mixing Plane approach with differences of 0.23-1.61% in the PR_{fan} . Similar differences with the experiments are observed for the 2D axisymmetric BFM pressure ratio radial distributions, with 0.07-1.45% difference at $\Omega/\Omega_{max} = 87.5\%$ and 0.31-1.61% at $\Omega/\Omega_{max} = 100\%$. For the two BFM approaches a higher variability of the radial pressure ratio can be observed with more apparent differences with the Mixing Plane approach radial profiles in the mid-span region, between 35% and 70% of the span.

The radial distributions of the isentropic efficiencies of the Mixing Plane model and the 3D sector BFM approach show a good agreement with the experiments (Fig.

6.1) with differences from 0.25-2.0% for the $\Omega/\Omega_{max} = 87.5\%$ point, and 0.8-2.6% for the $\Omega/\Omega_{max} = 100\%$ below the 80% of the span. The differences with the experiments increase for all the approaches on the near tip region (span 80-100%). The 2D axisymmetric BFM isentropic efficiency radial distributions have higher discrepancies with the experiments below the 35% of the span for both sampled operation points.

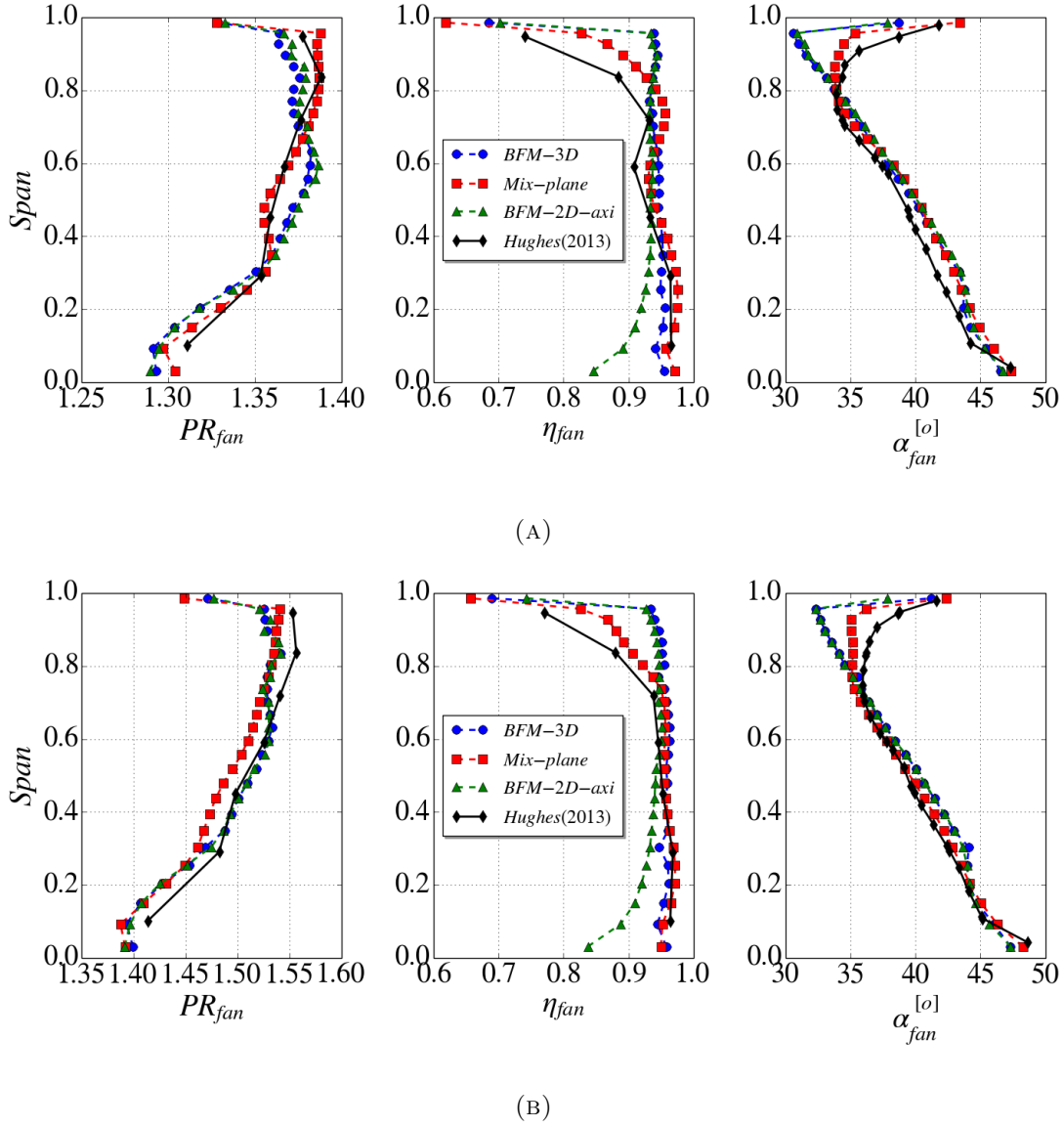


FIGURE 6.5: NASA R4 normal operation point Fan Exit Rake pressure ratio, isentropic efficiency, and swirl angle radial profiles: (a) 87.5% max rpm operation line, and (b) 100% max rpm operation line. Experimental results from Hughes et al.²⁴

The swirl angle radial distributions of the BFM approaches show a reasonable agreement, with a difference of less than 1.25 degrees for the $\Omega/\Omega_{max} = 87.5\%$ operation

point and 3 degrees for the $\Omega/\Omega_{max} = 100\%$ point under 80% of the span. Over the 80% of the span the discrepancy of the BFM approaches with the experiments increases with differences of up to 8 degrees. A better match is found for the Mixing Plane Model with discrepancies of 1 degree or less below 80% of the span.

Overall, the radial distributions of pressure ratio, isentropic efficiency, and swirl angle are in reasonable agreement with the experimental results for both the Mixing Plane and the BFM approaches. The discrepancy on the radial distributions on the selected metrics increase on the near tip region over 80% of the blade span. This indicates that the BFM approaches do not fully characterize the flow turning on that region and mass flow rate. This results on a misrepresentation of the work distribution. Additionally, the increase of the relative velocity over the 80% of the span leads to the emergence of transonic flow induced losses that are not fully captured by the Hall-Thollet model. These losses lead to a local rise on the differences with the experiments on the isentropic efficiency radial distributions of the two BFM approaches. However, these discrepancies with the experiments and the Mixing Plane approach are considered sufficiently small for a lower order model such as the Hall-Thollet BFM. Hence, the model provides an adequate representation of the turbomachinery throughflow in cases where some loss in the accuracy is permitted in favor of a reduction of the computational cost. This may be of use within the context of preliminary evaluations of propulsion integration for novel configurations.

In addition to the comparison with the experimental reference, the two body force model approaches have been compared with the mixing plane model at different rotational speeds and fan capacities. The results of that comparison are summarised in the Appendix [E](#).

6.5 Conclusions

The implementation of the Hall-Thollet Body Force Model in Fluent has been compared against the experimental results of the NASA Rotor 4 geometry. A RANS Mixing Plane model has also been used to evaluate the BFM performance. Two different implementations of the BFM have been evaluated, a 3D implementation of the model tested on a sector encompassing one blade passage, and a 2D axisymmetric planar implementation of the model. The BFM models have been calibrated for four speed lines of the R4 fan map.

The three numerical approaches show reasonable agreement with the stage pressure ratio characteristics across the fan map. The discrepancy with the experimental results increases slightly on the direction of the surge limit, and with the increase of the fan rotational speed and the increase of the Mach number. The BFM models have been calibrated to match the isentropic efficiency stage experimental results where possible. The discrepancy of the 3D sector BFM implementation increases with the rotational speed, compared with the 2D axisymmetric implementation. However, the results still show a good agreement with the experimental results and the mixing plane approach at the higher fan revolutions.

Overall, the radial distributions of the isentropic efficiency and swirl angle of the three numerical approaches are in reasonable agreement with the experimental results. An increase on the discrepancies with the experiments is observed over 80% of the blade span. These discrepancies account for changes on the work distribution on the captured flow rotation, small variations on the mass flow distributions, and an increase on the losses due to transonic flow mechanisms. However, these discrepancies with the experiments and the Mixing Plane approach may be considered sufficiently tolerable for such a lower order model depending on the application.

The Hall-Thollet model represents adequately the changes on the stage performance across different operation points. Some discrepancies with the higher fidelity references have been observed on the radial distributions of the pressure ratio, swirl angle, and isentropic efficiency for different operation points. However, these differences on the pressure ratio may be considered sufficiently small for a lower order method depending on the application. Therefore, the Hall-Thollet model provides an adequate representation of the turbomachinery throughflow in cases where some loss in the accuracy is permitted in favor of a reduction of the computational cost (40% and a 0.11% save on the computational cost from the mixing plane reference). Hence, this model can be applied within the context of preliminary evaluations of propulsion integration for novel configurations.

CHAPTER 7

Axisymmetric propulsion integration aerodynamics with through-flow turbomachinery

In previous chapters, a simplified one-dimensional model to evaluate the turbomachinery was used to model the fan stage of the embedded propulsor (Chapter 5). It was found that the fan outlet conditions have a notable impact on the overall efficiency of the system. For example, in embedded propulsive systems, the fan outlet conditions are defined by the propagation of the intake distortion through the turbomachinery stage. To evaluate the distortion transfer effects while a reasonable computational cost is maintained, a low order through-flow turbomachinery model was implemented on the design methodology (Chapter 6).

This chapter includes the studies carried with the 2D axisymmetric methodology including the through-flow fan model. The integration of the through-flow fan model enables a more complete study of the aerodynamic performance of the aft-mounted annular BLI propulsor. The intake and exhaust operation is linked with the change of the flow properties across the fan stage. Hence, the impact of the aerodynamic coupling between the fan and the aerodynamic design of the different housing components can be accounted. To understand the impact of the fan model on the performance prediction, a comparison the differences between the previous approaches and the coupled fan model is carried (Section 7.1)

In previous chapters, limited attention has been given to the exhaust design of embedded systems and its effect on the performance. This also has been found to be extended to the open literature, where the importance of the exhaust on the aerodynamic performance of the embedded propulsion systems has been largely neglected, with a few notable exceptions such as Kim and Liou⁴¹ and Lee et al¹⁸¹. However, for low specific thrust systems such as the embedded distributed propulsion

the aerodynamic performance of the exhaust has a significant impact on the whole power plant operation, since small penalties on the velocity coefficients can lead to a significant increase in the fuel consumption³⁶. The introduction of the through-flow fan model allows to include more representative flow operating conditions for the exhaust. Thus, in this chapter an analysis of the design requirements of the exhaust systems is included (Section 7.2).

The through-flow fan model is expected to also affect the exhaust requirements of the other housing components (aft fuselage, intake and exhaust). For this reason, the characteristics of the design space for these components are also explored in this chapter (Section 7.3).

7.1 Effect of the fan stage modelling in the aerodynamic performance

In this chapter, the Hall-Thollet low order through-flow model¹⁷ is applied to evaluate the interaction between the aerodynamic performance of the housing components and the turbomachinery. Before the application of this model, it is of interest to evaluate the differences between this approach and the semi-coupled one-dimensional fan approach applied in Chapter 5. For this purpose, a comparison between both approaches (one-dimensional and through-flow) for a given configuration is proposed performed (Tab. 7.1).

TABLE 7.1: Summary of the test configurations for the comparison between the 2D axisymmetric approaches

Test configurations			
Case	Dimensionality	Fan model	Fan efficiency
A	2D axisymmetric	Through-flow	From CFD ($\eta_{is} \sim 0.81$)
B	2D axisymmetric	Semi-coupled	$\eta_{is} \sim 0.81$
C	2D axisymmetric	Semi-coupled	$\eta_{poly} = 0.91$

A fixed propulsion integration aerodynamic design is evaluated (Fig. 7.1) under Sample mid-cruise operating conditions (Tab. 5.1). The rotational speed of the fan is fixed to 87.5% of the maximum rpm of the fan to obtain a $FPR \sim 1.3$ (Chapter 6). The configuration was then sized based on the boundary layer properties (momentum flux deficit and mass flow) and the target capacity of the NASA R4²³

stage (Appendix B.4). Given the length constraints of the aircraft (l_{ac}) and the increase in the nacelle length (l_{nac}) from the cases in Chapter 5 due to NASA R4 stage length²³, the increase in the nacelle area is bounded by the axial location. In this case, the location was fixed at $l_{aft}/l_{ps} = 0.7$ axially and $r_{hub}/r_{fus} = 0.1$ radially, with a hub-to-tip ratio of 0.3 to match the R4 requirements. This required the reduction of the BLR from 0.415 in previous chapters to 0.39 to match the maximum length constraint. This results in an expected TS of approximately 0.17-0.18 with power savings between 2-9%.

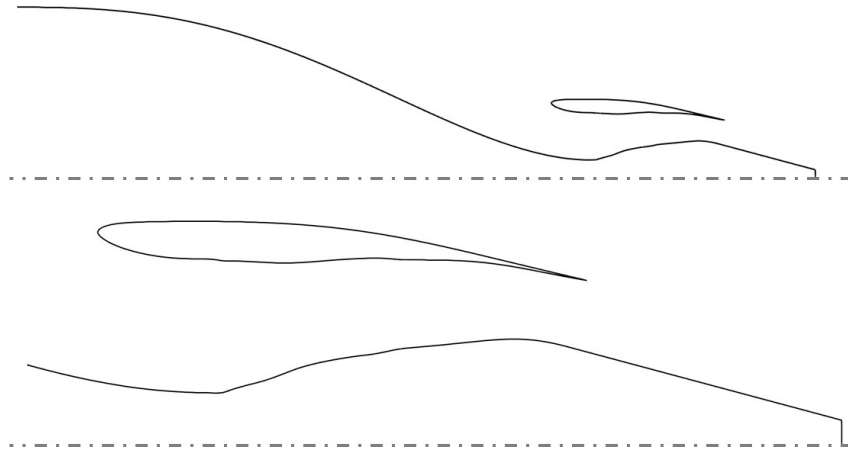


FIGURE 7.1: View of the sample geometry applied for the comparison between the through-flow fan model and the one-dimensional fan model ($l_{aft}/l_{ps} = 0.7$ and $r_{hub}/r_{fus} = 0.1$)

The mass flow rate of the through-flow fan model cannot be fixed as in the one-dimensional fan model (Cases B and C in Tab. 7.1). To allow a fair comparison between both approaches, the mass flow rate requirements of the through-flow fan model (Case A) are imposed into the one-dimensional fan model. The isentropic efficiency of the stage with the through-flow approach is determined by the stage geometry, and the local flow conditions at the fan face (total pressure and temperature) and the exhaust (static pressure at the discharge). For this particular design, this results in a loss in the fan isentropic efficiency (approximately 8%) from the NASA R4²³ peak efficiency value for the selected fan rotational speed ($\eta_{is} = 0.89$). The loss in the fan efficiency is a consequence of the flow distortion of the intake⁴³. This distortion propagates through the stage and results on changes on the fan work distributions with respect to the design point with uniform flow conditions. These losses in the stage isentropic efficiency are expected to be reduced by future distortion tolerant fan stages⁷⁸. However, the design of such distortion tolerant fans is considered out of the scope of the present work.

The one-dimensional fan model requires a fixed value of the fan stage isentropic or polytropic efficiency. Due to the loss in the isentropic efficiency of the fan stage with the through-flow approach, two different one-dimensional fan configurations are applied (Tab. 7.1). The first configuration (Case B) matches the isentropic efficiency of the stage obtained with the through-flow (Case A) method ($\eta_{is} \sim 0.81$). The second configuration (Case C) maintains a fixed value of polytropic efficiency as defined in Chapter 5 ($\eta_{poly} = 0.91$). These two configurations are used to determine the differences on the prediction of the aerodynamic performance of the propulsion integration due to the fan model, and to understand the limitations of the current fan technology.

7.1.1 Effect of the fan model in the thrust contribution and propulsive efficiency

The through-flow fan model provides a higher prediction of the thrust contribution of the aft mounted propulsor when compared with the semi-coupled one-dimensional fan approaches (Fig. 7.2). Based on the the through-flow approach (Case A), this results in a difference of 0.001 and 0.006 units of Thrust Split with the one-dimensional fan model with matched η_{is} (Case B) and with the one-dimensional model with fixed polytropic efficiency (Case C) configurations, respectively. This represents approximately 0.1-0.9% difference on the power savings predicted ($\sim 2 - 9\%$). This is a consequence of two factors: a variation on the Gross Propulsive Force, and the change on the local pressure forces at the intake and aft-fuselage surfaces.

The variation of the GPF with the fan model (Fig. 7.3) is mainly a consequence of the changes in the total pressure and total temperature in the exhaust inlet. The semi-coupled one dimensional fan adjusts the fan pressure ratio of the exhaust to match the target mass flow rate based on the fan face conditions and the prescribed isentropic or polytropic efficiency of the stage. Given that uniform flow conditions are imposed at the exhaust inlet boundary, this results in changes in the fan pressure ratio value with respect to the through-flow case. As a consequence, the total pressure and total temperature of the stage varies. This changes the effective fan nozzle pressure ratio (i.e. based on the static pressure at the discharge), from 1.75 in the through-flow case to 1.73 and 1.74 in cases B and C, respectively. The effect of these changes can be clearly observed in the changes in the Ideal Propulsive Force (Fig. 7.3).

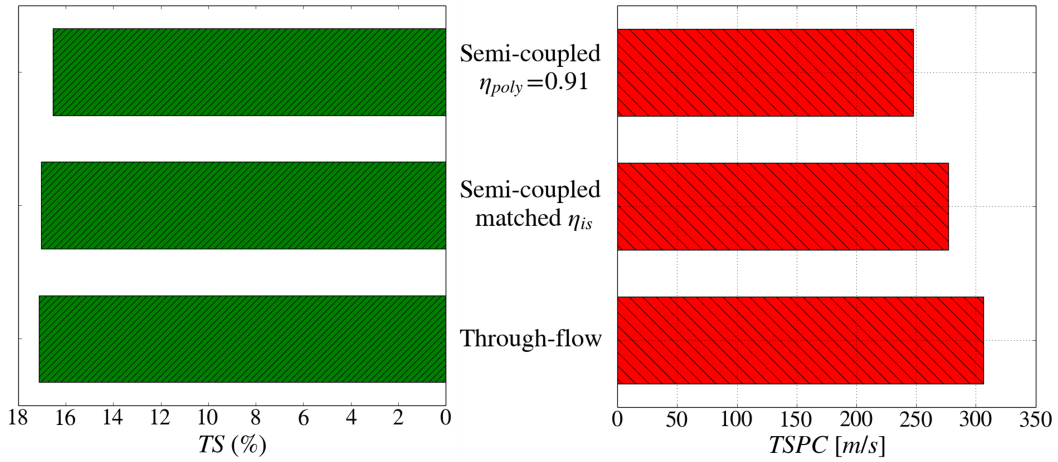


FIGURE 7.2: Effect of the one dimensional fan model in the prediction of the Thrust Split and the Thrust Specific Power Consumption for the Datum configuration

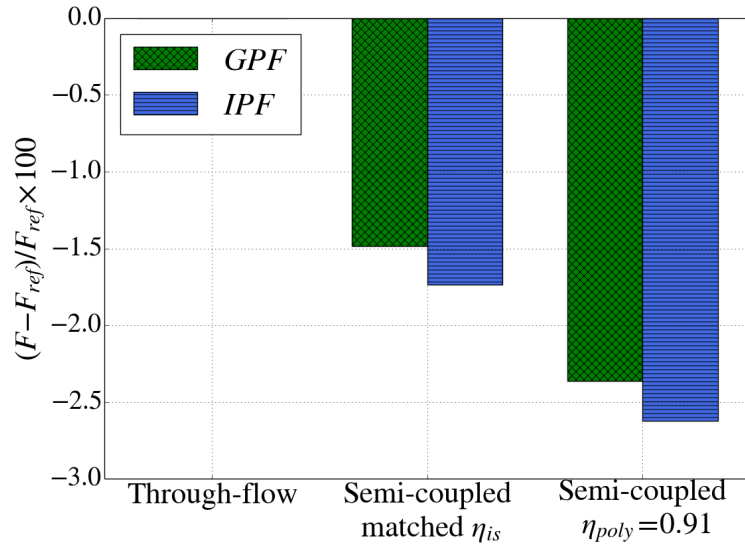


FIGURE 7.3: Effect of the one dimensional fan model on the prediction of the Gross Propulsive Force and the Ideal Propulsive Force. Variation of the metrics based on the values with the through-flow configuration

In addition to this, the required fan pressure ratio for the one-dimensional fan approach is sensitive to the changes in the total pressure values at the fan face. Thus, differences between the predicted total pressure at the reference plane increase the discrepancies on the *GPF*. The effect of the fan model on the prediction of the total pressure at the fan face is studied in more detail in Section 7.1.2.

The flow conditions at the exhaust vary substantially between the one-dimensional

fan approach and the case with the through-flow model (Figs. 7.4-7.5). A feature of the through-flow fan model is that the exhaust conditions are determined by the flow at the fan face, the stage geometry and the fan rotational speed. Due to the flow characteristics around the embedded propulsor, a clear radial gradient of the velocity is observed across the propulsor stage (Fig. 7.5) in the through-flow approach. The flow enters the intake duct with a velocity distribution defined by the boundary layer properties and the aft fuselage geometry. The flow direction then is modified by the intake duct geometry before it reaches the fan face. At the fan, the radial distortion is mitigated by the fan work (Fig. 7.5). This also results in some changes in the radial distribution of the mass flow. Then, the flow enters the exhaust duct with a non-uniform velocity, total pressure, and total temperature radial distribution. In contrast to the uniform conditions defined by the one-dimensional fan model, this results in a lower mass flow concentration at the inner surface of the exhaust duct for the through-flow fan approach (Fig. 7.5). This generates a reduction in the viscous forces in the inner walls of the exhaust⁹⁹. This also contributes to the increase in *GPF* by the reduction of the surface friction.

The changes on the pressure forces at the aft-fuselage and intake surfaces is another contributor to the differences in the force output between the one-dimensional and the through-flow model. The through-flow fan model induces a pressure gradient on the flow based on the fan geometry and operating conditions, while the one-dimensional flow determines the pressure gradient based on the target mass flow requirements (Figs. 7.4-7.5). This results in differences in the diffusion patterns at the aft-fuselage and intake surfaces, with a change on the local force contribution (Fig. 7.6).

The fan model also affects the predicted value of the *TSPC* (Fig. 7.2). The one-dimensional method under-predicts the *TSPC* value in comparison with the through-flow approach. This under-prediction of the *TSPC* is observed for the two input fan efficiencies tested. The case with fixed polytropic efficiency at 0.91 resulted in the lowest values of *TSPC* (i.e. highest propulsive efficiency), with a 19.6% reduction based on the through-flow case. The case with the fixed fan isentropic efficiency matched with the through-flow output value also predicted a lower value of the *TSPC* than the through-flow approach, with a 9.5% difference. These changes are notable, since the increase on *TSPC* is also correlated with the reduction of the power savings (Chapter 4) and results in additional penalties.

The source of these discrepancies is again the differences in the *FPR* predicted by the two fan model approaches. The one-dimensional fan cases predicted a lower required

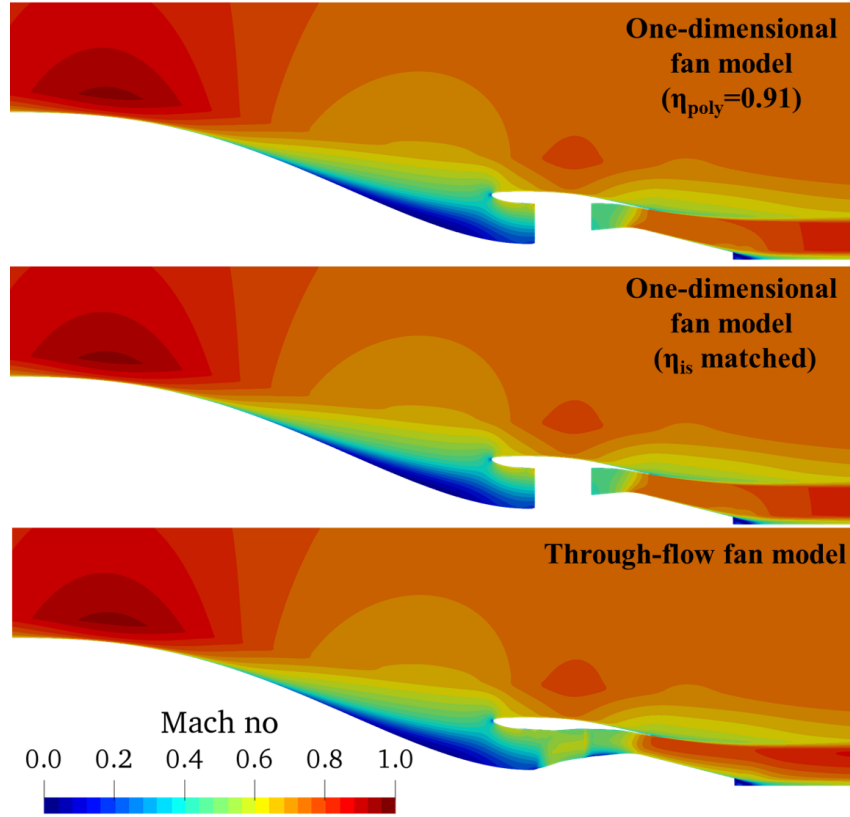


FIGURE 7.4: Mach number contours for the reference Datum design at mid-cruise conditions (Tab. 5.1). Comparison between the one-dimensional fan approach with two levels of fan efficiency and the through-flow low order fan model in 2D axisymmetric models

FPR to match the mass flow requirements at the exhaust duct. This resulted in a lower *GPF* prediction (Fig. 7.2), but also has as a consequence the reduction of the power required for the fan at a fixed isentropic or polytropic efficiency. The case with fixed polytropic efficiency has the highest efficiency value overall, this results in a further reduction of the *TSPC*. In the meantime, the fan pressure ratio and stage efficiency is determined by the local flow conditions, the fan rotational speed, and the stage geometry. This results in a variation of the power input based on the aerodynamic design of the propulsion integration and the propagation of the distortion at the intake through the fan stage (Fig. 7.5).

The differences observed between the three fan models are significant, since the combination of *TS* and *TSPC* is what defines the power savings from the integration of the aft-mounted BLI propulsor (Chapter 4). For example, if an independent power source for the aft-mounted propulsor is applied (Chapter 4.1.1), the power saving coefficient (*PSC*) reduces in 1.4% when the through-flow model is applied in comparison with the one-dimensional fan with matched isentropic efficiency. The

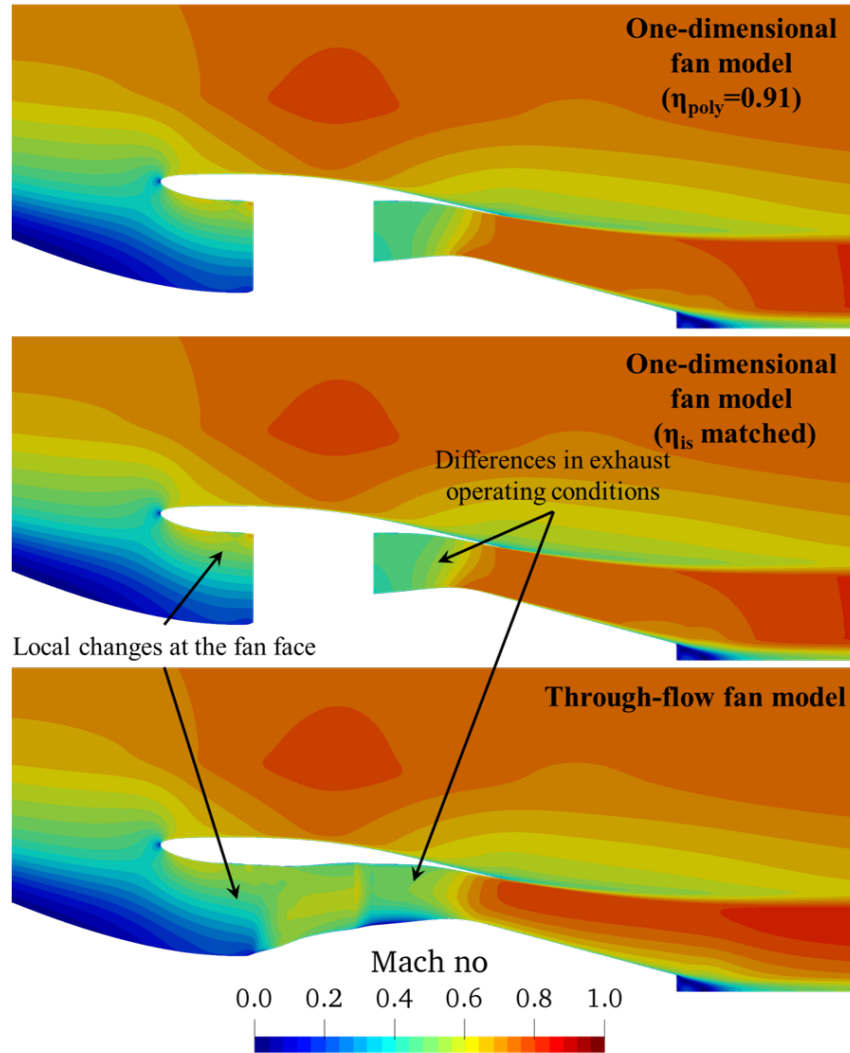


FIGURE 7.5: Mach number contours for the reference Datum design at mid-cruise conditions (Tab. 5.1). Detail around the fan cowl. Comparison between the one-dimensional fan approach with two levels of fan efficiency and the through-flow low order fan model in 2D axisymmetric models

difference in PSC increases with the case with fixed polytropic efficiency, with a reduction of the PSC of 3%. To put this in context, the expected PSC for this configuration is expected to be approximately 9-10%. This means 1.4% and 3% less fuel or power input reduction than expected. This has a significant economic and environmental impact. Thus, the application of a representative fan stage is essential to give an accurate representation of the power savings.

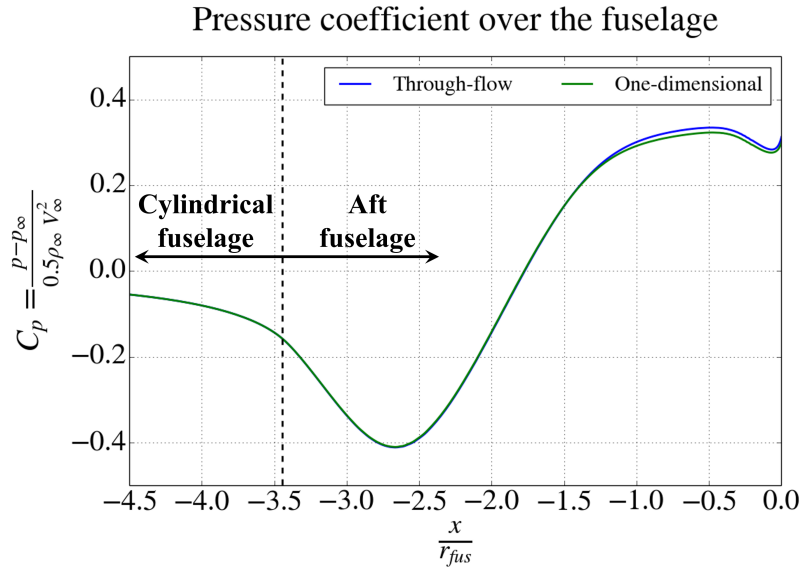
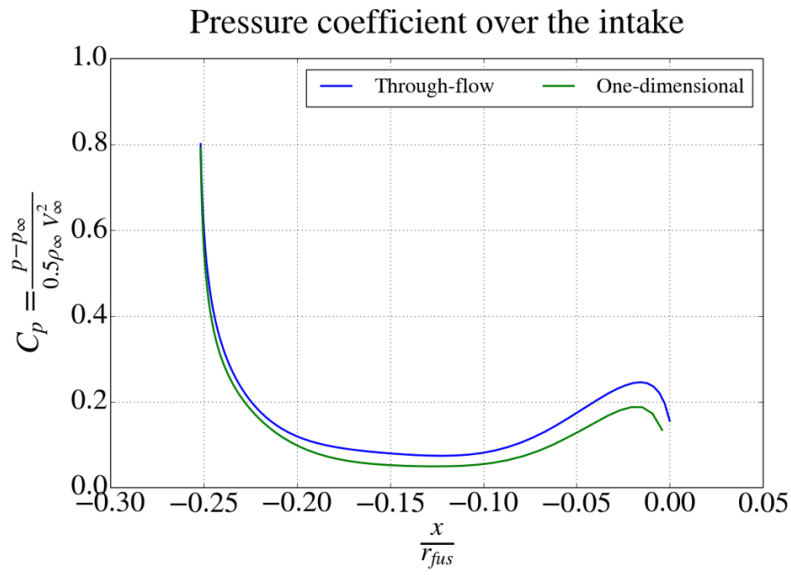


FIGURE 7.6: Pressure coefficient distributions over the fuselage (A) and the intake (B) for the the Datum design with the through-flow and the one-dimensional fan models

7.1.2 Effect of the fan model in the intake performance

The application of the through-flow fan model also introduces changes on the local flow field at the intake when compared with the one-dimensional fan approach (Figs. 7.4-7.5). This affects the predicted values of the intake performance metrics (IPR_{ing}

and RDI). To address the changes in the intake performance, the one-dimensional and through-flow models are compared (Fig. 7.7). In this case, the intake performance of the one-dimensional fan approaches is completely independent of the isentropic or polytropic efficiency selected for the fan. Thus, only the case with the fixed polytropic efficiency is displayed (Case B in Tab. 7.1).

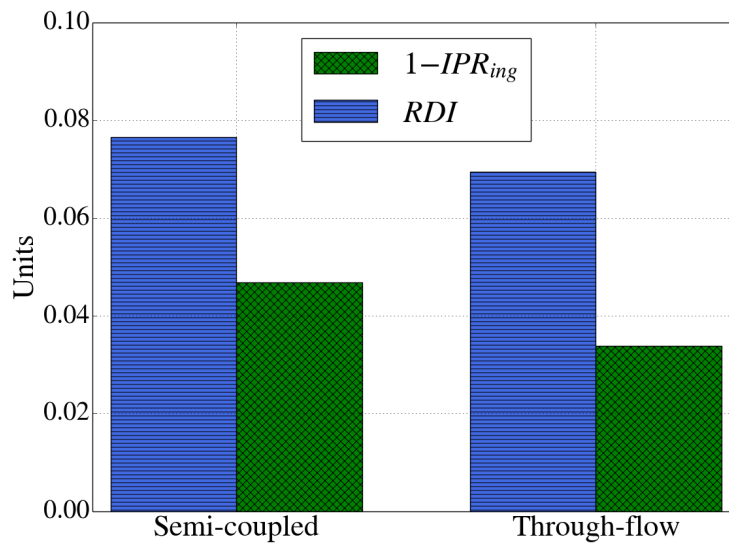


FIGURE 7.7: Effect of the one dimensional fan model on the prediction of the intake performance

A reduction in the Radial Distortion Index of approximately 0.008 units is observed for in the through-flow cases, when compared with the one-dimensional semi-coupled fan model (Fig. 7.7). This variation is not very significant, given that the radial distortion remains at relatively high values (above 0.05¹⁷⁷). At the same time, a reduction in the total pressure losses respective to the fuselage plane is observed in through-flow case, with an increase in IPR_{ing} of approximately 0.013 units (Fig. 7.7).

These differences on the RDI and IPR_{ing} are a consequence of the changes on the radial distribution of the total pressure and mass flow at the AIP (Fig. 7.8). The one-dimensional fan model at the fan face consist on a target mass flow rate boundary condition with a strong averaging of the static pressure¹⁵⁵. In this boundary, the local static pressure field is modified to match the mass flow rate requirements. The average value of static pressure is iterated until the boundary matches the requirements. This produces some discrepancies compared with a more representative fan model, such as the Hall-Thollet¹⁷ model, where the pressure gradient at the fan

face is defined by the fan geometry and operating conditions (rotational speed and capacity).

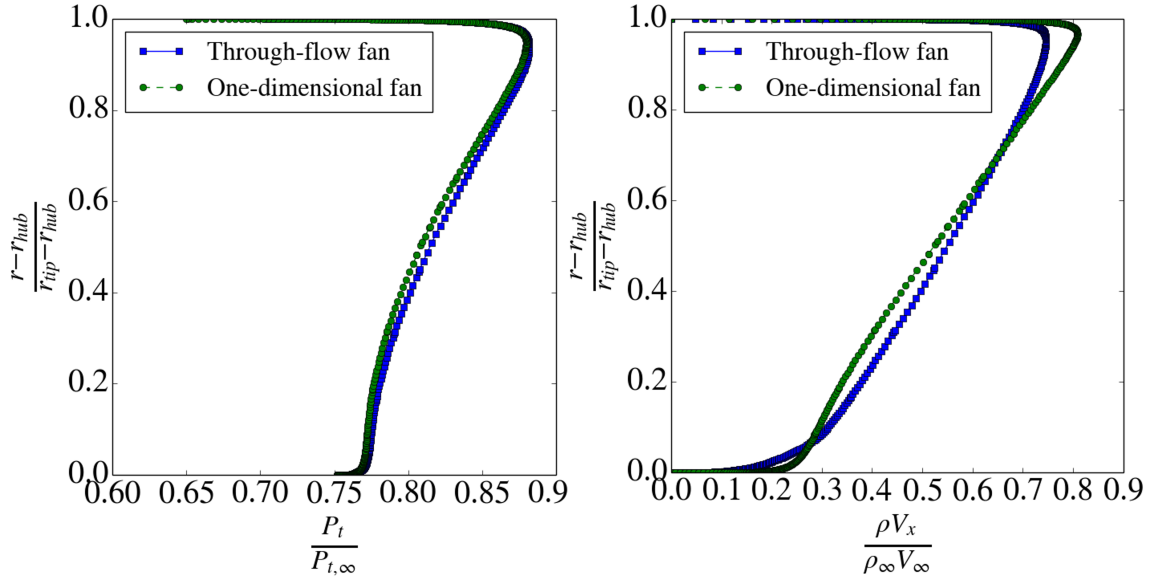


FIGURE 7.8: Pressure ratio and non-dimensional mass flow radial distributions at the AIP for the through-flow fan model and the one-dimensional fan model

As it has been observed, for the 2D axisymmetric configurations the application of a more representative fan stage with a through-flow model has a moderate impact on the prediction of the intake performance. Both the *RDI* and the total pressure losses at the intake are reduced with the throughflow model. Thus, both fan models could be applied for the prediction of the intake performance in 2D axisymmetric configurations. However, the application of the more representative through-flow model has the additional advantage of transferring the intake distortion through the fan stage. This results in more representative exhaust operating conditions in comparison with the uniform flow of the one-dimensional fan. The effects of the fan model in the exhaust performance are evaluated in the following section.

7.1.3 Effect of the fan model in the exhaust performance

The application of the through-flow fan model changes the operating conditions (total pressure and total temperature) at the exhaust duct. The change in the operating conditions is not only based on the average value of total pressure and total temperature, but also on the radial variation of these magnitudes (Fig. 7.4). This was observed to affect the values of the gross propulsive force and the ideal

propulsive force of the embedded propulsor (Fig. 7.3). The variation of these two magnitudes also changes the efficiency of the exhaust design (Fig. 7.9).

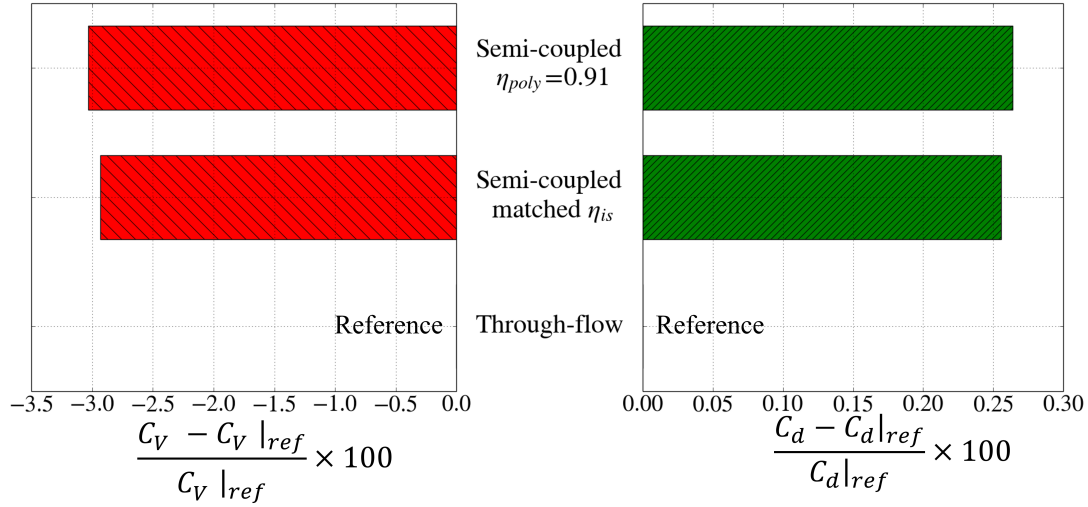


FIGURE 7.9: Effect of the one dimensional fan model in the prediction of the exhaust performance. Variation of the metrics based on the values with the through-flow configuration

The discharge coefficient (C_d) of the exhaust increases with the application of the one-dimensional fan models (Fig. 7.9). This is a consequence of the changes in the fan nozzle pressure ratio with the FPR and the change on total temperature with the stage efficiency. This affects the ideal discharge ratio (m/A) as defined in Eq. 3.41). At the same time, the velocity coefficient C_V increases in the one-dimensional models in comparison with the through-flow approach. This implies that while the GPF of the through-flow approach is higher (Fig. 7.3), there is a loss in efficiency on the exhaust duct.

The differences between both fan models in the predicted exhaust performance again show the importance of the application of a more representative fan stage model for the embedded propulsive systems. Thus, a representative fan stage model is necessary to evaluate the coupling between the embedded intake and the exhaust system, since the application of simplified exhaust conditions could lead to a misleading representation of the exhaust performance. Thus, for the analysis of the exhaust design it is required to have a coupled model that links the intake distortion with the exhaust conditions. In this thesis, this is achieved with the Hall-Thollet¹⁷ body force model. Hence, this through-flow fan approach will be applied in the subsequent chapters.

7.2 Optimisation of the exhaust of an embedded propulsor

The application of a through-flow fan model allows a more representative evaluation of the exhaust inlet conditions based on the fan response to the inlet distortion. This allows an evaluation of the aerodynamic design requirements of the exhaust system for the embedded propulsion systems, such as the aft-mounted annular BLI propulsor. The evaluation of such design requirements was found to be largely neglected in the open literature, despite some exceptions, such as Lee et al¹³⁹. Nevertheless, while some attention was given to the bulk exhaust design, the work of Lee et al¹³⁹ was mainly focused on the design requirements of the turbomachinery stage. The disregard of the exhaust design requirements for embedded systems could have a significant impact on the thrust contribution of the aft-mounted propulsor, since small penalties on the velocity coefficients can lead to higher power consumption in low specific thrust engines³⁶. Thus, it becomes crucial to evaluate the exhaust requirements of the embedded propulsion systems from an early design stage. For this reason, in this section the multi-objective optimisation methodology (Chapter 3.5) is used for the design of embedded exhaust systems with the aim of identifying its key design requirements.

7.2.1 Determination of the objective functions

The evaluation of the aerodynamic performance of the embedded propulsive systems requires the evaluation of several performance metrics (TS , $TSPC$, C_V , C_d , FPR , etc). The close integration of the propulsion system within the airframe increases the effect of the aerodynamic coupling between the different housing components. A possible consequence of this is the increase on the dependence between the performance metrics of the different housing components. Thus, before the optimisation of the exhaust system is performed, it is required to identify the sensitivity of the different metrics of interest to changes in the aerodynamic design of the exhaust system. For this reason, a design space exploration (DSE) is proposed to evaluate the sensitivities of the performance metrics to the exhaust system design.

As it was observed in Chapter 5, the integration of the propulsive system within the boundary layer of the airframe increases the aerodynamic coupling between the nacelle afterbody design and the exhaust system. This means that the changes in the nacelle afterbody design can affect the exhaust performance and design requirements.

Thus, if an accurate representation of the exhaust design characteristics is required, the exhaust system of the embedded propulsor should not be studied in isolation. For this reason, the geometry of the nacelle afterbody is included in the DSE of the exhaust geometry. The nacelle forebody and the geometry of the remaining housing components are fixed based on the Datum design defined in the previous section (Fig. 7.1).

The changes on the exhaust and nacelle afterbody aerodynamic design affect the turbomachinery performance (FPR and η_{is}) through variations in mass flow rate of the stage. These variations of the mass flow rate are driven by the change in the exhaust effective areas and the changes in the discharge static pressure. The change on the discharge static pressure is driven by the change on the flow diffusion patterns with the nacelle afterbody geometry (Chapter 5.2.2.2). Thus, in the DSE of the exhaust design, the fan pressure ratio (FPR) and isentropic efficiency (η_{is}) must be evaluated in conjunction with the thrust performance ($TSPC$ and TS) and the exhaust performance metrics (C_d and C_V).

For the DSE of the exhaust and nacelle afterbody geometry 9 degrees of freedom are required (l_{nac} , r_{te} , β_{nac} , β_{noz} , A_{noz} , $\theta_{up,noz}$, $\theta_{down,noz}$, β_{cone} , and f_{crop}) as defined in Chapter 3.2.2. A Latin Hypercube Sampling (LHS)¹⁶⁴ approach is used to evenly populate the design space. The aft-fuselage, intake, turbomachinery domain, and nacelle forebody geometries are kept constant based on the Datum design (Fig. 7.1). Sample mid-cruise operating conditions (Tab. 5.1) are applied. Geometry filters are applied for the nacelle afterbody to avoid geometries with inflections, as described in Chapter 3.5. A filter for the intersection between the nacelle afterbody geometry and the exhaust outer line was also included. Aerodynamic filters were also applied to remove designs with flow separation over the fan cowl (Chapter 3.5). The initial sample of the LHS was adjusted to obtain 144 valid designs after the filtering process.

From the designs of the DSE, the exhaust performance metrics (C_V and C_d), the turbomachinery performance metrics (FPR and η_{is}), and the propulsion integration aerodynamic performance metrics (TS and $TSPC$) are evaluated (Fig. 7.11). The Pearson and Spearman indices are applied to evaluate the correlation and the monotonicity of the different metrics, respectively. The correlation and monotonicity between pairs of metrics is then applied to identify the independent objective functions for the multi-objective optimisation (MOO) problem. This process is required to reduce the overall computational cost of the MOO by the reduction of the number of objective functions.

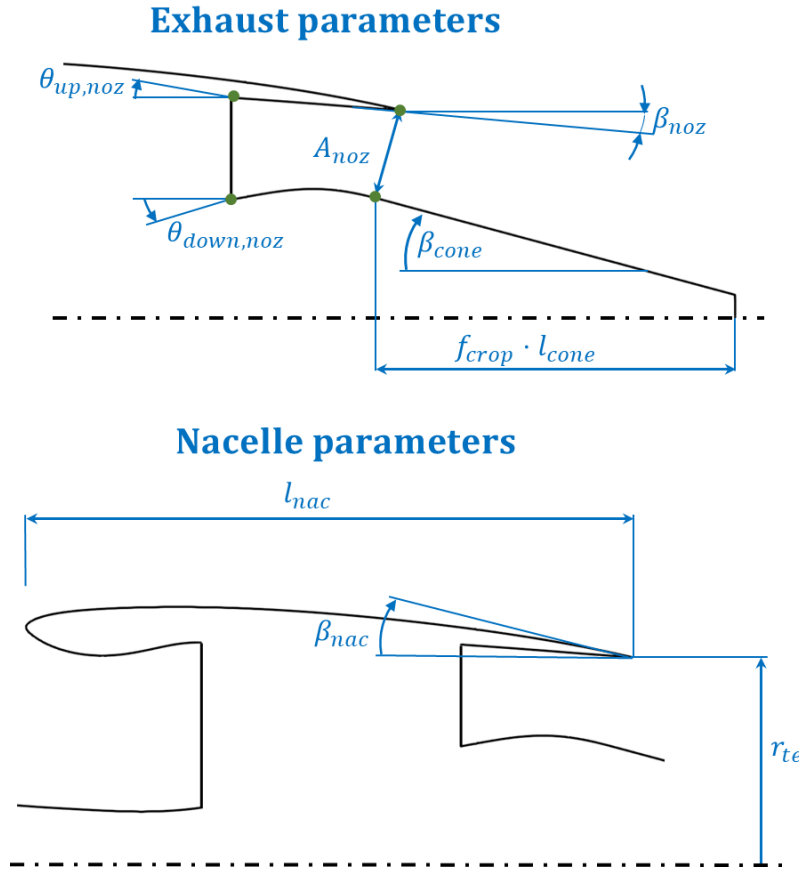


FIGURE 7.10: Definition of the 9 degrees of freedom for the design of the exhaust and the nacelle after-body

A very strong negative correlation¹⁸² and monotonicity was observed between the propulsive integration performance metrics (i.e. TS and $TSPC$), with Pearson and Spearman indices higher than -0.8 (Fig. 7.11). This is in agreement with the findings of Chapter 5.2 and a consequence of the inverse correlation between both metrics. The only variation between TS and $TSPC$ is derived from changes in W_{prop} , which depends mainly on the isentropic efficiency of the fan stage. While, the application of the through-flow fan model introduces some additional variability on the W_{prop} in comparison with the one-dimensional semi-coupled fan model applied in previous chapters, the correlation between TS and $TSPC$ is still sufficiently strong. Thus, any of these metrics can be applied to represent the aerodynamic performance of the aft-mounted propulsor.

The velocity coefficient (C_V) has also a very strong positive correlation and monotonicity with the TS , with Pearson and Spearman indices over 0.93 (Fig. 7.11). This suggests that the changes on velocity coefficient due to the design of the exhaust system contribute directly to the increase on the thrust contribution of the

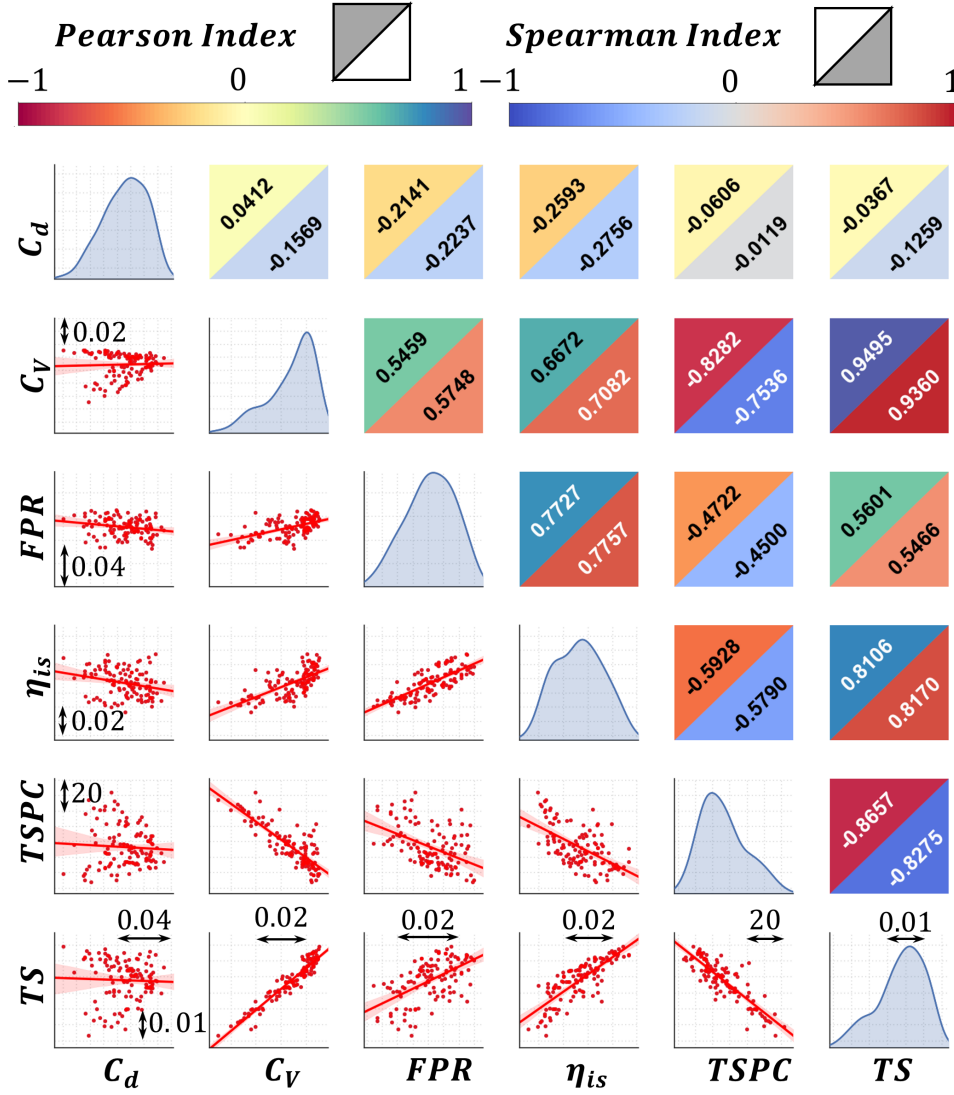


FIGURE 7.11: Comparison of the candidate objective functions for the optimisation of the exhaust aerodynamic design

system. This is true while the variations in C_v are driven by the change in the Gross Propulsive Force. Thus, it can be interpreted that the variations of the Ideal Propulsive Force with the exhaust inlet conditions is reduced in comparison with the increase of the thrust contribution. However, the main conclusion from the correlation and monotonicity between the TS and the C_v is that one of these metrics could be applied to represent the other in a MOO framework. Also, this means that the increase on the exhaust velocity coefficient contributes to the rise of the thrust contribution of the embedded propulsion system.

The turbomachinery performance metrics (FPR and η_{is}) have a strong correlation between them (Pearson index 0.77). However, when these metrics are compared with the TS , $TSPC$, and C_v ; only moderate levels of correlation and monotonicity

are observed (Fig. 7.11). Nevertheless, under this context, the correlation between TS and FPR can be considered sufficient to reduce the number of performance metrics on the study. Hence, both turbomachinery performance metrics (η_{is} and FPR) could be replaced by the TS in the context of an optimization. This is because this work is focused on the propulsion integration. If a detailed analysis of the performance of the turbomachinery design was required, both metrics should be included.

The only metric that maintains a clear independence of the rest of the candidate objective functions is the exhaust discharge coefficient, C_d (Fig. 7.11). No significant correlation and monotonicity is observed between this metric and the others. This means that the exhaust suppression is independent of the thrust contribution of the propulsive system. However, the analysis of the C_d is still of interest to understand the suppression losses of the exhaust systems of the embedded propulsion systems. Thus, this metric should be included in the definition of the multi-objective optimisation problem.

Under these circumstances, a multi-objective optimisation problem can be constructed based on two objective functions. To maintain compatibility with previous approaches, the TS has been selected as the metric to represent the main group of the propulsor aerodynamic performance. The other metric is the discharge coefficient. Thus, the problem consists of two objective functions to maximise (Eq. 7.1).

$$\begin{array}{l} \text{Objective} \\ \text{functions} \end{array} : \begin{cases} \text{maximise } (TS) \\ \text{maximise } (C_d) \end{cases} \quad (7.1)$$

7.2.2 Exhaust design optimisation

The objective functions for the optimisation of the exhaust of embedded propulsive systems have been defined (Eq. 7.1). In this section, these objective functions are used to optimise the aerodynamic design of the nacelle afterbody and the exhaust system of the Datum configuration defined in Section 7.1. The optimisation framework defined in Chapter 3.5 is applied with the Indicator-Based Evolutionary Algorithm (IBEA)¹⁶², with the Hypervolume Index as the indicator metric. The 9 degrees of freedom applied for the DSE of the previous section are used (l_{nac} , r_{te} , β_{nac} , β_{noz} , A_{noz} , $\theta_{up,noz}$, $\theta_{down,noz}$, β_{cone} , and f_{crop}). Furthermore, the LHS of the design space exploration is used as the seed for the optimisation problem. Further 45

generations of 36 individuals each are applied. The convergence criterion defined in Chapter 3.5 is used. For each generation, geometry filters are defined for the nacelle afterbody to avoid geometries with inflections, as described in Chapter 3.5. A filter for the intersection between the nacelle afterbody geometry and the exhaust outer line was also included. Aerodynamic filters were also applied to remove designs with flow separation over the fan cowl (Chapter 3.5).

The non-dominated design of the MOO result in a two-dimensional Pareto front (Fig. 7.12). Four reference non-dominated designs were selected to understand the limit of the design space. In addition, the Datum design (Fig. 7.1) defined in Section 7.1 was introduced for comparison. Two of these selected non-dominated designs (*A* and *D*) represent the limits of the two dimensional Pareto curve. The non-dominated design *B*, represents an intermediate design in which the increase on C_d does not penalise the TS as much as the design *A*. Some of the non-dominated designs have geometry tolerances very close to the manufacturing limits. The non-dominated design *C* represents an intermediate high TS in which the distance between the exhaust and the nacelle afterbody surfaces is not as restrictive as the other selected designs. Thus, it provides a higher flexibility for later design activities in this chapter.

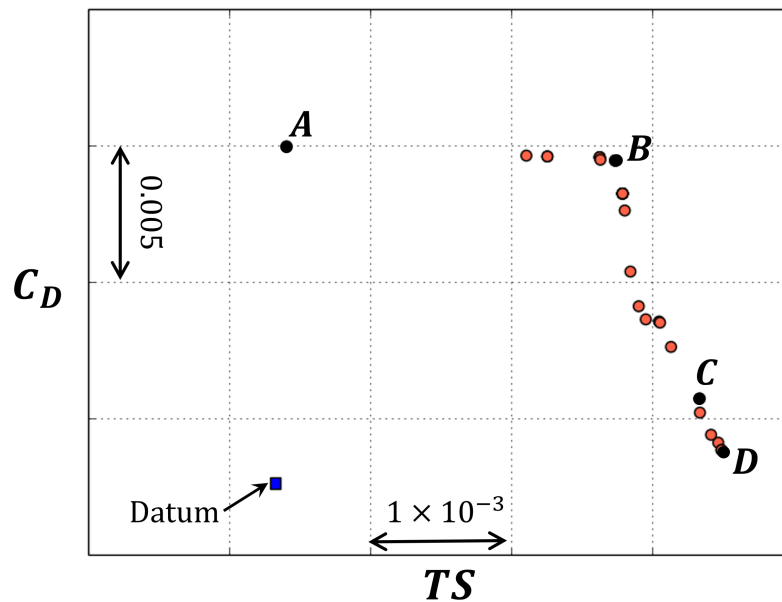


FIGURE 7.12: Pareto front of the multi-objective optimisation of the exhaust and nacelle after-body design

The non-dominated designs of the Pareto front extend on a range of approximately 0.003 units of TS and a 0.015 units of discharge coefficient. The changes on TS represent between 0.3-0.5% increase on the power savings. This is a second order

effect in comparison with the changes on power savings ($\sim 4-5\%$) due the propulsor location (Chapter 8.3). Thus, the exhaust design is not as significant as the bulk geometry parameters of the propulsion integration (i.e. system length and propulsor axial and radial location). However, it is still of interest to understand the design requirements for the exhaust systems of embedded propulsion systems.

Near the maximum C_d non-dominated designs, the variation of the discharge coefficient is negligible when compared with the variation of the TS . This is a consequence of the choked flow in the exhaust (Fig. 7.13). Under these conditions, the mass flow on the exhaust remains constant and the variations on the maximum C_d between non-dominated designs are defined by the effective static pressure at the discharge plane. Hence, there is no substantial reduction on the exhaust suppression (i.e. discharge losses in the exhaust) between the non-dominated design *A* and the non-dominated design *B* (Fig. 7.12), while a benefit from the Datum of approximately 0.0022 units of TS can be obtained by the application of the non-dominated design *B*.

The effective static pressure variation at the exhaust discharge plane is defined by the flow diffusion on the nacelle afterbody. The rise on the afterbody diffusion increases the effective nozzle pressure ratio (λ_{eff}). The increase on λ_{eff} chokes the exhaust for specific non-dominated designs (Fig 7.13). Under this condition, the discharge coefficient value cannot increase more for a given exhaust design. In addition, for an embedded propulsor, the variation on the exhaust static pressure is more significant than on a conventional podded exhaust. This is due to the effect of the fuselage boundary layer that adds a higher variation to the static pressure over the cowl compared with the freestream value. Thus, the nacelle design is an essential factor on the exhaust performance and operating conditions.

The Datum design (Fig. 7.1) was designed for a smooth curvature variation on the nacelle afterbody with an unchoked convergent nozzle. When compared with the non-dominated designs of the Pareto curve, a notable improvement on the TS and C_d values is observed. A maximum benefit of 0.0032 units in TS can be obtained with design *D*, while a benefit of 0.003 units is obtained with design *C*. As mentioned before, this translates to approximately a 0.3-0.5% increase on the power savings. This is not very significant compared with the expected values of approximately 9% in PSC . The increase on the TS is mainly driven by the reduction of the system length with shorter exhaust ducts and length of the cone (Fig. 7.13). The reduction of the exhaust duct length shortens the nacelle afterbody (Fig. 7.14). This diminishes the cowl wetted area and the drag contribution of the nacelle and

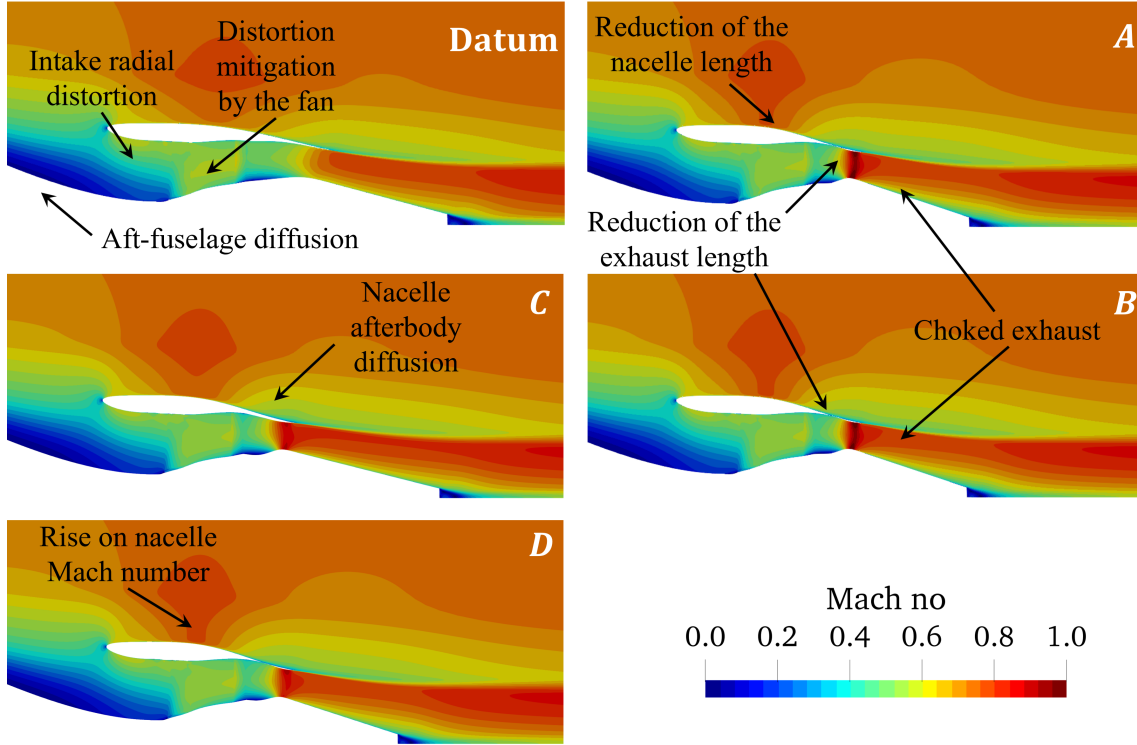


FIGURE 7.13: Mach number over the selected designs of the multi-objective optimisation of the exhaust and the nacelle afterbody. Datum design included for reference

the exhaust. An additional thrust contribution is obtained by the reduction of the nacelle trailing edge radius, r_{te} (Fig. 7.14). This increases the diffusion in the nacelle afterbody and increases the pressure forces in the thrust direction. Thus, there is a potential benefit on the application of compact exhaust systems (i.e. low r_{te} and l_{noz}) for the aft mounted embedded propulsor.

Additionally, for similar exhaust throat areas, the change on r_{te} moves the radial position of the cone downwards. This reduces the drag contribution of the cone surface, but the cone angle (β_{cone}) has to be adjusted to maximize the thrust contribution of the expansion of the nozzle exhaust flow (Fig. 7.14). Hence, the nacelle afterbody design has a notable effect on the thrust operation and defines the exhaust operation conditions.

A small increase between 0.0125 and 0.0119 units of C_d from the Datum configuration is obtained with the designs A and B, respectively (Fig. 7.12). However, a considerable penalty on the TS is found for the design A compared with the B (~ 0.003). This is indicative of the importance of the exhaust design on embedded systems. While a reduction of the exhaust length and trailing edge radial position is observed for all the configurations on the Pareto front (Fig. 7.14), the other design

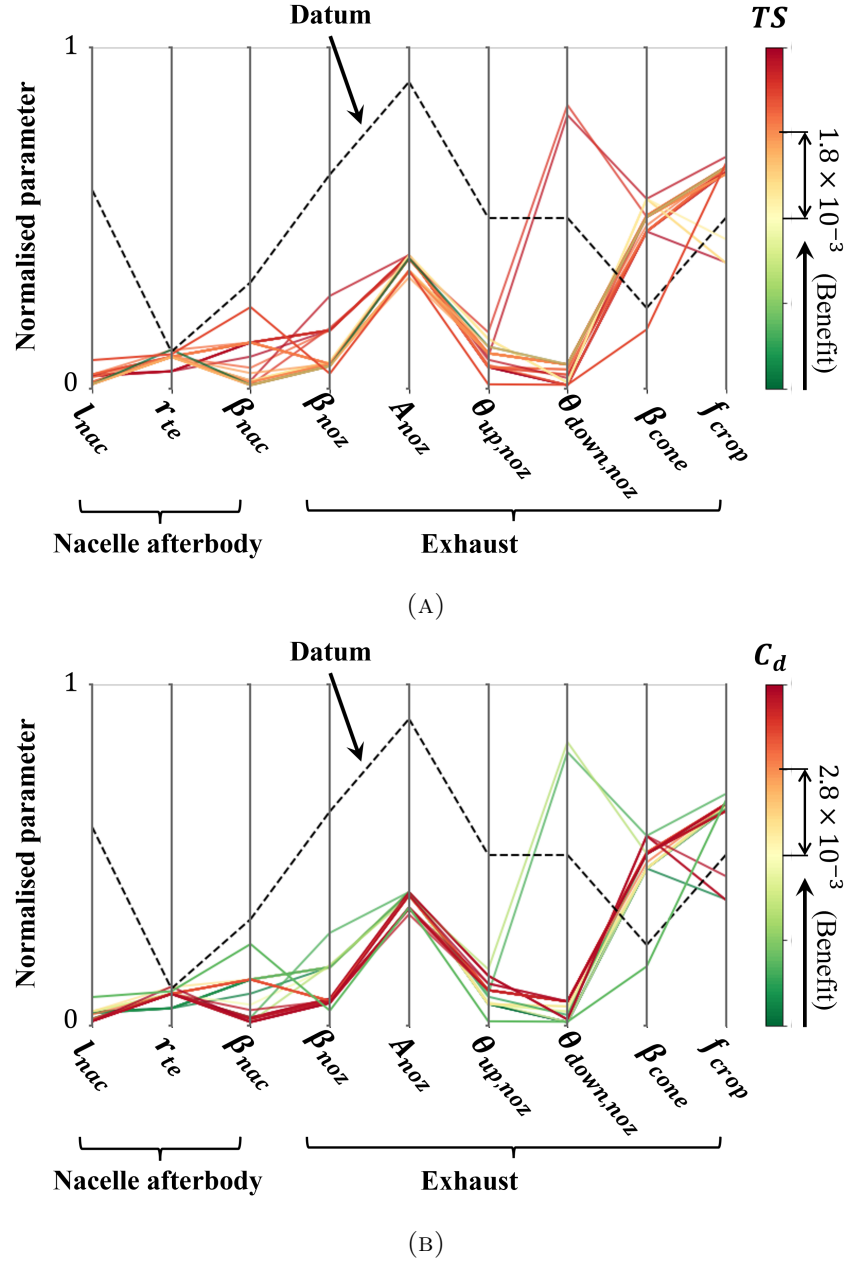


FIGURE 7.14: Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$) of the aerodynamic design of the exhaust. Coloured by TS (A), and C_d (B). Datum design as black dashed line. Values normalised with the optimisation limits

parameters can outweigh the benefit of the more compact system. Hence, the application of more compact exhaust designs for the embedded propulsive system will require detailed design of the exhaust characteristics to maximize the aerodynamic performance. Nevertheless, over the general benefits of the embedded system, the changes on the power savings due to the exhaust design have a secondary impact in comparison with other aspects of the design. The detailed analysis of the exhaust

will most-likely be of use in later design stages to maximise the overall benefits derived from the aft-mounted BLI propulsor.

7.3 Optimisation of the integration of an embedded propulsion system

The aerodynamic optimisation of the exhaust system produced an increase of approximately 0.003 units of TS with respect to the Datum design. However, to understand the actual contribution of that exhaust optimisation, it is necessary to compare this increase on TS with the maximum benefits that could be obtained with the optimisation of all the housing components of the propulsion integration. For this purpose, based on the exhaust design C (Ex-C), the multi-objective aerodynamic optimisation strategy for the aft-fuselage, intake and fan-cowl geometries of the embedded propulsor (Chapter 5.2) is used. This consists on 13 degrees of freedom to define the geometries of the different housing components, and three objective functions (Eq. 7.2). The Indicator-Based Evolutionary Algorithm¹⁶² is applied. An initial seed of 286 individuals is proposed with a Latin Hypercube Sampling. Further 50 generations of 65 individuals each are applied. The convergence of the optimisation problem was addressed with the criterion defined in Chapter 3.5.

$$\text{Objective functions} : \begin{cases} \text{maximise } (TS) \\ \text{maximise } (IPR_{ing}) \\ \text{minimise } (RDI) \end{cases} \quad (7.2)$$

The non-dominated designs of the MOO (Eq. 7.2) result in a three-dimensional Pareto surface (Fig. 7.15). As a reference, the Datum design and the exhaust design C (Ex-C) are included in the Pareto surface. A notable variation of the TS is observed across the Pareto surface, with a range of approximately 0.015 units of TS . This translates to approximately a 1.5-2.25% variation in PSC across the Pareto surface. Similarly, the RDI and IPR_{ing} have also a significant variation on the non-dominated designs of the Pareto surface, with a maximum difference of approximately 0.025 and 0.026 units, respectively. The wide range of non-dominated designs also results in an increase in the objective functions with respect to the Datum and the Ex-C designs (Fig. 7.15). On top of the 0.003 increase in TS from the Datum design to the Ex-C design, an additional maximum increase in TS of 0.006 units is observed from the Ex-C design. Thus, from a baseline aerodynamic design for the propulsion integration of the aft mounted propulsor (Datum), a maximum

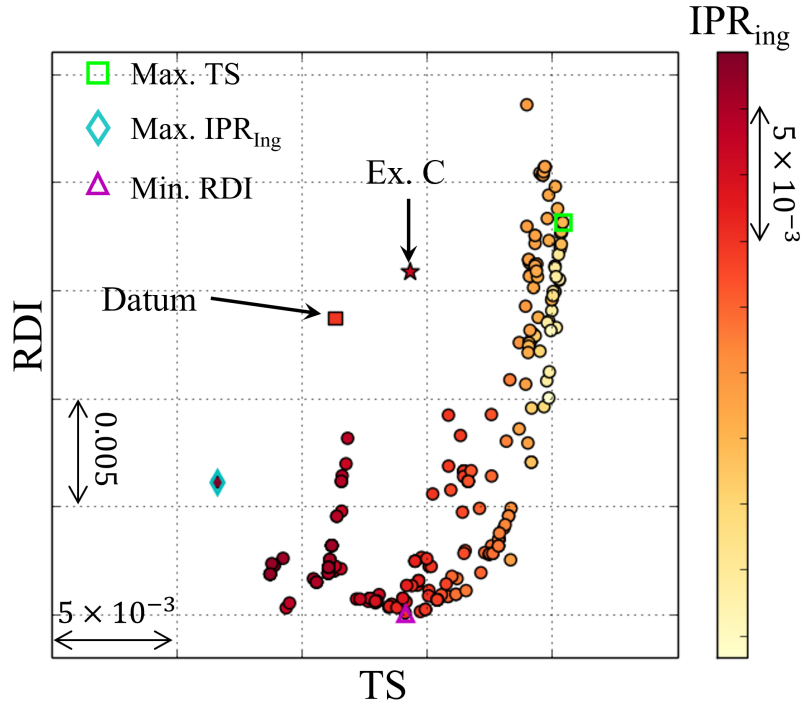


FIGURE 7.15: Pareto front of the multi-objective optimisation of the aft-fuselage, intake and nacelle with the downselected non-dominated exhaust design ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$). Comparison with the Datum design and the downselected non-dominated design (Ex-C)

increase of 0.009 units of TS is obtained. From this increase in TS , one third of the benefit is achieved by the redesign of the exhaust system. Based on the power savings estimation defined in Chapter 4, this increase on TS could imply an increase on the power saving coefficient (PSC) from the Datum of between 0.9% to 1.4%. While this has a lower impact than the location of the propulsor (i.e. 4 – 5% changes on power savings), the cumulative effect of the aerodynamic design housing components of the propulsion integration could have a notable impact on the overall power savings, which could be between 2% and 9% depending on the power configuration.

In addition, there is a potential increase in the intake performance with the reduction of the RDI from the Datum and Ex-C designs (Fig. 7.15). A maximum reduction approximately 0.014 units in RDI is observed. However, the radial distortion is still under relatively high values (i.e. $0.05 < RDI < 0.08$ ¹⁷⁷). The design with the minimum RDI has a lower TS than the reference designs. However, reductions of RDI of around 0.01 units can be obtained while an increase of approximately 0.0045 units in TS is obtained (Fig. 7.15). This means that the requirements for

a distortion tolerant fan could be reduced while a notable increase in the thrust contribution of the aft-mounted propulsor is maintained.

The non-dominated design with the maximum ingested pressure recovery (IPR_{ing}) penalises the TS in about 0.004 units, when compared with the Datum (Fig. 7.15). Some non-dominated designs of the Pareto surface maintain a higher IPR_{ing} , but the majority of them with thrust split values near the Datum and the Ex-C designs. This means that, for this particular configuration, the design requirements to reduce the total pressure losses at the intake and aft fuselage are in opposition to the reduction of the TS . This can be observed when the aerodynamics of the different designs are compared (Fig. 7.16). The design with the maximum IPR_{ing} presents a larger intake with a higher highlight curvature radius than the designs with the best values of TS and RDI . This results in a rounder highlight, with a thicker intake and nacelle. These features are also shared by the Datum and Ex-C designs, which also present higher values of IPR_{ing} than the design with the maximum TS .

Flatter fan cowls are observed to increase the thrust contribution of the aft-mounted propulsor (Fig. 7.16). This is similar to what it was observed in Chapter 5.2. The increase on the thrust contribution of the propulsion integration can be attributed to two main factors: the reduction of the wetted area and Mach number over the cowl, and the changes on diffusion in the nacelle afterbody. In this particular case, the effective fan nozzle pressure ratio increases and chokes the flow in the exhaust for the designs with maximum TS and minimum RDI (Fig. 7.16). This is a consequence of the changes on the diffusion in the nacelle afterbody

In Section 7.2, the optimisation of the exhaust and nacelle afterbody was constrained by a fixed forebody geometry, which limited the maximum thrust obtained with the exhaust design. However, as it can be observed the thrust contribution of the nacelle afterbody and the exhaust is highly dependent on the nacelle forebody design. At the same time, the nacelle forebody is constrained by the intake design requirements and the aerodynamic design of the aft-fuselage. Thus, for future work, it is recommended to perform a concurrent design of all the housing components of the embedded propulsive system. However, this will require to evaluate a very high-dimensional design space (20 degrees of freedom) with at least four objective functions (TS , IPR_{ing} , RDI , and C_d). This would result in a notable increase on the number of CFD evaluations and the overall computational cost. In order to facilitate the optimisation process under such conditions, more advanced design strategies would be required. For future work towards the optimisation of embedded

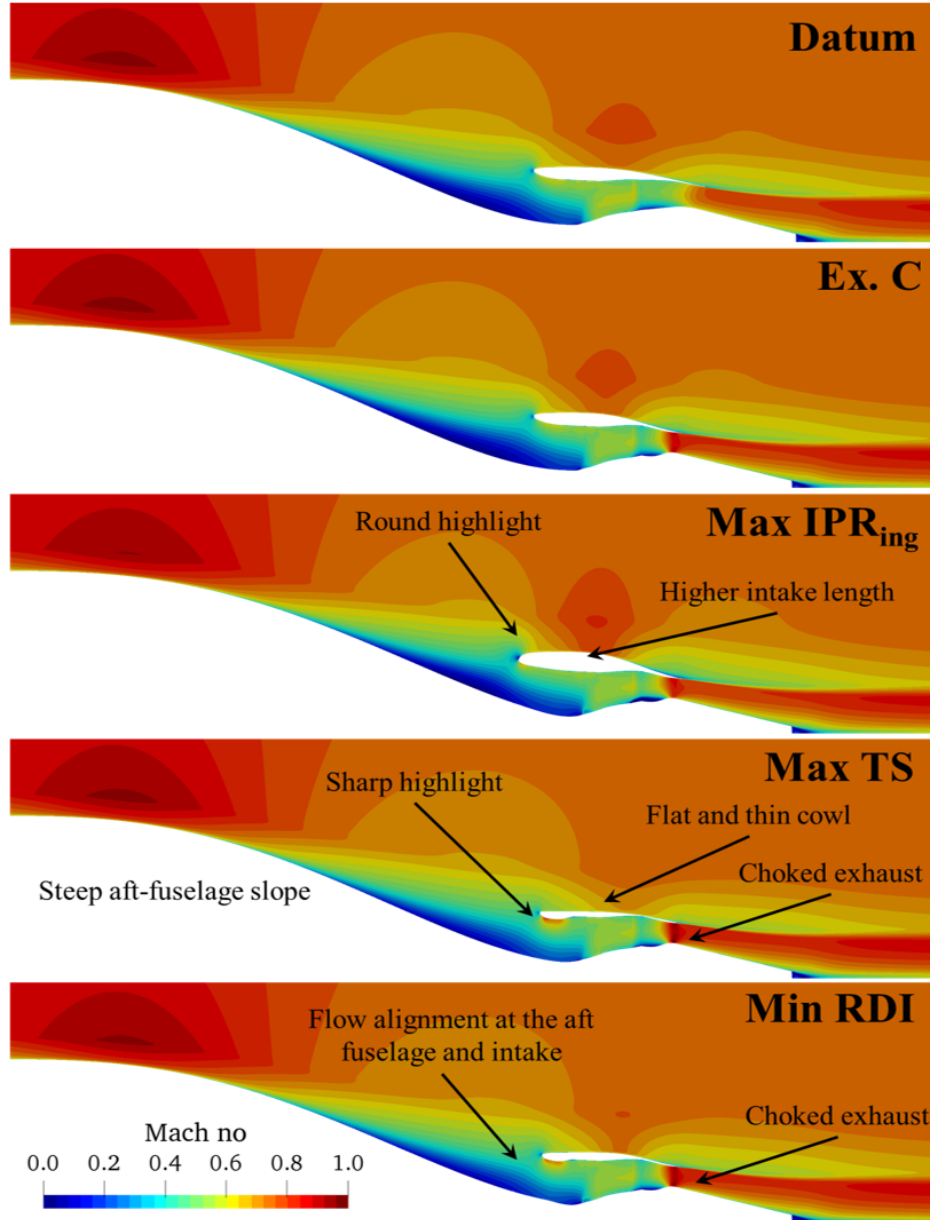


FIGURE 7.16: Mach number contours for the reference designs (Datum and Opt. C) compared with the best performing non-dominated designs for each objective function of the aft-fuselage, intake and fan cowl aerodynamic optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$)

systems, the author advises to explore dimensionality reduction methods such as active subspaces¹⁸³. Also, multi-fidelity design approaches¹²⁹ could help to reduce the overall computational requirements and facilitate the design and optimisation of the embedded propulsive system.

7.3.1 Impact of the design parameters in the aerodynamic performance

In this section, a more detailed analysis of the non-dominated designs of the Pareto surface is provided (Fig. 7.12). Following the approach presented in Chapter 5.2, the parameters of the non-dominated designs are normalised by the limits prescribed in the optimisation and presented in parallel coordinates plots (Figs. 7.17-7.18a). This is used to identify general groups of designs with common characteristics and the sensitivity of the objective functions to changes in these parameters.

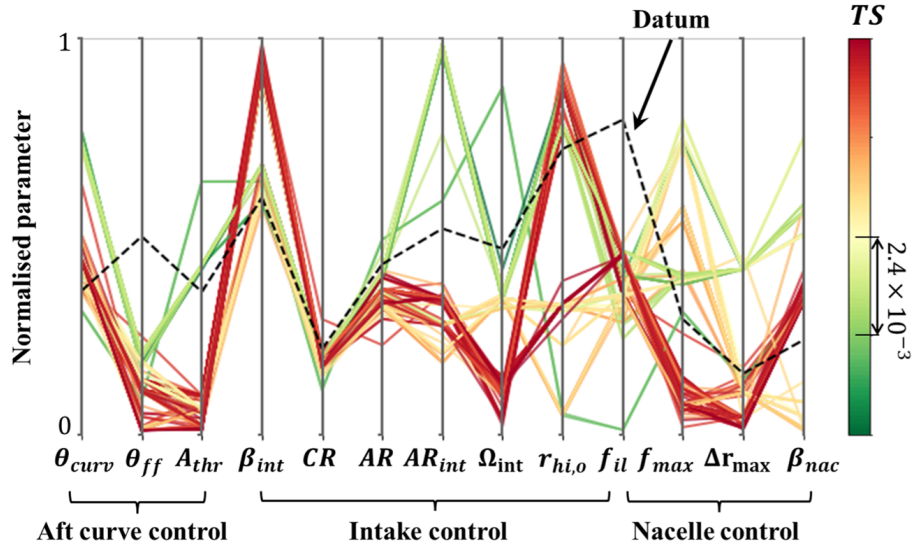


FIGURE 7.17: Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$) of the aerodynamic design of the aft-fuselage, intake and fan-cowl. Coloured by the Thrust Split value. Datum design as black dashed line. Values normalised with the optimisation limits

Some clear design groups with common characteristics can be observed for the non-dominated designs with the highest TS values (Fig. 7.17). Shorter intake (and cowl) lengths are preferred with lower AR_{int} than other non-dominated designs. At the same time, lower intake inclinations (Ω_{int}) with flatter nacelles (Δr_{max}) are preferred. These findings are different with the ones in Chapter 5.2, where higher intake inclinations were required to increase the TS . This is a result of the application of a more representative through-flow fan stage. This modifies the design requirements in two ways. First, the thrust provided with the through-flow fan model is less sensitive to changes in the effective pressure at the exhaust. Thus, the optimisation does not increase as much the flow diffusion in the nacelle afterbody to maximise the thrust. Second, the fan stage of the NASA R4 configuration increases the overall nacelle

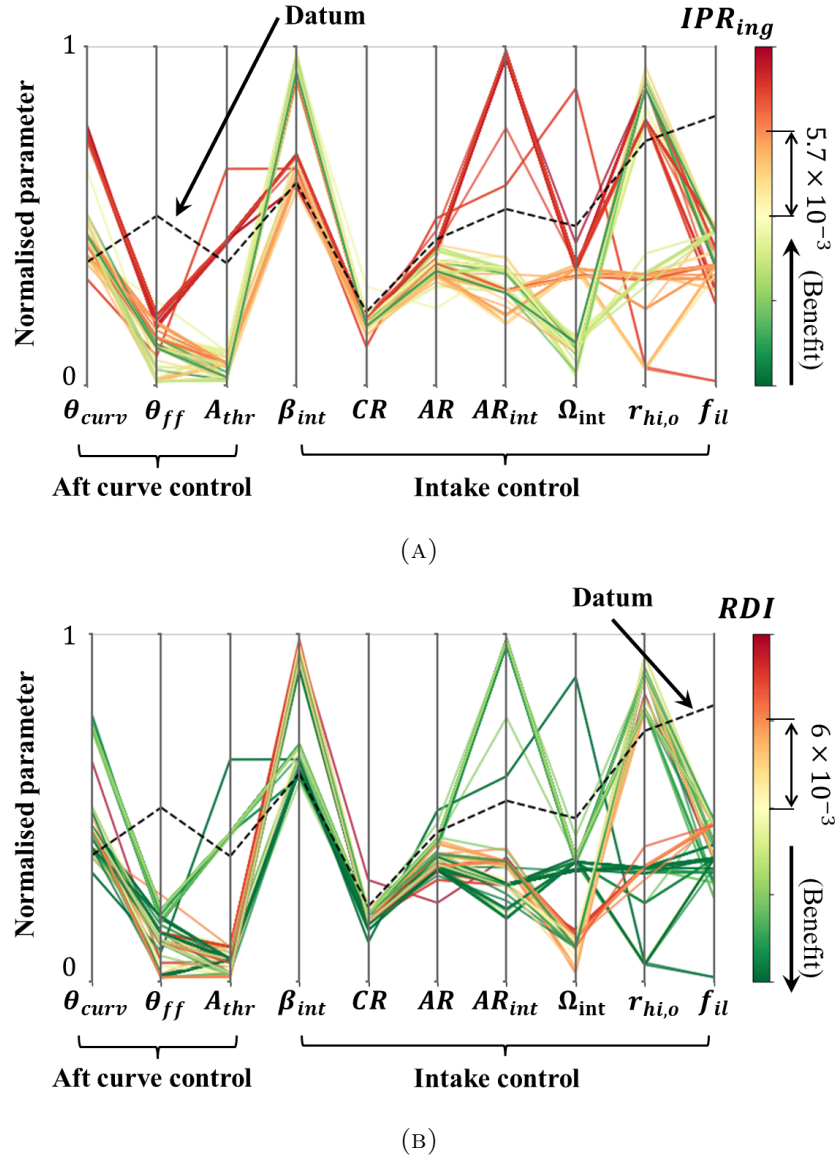


FIGURE 7.18: Parallel coordinates plot for the multi-objective optimisation ($r_{hub}/r_{fus} = 0.114$ and $l_{aft}/l_{ps} = 0.7$) of the aerodynamic design of the aft-fuselage, intake and fan-cowl. Coloured by the intake performance: IPR_{ing} (A), and RDI (B). Datum design as black dashed line. Values normalised with the optimisation limits

length, even with the application of the compact exhaust and intake. This results in a change in the design requirements of the fan cowl.

Similarly, some clusters of designs can be appreciated for the intake performance metrics (Fig. 7.18). The designs with the highest IPR_{ing} require larger intakes (AR_{int}) and higher intake inclination angles (Ω_{int}). At the same time, higher fuselage slopes (θ_{curv}) with higher values of θ_{ff} are also preferred. As observed in Chapter 5.2, this could be attributed to the orientation of the cowl towards the upcoming

flow in the aft-fuselage. The inclination of both surfaces (intake and aft-fuselage) changes to control the diffusion in the aft-fuselage and the orientation of the intake duct. On the other hand, the designs with the lowest RDI (Fig. 7.18b) require shorter intake lengths (AR_{int}) with lower intake inclinations.

7.4 Summary

In this chapter, the through-flow fan model was used for the 2D axisymmetric configurations. The changes on the aerodynamic performance between the through-flow model and the previously used one-dimensional fan were compared. A preliminary estimation determined the power savings for the test configuration between 2% and 9%. The application of the through-flow fan model results on a reduction of between 1.4% to 3% on the power saving coefficient in comparison with the one dimensional fan cases. This is mainly due to the changes on thrust specific power consumption with the reduction of the fan isentropic efficiency. When the fan efficiency is matched with the one-dimensional model, the differences on the predicted power savings are reduced. Thus, a more representative fan stage model, such as a through-flow model or a mixing plane approach, is recommended to capture the changes in power savings.

The fan model was observed to have a low influence on the predicted intake performance. With small variations on the RDI and IPR_{ing} between the one-dimensional fan and the through-flow model. Thus, for the analysis of the intake performance both models can be applied. However, the through-flow model results on more representative exhaust conditions due to the distortion transfer. This has a moderate effect on the exhaust performance, with reduced variations on the discharge coefficient ($\sim 0.25\%$) and more significant variations on the velocity coefficient ($\sim 3\%$). Hence, to evaluate the exhaust design requirements it is recommended to use a more representative fan stage model.

The through-flow model was applied for an optimisation of the aerodynamic design of the propulsion integration of the aft-mounted BLI configuration. The more detailed design of the housing components (intake, nacelle and exhaust) of the aft-mounted propulsor has a lower impact in comparison with the propulsor location. At a fixed propulsor position, an increase of up to 1.6% of the power savings was obtained with the optimisation of the aerodynamic design of the propulsion integration. From these changes, approximately one-third was obtained with the optimisation of the

exhaust design, while the remaining benefits were obtained with the optimisation of the aft-fuselage, intake and nacelle geometries.

CHAPTER 8

Non-axisymmetric propulsion integration

Previous chapters have explored the design space for the propulsion integration of the case study based on a two-dimensional axisymmetric approach. With this approach, the bulk design requirements of the propulsion integration of an aft-mounted BLI propulsor have been identified. However, the three dimensional flow conditions are expected to have a significant impact on the propulsion integration design^{13,59,76}. Thus, a 3D non-axisymmetric design approach has been built to understand the effect of the propulsion integration design on the aircraft-propulsor system.

The first part of this Chapter (Section 8.1) is dedicated to explore the design constraints for the integration of the propulsive system in a 3D reference airframe. In the second section, the 3D methodology is compared with the previous 2D axisymmetric methods to determine the discrepancies between both approaches (Section 8.2). The third section (Section 8.3), consists of a study of the importance of the position of the aft-mounted propulsor relative to the fuselage wake. This study is followed by an analysis of the importance of the bulk dimensions of the propulsor integration in the thrust contribution of the aft-mounted propulsor for a given propulsor position (Section 8.4). The information gathered on these sections and the previous chapters is then used to define a non-axisymmetric cowl design to improve the aerodynamic performance with respect to the axisymmetric baseline (Section 8.5). Finally, the main conclusions of this chapter are summarised in Section 8.6.

8.1 Design requirements

Before the propulsion integration design is addressed, it is important to understand the aerodynamic requirements and geometry constraints of the 3D aircraft. In the

present work, the FLR-F6 with FX2B fairing¹⁹ (F6) was selected (Fig. 8.1). A validation of the CFD methodology for this aircraft can be found in the Appendix D. To perform the aerodynamic studies, the F6 was scaled to the reference aircraft length. The cruise Mach number was maintained to the nominal design values ($M_\infty = 0.75$), but the operating altitude was corrected to 32,000 feet to match the reference cruise conditions of the previous studies. The scaling of the aircraft does preserve the Reynolds number from the reference configuration (from 5×10^6 to 2.6×10^7). However, the general flow topology derived from the transonic operation of the aircraft is maintained. The scaled aircraft at the corrected altitude was trimmed for the target lift coefficient of the experimental reference ($C_L = 0.5$) to obtain the reference drag values. The uncertainty derived from the mesh for this configuration under these operating conditions can be found in the Appendix C.3.1.

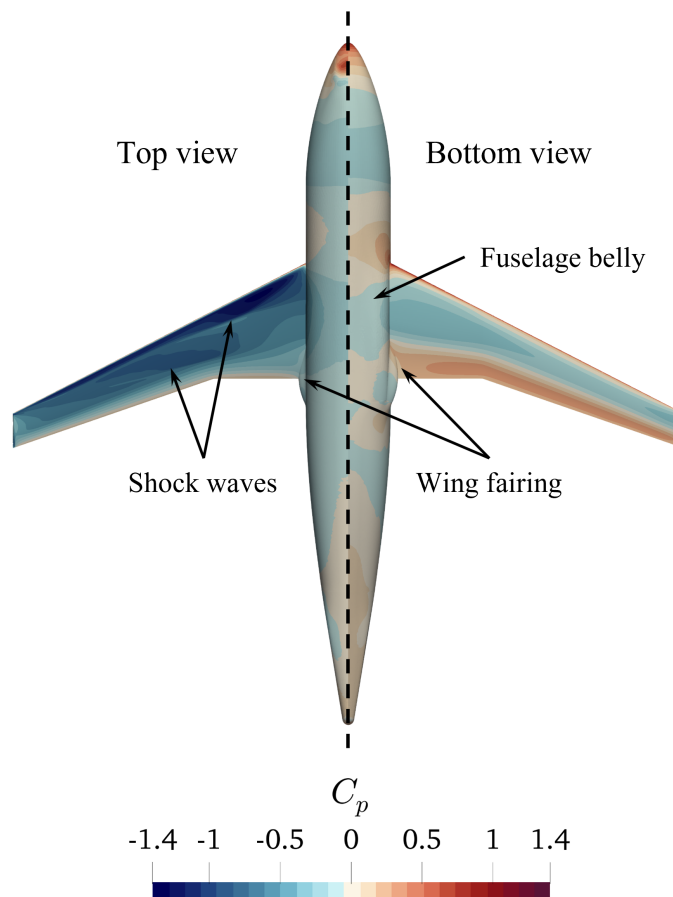


FIGURE 8.1: Pressure coefficient contours for the scaled DLR-F6 with FX2B body fairing¹⁹ at cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$, $Re = 2.6 \times 10^7$) and nominal angle of attack ($C_L = 0.5$)

The FX2B belly and fairing was introduced to the initial DLR-F6 definition in NASA's 3rd Drag Prediction Workshop¹⁸ to mitigate a horseshoe separation on the joint between the wing trailing edge and the fuselage. The presence of these bodies

was observed to reduce the drag by approximately a 1.5% from the initial baseline (Fig. D.7 in Appendix D) at wing tunnel conditions. For this case (scaled to the reference aircraft dimensions), the FX2B fairing is maintained to avoid the flow separation at the wing trailing edge. However, the introduction of the belly and the fairing is expected to add some additional three-dimensional flow effects that will affect the flow field around the aft-mounted propulsor. Besides the fairing and the belly, other sources of flow non-uniformities are expected to arise from the presence of the wings and the fuselage upsweep angle^{59,76}. For these reasons, before the integration of the propulsion system is included, an analysis of the aerodynamics around the fuselage afterbody is performed.

8.1.1 Aerodynamic constraints

It is of interest to evaluate the flow at the fuselage afterbody focusing on the boundary layer distribution around the fuselage. However, under non-homogeneous three-dimensional flows the definition of the boundary layer bounds is not trivial. As an alternative, a formulation based on the momentum flux deficit (Eq. 8.1) is proposed. This metric defines the loss of momentum from the freestream value due to the presence of the boundary layer around the aerodynamic body. The momentum flux deficit is an integral value derived from the boundary layer momentum thickness definition¹⁵⁰. For the boundary layer ingestion systems, this metric has great relevance. The main goal of the boundary layer ingestion is to compensate the momentum deficit (or kinetic energy) generated by the presence of the aircraft. Hence, it is essential to quantify the amount of momentum flux deficit ingested by the system to determine the potential aerodynamic gains of the configuration. A more detailed analysis of the trade between boundary layer ingestion and the momentum flux deficit can be found in Appendix B.

$$\dot{q} = \rho_{\infty} V_{\infty}^2 \Theta = \int_0^{\infty} \rho(\mathbf{x}) V(\mathbf{x}) (V_{\infty} - V(\mathbf{x})) dA \quad (8.1)$$

The integral value quantifies the boundary layer effects on the momentum at a local station, but it cannot be used to identify the boundary layer limits and the regions of maximum dissipation. For this reason, a normalised momentum flux deficit field, $\hat{q}$, was proposed (Eq. 8.2). This metric represents the momentum flux deficit per unit of area at a plane of reference and can also be identified as the defect on kinetic energy from the freestream value. The compressibility effects are also included with the changes on local density ($\rho(\mathbf{x})$). The local velocity passing through the plane

is defined as the normal velocity V_n . This metric is normalised with the freestream dynamic pressure value. With this definition of $\hat{q}$, the regions where the loss of momentum from the free-stream is more accused can be traced at specific stations of the fuselage afterbody.

$$\hat{q} = \frac{\frac{\partial \dot{q}}{\partial A}}{\frac{1}{2}\rho_\infty V_\infty^2} = \frac{V_n(\mathbf{x})(V_\infty - V_n(\mathbf{x}))}{\frac{1}{2}\rho_\infty V_\infty^2} \quad (8.2)$$

Five reference stations between 50% and 90% of the aircraft length from the fuselage nose were defined to study the evolution of the momentum flux deficit across the fuselage (Fig. 8.2). The first station is located just after the wing to include the effects of the wing wake downstream propagation. Despite of the DLR-F6 being a tail-less configuration, the gaps for the horizontal tail are present at the rear of the fuselage. This affects the local flow and it is not representative of the actual momentum flux deficit distribution of a real aircraft. Nevertheless, these sections of the fuselage afterbody were included to understand the evolution of the boundary layer across all the aircraft length.

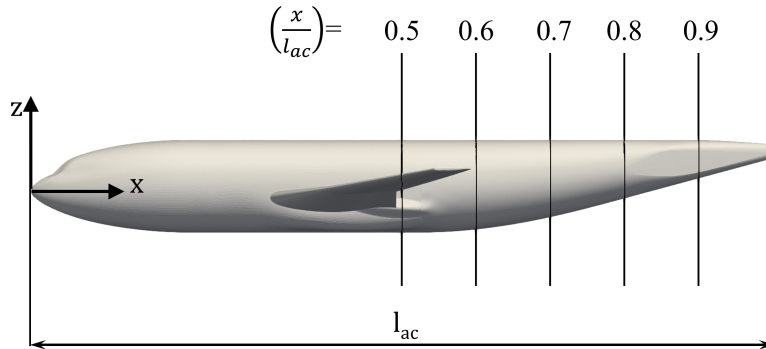


FIGURE 8.2: Side view of the DLR-F6 with FX2B body fairing and definition of the sample relative stations

In the station at 50% of the aircraft length ($x/l_{ac} = 0.5$), a thin boundary layer profile is observed on the top of the fuselage (Fig. 8.3). The thickness of the boundary layer increases locally towards the sides of the fuselage until the momentum thickness is reduced by the presence of a thinner boundary layer at the wing fairing. Another thin boundary layer is observed around the belly, with a similar reduction of the momentum flux deficit near the fairing. In the wake of the aircraft's wings, two regions of high momentum flux deficit are observed. First, the region with the highest $\hat{q}$ immediately after the wing trailing edge. This is where the main flux deficit regions of the wake are present and the ones that will propagate downstream. The second region, with a lower $\hat{q}$ contains the effect of the flow circulation through

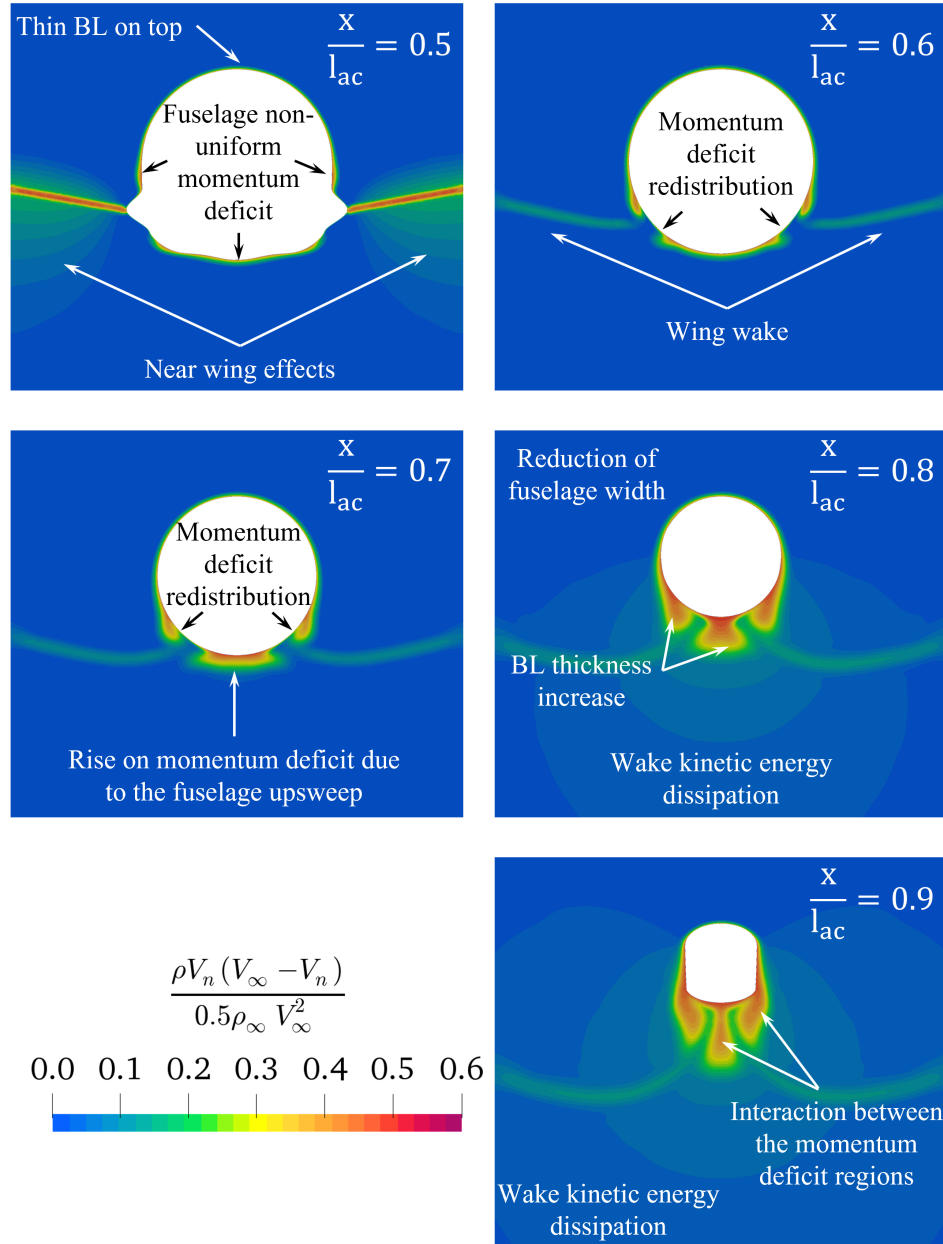


FIGURE 8.3: Momentum deficit distributions at the reference sample stations for the DLR-F6 at cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$, $Re = 2.6 \times 10^7$) and nominal angle of attack ($C_L = 0.5$)

the wing pressure and suction sides. At the 60% of the aircraft length both regions of the wing converge to a wing wake of lower intensity (Fig. 8.3). This is due to the redistribution of the momentum from the areas of lower $\hat{q}$ to the higher deficit region with lower mass concentration (Fig. 8.4). In more downstream stations ($x/l_{ac} = 0.7 - 0.8 - 0.9$), the momentum flux deficit of the wing wake decreases on intensity. However, at the same time the kinetic energy of the wake and the fuselage boundary layer is dissipated to the free-stream, where a region with higher area and

low momentum deficit concentration is observed.

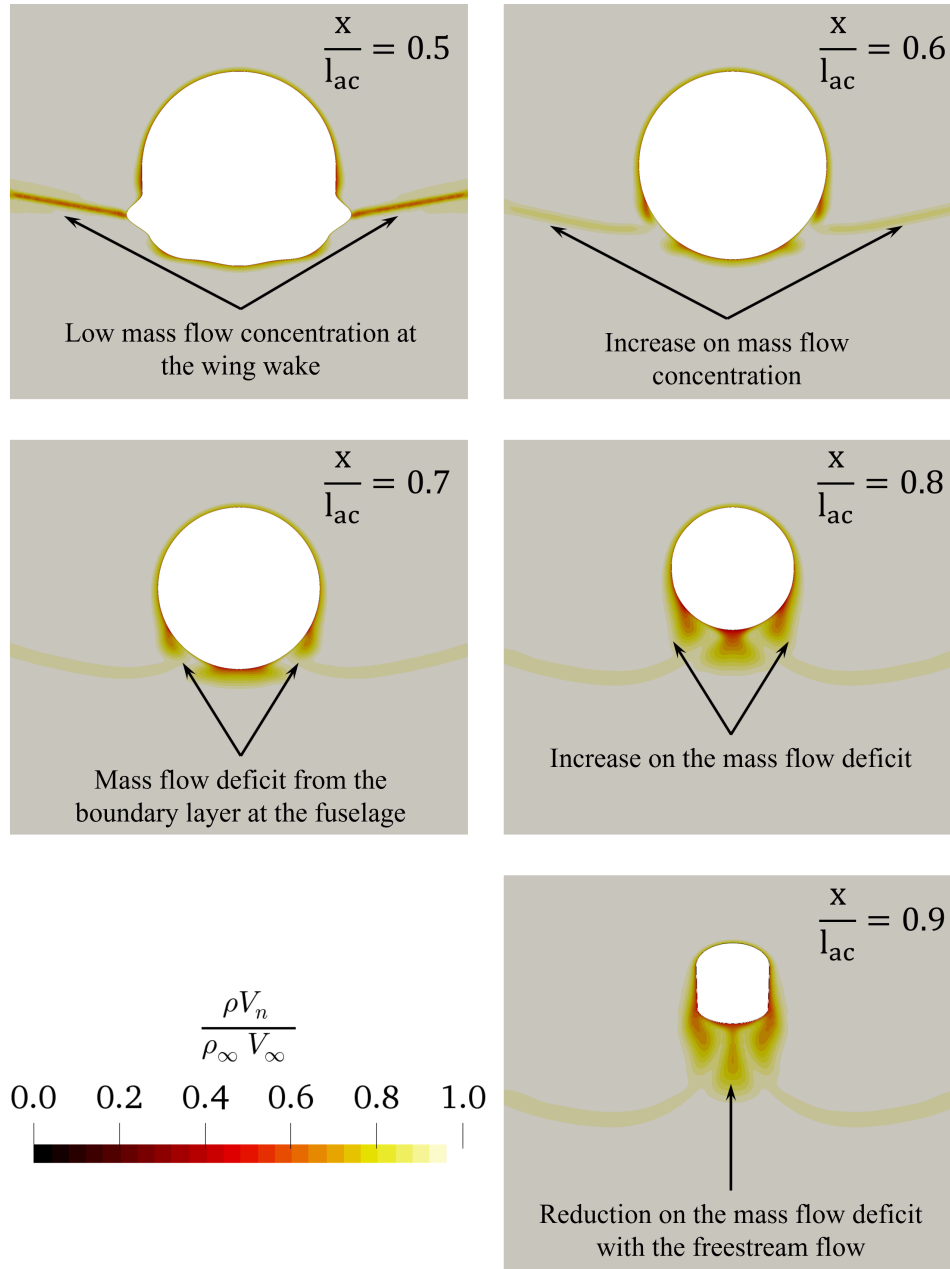


FIGURE 8.4: Distribution of mass flow per unit of area at the reference sample stations for the DLR-F6 at cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$, $Re = 2.6 \times 10^7$) and nominal angle of attack ($C_L = 0.5$)

At the same time, in the stations of 60% and 70% of the aircraft length, the momentum flux deficit regions at the fuselage belly and sides expand their area with the increase of the local boundary layer thickness (Fig. 8.3). The intensity of $\hat{q}$ increases also in these regions and further downstream (80% and 90% of the aircraft length). This is due to the viscous effects generated at the boundary layer. While the wing wake is allowed to dissipate into the free-stream, the presence of the fuselage has

a continuous contribution to the momentum flux deficit on the boundary layer regions. These regions expand downstream with the contraction of the fuselage and they propagate towards the lower part of the fuselage to the regions of lower mass flow concentration (Fig. 8.4).

As mentioned, important flow non-uniformities are expected to propagate downstream from the fuselage belly and wing fairing. These non-uniformities will affect the operation and design requirements for the integration of the aft-mounted propulsor. At the same time, the regions of the fuselage afterbody where the aft-mounted propulsor could be positioned ($x/l_{ac} \sim 0.8 - 0.9$ in Figs. 8.3-8.4) have the largest concentration of momentum deficit per unit of area. Thus, it is of interest to position the propulsive system on the regions where the momentum flux deficit concentration is higher, in order increase the amount of momentum deficit ingested.

For the propulsion integration with the present aircraft, the wing wake should not have a major contribution to the non-uniform flow perceived by the propulsion system in comparison with the other components of the aircraft. However, the dissipation of the kinetic energy in the wake of the wing and fuselage will affect the flow field around the fuselage and will contribute to the final net force of the vehicle.

In general, a large amount of the total momentum flux deficit is not recoverable with the boundary layer. This is due to the propagation of the deficit towards regions of lower intensity but large area extension. Given the space limitations for the aft mounted propulsion system, a significant part of the wake kinetic energy has to be wasted. The position of the propulsion system will have a major effect on the determination of which parts of the wake are ingested. However, the aircraft space limitations must be studied to determine the feasibility of the locations of interest. Also, the design of the propulsion integration of the aft-mounted powerplant requires the modification of the aft fuselage geometry. The change of this geometry is expected to have a major effect on the downstream propagation of the regions of higher momentum flux deficit. Thus, with the given spatial constraints, the aft-fuselage design will dictate the amount of the total momentum flux deficit that is recoverable by the BLI system.

8.1.2 Aircraft geometry constraints

From the propulsion integration of the aft-mounted annular propulsor, two main constraints were identified (Fig. 8.5): the aircraft length (l_{ac}) and the tail ground clearance or upsweep angle (δ_{ups}). First, when an existing airframe is used, the

reference length of the aircraft should not be exceeded. This is required to maintain the vehicle dynamics based on the position of the wing. If the airframe was to be designed along the propulsion integration of the embedded system, the relative position of the wings could be adjusted to compromise the vehicle length and the flight operation. At the same time, if the reference airframe remains unchanged, an excessive length of the aircraft would affect the second constraint: the ground clearance angle. This angle is derived from the airframe's take-off requirements to avoid tail-strikes.

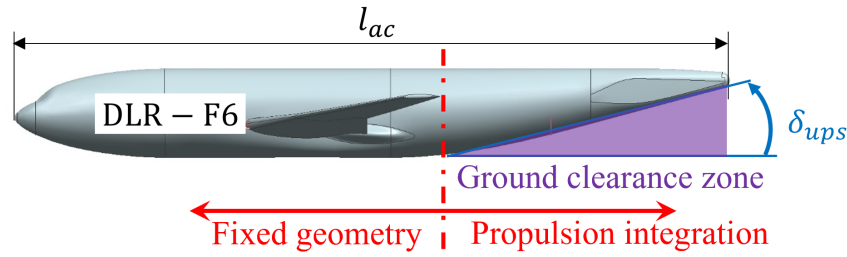


FIGURE 8.5: Schematic of the length and ground clearance constraints for the propulsion integration in the DLR-F6¹⁹ airframe configuration

The application of both constraints reduces the available positions for the aft mounted propulsor. Furthermore, the F6 fuselage wake regions with more concentrated momentum flux deficit are located close to the lower part of the fuselage where the positioning of the propulsor might be restricted. For this reason, it is of interest to identify the effect of the vertical and axial location of the aft-mounted propulsor on the amount of momentum deficit ingested. This will be addressed in Section 8.3.

8.2 Evaluation of an axisymmetric cowl design under 3D flow conditions

In a more representative approach, the aft-mounted propulsor will not be placed around the central axis of the aircraft due to the geometry constraints. This means that the 2D axisymmetric designs may be insufficient to define the design requirements of the aft mounted annular propulsor. However, some of the general design information obtained on previous chapters could still be applied to the preliminary design phase, before a more detailed analysis is performed. Thus, it is of interest to understand, as a first evaluation, the characteristics of the 2D axisymmetric models in comparison with a more realistic 3D approach.

In addition to the dimensionality of the problem, the approach selected to model the turbomachinery effect is also expected to have a significant impact on the prediction of the aerodynamic performance of the propulsion integration. As discussed in previous chapters, the flow distortion at the intake is ingested by the fan and propagate through the stage^{42,43,78}. Not to include these effects was found to misrepresent the exhaust operation (Chapters 5.4.2-7.1). In previous chapters, two different approaches for the model of the turbomachinery were introduced: the semi-coupled one dimensional fan, and the through-flow body force model. The through-flow body force model was defined as a more representative model for the fan operation under intake flow distortion. However, in the 3D configurations, the application of the through-flow fan model duplicates the computational cost of the analysis. This is because the rotation of the fan breaks the planar symmetry of the aircraft. For this reason, for any 3D studies that includes the low order through-flow fan model it is required to evaluate the full aircraft (port and starboard sides). For a similar type of analysis, while there are no lateral flows involved, the one-dimensional fan model could be used with only half of the domain due to the aircraft's symmetry in the XZ plane. Thus, the application of a one-dimensional approach of the fan could be advantageous for some specific applications in which the exhaust operation does not play a significant role (e.g. analysis of the intake operation under flow incidence).

To understand the effect of the dimensionality (2D axisymmetric and 3D) and the fan model (one-dimensional or through-flow) in the aerodynamic performance, a comparison between the models (Tab. 8.1) of a representative axisymmetric cowl design (Fig. 8.6) is proposed. This cowl design is modelled with the 2D axisymmetric methodology (cases A and B) with the reference axisymmetric fuselage and with the 3D methodology on the DLR-F6 fuselage (cases C and D). For the 2D axisymmetric configurations, the mean radius of the central section of the DLR-F6 was applied to define the 2D axisymmetric reference fuselage. In the 3D configuration the engine is installed vertically on the central axis defined by the fuselage's top line and bottom line. The symmetry plane of the engine also matches the symmetry plane of the fuselage. In both models, the change on the fuselage geometry (i.e. the origin of the aft fuselage design) starts at a distance from the fuselage nose of 60% of the reference aircraft length. Both geometry configurations are tested with both turbomachinery model approaches (one-dimensional or through-flow). For the cases with the semi-coupled one-dimensional model (A and C), the target mass flow at the intake is set to the values obtained with the equivalent through-flow model. The FPR values are adjusted to match the mass flow rate requirements at the exhaust for a fixed polytropic efficiency value ($\eta_{poly} = 0.91$). Uniform flow conditions are applied at the

exhaust. This obviously diverts from the actual values of fan efficiency and fan outlet conditions obtained with the through-flow method (cases B and D). Nevertheless, the objective of the test is not to emulate the same conditions obtained with the 3D through-flow model, but to understand if the one dimensional fan model can be applied under some specific conditions.

TABLE 8.1: Summary of the test configurations for the comparison between the 2D axisymmetric and the 3D modelling approaches

Test configurations				
Case	Dimensionality	Fan model	Fan efficiency	AoA [$^\circ$]
A	2D axisymmetric	Semi-coupled	$\eta_{poly} = 0.91$	N.A
B	2D axisymmetric	Through-flow	From CFD	N.A
C	3D	Semi-coupled	$\eta_{poly} = 0.91$	$-0.5 - 0.0 \ C_L = 0.5$
D	3D	Through-flow	From CFD	$-0.5 - 0.0 \ C_L = 0.5$

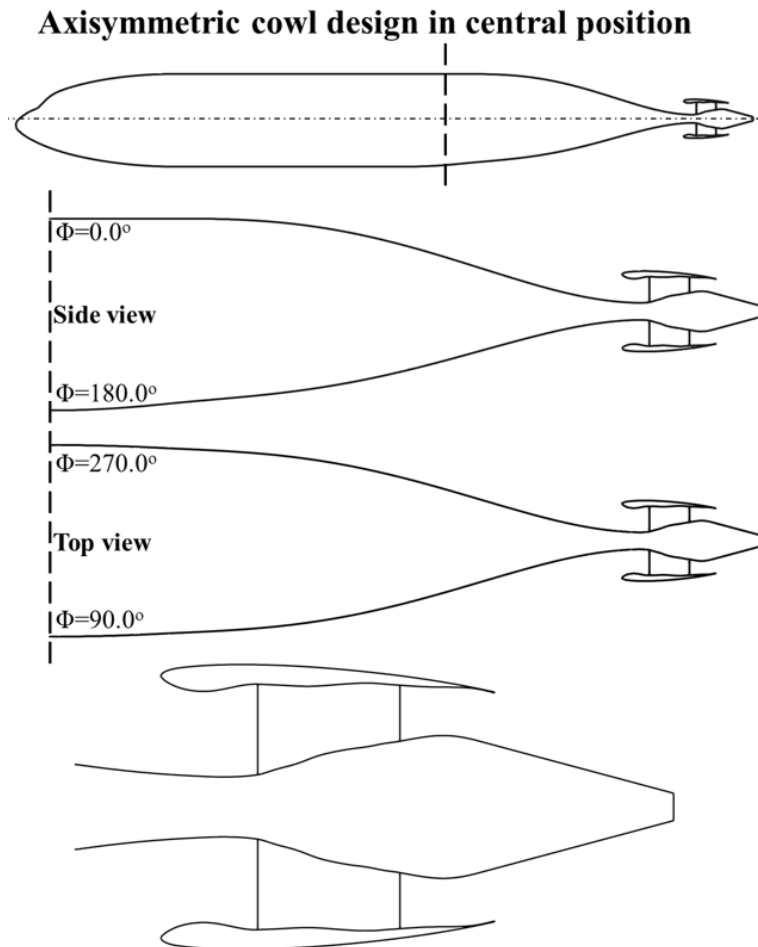


FIGURE 8.6: View of the baseline axisymmetric cowl positioned on the central axis of the DLR-F6 airframe

All configurations (Tab. 8.1) were run at nominal operating conditions for the scaled DLR-F6, with an operating altitude of 32,000 feet and a freestream Mach number of 0.75. The aerodynamic performance of the 3D configurations is performed at the nominal angle of attack for the target lift coefficient ($C_L = 0.5$). For the DLR-F6 at the reference operating conditions, this angle of attack is always obtained between 0 degrees and negative 0.5 degrees.

8.2.1 Comparison of the aerodynamic performance

When the thrust contributions (TS) of the 2D axisymmetric and 3D models of the cowl are compared (Tab. 8.1), a monotonic increase of the TS is observed with the increment on the model complexity (Fig. 8.7). This means that the application of a lower fidelity model (i.e. a 2D axisymmetric model or a one-dimensional fan model) can under predict the total thrust contribution of the aft-mounted propulsor. A maximum rise of 0.032 units in TS is observed between the 3D with the through-flow case (case D) and the 2D axisymmetric case with the one-dimensional semi-coupled propulsor (case A). Between approaches with the same fan model (one-dimensional or through-flow), the differences due the dimensionality of the problem is also important. The 3D configurations predict a higher TS than the 2D axisymmetric, with a difference of 0.024 units in the cases with one dimensional fan (A and C) and a difference of 0.02 units of TS in the cases with the through-flow fan model (B and D).

At the same time, for the same type of geometric configuration, the thrust contribution of the system increases for the through-flow models with respect to the one-dimensional approaches. In the 3D cases (C and D) the TS increases in 0.0084 units with the through-flow approach with respect to the one-dimensional fan model. The difference increases in the 2D axisymmetric cases, with an increase of 0.012 units with the through-flow fan model in comparison with the 2D axisymmetric case.

One of the main reasons behind the increase of TS with the through-flow models is the change of the local pressure forces at the aft-fuselage and the intake due to the differences in the propulsor model (Fig. 8.8). The semi-coupled one dimensional approach is modelled with a variable static pressure boundary to match the fan mass flow requirements without any influence of the downstream stage operation. On the other side, the through-flow model changes the local pressure gradients on the fan based on the geometry and operating conditions of the fan (relative velocity and mass flow rate). The differences between both approaches result in notable

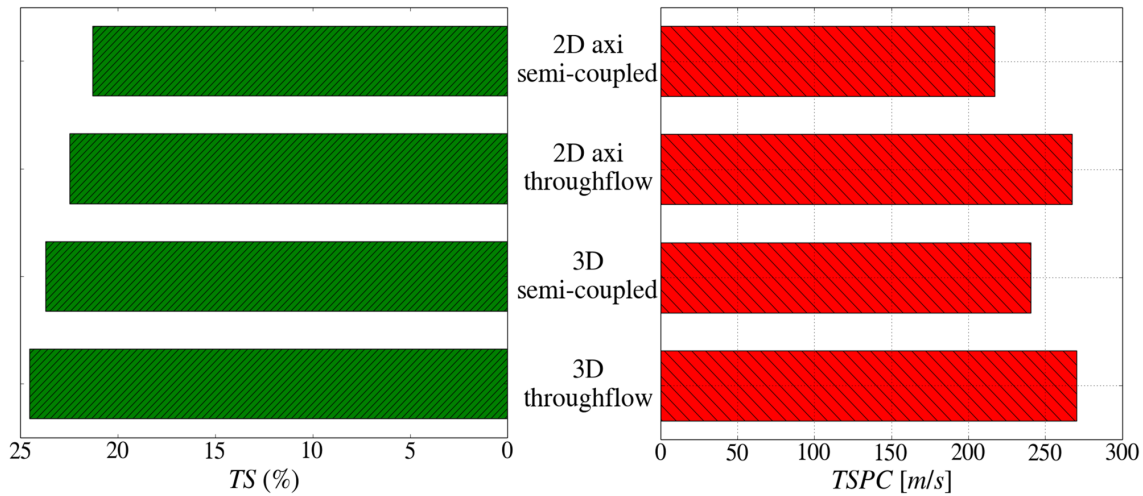


FIGURE 8.7: Effect of the model on the TS and $TSPC$ prediction ($M_\infty = 0.75$, $h = 32000\text{ft}$)

variations of the pressure gradients in the aft fuselage and the intake (Fig. 8.8). This produces local changes on the force contributions of the aft-fuselage and the intake that affect the final TS value. In addition, for the 3D configuration with the through-flow fan approach, the fan induces a rotation of the flow within the intake duct in the proximities of the blade (Fig. 8.8). This rotation also modifies the local pressure distributions in the proximities of the fan and introduces a lateral force component to the aft-mounted propulsor. This lateral force is expected to affect the vehicle dynamics and will require to be accounted in early design stages to maintain the aircraft's control and stability¹⁸⁴. Nevertheless, the analysis of this lateral force and any change on the flight dynamics is considered beyond the scope of the present work.

In addition to the changes on the local pressure forces in the intake duct, the fan model also affects the gross propulsive force of the system (Fig. 8.9). These changes have also a significant influence on the final TS prediction. The modified gross propulsive force (GPF^*) increases with the application of the through-flow fan model for the 2D axisymmetric and the 3D approaches. In this case this is attributed to a combination of two factors. First, the change in the exhaust operating conditions due to the loss of fan efficiency in the case with the through-flow model. This results in higher total temperature values at the exhaust duct for similar fan nozzle pressure ratios. These changes produce higher discharge velocities. This leads to an increase in the ideal propulsive force (Fig. 8.9). The second factor that contributes to the change on GPF^* is the reduction of the friction force in the exhaust walls. This is

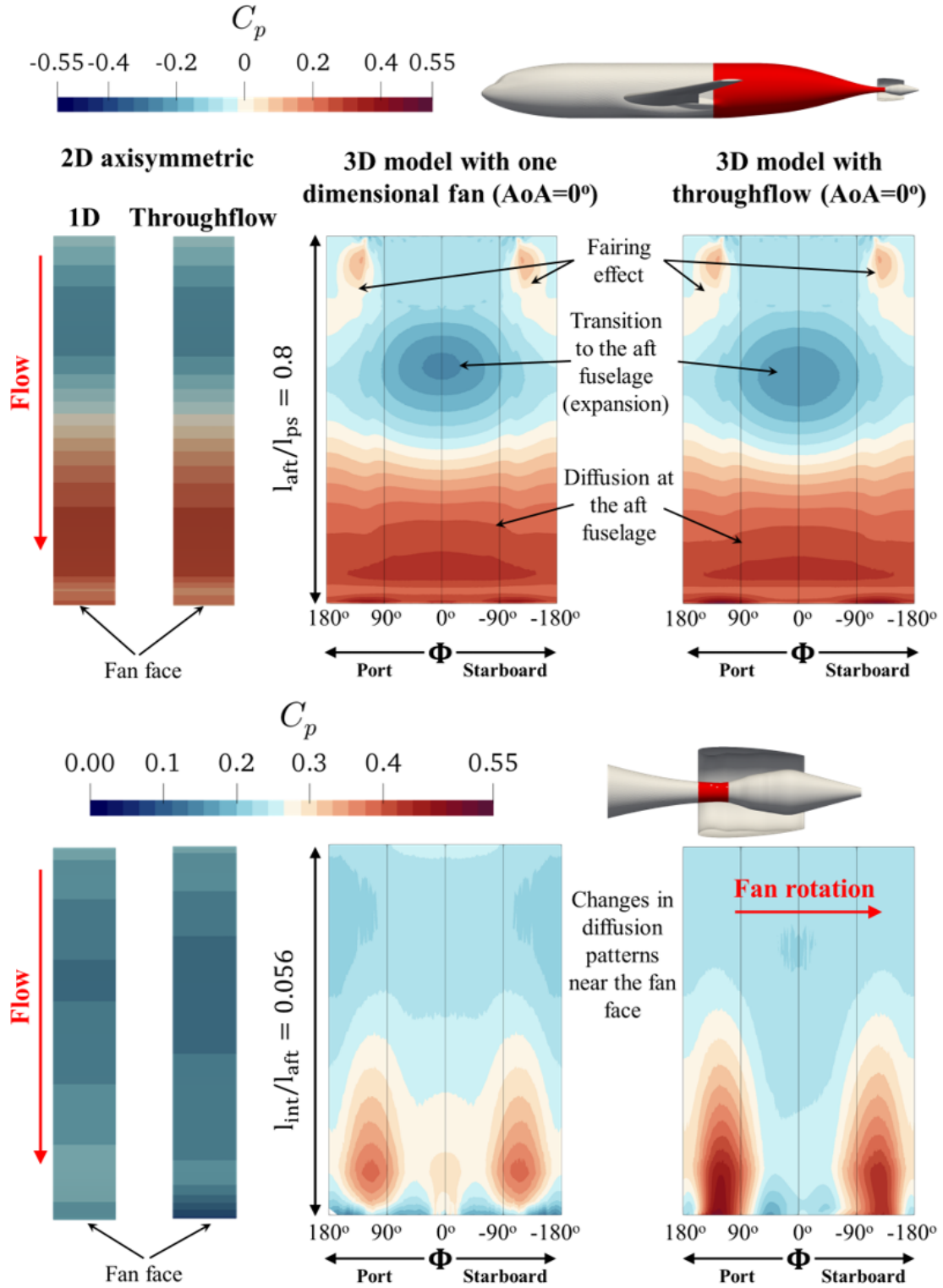


FIGURE 8.8: Pressure coefficient over the aft fuselage with detail of the intake duct region ($M_\infty = 0.75$, $h = 32000\text{ft}$). Surfaces unwrapped from the engine central axis. Axial coordinate normalised with the total length

due to the change on the mass flow distributions at the OGV outlet⁹⁹ in comparison with the one dimensional fan model with uniform flow conditions at the exhaust inlet (Figs. 8.10-8.11).

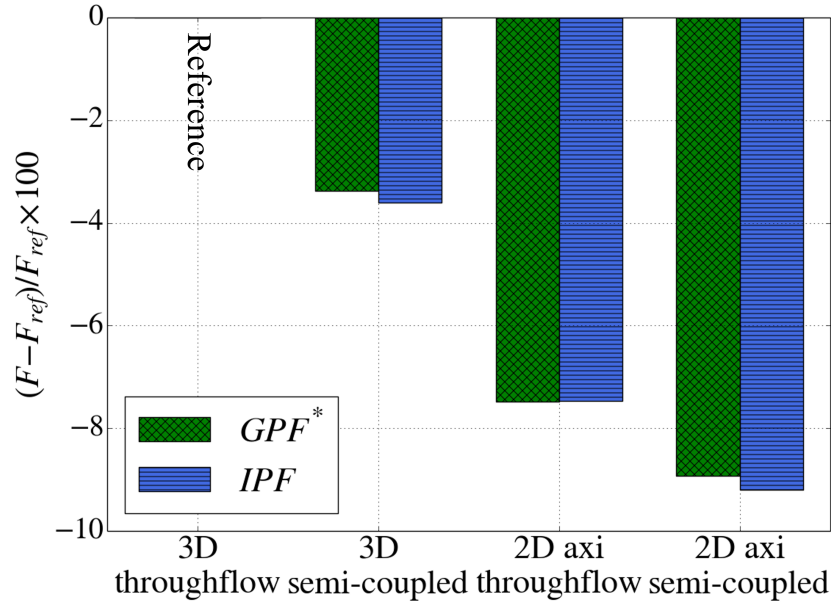


FIGURE 8.9: Comparison of the modified gross propulsive force GPF^* and the ideal propulsive force IPF for the different fan models and dimensions ($M_\infty = 0.75$, $h = 32000\text{ft}$). Values referenced to the 3D configuration with through-flow fan model

While the thrust contribution increases with the model complexity, the actual propulsive efficiency of the system is reduced substantially (Fig. 8.7), with an increase in $TSPC$ of approximately 50 m/s ($\sim 24.5\%$) in the 3D configuration with the through-flow (D) with respect to the 2D axisymmetric case with one dimensional fan (A). Also, the application of the through-flow fan model results in a higher predicted value of the Thrust Specific Power Consumption compared with the one-dimensional fan model for the 2D axisymmetric and 3D configurations. This is due to a more realistic definition of the loss of isentropic efficiency due to the distortion transfer. The NASA Rotor 4²³ configuration is not designed to maintain the nominal performance under the continuous radial and sectoral distortion operation of the aft mounted propulsor. Thus, notable reductions on the isentropic efficiency are to be expected. At the same time, the one-dimensional model was defined to operate under a nominal polytropic efficiency value (η_{poly}). This is expected to be more representative of the future generations of distortion tolerant fans⁷⁸. The design and evaluation of distortion tolerant fans is out of the scope of this project. However, for future work it is recommended to apply a more representative fan stage for this aerodynamic

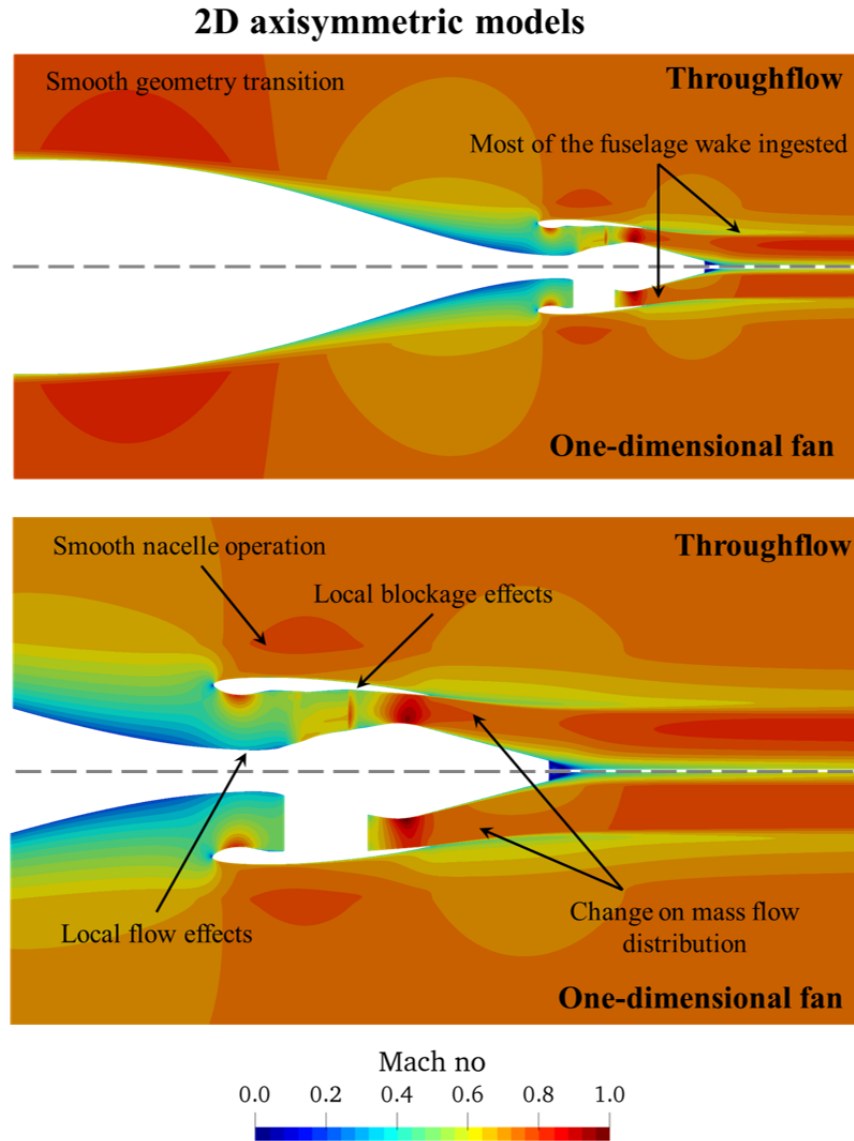


FIGURE 8.10: Mach number distributions over the 2D axisymmetric aft mounted annular propulsor design ($M_\infty = 0.75$, $h = 32000\text{ft}$). Upper figure: View of the flow at the aft-fuselage and power plant (through-flow fan on top, one-dimensional fan on bottom). Lower figure: View of the flow over the power plant (through-flow fan on top, one-dimensional fan on bottom)

requirements. This will reduce the TS_{PC} values for a given TS and will increase the power savings derived from the aft-mounted propulsor (Chapter 4).

In addition to the differences on the predicted TS with the two fan models, a notable increase in TS (~ 0.02 units) is observed for the 3D configurations compared with the 2D axisymmetric cases (Fig. 8.7). This has been attributed to the differences in the reference airframe. The 2D axisymmetric models are based on a baseline axisymmetric fuselage without any azimuthal change in the boundary layer

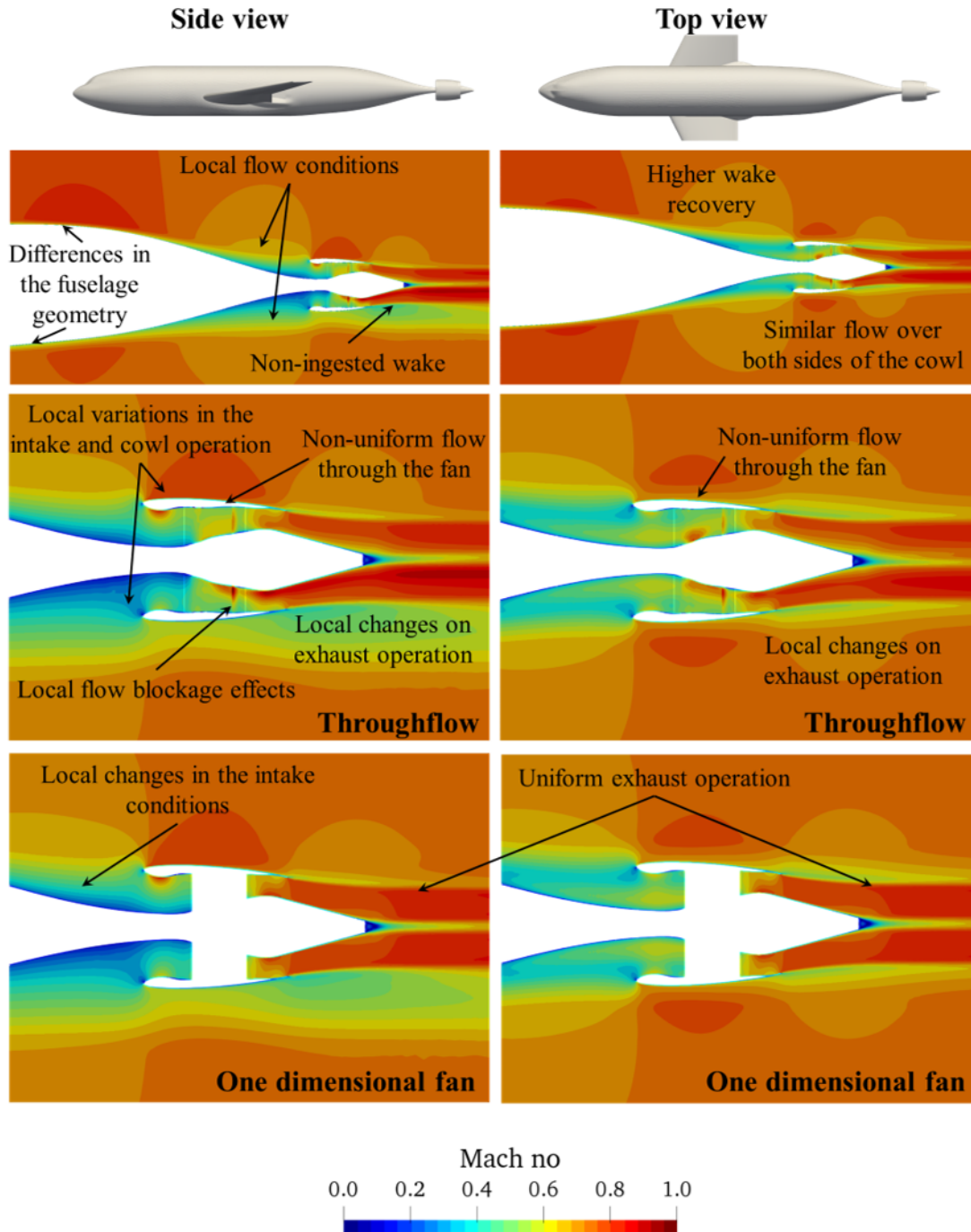


FIGURE 8.11: Mach number distributions over the three-dimensional aft mounted annular propulsor with axisymmetric cowl design ($M_\infty = 0.75$, $h = 32000\text{ft}$) and angle of attack 0°

distribution. However, the DLR-F6 geometry is composed by a more representative airframe that also includes the wings. This implies that there is a higher potential for energy recovery on specific regions of the wake with higher concentrations of the momentum flux deficit (Fig. 8.3). In addition, the aft mounted propulsor not only

compensates the kinetic energy defect produced by the fuselage, but also includes some parts of the wing and wing-joint. Hence, even if the aft mounted propulsor was mounted in a position in which the fuselage wake is not fully ingested, there could still have a higher margin for drag reduction when a more representative fuselage that includes non-axisymmetric shapes such as the belly and the wing junctions is applied.

Nevertheless, the existence of regions of higher momentum flux deficit in the wake of the aircraft means that the achievable TS could be highly conditioned by the propulsor location. In this case with the 3D centred positioned engine, there is a significant variation of the amount of the fuselage wake ingested from the top region of the airframe in which most of the wake is ingested to the bottom of the aircraft, where a significant part of the wake does not enter the intake duct (Fig. 8.11). The effects of the propulsor location relative to the fuselage wake are studied with more attention in Section 8.3.

Even in this case with the aft-mounted propulsor placed on the central axis of the fuselage, significant flow non-uniformities are observed over the housing components of the propulsion integration (Fig. 8.11). These changes are a consequence of the non-axisymmetric design of the airframe. This results in local changes in operation of the intake and fan cowl based on the fuselage afterbody flow conditions. Under these conditions, the preliminary analysis with the 2D axisymmetric approaches (Fig. 8.10) are insufficient to represent the actual aerodynamic requirements of the integration of the aft mounted propulsor, and they underpredict the TS in approximately 0.02-0.024 units when compared with 3D models. Furthermore, an axisymmetric cowl design applied to a three-dimensional aircraft is insufficient to fit the requirements of the flow in the fuselage wake, where the flow conditions over the propulsor housing varies azimuthally. For this reason, it is necessary to move towards more complex non-axisymmetric design approaches for the housing components and adapt the propulsion integration aerodynamic design to the local flow features. To evaluate better the design requirements, the possible benefits of the application of a bespoke non-axisymmetric propulsion integration design are evaluated in Section 8.5.

8.2.2 Comparison of the exhaust performance

The local flow conditions at different azimuthal coordinates also affect the operation of the fan for the through-flow model (Figs. 8.11-8.11). The changes in the mass

flow distributions across the stage result in a non-uniform exhaust operation with azimuthal variations in the jet characteristics. In addition to this, the local variation of the mass flow on the fan produces significant azimuthal variations on the blockage effects at the OGVs (Figs. 8.11-8.11). This increases the Mach number locally (e.g. in the bottom line of the fan in Fig. 8.11). This produces a rise on the viscous dissipation and total pressure losses at the OGVs with a penalty in the fan isentropic efficiency ($\sim 8\%$ from the nominal value of the NASA R4²³). In contrast, the 3D model with the one-dimensional fan presents a uniform exhaust operation across the different azimuthal lines. In combination with the uniform inlet boundary conditions for the total pressure and temperature, this misrepresents the system's exhaust and jet operation with changes on the ideal propulsive force and gross propulsive force (Fig. 8.9) with approximately a 3% difference between the through-flow (B and D) and the one-dimensional cases (A and C).

The effect of the fan model on the exhaust performance can be observed when the velocity and discharge coefficients of the different models are compared (Fig. 8.12). The velocity and discharge coefficients are evaluated based on the 3D configuration with the through-flow fan model as a reference ($C_{d_{ref}}$ and $C_{V_{ref}}^*$). Small variations on the modified velocity coefficient (C_V^*) are observed between the different fan approaches. The modified velocity coefficient is greater with the one-dimensional fan models in comparison with the through-flow approaches (approximately 0.25-0.33%). This means that while the GPF^* increases with the through-flow approach (Fig. 8.9) the losses in the exhaust duct are higher with respect to the ideal propulsive force value. Thus, to have a more accurate definition of the exhaust performance it is required to include a more representative evaluation of the fan stage and the transfer of the intake distortion through the stage.

The discharge coefficient (C_d) decreases with the complexity of the model, with higher predicted values for the 2D axisymmetric model with the semi-coupled one dimensional fan (Fig. 8.12). This is a consequence of the simplification of the exhaust operation in these models. The semi-coupled models do not include the azimuthal variation of the exhaust properties and mass flow distributions. This results on a higher uncertainty on the predicted discharge efficiency of the exhaust duct. The predicted exhaust suppression is lower for the 2D axisymmetric configurations, with approximately a 3% reduction in the C_d . The fuselage wake (Fig. 8.11) produces an azimuthal variation of the static pressure at the discharge plane of the exhaust. This has some implications for the exhaust design, since the definition of the aft mounted propulsor location relative to the fuselage wake could produce significant

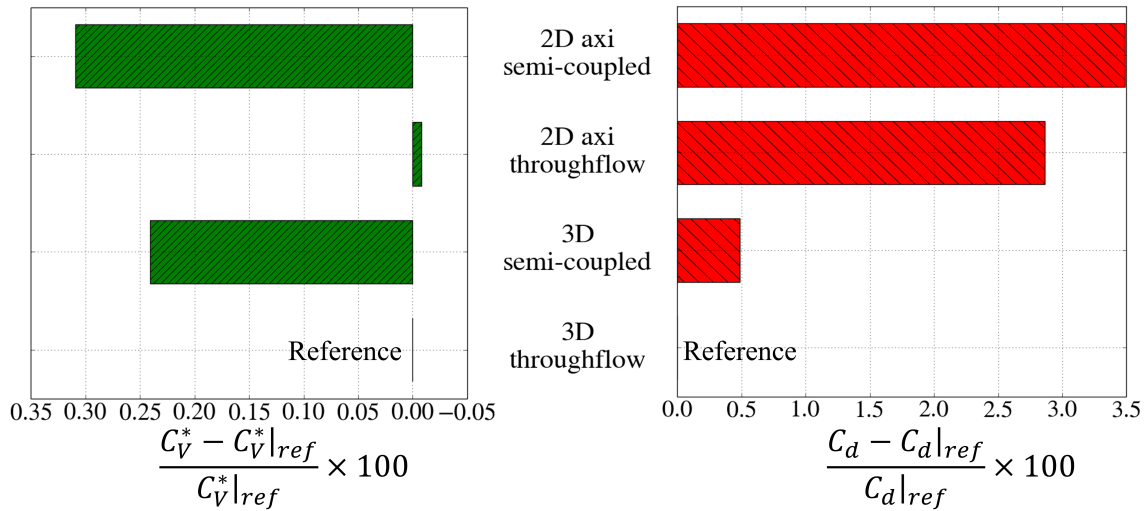


FIGURE 8.12: Effect of the dimensionality and fan approach in the prediction of the exhaust performance ($M_\infty = 0.75$, $h = 32000\text{ft}$). Values referenced to the 3D through-flow configuration

changes on the exhaust operation. This will require to correct the exhaust dimensions to account for the mass flow losses from the original design value¹⁵. Also, this could result in notable changes in the exhaust performance with the increase of the angle of attack. Under these circumstances, more advanced exhaust design approaches may be required. For example, future research may include the application of non-axisymmetric exhaust systems¹²² to adapt the exhaust design to the local flow conditions of the aft-mounted propulsor.

8.2.3 Comparison of the intake performance

In previous chapters, the information about the intake performance obtained with the 2D axisymmetric models was quite limited in comparison with the 3D approaches. The 2D axisymmetric approaches are constrained to the evaluation of the radial distortion (RDI) and the total pressure recovery from either the free-stream value (IPR) or a reference station of the fuselage (IPR_{ing}). For the 3D cases, the presence of sectoral distortion requires the evaluation of the circumferential distortion (CDI) and peak sectoral distortion (DC_{60}). The different fan models are also expected to have some influence^{92,93} on the prediction of the intake performance under high distortion. Under these conditions, the induced rotation of the flow near the fan and the pressure gradients prescribed by the mass flow radial and azimuthal distributions could affect the prediction of the intake performance values. To understand the differences in the predicted intake performance between

the different models, the available performance metrics for each configuration are compared (Fig. 8.13). As discussed in Chapter 3.3.6, in 3D configurations with non-uniform flows over the fuselage is more convenient to evaluate the IPR than the IPR_{ing} . Since both metrics are analogous, the IPR is evaluated in this chapter.

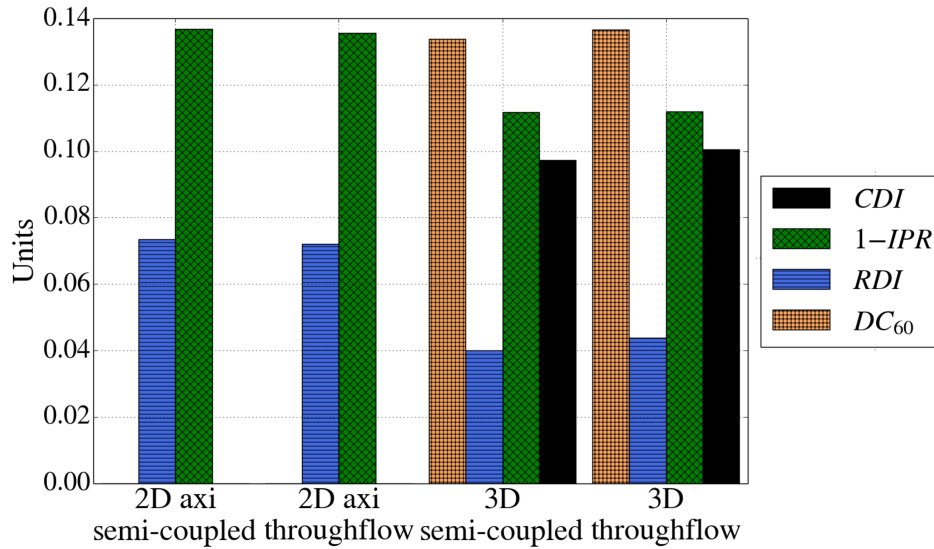


FIGURE 8.13: Effect of the model on the intake performance prediction ($M_\infty = 0.75$, $h = 32000\text{ft}$) at nominal angle of attack ($C_L = 0.5$)

The 2D axisymmetric configurations (A and B in Tab. 8.1) have similar values for both RDI and IPR . Very small differences of approximately 0.002 units are observed on the RDI and intake total pressure losses ($1 - IPR$), with higher values observed for the one-dimensional fan model. These variations of RDI and IPR are not significant given the values of the metrics, and they do not have a notable effect on the fan performance¹⁷⁷. This is consistent with the observations of Chapter 7.1.2. Similarly, the 3D configurations (C and D in Tab. 8.1) maintain consistent values on the IPR and RDI between them independently on the fan model (Fig. 8.13). Thus, the fan model does not have a significant effect on the prediction of the intake performance for the same type of configuration (2D axisymmetric or 3D).

However, when the 2D axisymmetric and the 3D configurations are compared some notable changes on the RDI and IPR are observed (Fig. 8.13). The RDI decreases from ~ 0.072 to ~ 0.4 in the 3D configurations in comparison with the 2D axisymmetric cases. This is a notable variation since the radial distortion is reduced approximately a 45% to more bearable limits (i.e. below 0.05¹⁷⁷). Similarly, the IPR increases in approximately 0.024 units for the 3D configurations in comparison

with the 2D axisymmetric cases. Hence, while the fan model does not have a notable effect on the predicted intake performance, the type of configuration does have a major impact on the prediction of the total pressure losses and radial distortion. For this reason, for the evaluation of the intake distortion of embedded systems a more representative fuselage model with 3D characteristics is recommended.

The changes on IPR and RDI is a consequence of the sectoral distortion in the 3D cases (Fig. 8.14). The non-uniform wake of the fuselage results in regions of the propulsor intake exposed to higher velocity flows where the total pressure losses at the boundary layer are less significant. This increases locally the total pressure values at the AIP with an rise on the average total pressure value, at the expense of the increase in sectoral distortion. This also reduces the annular average of the total pressure at different radial stations (Chapter 3.3) and results in lower radial distortion values in comparison with the 2D axisymmetric cases.

In the 3D configurations, the fan model has a minor effect on the predicted sectoral distortion values (Fig. 8.13). The through-flow approach predicts slightly higher values of RDI and DC_{60} , with an excess of approximately 0.003 and 0.004 units, respectively. These differences are not significant under the current values of distortion. Also, qualitatively the differences in mass flow distribution and sectoral distortion between both fan approaches show a good agreement (Fig. 8.14). This means that under specific conditions the semi-coupled one dimensional fan model could be used to model the intake operation. This could be used to cut in half the computational cost of the simulations, since the one dimensional fan maintains the aircraft symmetry. Nevertheless, this test has only been evaluated under cruise conditions. More demanding conditions, such as maximum take-off and crosswind operation will require further analysis. However, the detailed evaluation of off-design conditions were considered beyond the scope of this work. Future research may be focus on the effect of these off-design conditions on the intake performance of the embedded propulsion systems.

If the distortion levels are compared with references in the open literature, while the DC_{60} values are high (approximately 0.13) for a continuous operation of the intake, they are still below the 0.2 limit value for civil intakes and the 0.9 value for military configurations¹⁸⁵. The CDI with a value of approximately 0.1 units is on the limit of operation of conventional civil aero-engines¹⁸⁶. Both metrics imply that the fan operates under significant levels of distortion during most of the cruise stage. This will require the development of distortion tolerant fan stages that could perform

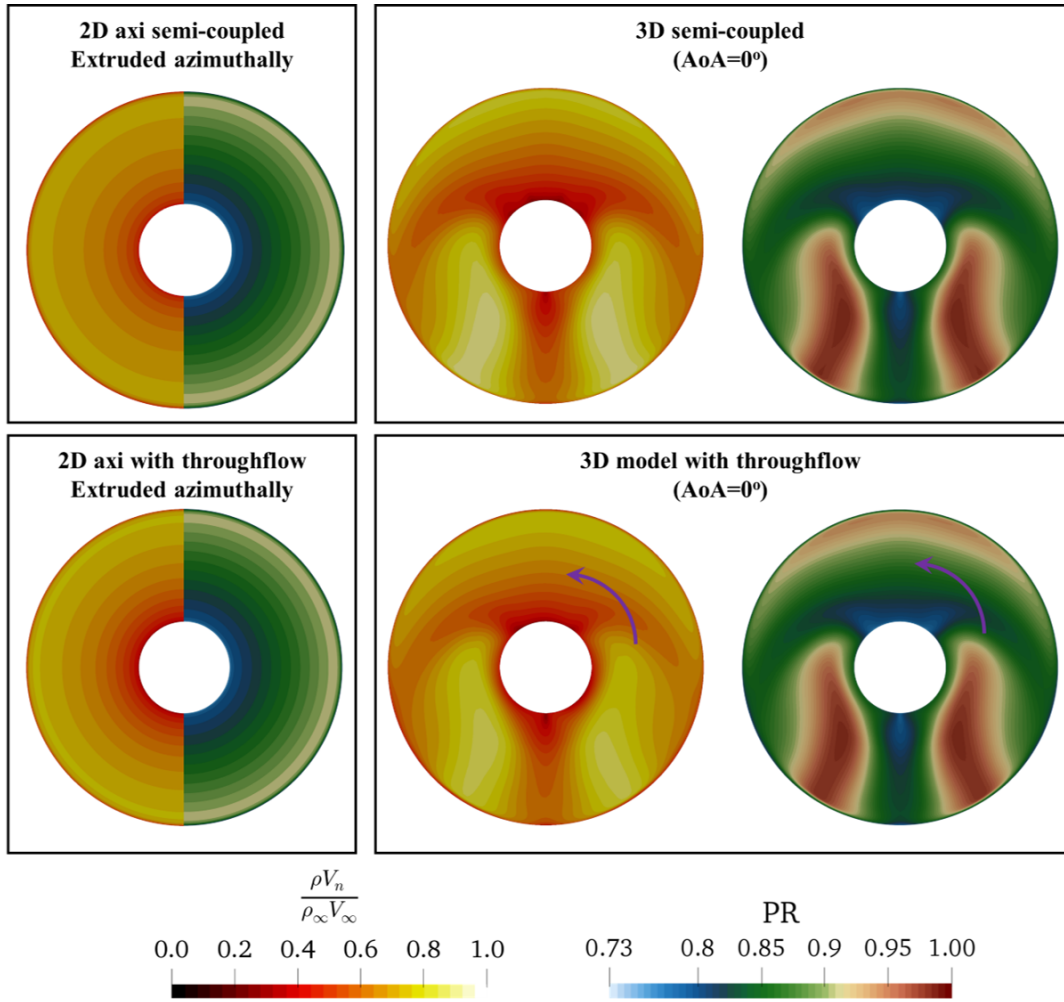


FIGURE 8.14: Non-dimensional mass flow and pressure ratio distributions at the AIP for the 2D axisymmetric case and the 3D flow with axisymmetric aft mounted propulsor ($M_\infty = 0.75$, $h = 32000\text{ft}$)

under these conditions. To the author's knowledge, there are no current open literature references on the limit distortion requirements for such fans. Nevertheless, it is still in the interest of the designer to reduce these distortion values with the intake design.

8.2.4 Influence of the number of outlet guided vanes

The relative position of the 3D propulsor with respect to the fuselage wake was observed to affect the local operation of the turbomachinery (Fig. 8.11). This lead to local rises on the Mach number over the outlet guided vanes which could result in higher isentropic efficiency loss in the fan stage. In particular, at the bottom line of the propulsor ($\Phi = 180^\circ$) the local Mach number in the vane exceeds 0.9

near the hub (Fig. 8.11). These near choke conditions are expected to penalise the performance in later studies. Thus, before moving forward, the local Mach number at the OGV row should be reduced.

This could be addressed with the redesign of the fan stage to increase the distortion tolerance. However, this is a complex process that was considered beyond the scope of the present work. Nevertheless, the acceleration of the flow at the OGVs could be partially addressed with a reduction of the number of vanes. This would reduce the blockage effects at the vanes and diminish the friction losses. This reduction of the number of vanes is easily achievable with the Hall-Thollet through-flow model¹⁷, where only a correction of the blockage factor is required.

To understand the possible benefits of the reduction of the number of OGVs, one test is proposed from the same 3D geometry baseline. From the reference Rotor 4 configurations with 54 vanes, other three configurations are proposed with the same geometries. The first configuration with 36 vanes leaves a 10° space between the vanes. The second configuration has the same number of vanes than fan blades. Finally, the third one presents a more extreme case with only 10 vanes. These numbers are an arbitrary definition of different design possibilities. However, the aim of this test is not to identify optimal vane configurations, but to find a configuration for later studies in this chapter with lower viscous losses at the OGV.

For this configuration, an increase in the stage isentropic efficiency (η_{is}) is observed (Fig. 8.15). The increase in η_{is} is inversely proportional to the number of vanes. This is a direct consequence of the reduction of the blockage effects in the OGV row. At the same time, no significant increase on the stage FPR is observed from the reference, with a maximum increase of less than 1% from the baseline in the configuration with 10 vanes. Under these conditions, the total temperature at the exhaust decreases, and so does the exhaust outlet velocity and the GPF^* . Hence, while the isentropic efficiency of the stage increases a $\sim 7.5\%$ with the reduction of the number of vanes to 10, the TS decreases ($\sim 14\%$). However, for the cases with 22 and 36 vanes, the $TSPC$ also decreases ($\sim 1 - 2.5\%$) with respect to the baseline. This means that while the thrust contribution of the aft mounted propulsor decreases, the propulsive efficiency increases. This leads to savings on the overall power consumption of the vehicle (Chapter 4).

The reduction of the number of vanes affects the radial and azimuthal distribution of mass flow across the fan stage. This has a direct effect on the intake performance. The total pressure recovery (IPR) remains almost unaltered between the different configurations (Fig. 8.16). However, the radial distortion and the sectoral distortion

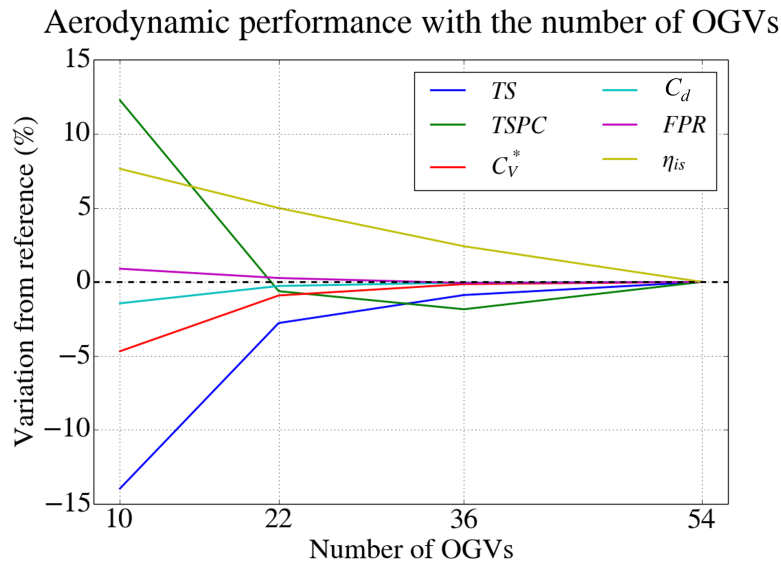


FIGURE 8.15: Effect of the number of OGVs on the aerodynamic performance for an axisymmetric intake and cowl in cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$) and nominal angle of attack ($C_L = 0.5$)

are affected by the change on the mass distributions with the number of outlet guided vanes. The DC_{60} decreases substantially with the reduction of the number of vanes in comparison with the reference, with a maximum difference of approximately 2.4% of the original value with 22 vanes. The RDI has a more modest reduction with a maximum reduction of approximately 0.4% with respect to the baseline in the case with 36 vanes.

Finally, the case with 36 vanes is selected for the rest of the configurations of this chapter. This configuration provides the best compromise between TS loss and $TSPC$ reduction, while also provides some benefits on the intake performance. This configuration also presents a lower variation of the exhaust performance in comparison with the baseline (Fig. 8.15).

Future work should include a more detailed analysis of the aerodynamics of the fan stage. The application of a distortion tolerant fan stage is highly recommendable to have a more representative definition of the overall system performance. Another possible solution could involve the application of non-axisymmetric OGVs rows as it was proposed by Hall¹¹¹. This would facilitate the mass flow redistribution in the fan stage and could be used to adapt the fan operation to the local flow conditions defined by the fuselage wake. In addition, another possibility of interest is the application of non-axisymmetric exhaust systems¹²² to reduce the exhaust suppression.

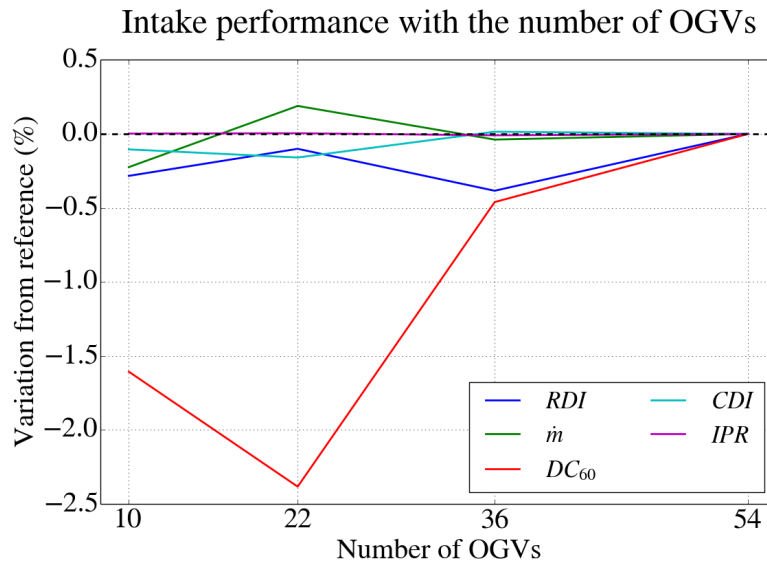


FIGURE 8.16: Effect of the number of OGVs on the intake performance for an axisymmetric intake and cowl in cruise conditions ($M_\infty = 0.75$, $h = 32000\text{ft}$) and nominal angle of attack ($C_L = 0.5$)

The local changes in the discharge area of the exhaust could also improve the control on the mass redistributions at the fan stage and reduce the loss in fan efficiency.

8.3 Impact of the aft mounted propulsor location

The position of the aft-mounted BLI propulsor relative to the fuselage wake is expected to have a significant effect on the aerodynamic performance of the propulsion integration. In some regions of the wake (Fig. 8.3), there is a higher concentration of momentum flux deficit. Thus, at some locations there is a higher wake recovery potential for the BLI system. At the same time the momentum distributions at the wake will modify the local static pressure and the discharge conditions of the exhaust duct. This will change the operating requirements of the unchoked exhaust and its performance. In addition, the local momentum distributions at the wake will also condition the level of flow distortion at the intake. For these reasons, it is of interest evaluate the effects of the location on the aerodynamic performance of the integration of the BLI propulsor.

At the operating conditions of the DLR-F6¹⁹, the regions with higher concentration of momentum flux deficit are found at lower vertical positions with respect to the fuselage belly. However, as it was discussed in Section 8.1.2, the position of the propulsor is limited by the ground clearance requirements in take-off conditions. This constraint defines the achievable wake recovery of the system, since the position of the propulsor may be restricted to some regions with lower concentration of momentum flux deficit. Nevertheless, the study of some positions of the propulsor that do not respect the ground clearance requirements is still of interest to understand all the requirements of the propulsion system.

To study this effects, an exploration of the impact of the propulsor location in the aerodynamic performance is proposed (Fig. 8.17). This exploration is based on 12 positions of the propulsor. This study covers three vertical positions with respect to the fuselage ($r/r_{fus} = 0 - 0.45 - 0.75$) and four axial positions ($l_{aft}/l_{ps} = 0.6 - 0.8 - 1.0 - 1.2$). Given the geometry design constraints, only three locations of the study (L1, L2 and L4) satisfy the ground clearance angle and length requirements. The remaining locations are applied to understand the general impact of the propulsor position relative to the fuselage. The axisymmetric intake, cowl and exhaust design of the previous sections is applied for all the locations. The span of the fan remains constant for the different locations, based on the fixed hub-to-tip ratio of the NASA R4 ($httr = 0.3$). The study is evaluated at nominal operating conditions for the airframe (altitude of 32,000 ft, $M_\infty = 0.75$, and $C_L = 0.5$).

8.3.1 Impact of the position on the aerodynamic performance

The thrust contribution of the aft mounted embedded propulsor decreases when the propulsor is located at aft-most locations and higher vertical positions (Fig. 8.18). Between both changes in geometry, the change in the vertical position of the aft-mounted propulsor has a more notable effect in the TS . For example, from the baseline presented in the previous section (Fig. 8.17) in location L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$), to a higher vertical position in location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) there is a reduction in TS of 0.046 units. At the same time, the increase in axial location of the embedded propulsor from L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$) to L12 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 1.2$) penalises the TS of the design in 0.025 units. The penalties derived from the vertical position have important consequences in the design of the embedded propulsive system, since the location L4 is one of the few locations that respect the geometry constraints of the airframe (Section 8.1.2). This

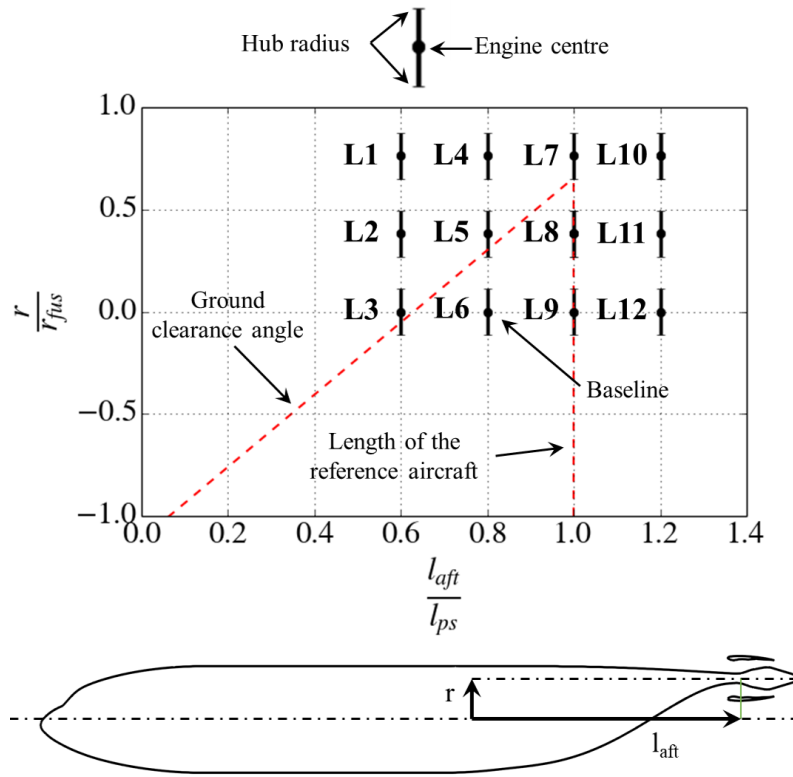


FIGURE 8.17: Definition of the different locations tested for the aft-mounted propulsor on the DLR-F6 aircraft

implies that the achievable TS is highly constrained by the location of the propulsor with respect to the wake. In this context, the aerodynamic design of the propulsor integration could have a significant impact on the final TS values. This will be studied in more detail in Section 8.5.

The changes in $TSPC$ have a similar evolution with the r/r_{fus} and the l_{aft}/l_{ps} to the changes in TS . Higher axial and vertical positions of the aft-mounted propulsor results in notable penalties on the $TSPC$. For example, from the baseline presented in the previous section (Fig. 8.17) in location L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$), to a higher vertical position in location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) there is an increase in $TSPC$ of approximately 60 m/s. At the same time, the increase in axial location of the embedded propulsor from L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$) to L12 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 1.2$) increases the $TSPC$ of the design in approximately 25 m/s. These penalties reflect the loss in propulsive efficiency with the location. Based on the trends observed, increase on $TSPC$ (i.e. penalties in the propulsive efficiency) is conditioned by the TS values. Also, the $TSPC$ value is affected by a second order effect based on the change in the fan isentropic efficiency. The increase in the vertical position was found to increase the fan isentropic efficiency. This has

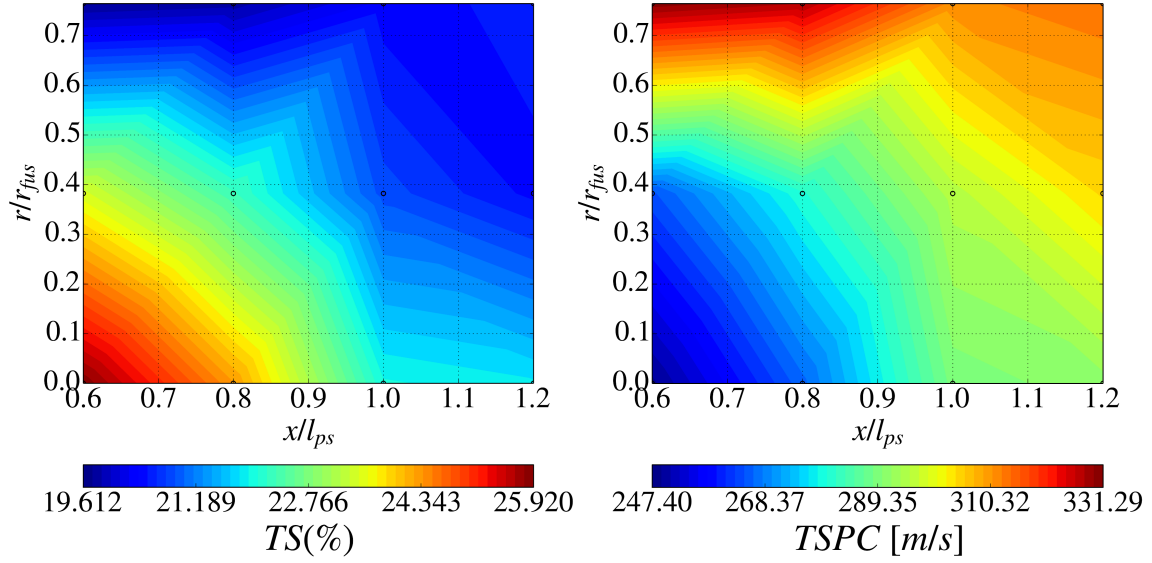


FIGURE 8.18: Effect of the propulsor position on the aerodynamic performance. Evaluation of the TS (left) and $TSPC$ (right) for an axisymmetric cowl design

been attributed to the changes in the intake pressure recovery (IPR). The intake performance will be analysed in more detail in Section 8.3.2.

To understand the influence of the vertical position on the system aerodynamics, it is of interest to evaluate the momentum flux deficit ingested by the aft-mounted propulsor (Fig. 8.19). At a central propulsor location ($r/r_{fus} = 0$) such as L6, most of the region of the wake with higher momentum flux deficit is ingested by the propulsor. This is due to a closer alignment of the flow with the intake position. Nevertheless, some sections of the wake with high momentum flux deficit concentrations are not ingested on the lower sections of the cowl. This is a consequence of the higher concentration of momentum flux deficit in the lower sections of the fuselage, being most of the wake originated in the fuselage belly and wing fairings (Fig. 8.3). At higher vertical propulsor positions such as L5 ($r/r_{fus} = 0.45$) and L6 ($r/r_{fus} = 0.75$), the extension of the non-ingested region of the airframe wake increases (Fig. 8.19). At the same time, the intensity of momentum flux deficit over the top section of the cowl decreases with the increase in r/r_{fus} . This means that while the lower section of the cowl ($\Phi = 180^\circ$) is deeply embedded on a lower momentum flow from the fuselage, the upper region of the cowl ($\Phi = 0^\circ$) is surrounded by higher velocity flows with a significantly lower impact of the fuselage boundary layer and wake.

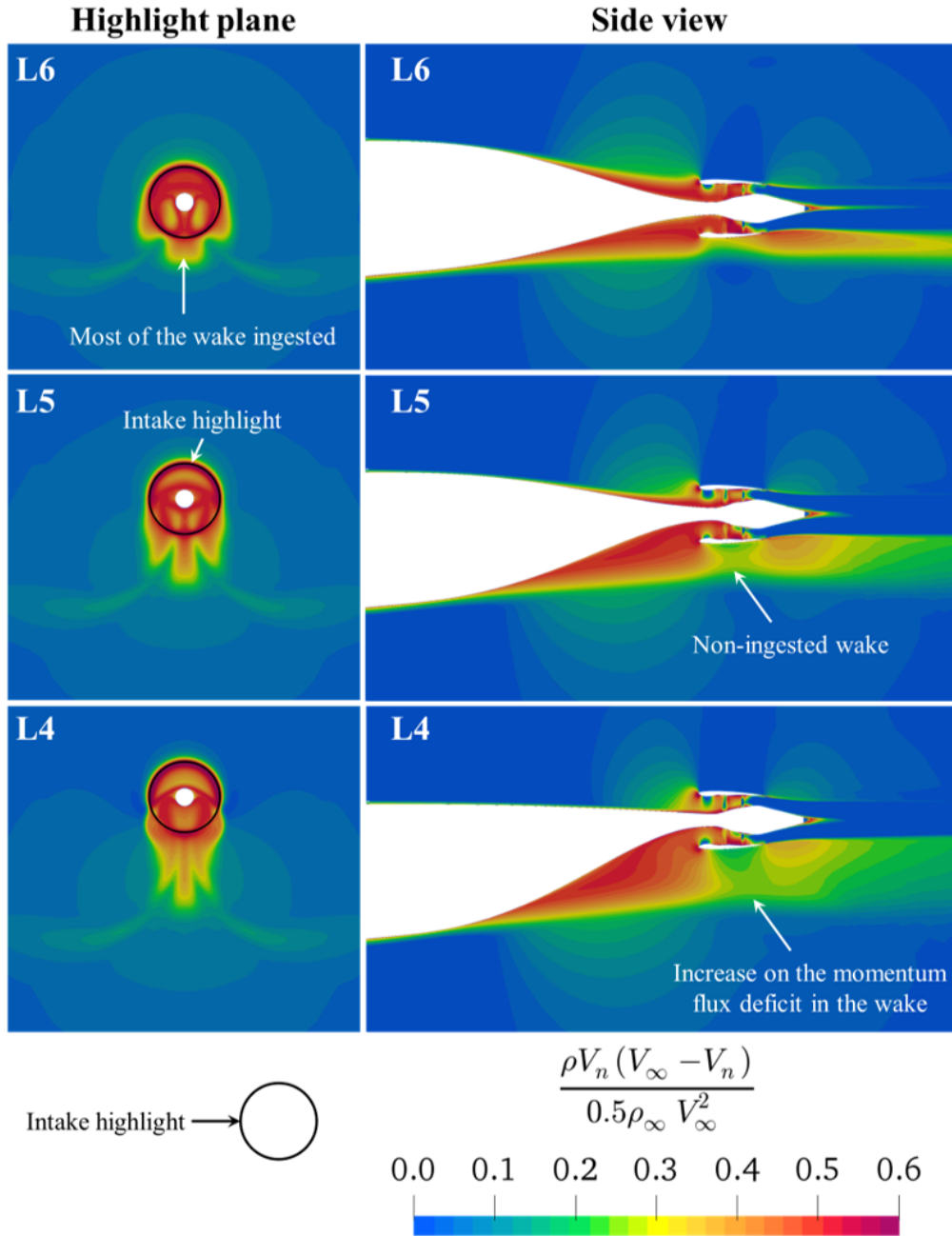


FIGURE 8.19: Changes on the momentum flux deficit with the change on vertical position of the propulsor. Evaluated in a plane tangent to the highlight, and in the symmetry plane of the airframe

The change in vertical position of the propulsor results in local variation of the propulsion integration aerodynamics (Fig. 8.20). This affects the design requirements of the propulsion integration. The increase in r/r_{fus} exposes the top region ($\Phi = 0^\circ$) of the fan cowl to the higher velocity flows from the freestream, while the bottom section operation is prescribed by the wake of the fuselage. At higher vertical positions such as L5 and L6, this results on an azimuthal variation of the

aerodynamic operation in which the top regions of the cowl ($\phi \sim 0^\circ$) are in transonic regime and the lower regions of the cowl ($\phi \sim 180^\circ$) in subsonic regime. This causes significant changes in the Mach number distributions over the nacelle forebody, with local accelerations between $\Phi = 90^\circ$ and $\Phi = -90^\circ$ (Fig. 8.20). At the same time, the change on local Mach number affects the diffusion pattern in the nacelle afterbody. This changes the nacelle contribution to the system drag and the operation of the exhaust due to the changes on static pressure at the trailing edge.

The increase on vertical location also requires significant changes on the aft-fuselage geometry. This affects the diffusion at the aft-fuselage surface (Fig. 8.20) which is displaced towards the lower section of the aft fuselage surface ($\Phi = 180^\circ$). The change on the diffusion pattern at the aft fuselage determines the forward force contribution of the aft fuselage and the TS . At the same time, the aft fuselage geometry shapes the azimuthal variation of the boundary layer. This defines the operating conditions and distortion levels at the intake and fan face.

While the reduction in TS with the propulsor vertical position could be traced to the reduction in the amount of momentum deficit ingested (Fig. 8.19), the effect of the axial location on the TS requires further attention. For example, for configurations at $r/r_{fus} = 0$ (e.g. L3, L6 and L9) the increase on l_{aft}/l_{ps} does not imply significant changes in the amount of momentum deficit ingested (Fig. 8.21). Furthermore, the increase in axial location results in a higher concentration of the momentum at the highlight plane. Hence, there must be another source for the loss in the thrust contribution from the aft-mounted propulsor with the axial location.

To understand the changes in TS with the axial location it is necessary to study the force contribution of the aft-fuselage surface to the TS (Fig. 8.22). The increase in the axial position of the propulsor requires an increase on the aircraft length. This results in an increase in the wetted surface of the aft fuselage. One of the consequences of this is the rise in the viscous drag. At the same time, the increase in the aft-fuselage length reduces the local slope of the aft fuselage surface. As an effect of this, there is a change in the diffusion of the flow at the aft fuselage in comparison for the positions with lower l_{aft}/l_{ps} (Fig. 8.22). Also, with the increase of the fuselage slope with lower l_{aft}/l_{ps} positions such as L3, the normal to the aft-fuselage surface in the axial component (n_x) increases. This, increases the pressure force contribution of the aft-fuselage to the overall propulsive force. Hence, the shorter aft-fuselage surfaces can increase the thrust contribution of the system by both mechanism, the reduction of viscous drag and the increase on the pressure forces in the thrust direction. This means that with an adequate aft-fuselage

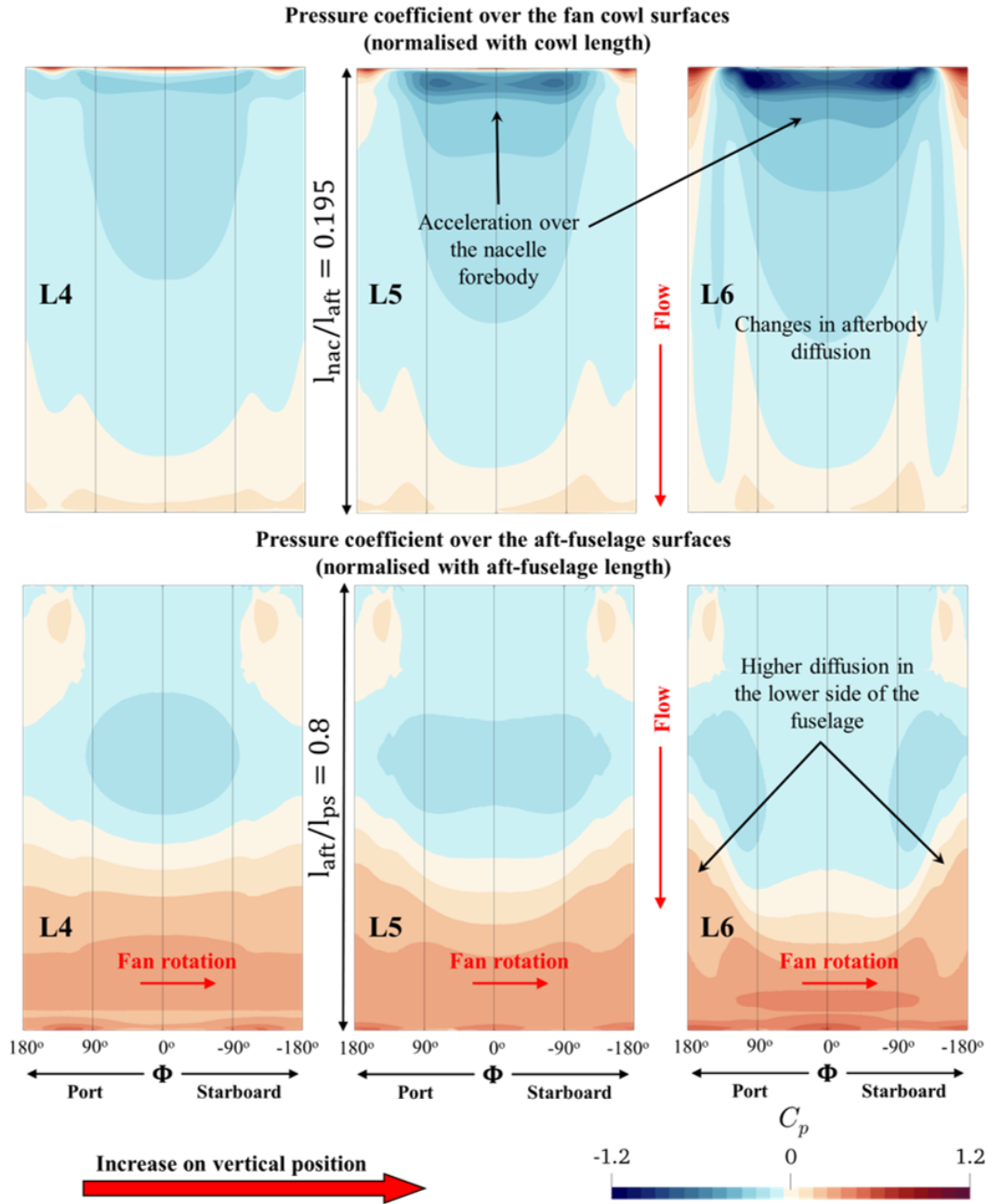


FIGURE 8.20: Changes on pressure coefficient over the fan cowl and aft fuselage surfaces with the change on the vertical position of the propulsor. Surfaces unwrapped from the central axis of the engine. Axial coordinate normalised with the total length of the surface

aerodynamic design, the kinetic energy losses in the fuselage wake can be reduced without significant changes in the amount of boundary layer ingestion. Thus, the aerodynamic design of propulsion integration can have a significant impact in the final TS values. In particular, the bulk geometric characteristics of the aft-mounted propulsor (aft fuselage length, propulsor location, cowl length...) are the major

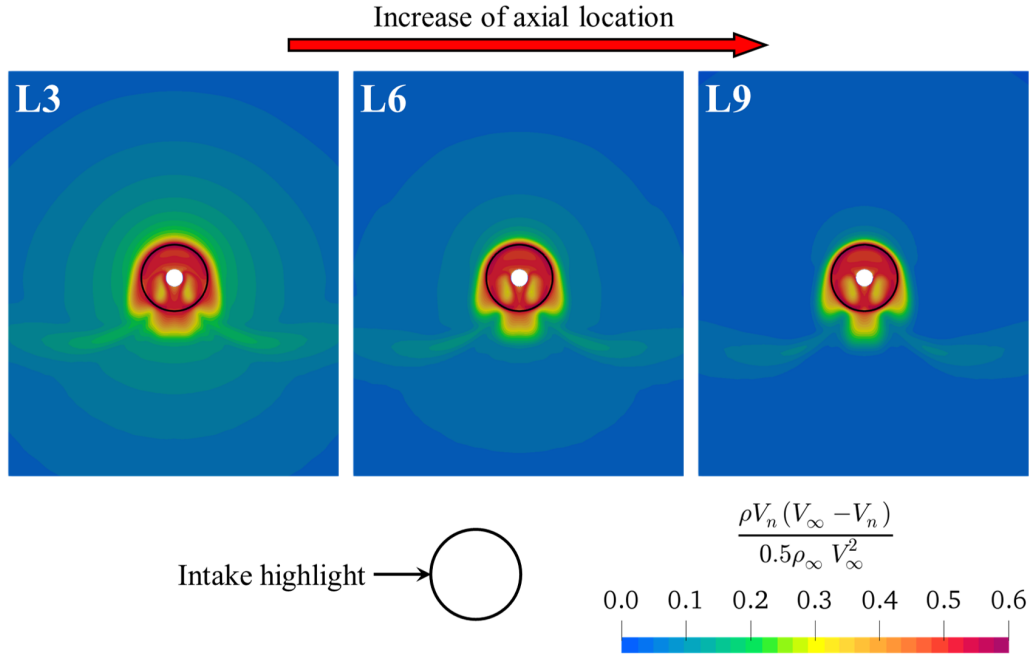


FIGURE 8.21: Changes on momentum flux deficit at the highlight plane with the change on the propulsor axial position

contributors to the changes in the thrust contribution of the aft-mounted propulsor.

8.3.2 Impact of the position on the intake performance

The vertical and axial position of the aft mounted propulsor requires of a modification of the aft-fuselage geometry. This is expected to influence the azimuthal distribution of the boundary layer over the aft fuselage and lead to changes on the intake operating conditions. Thus, in addition to the analysis of the thrust performance of the system, it is required to understand the effects of the propulsor location relative to the airframe in the aerodynamic performance of the intake (Fig. 8.23).

The intake pressure recovery (IPR) is observed to increase with r/r_{fus} and l_{aft}/l_{ps} (Fig. 8.23). For example, between the location L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$) and the location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) the IPR increases in 0.044 units. Also, based in location L6, the IPR increases by a maximum of 0.048 units at location L10 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 1.2$). At higher vertical locations, the top region of the intake ($\Phi = -45^\circ$ to $\Phi = 45^\circ$) is less embedded within the airframe boundary layer. Hence, it is exposed to higher velocity flows near to the freestream values. This increases the total pressure values at the fan face with respect to more embedded configurations at lower vertical and axial positions (Fig. 8.24).

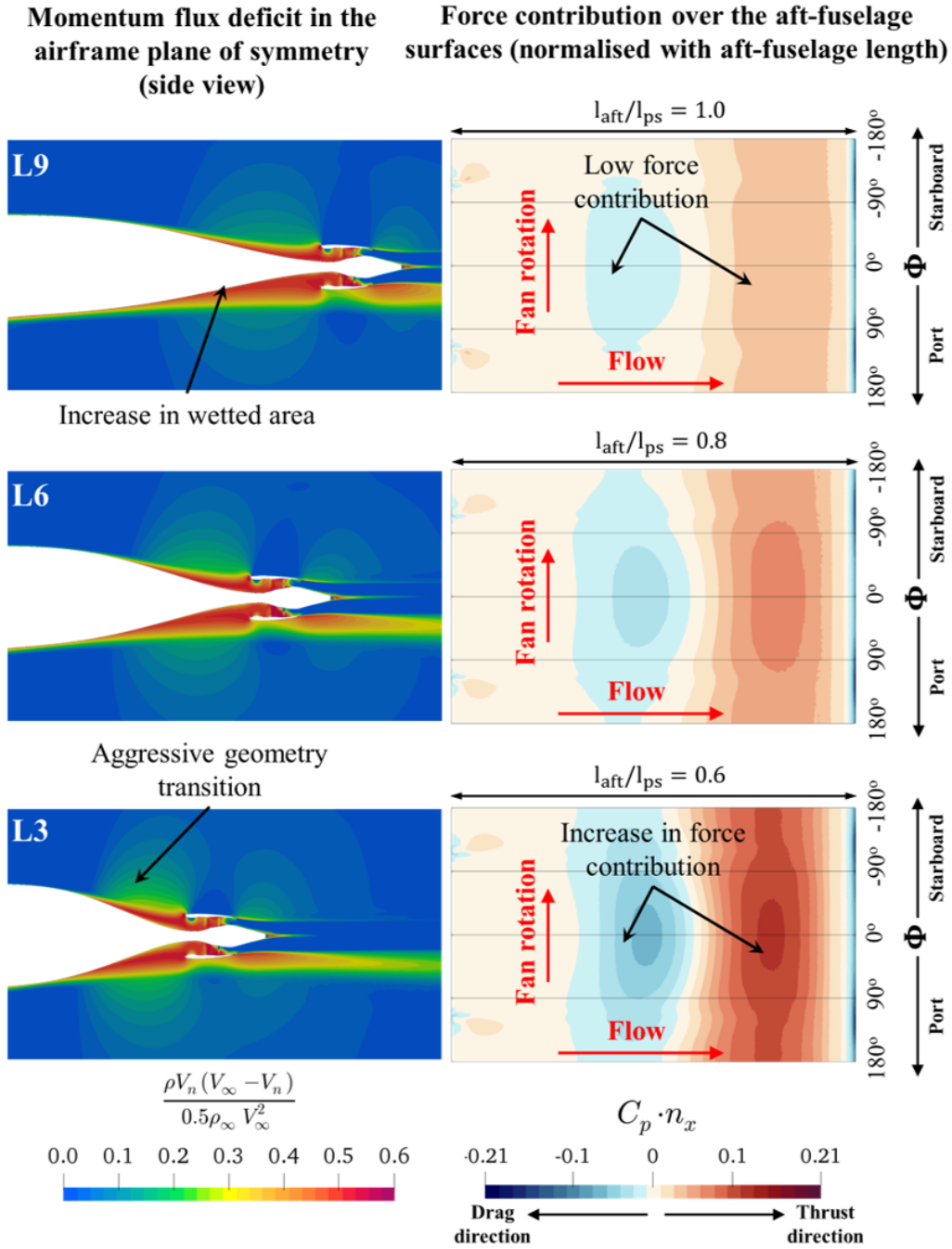


FIGURE 8.22: Effects of the axial position of the aft-mounted propulsor in the momentum flux deficit (symmetry plane of the airframe), and force contribution of the aft-fuselage surface (positive on the thrust direction)

However, the increase on the vertical position of the propulsor results on higher levels of sectoral distortion, with a notable increase in the DC_{60} (Fig. 8.23). For example, between the location L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$) and the location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) the DC_{60} increases in 0.19 units. The increase on the average total pressure with the vertical position results in a higher sensitivity

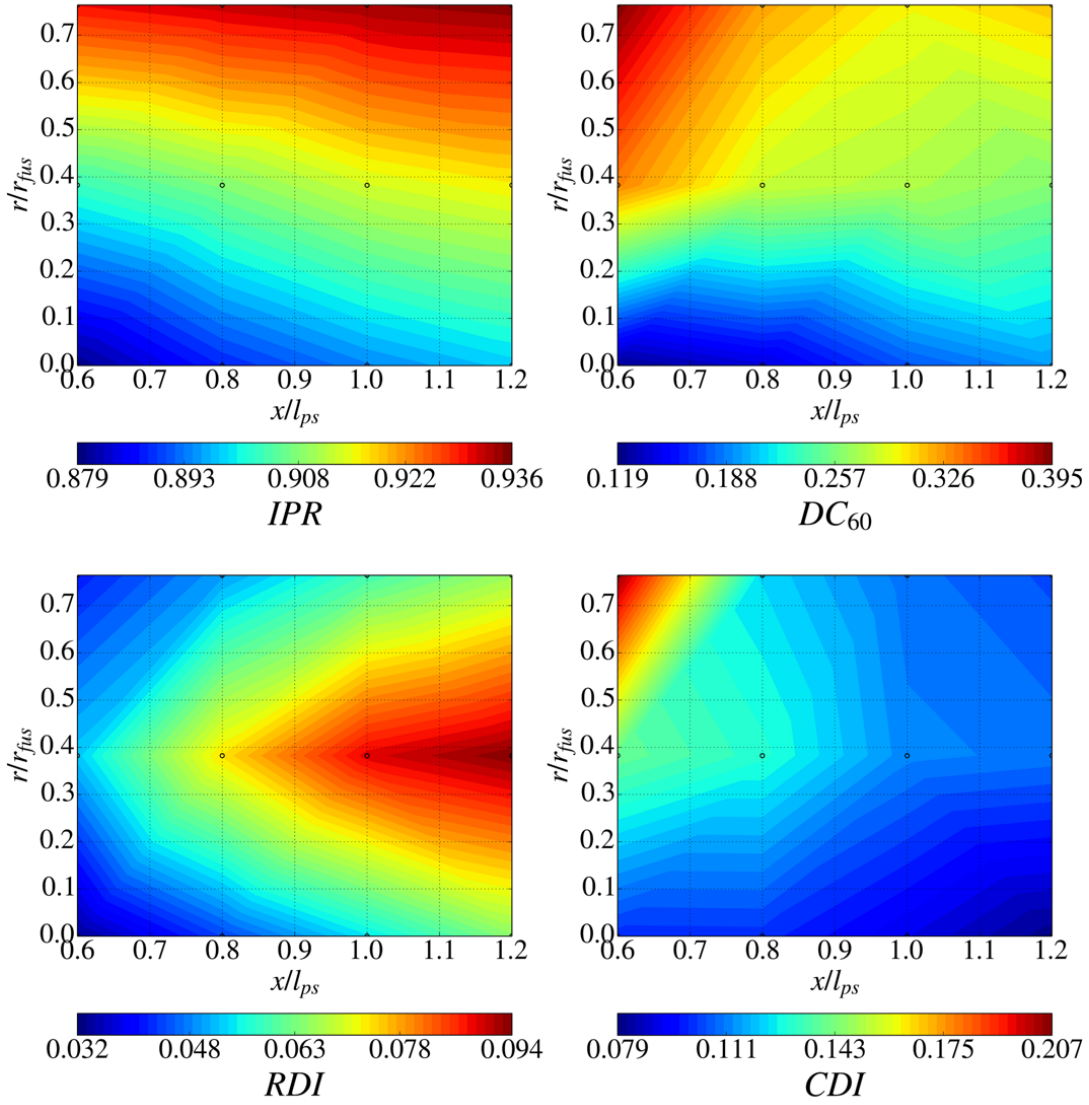


FIGURE 8.23: Effect of the propulsor position on the exhaust performance. Evaluation of the IPR , DC_{60} , RDI , and CDI for an axisymmetric intake design

of the metric to local sectoral variations of the total pressure. The increase on the vertical position of the propulsor results on more defined sectoral distortion regions (Fig. 8.24). These distortion sectors also present an adequate symmetry from the central plane of the AIP (i.e. the line between $\Phi = 0^\circ - 180^\circ$). For boundary layer ingestion configurations, the presence of these sectoral distortion regions is expected and found commonly in the literature^{41,42,187}. Under these conditions, while the DC_{60} values at higher vertical positions, the presence of clear, localised and more stable sectoral distortion regions is preferred to improve the operation of the fan. Thus, for the fan operation the application of higher vertical propulsor positions presents some advantages. However, as it was observed before this will come at the

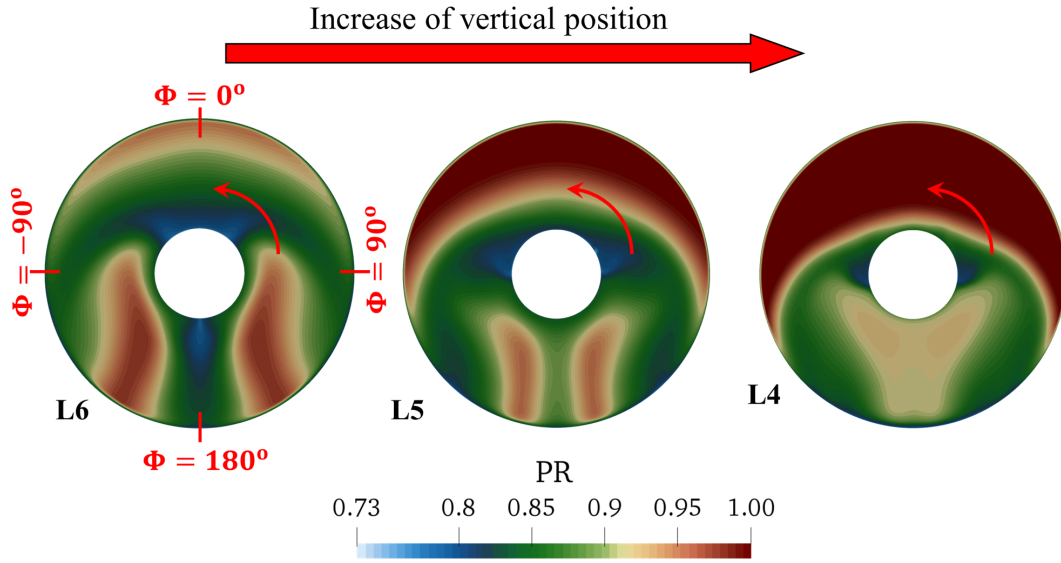


FIGURE 8.24: Variation of the total pressure recovery from the freestream value with the change on vertical position for a fixed $l_{aft}/l_{ps} = 0.8$. Sampled at the fan AIP

expense of a loss in the TS values. Nevertheless, it must be reminded that this intake design is based on a reference axisymmetric cowl design. A more refined intake design should be applied to reduce the intensity of the distortion at the key sectors. This is partially addressed in Section 8.5. Also, the TS values could be improved by a redesign of the aft-fuselage geometry and the fan cowl. This will be addressed in more detail in Section 8.5.

At the highest vertical location of the test ($r/r_{fus} = 0.75$), a notable increase on the distortion levels ($DC_{60} = 0.395$ and $CDI = 0.207$) is observed with the reduction of the axial position to $l_{aft}/l_{ps} = 0.6$ (Fig. 8.23). This is a consequence of the shock induced flow separation (Fig. 8.25) at the lower region of the intake (i.e. $\Phi = 180^\circ$). This separation emerges from the change on the local flow requirements defined by the propulsor location. At this axial and vertical position, the aft fuselage at $\Phi = 180^\circ$ has a steep geometry transition from the fuselage mid-body to the intake. This affects the local angle of the flow at the intake and increases the relative incidence of the intake (Fig. 8.26). The local change on the incidence moves the stagnation point towards the nacelle side. This requires the flow ingested by the intake to accelerate locally around the highlight, which leads to a shock induced separation at the intake lip (Fig. 8.26). This is a clear example of the importance of the propulsor position on the design requirements. The definition of the propulsor location has a critical impact on the design requirements of the intake and the fan

cowl. Thus, the propulsion integration aerodynamic design will be conditioned by the space and position constraints of the system.

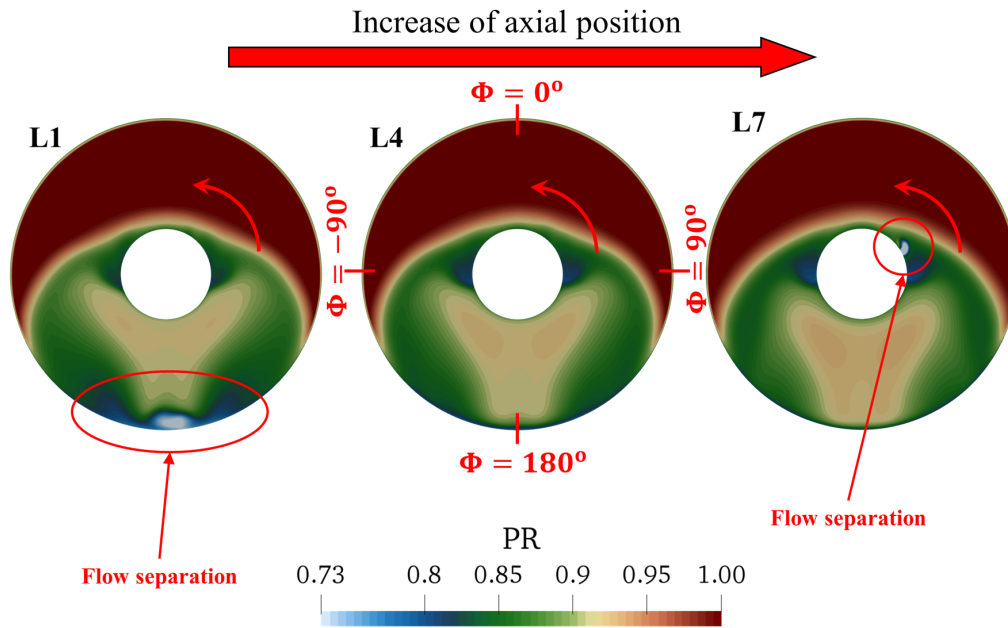


FIGURE 8.25: Variation of the total pressure recovery from the freestream value with the change on axial position for a fixed $r/r_{fus} = 0.75$. Sampled at the fan AIP

The increase of the axial position of the power-plant reduces the sectoral distortion levels (Fig. 8.17). However, this comes at the expense of the radial distortion. In addition, for the higher vertical propulsor positions the increase on axial positions can result in small regions of flow separation in the hub (Fig. 8.25). This is a consequence of the aft fuselage geometry and the local flow conditions and the induced flow rotation by the fan. The flow in the aft fuselage geometry moves from the regions with higher mass concentration at the top section of the fuselage ($\Phi = -90^\circ - 90^\circ$) to the lower regions with higher mass flow deficit (Fig. 8.27). This produces a rotation of the flow towards the lateral regions of the AIP ($\Phi = -90^\circ$ and $\Phi = 90^\circ$). When the flow approaches the fan, the rotation of the flow in the port side of the AIP $\Phi = -90^\circ$ is opposite to the fan rotation. This forces the flow to change in direction and creates a local separation at the fan face (Fig. 8.27). This condition can be avoided by the redesign of the aft fuselage geometry and the intake.

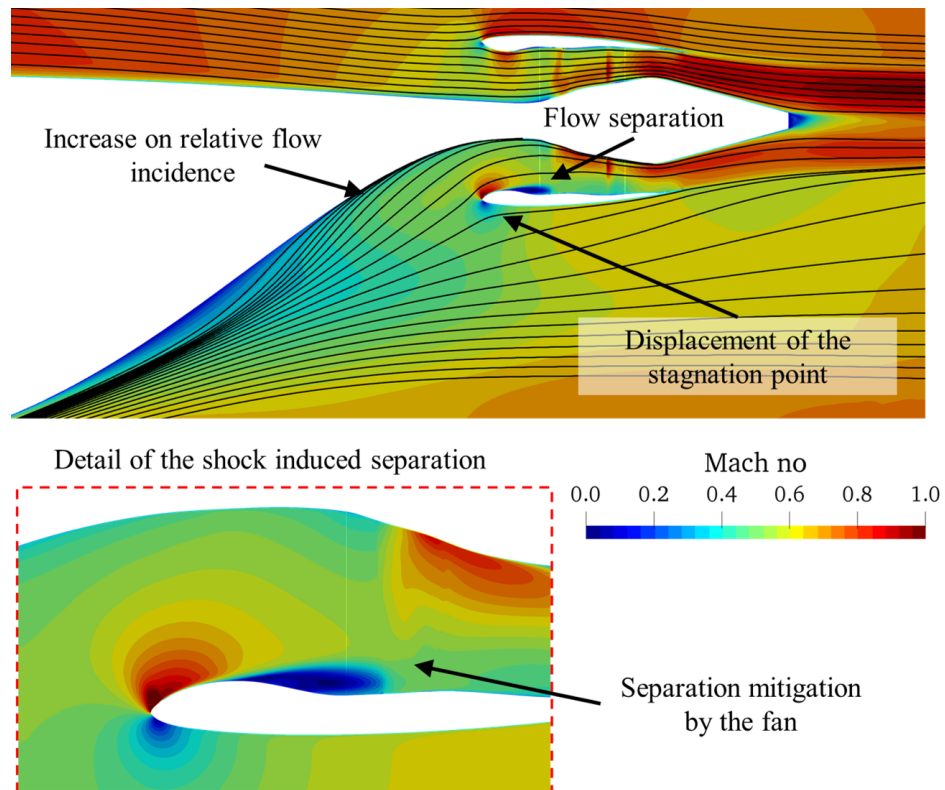


FIGURE 8.26: Change on the relative incidence of the flow with the aft fuselage geometry. Intake separation in location L1

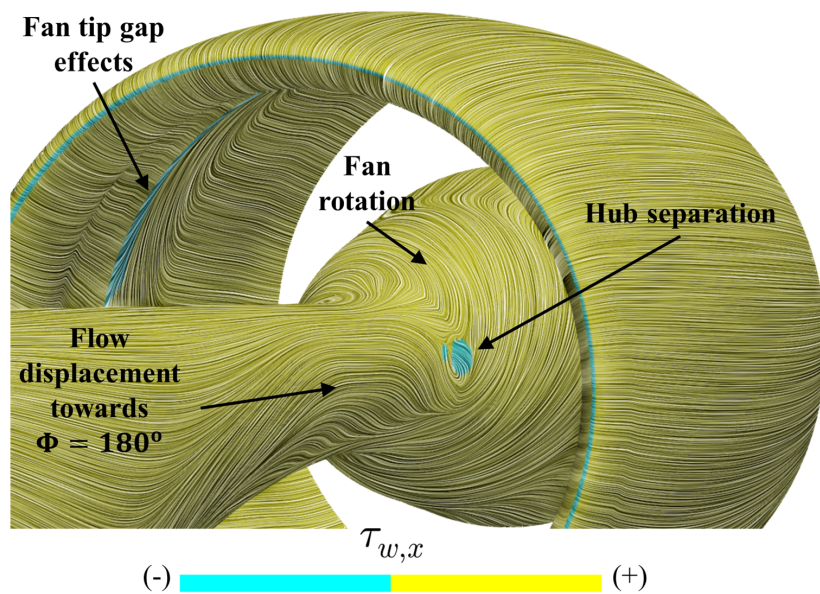


FIGURE 8.27: View of the flow separation at the hub for the location L7. Contour of axial wall shear stress. Oil flow lines (clear color) to show the wall shear stress gradient on the surfaces

8.3.3 Impact of the position on the exhaust performance

As it has been observed, the position of the propulsor with respect to the fuselage wake affects the operating conditions of the propulsor and the requirements for the aerodynamic design. This produces azimuthal variations of the Mach number over the fan cowl and produces notable changes in the pressure distributions at the nacelle afterbody (Fig. 8.20). This affects the exhaust operation at low fan nozzle pressure ratios (Fig. 8.28) with the azimuthal variation of the discharge conditions.

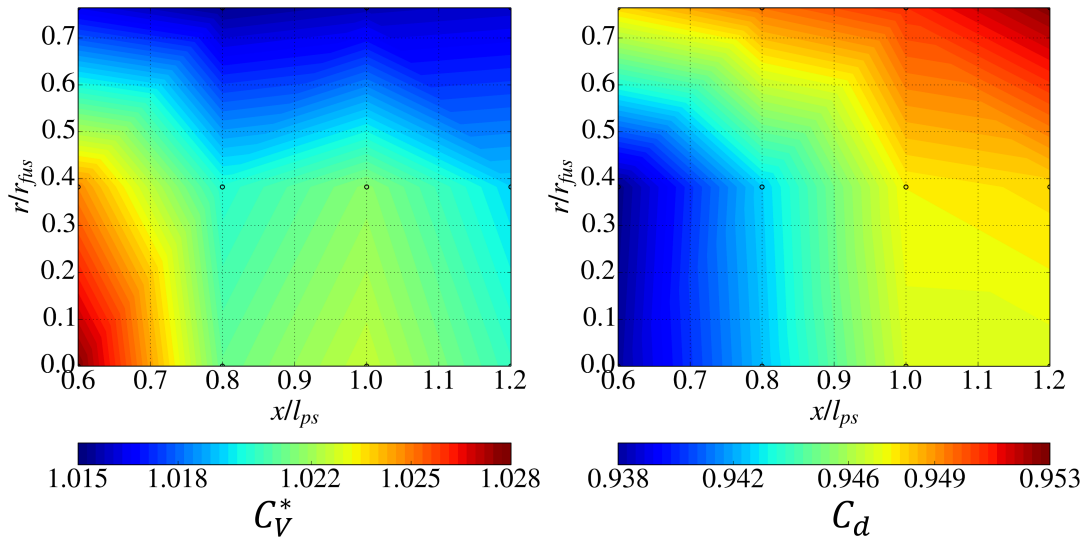


FIGURE 8.28: Effect of the propulsor position on the exhaust performance. Evaluation of the C_V^* and C_d for an axisymmetric cowl and exhaust design

The increase on the vertical position of the aft-mounted propulsor (r/r_{fus}) results in a reduction of the modified velocity coefficient, C_V^* (Fig. 8.28). For example, between the location L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$) and the location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) the C_V^* decreases in a 0.6%. This is understood to be an effect of the changes on the ideal propulsive force. The increase on the vertical location of the propulsor was found to increase the isentropic efficiency of the fan with a small variation of the FPR . This effectively increases the ideal pressure force and reduces the C_V^* . However, at the same time some changes on the GPF^* are present due to the changes on fan operation and the increase on intake pressure recovery.

The discharge coefficient increases with r/r_{fus} and l_{aft}/l_{ps} . For example, between the location L6 ($r/r_{fus} = 0.0$, $l_{aft}/l_{ps} = 0.8$) and the location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) the C_d increases in a 0.8%. This reduction of the exhaust suppression is a direct consequence of the changes on the local flow conditions at the exhaust

discharge plane. In the propulsors at $r/r_{fus} = 0$, the wake of the airframe affects a wider azimuthal region of the cowl (Fig. 8.22), while in cases at higher vertical positions the top region of the cowl is exposed to higher velocity flows from the freestream (Fig. 8.19). This results in changes in the diffusion patterns in the nacelle afterbody (Fig. 8.20) depending on the vertical and axial locations. This changes the local static pressure values at the nacelle trailing edge. In the cases with higher velocities over the cowl (i.e. higher r/r_{fus} and l_{aft}/l_{ps}), this results on a reduction of the static pressure at the discharge plane that reduces the exhaust suppression.

8.4 Impact of the propulsor dimensions on the performance

In the previous section, a fixed design for the housing of the propulsor (intake, nacelle and exhaust) was applied (Fig. 8.6) to evaluate the effects of the propulsor location. This was used to identify the sensitivity of the aerodynamic performance to changes on the aft-fuselage geometry. However, the impact on the aerodynamic performance of the bulk geometry parameters of the fan cowl (e.g. nacelle length, maximum radius, boat-tail angle, exhaust length...) has not yet been evaluated. In this section, the impact of the main dimensions and characteristics of the housing components in the aerodynamic performance is evaluated through a DSE. To simplify the analysis, the aft-fuselage and intake geometries are maintained constant. The axisymmetric intake design of the Datum design (Fig. 8.6) is used. The aft-fuselage geometry of the location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) is used. This location has been selected because it complies with the two main geometry constraints: aircraft length and ground clearance (Section 8.1.2).

An axisymmetric housing design is used (intake, nacelle, exhaust). The design space exploration consists of 7 degrees of freedom for the main parameters that determine the nacelle and exhaust design (Fig. 8.29). These parameters define the bulk dimensions of the nacelle and the exhaust (l_{nac} , r_{max} , f_{max} , r_{te} , f_{crop}), and the nacelle boat-tail angle (β_{nac}) and the propulsor cone angle (β_{cone}). A Latin Hypercube Sampling¹⁶⁴ (LHS) was used to evenly populate the design space for 84 designs (12 per degree of freedom). The size of the LHS was adjusted to obtain the desired number of designs while some geometry constraints are applied. These constraints are the maximum length of the aircraft and the absence of surface inflections on the fan cowl (Chapter 3.5).

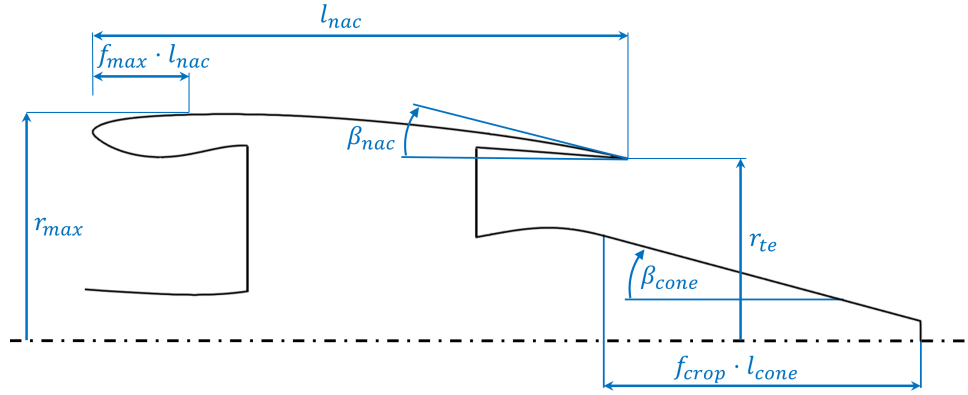


FIGURE 8.29: Definition of the geometry parameters applied for the design space exploration of the effects of the nacelle and exhaust design of the 3D models

Reference cruise operating conditions were used for the design space exploration (altitude of 32,000 ft, $M_\infty = 0.75$, and $C_L = 0.5$). The aerodynamic performance of the aft-mounted propulsor designs was evaluated with the thrust split (TS) and the thrust specific power consumption ($TSPC$). The discharge coefficient was included to understand the effects of the bulk geometry parameters on the exhaust performance. The results of the design space exploration are displayed in the Figure 8.30. The Baseline design (location L4) is included as a reference. The changes on the aerodynamic performance are based on the Baseline design.

A clear correlation between the TS and $TSPC$ is observed (Fig. 8.30). This has been observed in previous chapters of this thesis (Chapters 5.2 and 7.2) and it is due to the definition of the $TSPC$ as a function of the relative net thrust of the aft-mounted propulsor (Eq. 3.30). A notable increase on TS (approximately 0.02 units) was achieved between the best performing design of the DSE and the Baseline. The $TSPC$ is also reduced, with a difference of 28 m/s between the design with the highest TS and the Baseline. This translates to an additional increase of between 2-3% on the power saving coefficient from the Baseline value (approximately 9%).

In the cases with higher values of TS a relatively constant value of the discharge coefficient is observed (Fig. 8.30). Some significant penalties on C_d (up to 0.06 units) are observed for the designs with the lowest TS . Nevertheless, these designs are out of the region of interest. The design with the maximum TS (Max TS) of the DSE presents some penalties on the C_d with respect to the Baseline ($\sim 1.2\%$). However, these penalties could be addressed by the modification of the exhaust discharge area.

To understand the effect of the individual degrees of freedom of the DSE (Fig. 8.29) in the thrust contribution of the aft-mounted propulsor, the designs of the DSE have

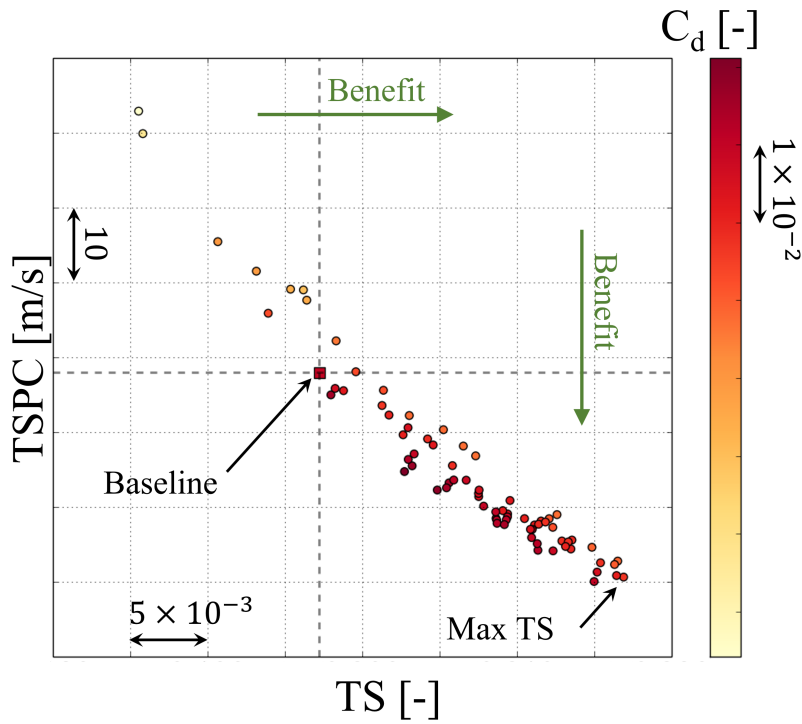


FIGURE 8.30: Changes on TS , $TSPC$ and C_d for the Design Space Exploration of the bulk geometry parameters of the nacelle and the exhaust. Baseline design for reference

been divided into three groups based on the range of values of TS (Fig. 8.31). Thus, the group with the worst performing designs contains the values of TS between 0 to 33% of the value and the best designs are in the group between 66% to 100% of the value. Then, the density distribution of the groups for each of the degrees of freedom is evaluated (Fig. 8.31). The degrees of freedom are normalised with the ranges for the proposed DSE. This is used to identify the regions of the design space where the design variables contribute to the increase on TS .

The DSE shows that more compact nacelle and exhaust designs are required to maximise the TS (Fig. 8.31). The best performing designs (66 – 100% group) have the lowest l_{nac} values and intermediate values of the nacelle trailing edge radius (r_{te}). As discussed in previous chapters (Chapters 5.2-8.3-7.2), the application of more compact nacelle and exhaust designs reduces the wetted area and reduces the drag of the propulsion system. Some additional thrust contribution is obtained with the increase on the diffusion in the nacelle afterbody due to the reduction of r_{te} (Fig. 8.32). For this last purpose, intermediate r_{max} and f_{max} values are desired. Under these conditions, the benefits on TS from the nacelle afterbody diffusion have to compensate the penalties on the increase of the wetted surface of the nacelle. The

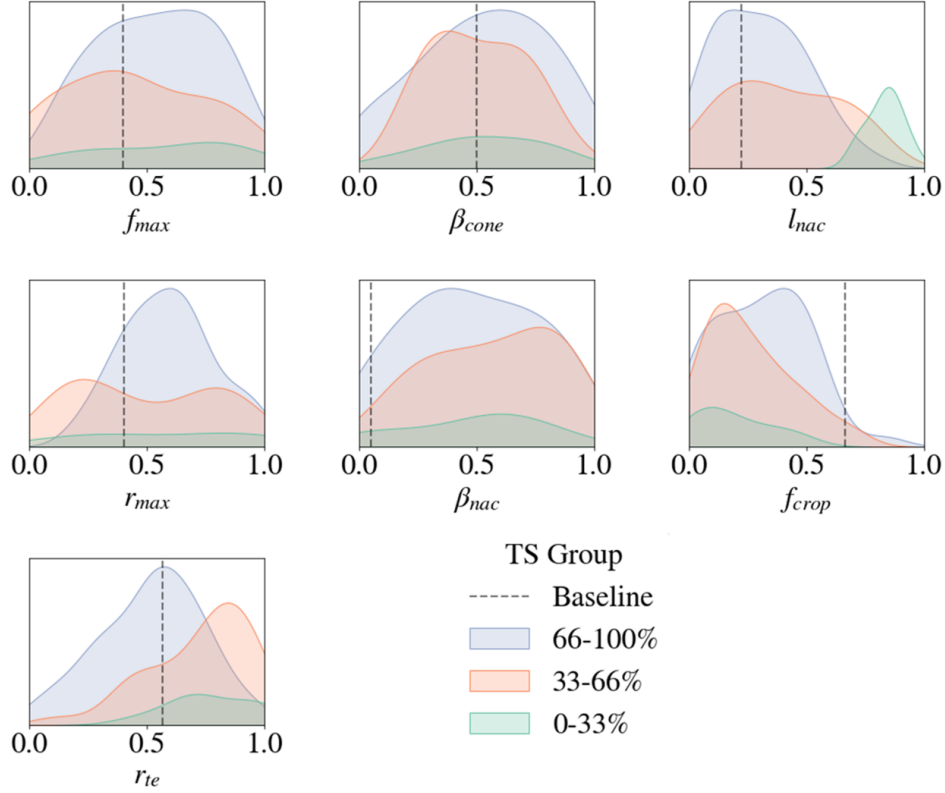


FIGURE 8.31: Histogram for the degrees of freedom of the DSE. Parameters normalised with the optimisation limits. Colour division based on the TS values. Thrust split range split into three equal intervals

TS can also benefit from a more compact exhaust cone, with intermediate high values of β_{cone} and low values of f_{crop} to make the exhaust system shorter.

As an example, the design with the maximum TS of the DSE (Max TS) is compared with the Baseline design (Tab 8.4). The Max TS design has a slightly more compact cowl (2.5% reduction in l_{nac} and 1.3% in r_{te}), with a higher r_{max} (3.7% increase), f_{max} (48% increase), and β_{nac} (21% increase). The exhaust cone inclination inclination, β_{cone} , increases (approximately from 15 to 16 degrees), and f_{crop} also increases (approximately 4%). The combination of all the parameters results in a reduction of approximately 7.5% in the exhaust cone length. Overall, these changes generate an increase of approximately 0.02 in TS from the Baseline design ($\sim 2 - 3\%$ change in power savings).

The increase on TS from the Baseline is partially the result on the increase on the propulsive force in the nacelle due to the geometry and the reduction of the friction losses over the cowl (Fig. 8.32). The reduction of the friction losses over the cowl is obtained by the reduction of the cowl length (2.5% reduction in l_{nac}) and the reduction of the velocity over the nacelle forebody (Fig. 8.33), which reduce the

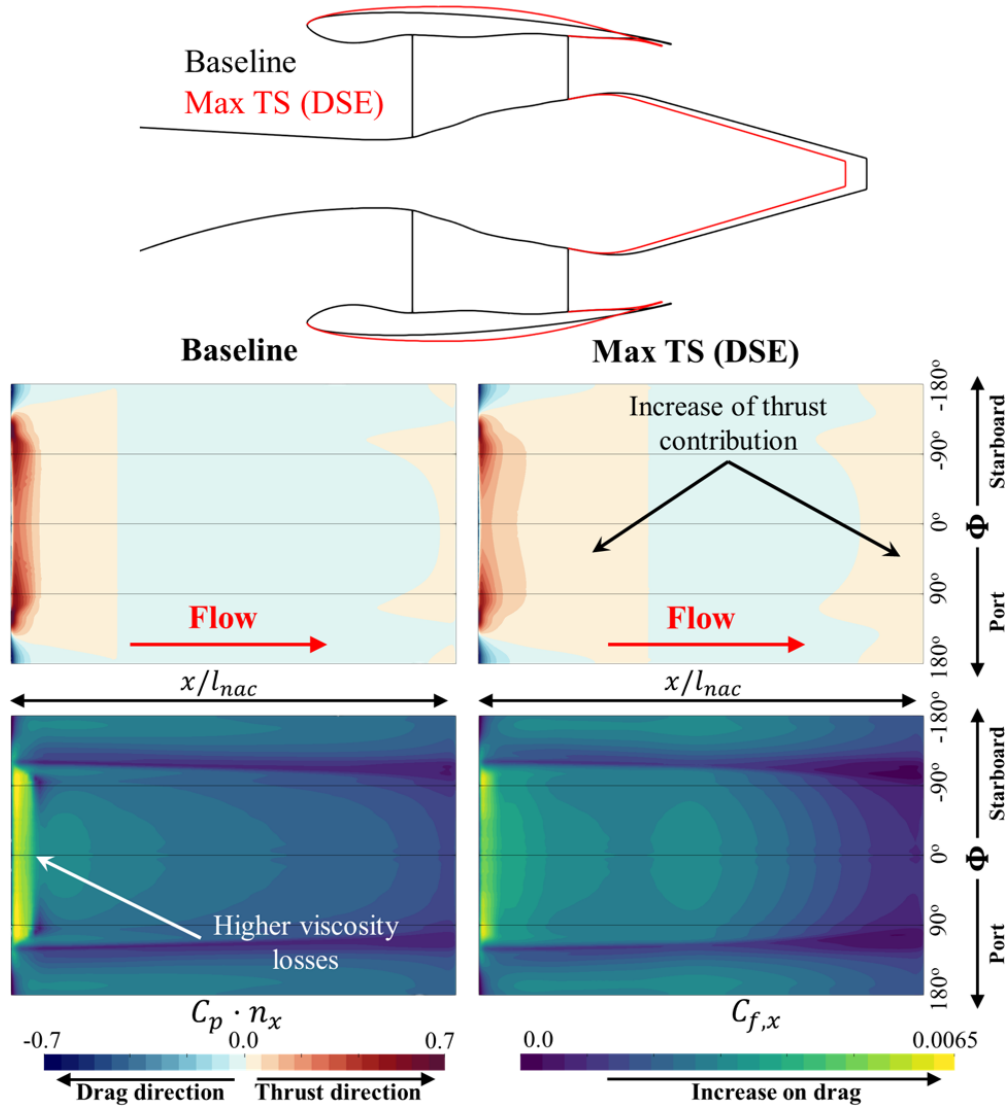


FIGURE 8.32: Projection of the pressure force and viscous force contribution over the fan cowl for the baseline design and the design with maximum TS of the Design Space Exploration. Length normalised with l_{nac}

wetted area and the wall shear stress, respectively. This reduction of the velocity is a consequence of the reduction of the curvature gradient in the nacelle forebody from the highlight. Another contributor to the increase on TS is the increase on the thrust force on the exhaust cone with the increase of β_{cone} .

The gains on TS (~ 0.02) with the changes on the bulk parameters (Fig. 8.29) of the propulsion integration highlight the importance of the propulsion integration design. Previously in this work (Chapter 7.3), the effect of the propulsion integration design was found to be approximately 0.01 units of TS for an axisymmetric configuration (1-1.6% changes in power saving coefficient). This indicates that there

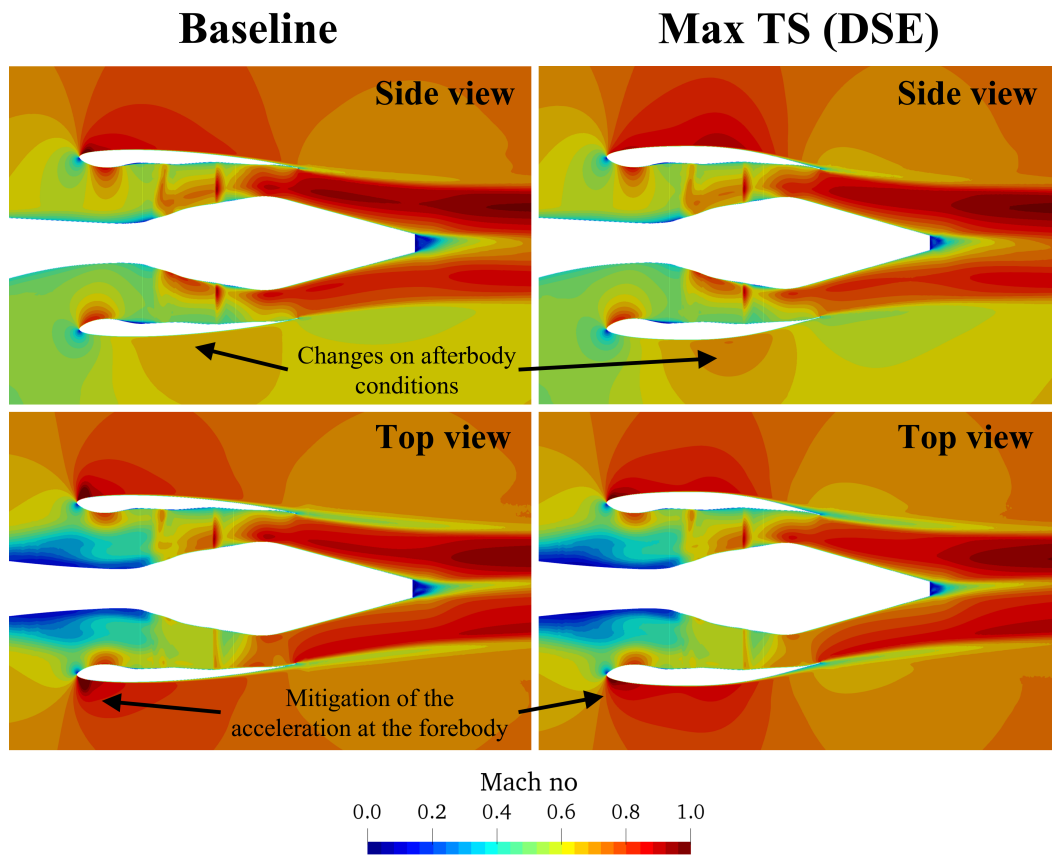


FIGURE 8.33: Mach number contours for the for the baseline design and the design with maximum TS of the Design Space Exploration. Top and side views

is a higher impact of the propulsion integration design in 3D conditions with non-axisymmetric flows. However, it is acknowledged that the Baseline design is not completely representative of a bespoke aerodynamic design on the housing components and has some adverse aerodynamic characteristics due to the axisymmetric cowl design (Fig. 8.33). Further research is required to fully quantify the impact of the three-dimensional design space on the aerodynamic performance. This requires the exploration of non-axisymmetric propulsion integration designs. However, this increases the number of degrees of freedom significantly (from 20 to 100), and will require additional CFD evaluations to cover the design space. Due to the intensive computational cost of each CFD evaluation, this activity becomes prohibitive. For this reason, future research should involve dimensionality reduction techniques, such as active subspaces¹⁸³. These methods could be applied in combination with multi-fidelity approaches as shown by Tejero et al¹²⁹ for the aerodynamic design of conventional podded power-plants. Despite these limitations, in the next section (Section 8.5), an attempt to the non-axisymmetric design of the aft-mounted

propulsor is introduced based on aerodynamic criteria and the knowledge obtained through this work.

TABLE 8.2: Summary of the geometry changes from the Baseline design for the design with the highest TS of the design space exploration

Change on the design parameters from the Baseline in the Max TS design						
l_{nac}	f_{max}	r_{max}	β_{nac}	r_{te}	β_{cone}	f_{crop}
-2.5%	48%	3.7%	21%	-1.3%	5.5%	4%

8.5 Non-axisymmetric cowl design

In this section, a non-axisymmetric bespoke design for the aft-mounted propulsor is defined. The design is based on the knowledge gathered through this project about the propulsion integration design in embedded propulsion systems. The aim of the activity is to define a configuration that increases the TS of the system without penalising the intake and exhaust performance.

The propulsor location is fixed at $r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$ (Location L4 in Fig. 8.17). As in Section 8.4, this location was selected because it complies with the geometry restrictions for the aircraft (i.e. maximum length and ground clearance). To simplify the design activity, the fuselage afterbody is fixed based on the Baseline design at this axial and vertical position of the propulsor. The non-axisymmetric geometry (Chapter 3.2.2.8) is defined with 5 control lines (Fig. 8.34). The intake, nacelle and exhaust designs of the configuration are modified. The exhaust design parameters of the system (r_{te} , A_{noz} , $\theta_{up,noz}$, $\theta_{down,noz}$, β_{cone} , β_{noz} , f_{crop}) are set to be consistent between the different control lines, to define an axisymmetric exhaust system (Fig. 8.34). This requires to define one parameter that will be repeated azimuthally. The parameters that define the length constraints of the cowl and the intake (l_{nac} , f_{max} and l_{int}) are also set constant azimuthally. However, the local cowl and intake length is modified with the scarf angle (Fig. 8.34), with the nominal length value set at $\Phi = 90.0^\circ$. This reduces the required design parameters for a non-axisymmetric configuration from a 100 to 45 (Fig. 8.34). This is still a considerable number, but this enables the definition of a bespoke configuration with less than half of degrees of freedom while a sufficient level of detail is maintained.

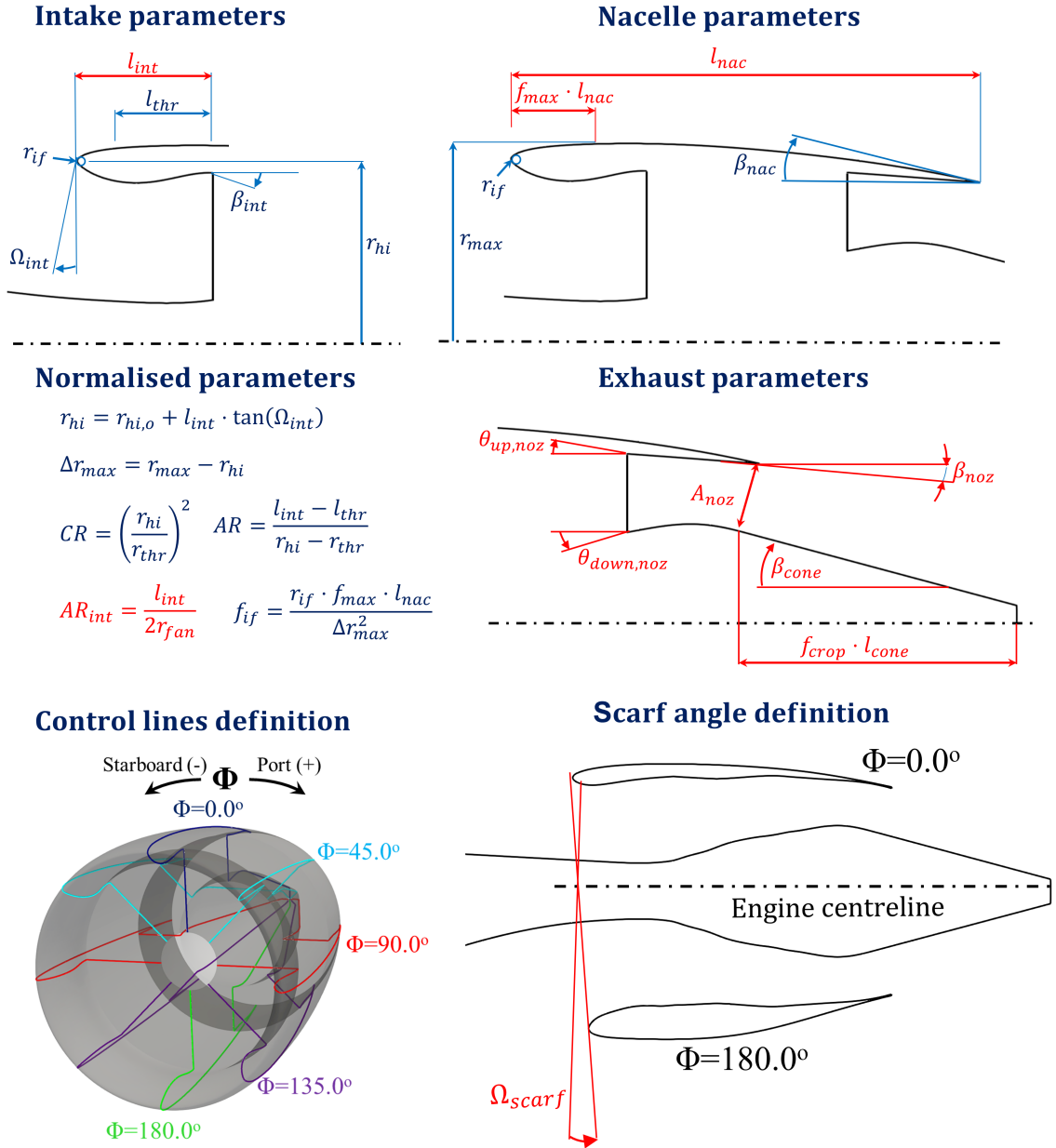


FIGURE 8.34: Definition of the degrees of freedom and reference control lines for the non-axisymmetric design of the housing components of the embedded propulsor. In red parameters set constant azimuthally (axisymmetric). In blue parameters applied to each control line. Control lines defined with left-right symmetry by the plane containing $\Phi = 0.0^\circ$ and $\Phi = 180^\circ$

The aerodynamic characteristics of the Baseline design are evaluated to identify the necessary aerodynamic improvements for the bespoke design (Fig. 8.35). As observed in Section 8.3, the highest concentration of momentum flux deficit at the highlight plane in the location L4 ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$) is found on the lower section of the propulsor ($\Phi \in (-135^\circ, 135^\circ)$) and it is not ingested by the axisymmetric cowl design (Fig. 8.35). This means that there is still some margin

to increase the energy recovery of the system with the design of the intake of the propulsor. However, this is limited by the vertical position of the propulsor. Also, in Section 8.4, it was observed that additional benefits on the TS are obtained with the change on the bulk dimensions and characteristics of the propulsion integration. In particular, with more compact nacelle and exhaust designs (i.e. lower l_{nac} and r_{te}). Thus, a bespoke design of the aft-mounted propulsor integration should focus in these two aspects: the amount of momentum flux deficit ingested, and the increase of the thrust with the cowl and exhaust design.

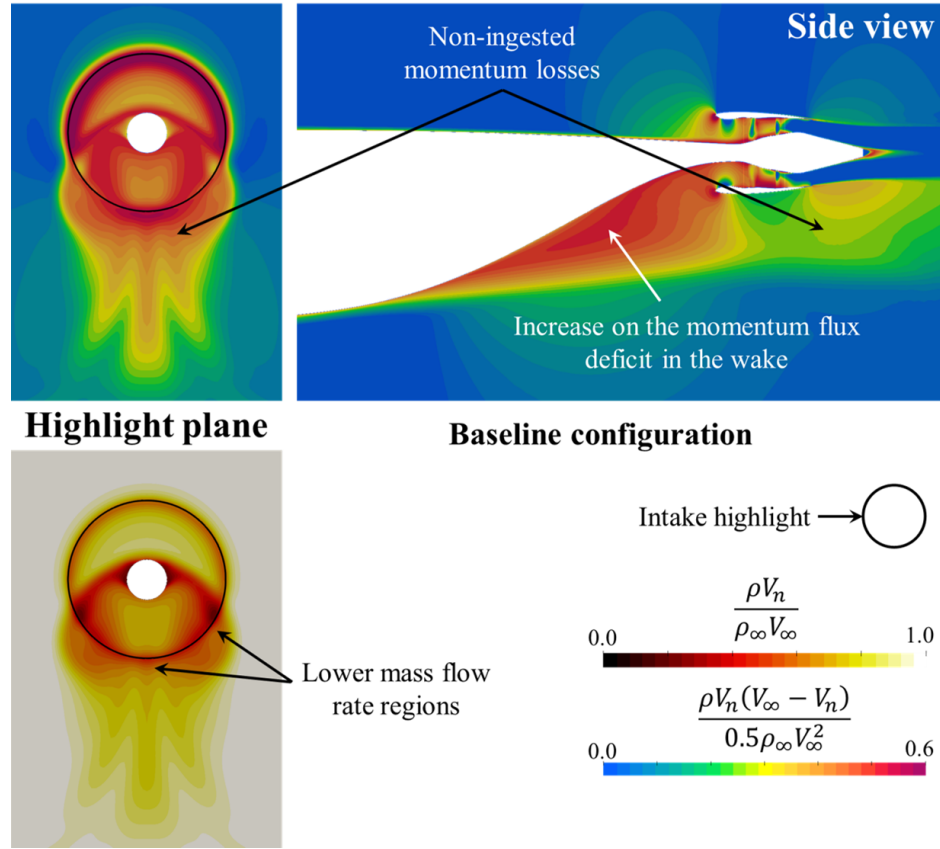


FIGURE 8.35: Momentum flux deficit ingested at the highlight plane (top) in the Baseline configuration ($r/r_{fus} = 0.75$, $l_{aft}/l_{ps} = 0.8$). Side view of the momentum flux deficit generated by the cowl and the losses and the non-ingested wake

The intake length (AR_{int}) was kept constant from the Baseline case in order to simplify the analysis. A more compact cowl is used, with a 7.6% reduction in l_{nac} and a 1.3% reduction in r_{te} (Tab. 8.3). To increase the alignment of the nacelle with the flow near $\Phi = 180^\circ$, the intake is scarfed (Ω_{scarf}) 3 degrees. This is complemented with a reduction of the intake inclination (Ω_{int}) in the control lines between 0.0° and 135° (Tab. 8.4), where the relative orientation of the incident flow is lower than on the vicinity of $\Phi = 180^\circ$, as it was observed for location L1 (Fig. 8.26). In that control line ($\Phi = 180^\circ$), the Ω_{int} is maintained but the initial value of the highlight

radius ($r_{hi,o}$) increases to allow a further ingestion of the momentum flux deficit (Figs. 8.35-8.36). The r_{max} and the f_{max} are also adjusted to increase the thrust contribution of the nacelle afterbody, given the local highlight radius at each control line (Tabs. 8.3-8.4).

TABLE 8.3: Summary of the geometry changes from the Baseline in the Bespoke design. Summary of the parameters modified axisymmetrically

Change on the axisymmetric design parameters from the Baseline for the bespoke design										
AR_{int}	l_{nac}	r_{te}	f_{max}	β_{wedge}	A_{noz}	$\theta_{up,noz}$	$\theta_{down,noz}$	β_{cone}	f_{crop}	Ω_{scarf}
0.0%	-7.6%	-1.3%	16.0%	8.7%	-3.7%	12.5%	-52.4%	0.0%	9.5%	3°

TABLE 8.4: Summary of the geometry changes from the Baseline in the Bespoke design. Summary of the parameters modified in each control-line

Change on the design parameters of the control-lines from the Baseline in the bespoke design								
Φ	$r_{hi,o}$	Ω_{int}	AR	CR	β_{int}	r_{if}	r_{max}	β_{nac}
0.0°	1.9%	-72.2%	102.2%	-10.4%	-1.0%	44.7%	-2.0%	11.5%
45.0°	1.9%	-72.2%	64.3%	-8.1%	-1.0%	60.8%	-2.0%	11.5%
90.0°	0.0%	-77.8%	44.1%	-8.8%	31.9%	68.8%	-2.0%	11.5%
135.0°	1.3%	-72.2%	26.4%	-9.6%	-100.0%	52.7%	-2.0%	11.5%
180.0°	16.9%	0.0%	-14.1%	2.2%	-67.0%	125.1%	13.1%	19.2%

While some additional momentum flux deficit could be ingested by the increase of r_{hi} in $\Phi = 135^\circ$, there is also a mass flow deficit in that region (Fig. 8.35). If r_{hi} was to be increased in that area, this will result in a displacement of the stagnation point towards the interior of the intake duct, with an increase on the spillage drag over the cowl and the local acceleration of the flow. Thus, the intake was designed to mitigate this problem (Fig. 8.36). An alternative solution could come from the redesign of the fuselage aft surface. However, to simplify the design activity this was kept constant.

The Bespoke 3D non-axisymmetric design results in an increase of 0.026 in TS , from the Baseline. This is a 0.007 increase from the best design of the axisymmetric cowl DSE (Fig. 8.30). This translates approximately to an increase of 2.6-3.9% in the power saving coefficient from the Baseline design ($\sim 9\%$). In addition, the $TSPC$ decreases a 10% (~ 35 m/s) compared with the Baseline, due to the increase of TS (Fig. 8.37). Nevertheless, these improvements cannot achieve a similar TS value to the obtained with lower radial locations of the propulsor with the Baseline

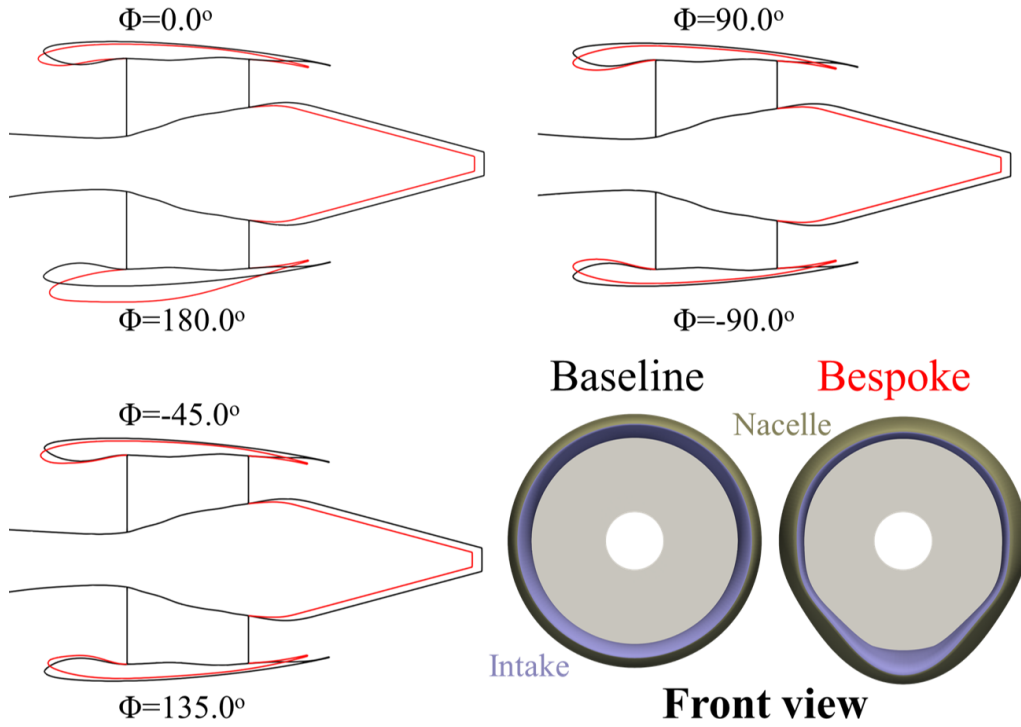


FIGURE 8.36: Comparison between the Baseline (black) and the bespoke (red) design in the 5 control-lines ($\Phi = 0.0^\circ - 45^\circ - 90^\circ - 135^\circ - 180^\circ$), and front views of the intake (purple) and fan cowl (khaki)

cowl design (Fig. 8.18). For example, the Baseline cowl design in location L6 (Fig. 8.17), placed in the same axial location but in the fuselage centreline, has a TS approximately 0.02 units higher than the Bespoke design ($\sim 2 - 3\%$ in power savings). Hence, the bulk geometry characteristics (i.e. length of the system and position) have a higher impact on the aerodynamic performance than the detail parameters of the propulsion integration design. In addition, this highlights the loss in the maximum performance obtained by the change on the propulsor position to comply with the geometry constraints of the aircraft (Section 8.1.2). Thus, the relaxation of these constraints is recommended while possible in the design of the future novel aircraft configurations to enable higher power savings derived from the aft-mounted propulsor.

These improvements on the aerodynamic performance are obtained partially by the reduction of the Mach number over the fan cowl and the increase on the ingested momentum flux deficit of the fuselage (Fig. 8.38). The reduction of the nacelle length (Fig. 8.36) and the increase on the diffusion in the nacelle afterbody (Fig. 8.38) are also contributors to the increase on TS .

The Bespoke design maintains similar values of the intake performance than the

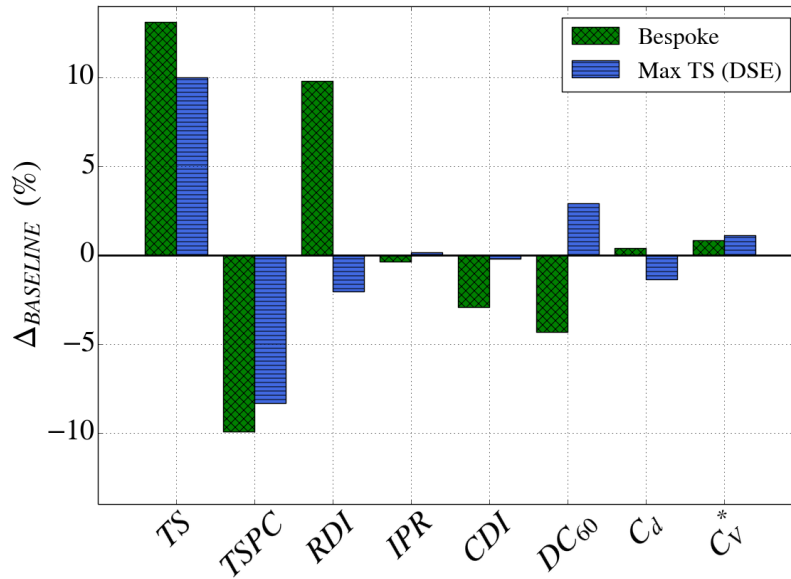


FIGURE 8.37: Increase of the aerodynamic performance metrics from the Baseline design. Comparison between the Bespoke design and the design with the maximum TS of the DSE

Baseline, while the thrust contribution increases (Fig. 8.37). The RDI increases from 0.049 to 0.054. Nevertheless, this increase is not significant on the context of the RDI ¹⁷⁷. Similarly, the IPR decreases in approximately 0.4% which is not significant in this case. Small benefits in the annular distortion are observed, with a reduction of 2.5% and 4% in the CDI and DC_{60} , respectively. These are also not significant under the current levels of distortion ($DC_{60} \sim 0.3$). These changes are due to the changes on total pressure at the AIP (Fig. 8.39), where the intake design moves slightly the regions with lower total pressure towards the upper region of the fan face ($\Phi = -45^\circ$ - 45°). To further reduce this distortion a combined detailed design of the fuselage afterbody and the intake geometries will be required.

The exhaust performance is also maintained at similar levels in comparison with the Baseline and the Max TS design of the DSE. With an increase from the Baseline of approximately 1% and 2% in C_d and C_v^* , respectively. This means that the thrust contribution of the system can be achieved without penalising the exhaust performance. Furthermore, a more detailed analysis of the exhaust system under three dimensional flows could provide further improvements on the exhaust performance and the TS . Future work should involve a more detailed analysis of the exhaust design of the embedded system with three dimensional flows. More advanced exhaust designs, such as non-axisymmetric exhaust designs¹²², could be applied to

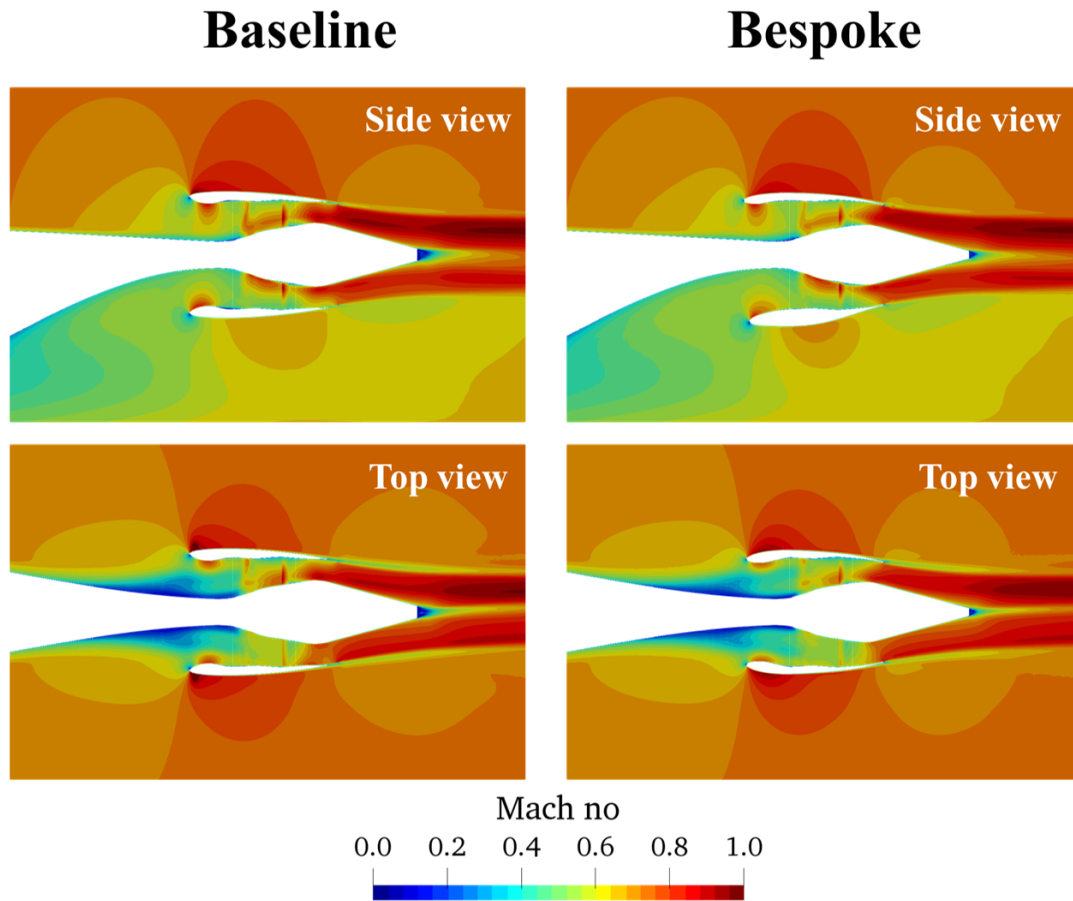


FIGURE 8.38: Mach number contours over the Baseline and the Bespoke design. Side and top views of the flow in the engine symmetry axis

further increase the overall thrust contribution and power savings obtained with the aft-mounted propulsion system.

8.6 Summary

This chapter included the studies carried out with a three-dimensional airframe and aft-mounted propulsor. First, the 3D approach with a through-flow fan model was compared with lower levels of modelling fidelity, such as 2D axisymmetric configurations and one-dimensional fan models. The comparison of the modelling fidelity of the CFD approach shown a reduction of 0.02 units of TS when 2D axisymmetric models are applied instead of more representative 3D approaches, but with less than 1% of the computational cost. This translates to approximately a 2 – 3% variation in the power savings from the preliminary prediction of 11%. The application of a more representative through-flow approach also increased the predicted TS is

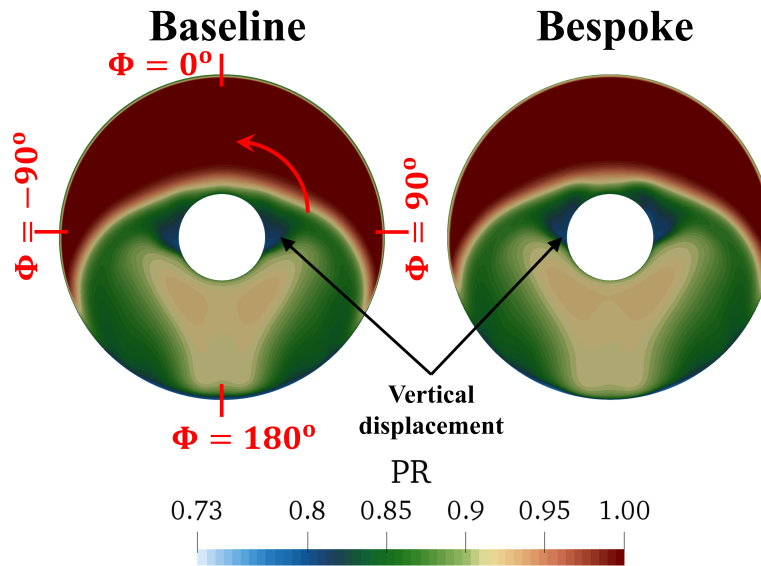


FIGURE 8.39: Intake pressure ratio (based on freestream total pressure) at the AIP for the Baseline and Bespoke configurations

approximately 0.01 ($\sim 1 - 1.5\%$ in power savings), when compared with a one dimensional fan model. The exhaust performance was consistent between the different levels of fidelity. The intake distortion were consistent between the one-dimensional fan model and the through-flow fan models, but RDI and the IPR values improved when a more representative fuselage was applied.

The impact of the aft-mounted propulsor location was evaluated with a design space exploration based on a fixed axisymmetric cowl design. The highest TS values were found in positions aligned with the fuselage centreline. The TS was penalised up to 0.04 units with the increase on the vertical and axial location of the propulsor. This is important, since the position of the aft-mounted propulsor has to comply with some space restrictions due to the aircraft take-off requirements. In this case study, locations that comply these requirements are the ones with the lowest TS values. Thus, for the design of the future aircraft configurations with aft-mounted annular propulsors, it is advisable to reduce these geometry constraints with changes on the geometry of the airframe.

To reduce the penalties on TS with the increase on the vertical position of the propulsor, a Design Space Exploration of the main geometric characteristics of the aft-mounted propulsor was carried out. An increase on the TS of approximately 0.02 units was found relative to a Baseline design. To further increase the TS , a Bespoke design was defined based on the design criteria gathered through this work. This resulted on an increase of 0.026 units of TS from the baseline. This

indicates the potential improvement of the TS and power savings with the design of the propulsion integration. However, the increase on TS obtained with these design does not achieve the values obtained with the propulsor locations closer to the fuselage centreline.

Conclusions and future work

9.1 Contributions to knowledge and conclusions

The main contribution to knowledge of this work is the development of a systematic methodology to design and evaluate the aerodynamic performance of embedded propulsion systems. In addition, this methodology has been evaluated for different levels of fidelity across this work to determine the modelling requirements for the embedded propulsion systems. Finally, another significant contribution of this work is the definition of the relative importance of the propulsion integration of the embedded propulsion system on the overall power savings and aerodynamic performance of a novel aircraft configuration.

In addition to these key contributions, other significant novelties are derived from this work. These novelties have been divided into three groups based on the key contributions to knowledge of the work. First, the conclusions and contributions related to the methods required for the aerodynamic design of an embedded system are summarised in Section 9.1.1. Second, the conclusions from the modelling requirements to evaluate the aerodynamic performance of embedded propulsive systems are defined in Section 9.1.2. Finally, the design requirements for the embedded propulsion system obtained by the application of the design methodology are reported in Section 9.1.3.

9.1.1 Methods for the design of embedded propulsion systems

This research presented a systematic methodology for the aerodynamic design and evaluation of embedded propulsion systems for novel aircraft configurations, such as distributed propulsion systems and boundary layer ingestion. As a case study, an aft-mounted annular boundary layer ingestion propulsor was selected for a medium-sized single-aisle aircraft. As a summary, the main conclusions about the design methodology are as follows:

- A structured design framework for the aerodynamic design of the integration of embedded propulsive systems was proposed. This methodology is derived from existing methods and has been adapted to the requirements of the embedded propulsion systems. This methodology consists of:
 - A parametric approach for the geometry of the propulsion integration of embedded propulsion systems based on an intuitive Class-Shape Transformation method³⁸
 - Low order methods for the preliminary evaluation of the aerodynamic performance based on thrust-drag estimations
 - Turbomachinery stage modelling based on two levels of fidelity: a one-dimensional semi-coupled fan model, and a coupled through-flow approach
 - An adapted methodology to evaluate the aerodynamic performance of the embedded propulsive systems
 - A compressible, viscous CFD approach
- In the development of the design framework for the embedded propulsion systems, the main contributions of the research were as follows:
 - A novel parametrisation for the geometry definition of embedded propulsion systems has been developed. This is an extension from previous parametric definitions of the intake¹⁴³, nacelle³⁵, and the exhaust systems³⁶ originally defined for wing-mounted conventional engines
 - A tailored force accounting methodology for embedded propulsive systems was defined based on the Modified Near Field Method¹⁴

- A one-dimensional fan model was proposed with a novel mass flow matching capability. In embedded systems, the exhaust discharge conditions vary significantly with the propulsion integration design. This approach provides a consistent representation of the fan stage operation across the design space
 - The through-flow Hall-Thollet body force model¹⁷ was incorporated into the CFD analysis framework. This was validated for a reference fan configuration²³. The through-flow approach was found to provide an adequate representation of the fan stage performance at about 40% of the computational cost of higher fidelity methods, such as a mixing-plane model
 - As an improvement to previous approaches a low order model, based on thrust-drag estimations, has been proposed for the preliminary evaluation of the aerodynamic performance for configurations with a close propulsor-airframe coupling
 - An improved approach to account for the power savings derived from the integration of the aft-mounted propulsor was defined based on existing tools^{168,169}. This approach is based on the optimisation for minimum fuel consumption of the cycle requirements for the wing-mounted power plants based on the thrust and power requirements of the aft-mounted propulsor
- A novel contribution of this work is the generalisation of the design framework to allow the aerodynamic analysis of embedded propulsion systems at different levels of fidelity. This includes the performance prediction based on zero-dimensional low order models, and CFD based approaches for 2D axisymmetric and 3D non-axisymmetric configurations. Different models of fidelity for the fan representation are also included in the CFD methods
 - In this work, a multi-objective optimisation (MOO) approach based on CFD-in-the-loop is applied for the first time to evaluate the combined design requirements of the housing components in embedded propulsion systems. This is a significant advancement in the current state of the art in the optimisation of embedded systems, where single-objective optimisation for particular applications is the typical approach^{13,41,176}. From this, some additional contributions are derived:
 - A coupled MOO approach for the aft-fuselage, intake and nacelle geometries is proposed for the first time

- The first detailed evaluation of the exhaust design for an embedded propulsive system was performed applying the MOO approach

9.1.2 Modelling requirements for embedded propulsion systems in novel aircraft configurations

A hierarchical approach was used to determine the modelling requirements for the embedded propulsive systems, such as the BLI propulsor. This included low order models, and 2D axisymmetric and 3D CFD. The main conclusions from the application of this hierarchical approach are as follows:

- The net value of the power savings is strongly dependent on the power source for the embedded propulsor. From two power configurations for the aft-mounted propulsor, savings between 3% and 11% were predicted based on a reference without the embedded propulsor. These values are aligned with results in the open literature^{1,11,12}, where benefits in power consumption between 3% and 9% are reported. However, to provide a final number for the power savings, it is necessary to specify the power source for the embedded propulsive (turbo-mechanical, electric, turbo-electric, hybrid-electric...)
- The power savings obtained by the integration of the aft-mounted propulsor are strongly dependent on the thrust contribution of the embedded propulsive system (measured by the TS), and the propulsive efficiency (evaluated with the $TSPC$). For a fixed power input, the increase in power savings is achieved by the increase in the propulsive force contribution of the aft-mounted propulsor. If the fan stage design is also fixed, the increase on TS is only achievable by the aerodynamic design of the propulsion integration of the embedded system. Thus, based on the previous assumptions, the aerodynamic analysis of the propulsion integration of embedded systems can be evaluated based on the relative increase in thrust split. Based on a preliminary estimation for a nominal TS of 0.22, the increase of 0.01 in TS leads to an approximate increase in power savings of between 1% and 1.55% with respect to an equivalent aircraft without the embedded propulsor. This represents proximately a 10 – 30% of the overall expected power savings
- Low order force prediction models are insufficient to characterise the aerodynamic performance of close-coupled propulsive systems when compared with a higher fidelity approach. These low order methods have been commonly

applied in the literature for the preliminary evaluation of embedded propulsive systems^{26,44,135,179}. Compared with a 2D axisymmetric CFD approach ($TS \sim 0.22$), these methods over predicted the TS in approximately 50%

- A novel low order approach was introduced, this reduced the differences in predicted TS relative to the 2D axisymmetric CFD to approximately 20%. The computational cost of these low order models is negligible in comparison with the 2D axisymmetric CFD. Nevertheless, all low order force models failed to capture the variation in the thrust contribution of the embedded propulsive system with the changes in the aerodynamic design of the propulsion integration
- A 2D axisymmetric approach can be applied in the preliminary design of the integration of the aft-mounted annular BLI propulsor to reduce the computational cost (about 1% of the cost of the equivalent 3D approach). This is of particular interest on computationally expensive design strategies such as multi-objective optimisation. However, this is restricted to the configurations where the propulsor is placed along the fuselage centreline
- While more economical computationally, the 2D axisymmetric CFD methods have some discrepancies with higher fidelity approaches such as the 3D models. An equivalent axisymmetric cowl design with a more representative 3D airframe increased the TS predicted in approximately 0.02 units ($TS \sim 0.24$), compared with a fully 2D axisymmetric case. This means that the 2D axisymmetric approach should only be applied on an early design stage to understand the characteristics of the design space. However, to obtain a representative value of the aerodynamic performance of the propulsion integration a 3D CFD approach is required
- The fan model has an impact on the prediction of the aerodynamic performance for embedded propulsion systems. The application of the through-flow fan approach lead to an increase of about 0.01 in TS compared with a one-dimensional fan model. This is approximately a 10 – 30% change in the overall expected power savings (3 – 11%)
- The fidelity of the fan model does not have a notable impact on the prediction of the intake performance for a given configurations (2D axisymmetric or 3D). However, independently of the fan model, the 3D models present significant differences on IPR and RDI when compared with an equivalent 2D axisymmetric case. This is a result of the sectoral distortion in the 3D configurations.

Thus, to evaluate the intake performance of the embedded propulsor a 3D configuration with a representative fuselage geometry is required

The Table 9.1 is presented as a summary of the modelling requirements. The changes on thrust split (TS) on the table are based on a reference value of 0.22. The power saving coefficient in this case is $PSC \sim 13\%$. To evaluate the changes of PSC with the TS , a exchange rate of 1% of the PSC value per 0.01 units of TS is selected.

TABLE 9.1: Summary of the changes on the predicted performance with the models applied. Changes on performance based on $TS \sim 0.22$ and $PSC \sim 13\%$. Exchange rate equal to 1% of the PSC value per 0.01 units of TS

Approach	Fan model	Intake performance	Aerodynamic performance
LOM 1D	LOM 1D	<ul style="list-style-type: none"> • Not modelled 	<ul style="list-style-type: none"> • Over-predicts the TS between 20-50% compared with 2D axisymmetric CFD • Cannot capture effects of detailed aerodynamic design
CFD 2D axi	1D	<ul style="list-style-type: none"> • RDI and IPR 	<ul style="list-style-type: none"> • 1% of the computational cost of 3D
	Throughflow	<ul style="list-style-type: none"> • RDI and IPR • No significant differences with 1D fan 	<ul style="list-style-type: none"> • Higher TS (~ 0.01) compared with equivalent 1D fan ($\sim 1\%$ in PSC)
CFD 3D	1D	<ul style="list-style-type: none"> • IPR, sectoral, and radial distortion 	<ul style="list-style-type: none"> • Higher TS (~ 0.02) compared with equivalent 2D axi ($\sim 2\%$ in PSC)
	Throughflow	<ul style="list-style-type: none"> • IPR, sectoral, and radial distortion • No significant differences with 1D fan • Notable changes from 2D axi 	<ul style="list-style-type: none"> • Higher TS (~ 0.02) compared with equivalent 2D axi ($\sim 2\%$ in PSC) • Higher TS (~ 0.01) compared with equivalent 1D fan ($\sim 1\%$ in PSC)

9.1.3 Design of the propulsion integration of the aft-mounted boundary layer ingestion propulsor

The design framework for the propulsion integration of embedded propulsion systems was also applied to evaluate the design requirements of the case study (aft-mounted

annular BLI propulsor). The main conclusions of these studies are summarised as the following:

- In the aerodynamic design of the integration of the embedded propulsor, the main contributors to changes on the TS are the parameters that define the system's dimensions and position (i.e. cowl and aft-fuselage length, cowl radius, fan radius, and hub to tip ratio). In the 2D axisymmetric configurations, the position with the best performance ($TS \sim 0.22$) was radially at 10% of the fuselage radius and axially at 70% of the reference fuselage afterbody length (4.75 times the fuselage radius). This corresponds to a hub-to-tip ratio of about 0.3, similar to current fan configurations²³. The increase on the radial position of the propulsor in about 30% of the fuselage radius results in a penalty of 0.03 units in the TS compared with the best location ($TS \sim 0.22$). The increase of the axial position has a lower impact with in the TS values, with a 0.01 reduction in TS with an increase of 15% of the reference fuselage length. However, this further aft position can be used to improve the total pressure recovery in about 0.005 units ($IPR_{ing} \sim 0.975$). This could be beneficial to facilitate the fan design and operation. Overall, from the location of the propulsor with the highest TS to the location with the lowest TS , the TS is reduced in 0.04. This has a significant effect on the power savings. For example, if nominal power savings of $\sim 11\%$ are assumed, the reduction of TS with the propulsor location represents a $\sim 36\%$ reduction of the predicted power savings
- In the 3D configurations, where the $httr$ is fixed ($httr = 0.3$), the configurations with the aft mounted propulsor located along the fuselage centreline produce the highest thrust contributions with up to $TS \sim 0.26$. Based on this centreline just with changes on the aft-fuselage geometry with the axial position, the most compact propulsor locations (shorter aft fuselage length) provided the highest TS values. Also, the increase on the aft fuselage length penalised the TS values with a similar exchange rate than the 2D axisymmetric cases, with a reduction of 0.01 units of TS with an increase of 15% of the reference fuselage length
- Installation positions further from the fuselage centreline are required to comply with the ground clearance restrictions. Compared with the centreline position, these locations lead to a reduction in about 0.05 units in the TS . This could represent a loss of about 40% of the originally predicted power savings

for the 3D configurations ($\sim 13\%$). This is understood to depend on the airframe design and the ground clearance restrictions of that particular design. As a consequence, in the design of future embedded propulsion systems, the airframe and the propulsion integration must be designed concurrently

- Additional gains on the Thrust Split can be achieved by the detailed design of the propulsive system. In 2D axisymmetric configurations, a total increase of about 0.009 in TS was obtained by the application of the multi-objective optimisation methodology. From this, one third of the increase in TS was obtained from the optimisation of the exhaust system, and the remaining part from the optimisation of the aft-fuselage, nacelle, and intake aerodynamic designs. This represents an increase of about 0.9-1.4% in the power savings, on top of the predicted power saving values. Given the expected power savings found in the literature^{1,11,12} (3-9%), the aerodynamic design of the integration could have a significant impact on the final power savings. Thus, the detailed design of the integration of the embedded power-plant should not be overlooked in the preliminary design. However, in comparison with the bulk parameters (i.e. system lengths, and axial and radial location of the propulsor) these changes on power savings have a second order contribution
- The MOO approach required the evaluation of a high-dimensional design space (20 degrees of freedom) and a high number of individuals. This is not suitable for the current 3D design framework, given the increase of computational cost (150 times higher) and increase of degrees of freedom (45 to 100). Thus, a more traditional design approach was required for the 3D configurations. As a baseline, a propulsor location that complied with the ground clearance restriction is selected, with power savings of approximately 9%. First, a design space exploration identified an increase of 0.02 units in TS . Later a manual bespoke design for the propulsion integration was defined with an increase of 0.026 units of TS from the baseline. This results in approximately a 2-4% increase on the power savings from the baseline. These improvements on TS reduced the penalties associated to the propulsor location, but could not match the power savings obtained with more favourable propulsor locations. For example, at the same axial position, a generic axisymmetric cowl design placed along the fuselage centreline predicted power savings of approximately 14% with respect to an equivalent aircraft without the aft-mounted embedded propulsor

9.2 Future work

The main recommendations for future research activities are the following:

1. A more detailed analysis of the contribution of hybrid-electric configurations should be carried to obtain a clear representation of the power savings due to the distributed propulsion. This should be coupled with a weight estimation methodology to determine the most suitable thrust requirement for the aft mounted propulsor. The application of both approaches would reduce any uncertainty on the quantification of the power savings derived from the BLI propulsor
2. The 2D axisymmetric models allowed the use of a multi-objective optimisation design methodology while a reasonable computational cost was maintained. However, the high dimensionality of the problem still required a substantial computational effort. For this reason, it is of interest to evaluate the utilisation of surrogate modelling approaches to reduce the computational cost. These models would reduce the computational cost and time requirements for the design of the embedded propulsion system and will facilitate the design and decision making on the preliminary design stage of the novel aircraft configurations
3. A multi-point design strategy should be applied to integrate the impact of the off-design conditions in the aerodynamic design of the propulsion integration from an early design stage
4. The impact of the intake distortion in the fan aerodynamic and mechanical performance should be studied in more detail. This should involve an analysis of the requirements for the development of a distortion tolerant fan stage. The application of such studies would enable the application of a more representative fan stage to define the thrust contribution and power savings from the aft mounted propulsor and other embedded propulsion systems
5. The present work was limited to the application of axisymmetric exhaust systems. However, as it was noted, significant variations in the local flow conditions around the nacelle trailing edge are expected for the embedded propulsion system. Under these conditions, it is of interest to explore the application of non-axisymmetric exhaust designs to improve the aerodynamic performance of the propulsion integration. In addition, the application of non-axisymmetric

OGV rows could provide additional benefits by improving the mass flow distribution across the fan stage

6. The optimisation of the aerodynamic design of the propulsion integration for 3D non-axisymmetric cowl designs was considered unfeasible due to the high dimensionality of the design space and the elevated computational cost of the 3D methodology. Under these circumstances, it would be of interest to evaluate the applicability of dimensionality reduction methods to simplify the design space. These methods could be complemented with a multi-fidelity approach to reduce the computational cost. The application of these approaches would enable more advanced studies for the aerodynamic design and evaluation of the embedded propulsion systems
7. The following text was added: To validate the design methodologies and the performance accounting of the aft-mounted BLI propulsor, the construction of a scaled wind-tunnel experimental configuration is recommended. This configuration should be evaluated at representative transonic conditions. A small-scale fan configuration should be included to account for the turbomachinery effects. The tools developed on the present work could be used to assist the design of the experimental benchmark case

References

- ¹Jason Welstead and James L. Felder. Conceptual Design of a Single-Aisle Turboelectric Commercial Transport with Fuselage Boundary Layer Ingestion. In *54th AIAA Aerospace Sciences Meeting*, pages 1–17, San Diego, California, jan 2016. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2016-1027.
- ²Mark Drela. Development of the D8 Transport Configuration. In *29th AIAA Applied Aerodynamics Conference*, Honolulu, Hawaii, jun 2011. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2011-3970.
- ³Ludovic Wiart, Olivier Atinault, Richard Grenon, Bernard Paluch, and David Hue. Development of NOVA Aircraft Configurations for Large Engine Integration Studies. In *33rd AIAA Applied Aerodynamics Conference*, pages 1–30, Dallas, TX, 2015. doi:10.2514/6.2015-2254.
- ⁴Peter Schmollgruber, David Donjat, Michael Ridel, Italo Cafarelli, Olivier Atinault, Christophe François, and Bernard Paluch. Multidisciplinary Design and performance of the ONERA Hybrid Electric Distributed Propulsion concept (DRAGON). In *AIAA Scitech 2020 Forum*, Orlando, Florida, jan 2020. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2020-0501.
- ⁵Hyun Dae Kim. Distributed Propulsion Vehicles. *27th International Congress of the Aeronautical Sciences, ICAS 2010*, pages 1–11, 2010.
- ⁶Andrew Rolt and Whurr John. Optimizing Propulsive Efficiency in Aircraft with Boundary Layer Ingesting Distributed Propulsion. In ISABE, editor, *ISABE-2015-20201; International Society for Airbreathing Engines (ISABE)*, Phoenix, AZ, 2015.
- ⁷Hyoungjin Kim and Meng-Sing Liou. Optimal Shape Design of Mail-Slot Nacelle on N3-X Hybrid Wing Body Configuration. In *31st AIAA Applied Aerodynamics Conference*, San Diego, California, jun 2013. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2013-2413.
- ⁸Mark Moore. NASA N3-X from OpenVSP Hangar. ID# 59 NASA (<http://hangar.openvsp.org/vspfiles/59>), 2022.
- ⁹Alejandra Uranga, Mark Drela, Edward Greitzer, Neil Titchener, Michael Lieu, Nina Siu, Arthur Huang, Gregory M. Gatlin, and Judith Hannon. Preliminary Experimental Assessment of the Boundary Layer Ingestion Benefit for the D8 Aircraft. *52nd Aerospace Sciences Meeting*, (January):1–25, 2014, doi:10.2514/6.2014-0906.
- ¹⁰D. K. Hall, E. M. Greitzer, A. Uranga, M. Drela, and S. A. Pandya. Inlet Flow Distortion in an Advanced Civil Transport Boundary Layer Ingesting Engine Installation. *Journal of Turbomachinery*, 144(10), oct 2022, doi:10.1115/1.4054035.
- ¹¹Justin S. Gray and Joaquim R.R.A. Martins. Coupled aeropropulsive design optimisation of a boundary-layer ingestion propulsor. *Aeronautical Journal*, 123(1259):121–137, 2019, doi:10.1017/aer.2018.120.
- ¹²Arne Seitz, Anaïs Luisa Habermann, Fabian Peter, Florian Troeltsch, Alejandro Castillo Pardo, Biagio Della Corte, Martijn Van Sluis, Zdobyslaw Goraj, Mariusz Kowalski, Xin Zhao, Tomas Grönstedt, Julian Bijewitz, and Guido Wortmann. Proof of concept study for fuselage boundary layer ingesting propulsion. *Aerospace*, 8(1):1–65, 2021, doi:10.3390/aerospace8010016.
- ¹³Justin Gray, Charles Mader, Gaetan Kenway, and Joaquim Martins. Coupled Aeropropulsive Optimization of a Three-Dimensional Boundary-Layer Ingestion Propulsor Considering Inlet Distortion. *Journal of Aircraft*, pages 1–12, 2020, doi:10.2514/1.C035845.

¹⁴MIDAP Study group. Guide to in-Flight Thrust Measurement of Turbo- jets and Fan Engines. Agardograph No. 237, Technical Report No. AG-237. Technical report, AGARD (Advisory Group for Aerospace Research and Development), Neuilly sur Seine, France, 1979.

¹⁵Ioannis Goulos, John Otter, Fernando Tejero, Josep Hueso Rebassa, David MacManus, and Christopher Sheaf. Civil turbofan propulsion aerodynamics: Thrust-drag accounting and impact of engine installation position. *Aerospace Science and Technology*, 111:106533, apr 2021, doi:10.1016/j.ast.2021.106533.

¹⁶Aurélien Arntz and Olivier Atinault. Exergy-Based Performance Assessment of a Blended Wing-Body with Boundary-Layer Ingestion. *AIAA Journal*, 53(12):3766–3776, 2015, doi:10.2514/1.J054072.

¹⁷William Thollet. *Body force modeling of fan – airframe interactions*. Phd thesis, ISAE-SUPAERO, Toulouse, France, 2017.

¹⁸NASA.gov. 2nd AIAA CFD Drag Prediction Workshop (<https://aiaa-dpw.larc.nasa.gov/Workshop2/>), 2003.

¹⁹NASA.gov. 3rd AIAA CFD Drag Prediction Workshop (<https://aiaa-dpw.larc.nasa.gov/Workshop3/workshop3.html>), 2006.

²⁰NASA.gov. 4th AIAA CFD Drag Prediction Workshop (<https://aiaa-dpw.larc.nasa.gov/Workshop4/>), 2009.

²¹Kalyan Deb. *Multiobjective Optimization Using Evolutionary Algorithms*. Wiley, New York, NY, 2001.

²²Matthew H. Robinson. *The Aerodynamics of Aero-Engine Nacelles*. Phd thesis, Cranfield University, 2018.

²³Christopher E Hughes. Aerodynamic Performance of Scale-Model Turbofan Outlet Guide Vanes Designed for Low Noise. *40th AIAA Aerospace Sciences Meeting & Exhibit - Cleveland , OH*, 2002, doi:<https://doi.org/10.2514/6.2002-374>.

²⁴Christopher E Hughes, Richard P Woodward, Gary G Podboy, and Robert J Jeracki. The effect of bypass nozzle exit area on fan aerodynamic performance and noise in a model turbofan simulator. In *Proceedings of the ASME Turbo Expo*, volume 6 PART B, pages 1241–1264, 2013, doi:10.1115/GT2005-68573.

²⁵O. Brodersen and A. Stürmer. Drag prediction of Engine-Airframe interference effects using unstructured Navier-Stokes calculations. *19th AIAA Applied Aerodynamics Conference*, (June), 2001, doi:10.2514/6.2001-2414.

²⁶Arne Seitz, Julian Bijewitz, Sascha Kaiser, and Guido Wortmann. Conceptual investigation of a propulsive fuselage aircraft layout. *Aircraft Engineering and Aerospace Technology*, 86(6):464–472, 2014, doi:10.1108/AEAT-06-2014-0079.

²⁷M Langley. The design of axisymmetric cowls for podded nacelles for high by-pass ratio turbofan engines. R&M no. 3846. Technical report, Aeronautical Research Council, 1979.

²⁸Matthew Robinson, David G. MacManus, and Christopher Sheaf. Aspects of aero-engine nacelle drag. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 233(5):1667–1682, 2019, doi:10.1177/0954410018765574.

²⁹Andrei-Silviu Milea. *Aero-engine installation aerodynamics and design for novel aircraft configurations - MSc Thesis*. Msc thesis, Cranfield University, 2020.

³⁰Eckart Zitzler, Kalyan Deb, and Lothar Thiele. Comparison of Multiobjective Evolutionary Algorithms: Empirical Results. *Evolutionary computation*, 8:173–195, 2000, doi:10.1162/106365600568202.

³¹ACARE. Flightpath 2050. Europe’s Vision for Aviation. Technical report, European Union, 2011.

³²Pedro Argüelles, John Lumsden, Manfred Bischoff, Denis Ranque, Philippe Busquin, Søren Rasmussen, B. A. C Droste, Paul Reutlinger, Sir Richard Evans, Sir Ralph Robins, Walter Kröll, Helena Terho, Jean-Luc Largardère, Arne Wittlöv, and Alberto Lina. European Aeronautics: A vision for 2020 - Report of the group of personalities. Technical report, European Commission, Belgium, 2001.

³³Amir S. Gohardani, Georgios Doulgeris, and Riti Singh. Challenges of future aircraft propulsion: A review of distributed propulsion technology and its potential application for the all electric commercial aircraft. *Progress in Aerospace Sciences*, 47(5):369–391, 2011, doi:10.1016/j.paerosci.2010.09.001.

³⁴Hans-Jörg Steiner, Arne Seitz, Kerstin Wiecek, Kay Plötner, Askin Isikveren, and Mirko Hornung. Multi-disciplinary Design and Feasibility Study of Distributed Propulsion Systems. 2012-1.7.5. In *28th Congress of the International Council of the Aeronautical Sciences 2012, ICAS 2012*. American Institute of Aeronautics and Astronautics, 2012.

³⁵Fernando Tejero, Matthew Robinson, David G. MacManus, and Christopher Sheaf. Multi-objective optimisation of short nacelles for high bypass ratio engines. *Aerospace Science and Technology*, 91:410–421, 2019, doi:10.1016/j.ast.2019.02.014.

³⁶Ioannis Goulos, Tomasz Stankowski, John Otter, David MacManus, Nicholas Grech, and Christopher Sheaf. Aerodynamic Design of Separate-Jet Exhausts for Future Civil Aero-engines - Part I: Parametric Geometry Definition and Computational Fluid Dynamics Approach. *Journal of Engineering for Gas Turbines and Power*, 138(8):1–14, 2016, doi:10.1115/1.4032649.

³⁷Ioannis Goulos, John Otter, Tomasz Stankowski, David MacManus, Nicholas Grech, and Christopher Sheaf. Aerodynamic Design of Separate-Jet Exhausts for Future Civil Aero-engines - Part II: Design Space Exploration, Surrogate Modeling, and Optimization. *Journal of Engineering for Gas Turbines and Power*, 138(8):1–12, 2016, doi:10.1115/1.4032652.

³⁸Robert Christie, Matthew Robinson, Fernando Tejero, and David G. MacManus. The use of hybrid intuitive class shape transformation curves in aerodynamic design. *Aerospace Science and Technology*, 1:105473, 2019, doi:10.1016/j.ast.2019.105473.

³⁹David G. MacManus, Nicola Chiereghin, Daniel Gil Prieto, and Pavlos Zachos. Complex aero-engine intake ducts and dynamic distortion. *33rd AIAA Applied Aerodynamics Conference, Dallas, TX*, (June):1–28, 2015, doi:10.2514/6.2015-3304.

⁴⁰Justin Gray, Charles Mader, Gaetan Kenway, and Joaquim Martins. Coupled Aeropropulsive Optimization of a Three-Dimensional Boundary-Layer Ingestion Propulsor Considering Inlet Distortion. *Journal of Aircraft*, pages 1–12, 2020, doi:10.2514/1.C035845.

⁴¹Hyoungjin Kim and Meng Sing Liou. Flow simulation and optimal shape design of N3-X hybrid wing body configuration using a body force method. *Aerospace Science and Technology*, 2017, doi:10.1016/j.ast.2017.09.046.

⁴²Ewan J. Gunn, Sarah E. Tooze, Cesare A. Hall, and Yann Colin. An experimental study of loss sources in a fan operating with continuous inlet stagnation pressure distortion. *Journal of Turbomachinery*, 135(3), 2013, doi:10.1115/1.4007835.

- ⁴³J. J. Defoe, M. Etemadi, and D. K. Hall. Fan performance scaling with inlet distortions. *Journal of Turbomachinery*, 140(7):1–11, 2018, doi:10.1115/1.4039433.
- ⁴⁴Arne Seitz and Corin Gologan. Parametric design studies for propulsive fuselage aircraft concepts. *CEAS Aeronautical Journal*, 6(1):69–82, 2015, doi:10.1007/s13272-014-0130-3.
- ⁴⁵S-16 Turbine Engine Inlet Flow Distortion Committee. Inlet Total-Pressure-Distortion Considerations for Gas-Turbine Engines. No. AIR1419. Technical report, Society of Automotive Engineers, USA, 1999.
- ⁴⁶Jesús Matesanz-García, Tommaso Piovesan, and David G. MacManus. Aerodynamic optimization of the exhaust system of an aft-mounted boundary layer ingestion propulsor. *International Journal of Numerical Methods for Heat & Fluid Flow*, dec 2022, doi:10.1108/HFF-06-2022-0362.
- ⁴⁷Fernando Tejero, David MacManus, Jesús Matesanz García, Avery Swarthout, and Christopher Sheaf. Towards the design and optimisation of future compact aero-engines: intake/fancowl trade-off investigation. *International Journal of Numerical Methods for Heat & Fluid Flow*, dec 2022, doi:10.1108/HFF-06-2022-0366.
- ⁴⁸Jesús Matesanz-García, Robert Christie, Fernando Tejero, David MacManus, and Alexander Heidebrecht. An automated approach for the aerodynamic design of close-coupled propulsion/airframe configurations. *1st Aerospace Europe Conference, Bordeaux, France*, 2020.
- ⁴⁹Jesús Matesanz-garcía, Tommaso Piovesan, and David G MacManus. Aerodynamic optimization of the exhaust system of an aft-mounted boundary layer ingestion propulsor. FP06-2022-matesanz-garcia. In 3AF, editor, *56th 3AF International Conference on Applied Aerodynamics 28 — 30 March 2022, Toulouse – France*, Toulouse, France, 2022.
- ⁵⁰Fernando Tejero, Jesús Matesanz-García, Avery Swarthout, and David MacManus. Towards the design and optimisation of future compact aero-engines: intake/fancowl trade-off investigation. FP04-AER. In *56th 3AF International Conference on Applied Aerodynamics*, Toulouse, France, 2022. 3AF: Association Aéronautique et Astronautique de France.
- ⁵¹Francisco Sanchez Moreno, David MacManus, Fernando Tejero, Jesús Matesanz-García, and Christopher Sheaf. Robustness of optimisation algorithms for transonic aerodynamic design. In *9th European Conference for Aeronautics and Space Sciences (EUCASS-3AF 2022)*, Lille, France, 2022.
- ⁵²Avery Swarthout, David MacManus, Fernando Tejero, Jesús Matesanz-García, Luca Boscagli, and Christopher Sheaf. A Comparative Assessment of Multi-Objective Optimisation Methodologies for Aero-Engine Nacelles. In *33rd Congress of the International Council of the Aeronautical Sciences (ICAS)*, Stockholm, Sweden, 2022.
- ⁵³Kenneth Suder. Overview of the NASA Environmentally Responsible Aviation Project’s Propulsion Technology Portfolio. In *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Atlanta, Georgia, jul 2012. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2012-4038.
- ⁵⁴Pavlos Rompokos, Sajal Kisson, Ioannis Roumeliotis, Devaiah Nalianda, Theoklis Nikolaidis, and Andrew Rolt. Liquefied Natural Gas for Civil Aviation. *Energies*, 13(22):5925, nov 2020, doi:10.3390/en13225925.
- ⁵⁵Pavlos Rompokos, Andrew Rolt, Devaiah Nalianda, Askin T. Isikveren, Capucine Senné, Tomas Gronstedt, and Hamidreza Abedi. Synergistic Technology Combinations for Future Commercial Aircraft Using Liquid Hydrogen. *Journal of Engineering for Gas Turbines and Power*, 143(7), jul 2021, doi:10.1115/1.4049694.

⁵⁶Andrew R. Gibson, David W. Hall, Mark H. Waters, Philippe J. Masson, Benjamin T. Schiltgen, Trevor J. Foster, and Jonathan D. Keith. The Potential and Challenge of TurboElectric Propulsion for Subsonic Transport Aircraft. *48th AIAA Aerospace Sciences Meeting*, (January):1–22, 2010, doi:10.2514/6.2010-276.

⁵⁷Michiel H Straathof, Michel J. L. van Tooren, Mark Voskuil, and Barry Koren. Aerodynamic Shape Parameterisation and Optimisation of Novel Configurations. *Proceedings of the 2008 Royal Aeronautical Society Annual Applied Aerodynamics Research Conference*, 2008.

⁵⁸David L. Rodriguez. Multidisciplinary Optimization Method for Designing Boundary-Layer-Ingesting Inlets. *Journal of Aircraft*, 46(3):883–894, 2009, doi:10.2514/1.38755.

⁵⁹Jai Ahuja. *A methodology for capturing the aero-propulsive coupling characteristics of boundary layer ingesting aircraft in conceptual design*. Phd thesis, Georgia Institute of Technology, 2020.

⁶⁰James Felder, Hyun Kim, Gerald Brown, and Joseph Kummer. An Examination of the Effect of Boundary Layer Ingestion on Turboelectric Distributed Propulsion Systems. *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, (January):1–26, 2011, doi:10.2514/6.2011-300.

⁶¹A. M. O. Smith and Howard E. Roberst. The Jet Airplane Utilizing Boundary Layer Air for Propulsion. *Journal of the Aeronautical Sciences*, 14(2):97–109, feb 1947, doi:10.2514/8.1273.

⁶²Dimitra Eirini Diamantidou, Md Lokman Hosain, and Konstantinos G. Kyprianidis. Recent Advances in Boundary Layer Ingestion Technology of Evolving Powertrain Systems. *Sustainability*, 14(3):1731, feb 2022, doi:10.3390/su14031731.

⁶³Leroy H. Smith. Wake ingestion propulsion benefit. *Journal of Propulsion and Power*, 9(1):74–82, jan 1993, doi:10.2514/3.11487.

⁶⁴Drewan Sanders. *Boundary Layer Ingestion Performance Assessment with Application to Business Jets*. Phd thesis, Cranfield University, 2018.

⁶⁵R. H. Liebeck. Design of the Blended Wing Body Subsonic Transport. *Journal of Aircraft*, 41(1):10–25, jan 2004, doi:10.2514/1.9084.

⁶⁶May-fun Liou, Hyoungjin Kim, Byungjoon Lee, Meng-sing Liou, and Nasa Glenn. Aerodynamic Design of the Hybrid Wing Body Propulsion- Airframe Integration. In *35th AIAA Applied Aerodynamics Conference*, number June, pages 1–17, Denver, CO, 2017. doi:10.2514/6.2017-3411.

⁶⁷Hyun Dae Kim, James L. Felder, Michael T. Tong, Jeffrey J. Berton, and William J. Haller. Turboelectric distributed propulsion benefits on the N3-X vehicle. *Aircraft Engineering and Aerospace Technology*, 86(6):558–561, sep 2014, doi:10.1108/AEAT-04-2014-0037.

⁶⁸J. I. Hileman, Z. S. Spakovszky, M. Drela, M. A. Sargeant, and A. Jones. Airframe Design for Silent Fuel-Efficient Aircraft. *Journal of Aircraft*, 47(3):956–969, may 2010, doi:10.2514/1.46545.

⁶⁹Michael Tong, Scott M. Jones, William J. Haller, and Robert F. Handschuh. Engine Conceptual Design Studies for a Hybrid Wing Body Aircraft. In *Volume 1: Aircraft Engine; Ceramics; Coal, Biomass and Alternative Fuels; Controls, Diagnostics and Instrumentation; Education; Electric Power; Awards and Honors*, pages 109–117. ASMEDC, jan 2009, doi:10.1115/GT2009-59568.

⁷⁰Ronald T. Kawai, Douglas M. Friedman, and Leonel Serrano. Blended Wing Body (BWB) Boundary Layer Ingestion (BLI) Inlet Configuration and System Studies. Technical report, NASA, 2006.

⁷¹Hyoungjin Kim and Meng-Sing Liou. Flow Simulation of N2B Hybrid Wing Body Configuration. In *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace*

Exposition, number January 2012, Nashville, Tennessee, jan 2012. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2012-838.

⁷²Irian Ordaz, Sriram K. Rallabhandi, Eric J. Nielsen, and Boris Diskin. Mitigation of Engine Inlet Distortion through Adjoint-Based Design. *35th AIAA Applied Aerodynamics Conference*, (June):1–12, 2017, doi:10.2514/6.2017-3410.

⁷³Tze Sing Tse and Cesare A. Hall. Flow Field and Power Balance of a Distributed Aft-fuselage Boundary Layer Ingesting Aircraft. In *AIAA Propulsion and Energy 2020 Forum*, VIRTUAL EVENT, aug 2020. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2020-3779.

⁷⁴Stefan Stückl, Artur Mirzoyan, and Askin T. Isikveren. DisPURSAL D1.2 – Report on the Technology Roadmap for 2035. Technical Report April, 2015.

⁷⁵Drewan S. Sanders and Panagiotis Laskaridis. Full-Aircraft Energy-Based Force Decomposition Applied to Boundary-Layer Ingestion. *AIAA Journal*, pages 1–17, 2020, doi:10.2514/1.j058695.

⁷⁶Anaïs Luisa Habermann, Anubhav Gokhale, and Mirko Hornung. Numerical investigation of the effects of fuselage upsweep in a propulsive fuselage concept. *CEAS Aeronautical Journal*, 12(1):173–189, jan 2021, doi:10.1007/s13272-020-00487-2.

⁷⁷E. J. Gunn and C. A. Hall. Aerodynamics of boundary layer ingesting fans. *Proceedings of the ASME Turbo Expo*, 1A:1–13, 2014, doi:10.1115/GT2014-26142.

⁷⁸Alejandro Castillo Pardo and Cesare A Hall. Aerodynamics of Boundary Layer Ingesting Fuselage Fans. *Journal of Turbomachinery*, 143(4), 2021, doi:10.1115/1.4049918.

⁷⁹Mark Drela. Power Balance in Aerodynamic Flows. *AIAA Journal*, 47(7):1761–1771, 2009, doi:10.2514/1.42409.

⁸⁰A. L. Habermann, J. Bijewitz, A. Seitz, and M. Hornung. Performance bookkeeping for aircraft configurations with fuselage wake-filling propulsion integration. *CEAS Aeronautical Journal*, 11(2):529–551, 2020, doi:10.1007/s13272-019-00434-w.

⁸¹Miguel Ángel Aguirre. *Exergy analysis of innovative aircraft with aeropropulsive coupling*. Phd, Institut Supérieur de l’Aéronautique et de l’Espace, SUPAERO, 2022.

⁸²Sven Schmitz. Drag Decomposition Using Partial-Pressure Fields in the Compressible Navier–Stokes Equations. *AIAA Journal*, 57(5):2030–2038, may 2019, doi:10.2514/1.J057701.

⁸³Kazuhiro Kusunose and James P. Crowder. Extension of Wake-Survey Analysis Method to Cover Compressible Flows. *Journal of Aircraft*, 39(6):954–963, nov 2002, doi:10.2514/2.3048.

⁸⁴Luigi Paparone and Renato Tognaccini. Computational Fluid Dynamics-Based Drag Prediction and Decomposition. *AIAA Journal*, 41(9):1647–1657, sep 2003, doi:10.2514/2.7300.

⁸⁵K. Oswatitsch. *Gas dynamics*. New York: Academic Press, 1956.

⁸⁶E. C. Maskell. Progress towards a method for the measurement of the components of the drag of a wing of finite span, RAE Technical Report No. 72232. Technical report, RAE, 1973.

⁸⁷J. E. Hackett, J. E. Williams, and J. Patrick. Wake Traverses Behind Production Cars and Their Interpretation. feb 1985, doi:10.4271/850280.

⁸⁸Kazuhiro Kusunose. Drag prediction based on a wake-integral method. In *16th AIAA Applied Aerodynamics Conference*. American Institute of Aeronautics and Astronautics, jun 1998, doi:10.2514/6.1998-2723.

⁸⁹Alejandra Uranga, Mark Drela, David K. Hall, and Edward M. Greitzer. Analysis of the aerodynamic benefit from boundary layer ingestion for transport aircraft. *AIAA Journal*, 56(11):4271–4281, 2018, doi:10.2514/1.J056781.

⁹⁰Brennan T. Blumenthal, Alaa A. Elmiligui, Karl A. Geiselhart, Richard L. Campbell, Mark D. Maughmer, and Sven Schmitz. Computational Investigation of a Boundary-Layer-Ingestion Propulsion System. *Journal of Aircraft*, 55(3):1–13, 2018, doi:10.2514/1.C034454.

⁹¹D K Hall, E. M. Greitzer, and C. S. Tan. Analysis of fan stage conceptual design attributes for boundary layer ingestion. *Journal of Turbomachinery*, 139(7):1–10, 2017, doi:10.1115/1.4035631.

⁹²Teng Cao, Nagabhushana Rao Vadlamani, Paul G. Tucker, Angus R. Smith, Michal Slaby, and Christopher T.J. Sheaf. Fan-Intake interaction under high incidence. *Journal of Engineering for Gas Turbines and Power*, 139(4):1–10, 2017, doi:10.1115/1.4034701.

⁹³Luca Boscagli, Robert Christie, and David G MacManus. Methodology assessment for the design and analysis of aero-engine short intakes. In 3AF, editor, *Aerospace Europe Conference, Bordeaux, France*, Bordeaux, 2020.

⁹⁴B. J. Deneys Schreiner, Fernando Tejero, David G. MacManus, and Christopher Sheaf. Robust aerodynamic design of nacelles for future civil aero-engines. *Proceedings of the ASME Turbo Expo*, 1(M):1–11, 2020, doi:10.1115/GT2020-14470.

⁹⁵Andrea Magrini, Denis Buosi, and Ernesto Benini. Maximisation of installed net resulting force through multi-level optimisation of an ultra-high bypass ratio engine nacelle. *Aerospace Science and Technology*, 119:107169, dec 2021, doi:10.1016/j.ast.2021.107169.

⁹⁶Justin S. Gray, Charles A. Mader, Gaetan K.W. Kenway, and Joaquim R.R.A. Martins. Modeling boundary layer ingestion using a coupled aeropropulsive analysis. *Journal of Aircraft*, 55(3):1191–1199, 2018, doi:10.2514/1.C034601.

⁹⁷Anais Luisa Habermann, Rebecca Zahn, Arne Seitz, and Mirko Hornung. Multidimensional Parametric Study of a Propulsive Fuselage Concept Using OpenFOAM. AIAA 2020-2754. In *AIAA AVIATION 2020 FORUM, VIRTUAL EVENT*, jun 2020. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2020-2754.

⁹⁸Byung Joon Lee, Meng Sing Liou, and Chongam Kim. Adjoint Based Design Approach for Boundary Layer Ingestion Offset Intake. In *19th AIAA Computational Fluid Dynamics*, number June, pages 1–19, San Antonio, Texas, jun 2009. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2009-3804.

⁹⁹Ioannis Goulos, David MacManus, and Christopher Sheaf. Civil turbofan engine exhaust aerodynamics: Impact of fan exit flow characteristics. *Aerospace Science and Technology*, 93:105181, 2019, doi:10.1016/j.ast.2019.05.033.

¹⁰⁰William T. Cousins and Milt W. Davis. Evaluating complex inlet distortion with a parallel compressor model: Part 1 - Concepts, theory, extensions, and limitations. *Proceedings of the ASME Turbo Expo*, 1:1–12, 2011, doi:10.1115/GT2011-45067.

¹⁰¹Milt W. Davis and William T. Cousins. Evaluating Complex Inlet Distortion With a Parallel Compressor Model: Part 2—Applications to Complex Patterns. In *Volume 1: Aircraft Engine; Ceramics; Coal, Biomass and Alternative Fuels; Wind Turbine Technology*, pages 13–23. ASMEDC, jan 2011, doi:10.1115/GT2011-45068.

¹⁰²Vinicius T. Silva, Anders Lundblad, Olivier Petit, and Carlos Xisto. Multipoint Aerodynamic Design of Ultrashort Nacelles for Ultrahigh-Bypass-Ratio Engines. *Journal of Propulsion and Power*, pages 1–18, jan 2022, doi:10.2514/1.B38497.

¹⁰³Frank E Marble. Section C. Three-Dimensional Flow in Turbomachines. In Princeton University, editor, *Aerodynamics of Turbines and Compressors. (HSA-1)*, pages 83–166. Princeton University Press, dec 1964.

- ¹⁰⁴Vassilios Pachidis, Pericles Pilidis, Ioannis Templalexis, Theodosios Korakianitis, and Petros Kotsiopoulos. Prediction of engine performance under compressor inlet flow distortion using streamline curvature. *Journal of Engineering for Gas Turbines and Power*, 129(1):97–103, 2007, doi:10.1115/1.2363414.
- ¹⁰⁵Simon J. Gallimore. Viscous Throughflow Modeling of Axial Compressor Bladerows Using a Tangential Blade Force Hypothesis. *Journal of Turbomachinery*, 120(4):662–670, oct 1998, doi:10.1115/1.2841775.
- ¹⁰⁶Jean-François Simon. *Contribution to Throughflow Modelling for Axial Flow Turbomachines*. Phd thesis, Université de Liège, 2007.
- ¹⁰⁷George Kiwada. *Development of a body force description for compressor stability assessment*. Msc, Massachusetts Institute of Technology, 2008.
- ¹⁰⁸R. V. Chima. A Three-Dimensional Unsteady CFD Model of Compressor Stability. In *Volume 6: Turbomachinery, Parts A and B*, pages 1157–1168. ASMEDC, jan 2006, doi:10.1115/GT2006-90040.
- ¹⁰⁹Maximilian Lewis Brand. *An improved blade passage model for estimating off-design axial compressor performance*. Msc, Massachusetts Institute of Technology, 2013.
- ¹¹⁰Y. Gong. *A Computational Model for Rotating Stall and Inlet Distortion in Multistage Compressors*. Phd thesis, Massachusetts Institute of Technology, 1998.
- ¹¹¹D. K. Hall. *Analysis of civil aircraft propulsors with boundary layer ingestion*. Phd thesis, Massachusetts Institute of Technology, 2015.
- ¹¹²Emmanuel Benichou, Guillaume Dufour, Yannick Bousquet, Nicolas Binder, Aurélie Ortolan, and Xavier Carbonneau. Body force modeling of the aerodynamics of a low-speed fan under distorted inflow. *International Journal of Turbomachinery, Propulsion and Power*, 4(3), 2019, doi:10.3390/ijtp4030029.
- ¹¹³Teng Cao, Paul Hield, and Paul G. Tucker. Hierarchical immersed boundary method with smeared geometry. *Journal of Propulsion and Power*, 33(5):1151–1163, 2017, doi:10.2514/1.B36190.
- ¹¹⁴Martin O. L. Hansen. *Aerodynamics of Wind Turbines*. Earthscan, London, second edition, 2008.
- ¹¹⁵Arne Seitz, Antes Lisa Habermann, and Martijn van Sluis. Optimality considerations for propulsive fuselage power savings. *Journal of Aerospace Engineering*, 2020, doi:https://journals.sagepub.com/doi/10.1177/0954410020916319.
- ¹¹⁶Hans Mårtensson and Martin Laban. Design and Performance of a Boundary Layer Ingesting Fan. In *Volume 2A: Turbomachinery*, pages 1–9. American Society of Mechanical Engineers, sep 2020, doi:10.1115/GT2020-15479.
- ¹¹⁷David L Daggett and Karl A Geiselhart. Ultra efficient engine technology systems integration and environmental assessment (No. NASA/CR-2002-211754). Technical report, NASA, 2002.
- ¹¹⁸David L. Daggett, Stephen T. Brown, and Ron T. Kawai. Ultra-efficient Engine Diameter Study (No. NASA/CR—2003-212309). Technical report, NASA, 2003.
- ¹¹⁹Andreas Peters. *Ultra-short nacelles for low fan pressure ratio propulsors*. Phd thesis, Massachusetts Institute of Technology, 2014.
- ¹²⁰John Seddon and E Laurie Goldsmith. *Intake aerodynamics*. Wiley–Blackwell, 2nd edition, 1998.

¹²¹G Jefford, C. *the Birth of Tornado Royal Air Force Historical Society 2*. Royal Air Force Historical Society, 2002.

¹²²John James Otter. *Aerodynamics and Performance of Civil Aero-Engine Exhaust Systems*. PhD thesis, Cranfield University, 2018.

¹²³Fernando Tejero, David MacManus, and Christopher Sheaf. Impact of Droop and Scarf on the Aerodynamic Performance of Compact Aero-Engine Nacelles. In *AIAA Scitech 2020 Forum*, Orlando, Florida, jan 2020. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2020-1522.

¹²⁴Fernando Tejero, Ioannis Goulos, David MacManus, and Christopher Sheaf. Effects of Aircraft Integration on Compact Nacelle Aerodynamics. In *AIAA Scitech 2020 Forum*, Orlando, Florida, jan 2020. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2020-2225.

¹²⁵Abhijit Guha. Optimum Fan Pressure Ratio for Bypass Engines with Separate or Mixed Exhaust Streams. *Journal of Propulsion and Power*, 17(5):1117–1122, sep 2001, doi:10.2514/2.5852.

¹²⁶I. Goulos, J. Otter, T. Stankowski, D. MacManus, N. Grech, and C. Sheaf. Design optimisation of separate-jet exhausts for the next generation of civil aero-engines. *Aeronautical Journal*, 122(1256):1586–1605, 2018, doi:10.1017/aer.2018.95.

¹²⁷John J. Otter, Ioannis Goulos, Robert Christie, and David G. MacManus. Design and analysis of non-axisymmetric installed aero-engine exhaust systems. *Aerospace Science and Technology*, 106:106210, nov 2020, doi:10.1016/j.ast.2020.106210.

¹²⁸Josep Hueso Rebassa, Fernando Tejero, John Otter, Ioannis Goulos, and David G MacManus. Multi-Fidelity Assessment of Exhaust Systems for Complete Engine-Airframe Configurations. In *Aerospace Europe Conference 2020 (AEC2020)*, Bordeaux, France, 2020.

¹²⁹Fernando Tejero, David G MacManus, Josep Hueso Rebassa, Ioannis Goulos, and Christopher Sheaf. Techniques, Aerodynamic optimisation of civil aero-engine nacelles by dimensionality reduction and multi-fidelity. In *56th 3AF International Conference on Applied Aerodynamics*, Toulouse, France, 2022. 3AF: Association Aéronautique et Astronautique de France.

¹³⁰Fernando Tejero, David G. MacManus, and Christopher Sheaf. Surrogate-based aerodynamic optimisation of compact nacelle aero-engines. *Aerospace Science and Technology*, 93:105207, 2019, doi:10.1016/j.ast.2019.05.059.

¹³¹Fernando Tejero, Robert Christie, David MacManus, and Christopher Sheaf. Non-axisymmetric aero-engine nacelle design by surrogate-based methods. *Aerospace Science and Technology*, 117:106890, oct 2021, doi:10.1016/j.ast.2021.106890.

¹³²Luca Menegozzo and Ernesto Benini. Boundary Layer Ingestion Propulsion: A Review on Numerical Modeling. *Journal of Engineering for Gas Turbines and Power*, 142(12), dec 2020, doi:10.1115/1.4048174.

¹³³Jonathan Gladin, Jonathan Sands, Brian Kestner, and Dimitri Mavris. Effects of Boundary Layer Ingesting (BLI) Propulsion Systems on Engine Cycle Selection and HWB Vehicle Sizing. In *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, Nashville, Tennessee, jan 2012. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2012-837.

¹³⁴Esteban A. Valencia, Devaiah Nalianda, Panagiotis Laskaridis, and Riti Singh. Methodology to assess the performance of an aircraft concept with distributed propulsion and boundary layer ingestion using a parametric approach. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 229(4):682–693, 2015, doi:10.1177/0954410014539291.

- ¹³⁵C. Goldberg, D. Nalianda, D. MacManus, P. Pilidis, and J. Felder. Method for simulating the performance of a boundary layer ingesting propulsion system at design and off-design. *Aerospace Science and Technology*, 78:312–319, 2018, doi:10.1016/j.ast.2018.04.026.
- ¹³⁶Shishir A. Pandya, Alejandra Uranga, Alejandro Espitia, and Arthur Huang. Computational Assessment of the Boundary Layer Ingesting Nacelle Design of the D8 Aircraft. In *52nd Aerospace Sciences Meeting*, National Harbor, Maryland, jan 2014. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2014-0907.
- ¹³⁷Alejandra Uranga, Mark Drela, Edward M. Greitzer, David K. Hall, Neil A. Titchener, Michael K. Lieu, Nina M. Siu, Cécile Casses, Arthur C. Huang, Gregory M. Gatlin, and Judith A. Hannon. Boundary Layer Ingestion Benefit of the D8 Transport Aircraft. *AIAA Journal*, 55(11):3693–3708, nov 2017, doi:10.2514/1.J055755.
- ¹³⁸Jai Ahuja and Dimitri N. Mavris. Sensitivity of Boundary Layer Ingestion Effects to Tube and Wing Airframe Design Features. In *AIAA Scitech 2020 Forum*, Orlando, Florida, jan 2020. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2020-1523.
- ¹³⁹B J Lee, May-Fun Liou, and Meng-Sing Liou. Conceptual Aerodynamic Design of a Tail-Cone Thruster System Under Axi-Symmetric Inlet Distortion. In *Volume 2A: Turbomachinery*, pages 1–17. American Society of Mechanical Engineers, jun 2018, doi:10.1115/GT2018-75861.
- ¹⁴⁰European Aviation Safety Agency (EASA). Type Certificate Data Sheet for AIRBUS A318 - A319 - A320 - A321. No. EASA.A.064. Technical Report 34, European Aviation Safety Agency, Blagnac, France, 2018.
- ¹⁴¹Feng Zhu and Ning Qin. Intuitive class/shape function parameterization for airfoils. *AIAA Journal*, 52(1):17–25, 2014, doi:10.2514/1.J052610.
- ¹⁴²Brenda Kulfan and John Bussioletti. "Fundamental" Parametric Geometry Representations for Aircraft Component Shapes. *11th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference*, (September):1–45, 2006, doi:10.2514/6.2006-6948.
- ¹⁴³Robert Christie, Alexander Heidebrecht, David MacManus, and Alexander Heidebrecht. An Automated Approach to Nacelle Parameterization Using Intuitive Class Shape Transformation Curves. *Journal of Engineering for Gas Turbines and Power*, 139(6):062601, 2017, doi:10.1115/1.4035283.
- ¹⁴⁴Yuan Shin Lee, Yawei Ma, and George Jegadesh. Rolling-ball method and contour marching approach to identifying critical regions for complex surface machining. *Computers in Industry*, 41(2):163–180, 2000, doi:10.1016/S0166-3615(99)00042-1.
- ¹⁴⁵ISO International Standard 2533-1975. Standard Atmosphere First Edition. Technical report, ISO, Geneva, Switzerland, 1978.
- ¹⁴⁶Daniel Raymer. *Aircraft Design: A Conceptual Approach*. American Institute of Aeronautics and Astronautics, Inc., Washington, DC, sixth edit edition, sep 2018.
- ¹⁴⁷Egbert Torenbeek. *Advanced Aircraft Design*. John Wiley & Sons, Ltd, 2013.
- ¹⁴⁸Egbert Torenbeek. *Synthesis of Subsonic Airplane Design: An introduction to the preliminary design of subsonic general aviation and transport aircraft, with emphasis on layout, aerodynamic design, propulsion and performance*. Springer Netherlands, Dordrecht, 1982.
- ¹⁴⁹David K. Hall, Arthur C. Huang, Alejandra Uranga, Edward M. Greitzer, Mark Drela, and Sho Sato. Boundary layer ingestion propulsion benefit for transport aircraft. *Journal of Propulsion and Power*, 33(5):1118–1129, 2017, doi:10.2514/1.B36321.

- ¹⁵⁰Frank M. White. *Viscous Fluid Flow*. McGraw-Hill Education, Singapore, third edit edition, 2006.
- ¹⁵¹Julian Bijewitz, Arne Seitz, Askin T. Isikveren, and Mirko Hornung. Multi-disciplinary design investigation of propulsive fuselage aircraft concepts. *Aircraft Engineering and Aerospace Technology*, 88(2):257–267, mar 2016, doi:10.1108/AEAT-02-2015-0053.
- ¹⁵²Definitions Panel of the Aeronautical Research Council. Report of the definitions panel on the definitions of the thrust of a jet engine and of the internal drag of a ducted body. Technical Report C.P. 190. Technical report, Aeronautical Research Council, London, UK, 1955.
- ¹⁵³Santiago Ramirez-Rubio and David G MacManus. Framework for estimation of nacelle drag on isolated aero-engines with separate jets. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 234(14):2025–2040, nov 2020, doi:10.1177/0954410020922928.
- ¹⁵⁴F. R. Menter. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA Journal*, 32:1598–1605, 1994, doi:10.2514/3.12149.
- ¹⁵⁵ANSYS INC. ANSYS Fluent User’s Guide. Release 19.1. Technical report, ANSYS Inc, Canonsburg, PA, USA, 2019.
- ¹⁵⁶N. A. Cumpsty and J. H. Horlock. Averaging Nonuniform Flow for a Purpose. *Journal of Turbomachinery*, 128(1):120, 2005, doi:10.1115/1.2098807.
- ¹⁵⁷ANSYS INC. ANSYS Fluent UDF Manual. Release 19.1. Technical report, ANSYS Inc, Canonsburg, PA, USA, 2019.
- ¹⁵⁸Ismail B. Celik, Urmila Ghia, Patrick J. Roache, Christopher J. Freitas, Hugh Coleman, and Peter E. Raad. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering, Transactions of the ASME*, 130(7):0780011–0780014, 2008, doi:10.1115/1.2960953.
- ¹⁵⁹Matthew Robinson, David G. MacManus, Alexander Heidebrecht, and Nicholas Grech. An optimization method for nacelle design. *AIAA SciTech Forum - 55th AIAA Aerospace Sciences Meeting*, (January):1–17, 2017, doi:10.2514/6.2017-0708.
- ¹⁶⁰Kalyanmoy Deb, Associate Member, Amrit Pratap, Sameer Agarwal, and T Meyarivan. A fast and elitist multi-objective genetic algorithm: NSGAII. 6(2):182–197, 2002.
- ¹⁶¹David Hadka. Platypus manual, 2018.
- ¹⁶²Eckart Zitzler and Simon Künzli. Indicator-based selection in multiobjective search. *Lecture Notes in Computer Science (including subseries Lecture Notes in Artificial Intelligence and Lecture Notes in Bioinformatics)*, 3242:832–842, 2004, doi:10.1007/978-3-540-30217-9_84.
- ¹⁶³Margarita Reyes Sierra and Carlos A. Coello Coello. Improving PSO-based Multi-Objective optimization using crowding, mutation and ϵ -dominance. *Lecture Notes in Computer Science*, 3410:505–519, 2005, doi:10.1007/978-3-540-31880-4_35.
- ¹⁶⁴J. C. Helton and F. J. Davis. Latin hypercube sampling and the propagation of uncertainty in analyses of complex systems. *Reliability Engineering and System Safety*, 81(1):23–69, 2003, doi:10.1016/S0951-8320(03)00058-9.
- ¹⁶⁵Kalyanmoy Deb. Multi-objective Genetic Algorithms: Problem Difficulties and Construction of Test Problems. *Evolutionary Computation*, 7(3):205–230, sep 1999, doi:10.1162/evco.1999.7.3.205.

- ¹⁶⁶James Felder, Hyun Kim, and Gerald Brown. Turboelectric Distributed Propulsion Engine Cycle Analysis for Hybrid-Wing-Body Aircraft. In *47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition*, Orlando, Florida, jan 2009. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2009-1132.
- ¹⁶⁷C. Pornet and A.T. Isikveren. Conceptual design of hybrid-electric transport aircraft. *Progress in Aerospace Sciences*, 79:114–135, nov 2015, doi:10.1016/j.paerosci.2015.09.002.
- ¹⁶⁸W. L. MacMillan. *Development of a modular type computer program for the calculation of gas turbine off design performance*. Thesis, Cranfield University, 1974.
- ¹⁶⁹Palmer. J. The Turbomatch Scheme For Aero/Industrial Gas Turbine Engine Design Point/Off Design Performance Calculation. Technical report, Cranfield University, Cranfield, Bedfordshire, MK43 0AL, UK, 1999.
- ¹⁷⁰Tomasz Piotr Stankowski. *Aerodynamics of Propulsion System Integration for Modern Aero-Engines*. Phd, Cranfield University, 2016.
- ¹⁷¹Dong C. Liu and Jorge Nocedal. On the limited memory BFGS method for large scale optimization. *Mathematical Programming*, 45(1-3):503–528, aug 1989, doi:10.1007/BF01589116.
- ¹⁷²Nicholas Cumpsty and Andrew Heyes. *Jet Propulsion. A simple guide to the aerodynamics and thermodynamic design and performance of jet engines*. Cambridge University Press, New York, third edit edition, 1998.
- ¹⁷³Alexios Alexiou, Nikolaos Aretakis, Ioannis Kolias, and Konstantinos Mathioudakis. Novel Aero-Engine Multi-Disciplinary Preliminary Design Optimization Framework Accounting for Dynamic System Operation and Aircraft Mission Performance. *Aerospace*, 8(2):49, feb 2021, doi:10.3390/aerospace8020049.
- ¹⁷⁴European Aviation Safety Agency (EASA). TYPE-CERTIFICATE DATA SHEET for CFM56-5 series engines. No. E.067. Technical Report 02, EASA, Paris, France, 2018.
- ¹⁷⁵Alan H. Epstein. Aeropropulsion for Commercial Aviation in the Twenty-First Century and Research Directions Needed. *AIAA Journal*, 52(5):901–911, may 2014, doi:10.2514/1.J052713.
- ¹⁷⁶Arne Seitz, Anaïs Luisa Habermann, and Martijn van Sluis. Optimality considerations for propulsive fuselage power savings. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 235(1):22–39, 2021, doi:10.1177/0954410020916319.
- ¹⁷⁷J. D. Hubble and R. E. Smith. Evaluation of an Airjet Distortion Generator Used to Produce Steady- State, Total-Pressure Distortion at the Inlet of a General Electric F101-GE-100 Turbofan Engine. ADA072743. Technical report, ARO INC ARNOLD AFS TN, USA Air Force, Tennessee, 1979.
- ¹⁷⁸Robert Christie. Internal communication 14-9-2022 (Cranfield, Bedfordshire, MK43 0AL, UK), 2022.
- ¹⁷⁹Julian Bijewitz, Arne Seitz, Mirko Hornung, and Askin T. Isikveren. Progress in Optimizing the Propulsive Fuselage Aircraft Concept. *Journal of Aircraft*, 54(5):1–11, 2017, doi:10.2514/1.C034002.
- ¹⁸⁰ANSYS Inc. ANSYS Turbogrid Tutorials. Release 19.1. Technical report, ANSYS Inc., Canonsburg, PA, USA, 2019.
- ¹⁸¹B. J. Lee, May Fun Liou, and Meng Sing Liou. Conceptual aerodynamic design of a tail-cone thruster system under axi-symmetric inlet distortion. *Proceedings of the ASME Turbo Expo*, 2A-2018:1–17, 2018, doi:10.1115/GT2018-75861.

- ¹⁸²J. R. Taylor. *An introduction to Error Analysis: The study of uncertainties in physical measurements*. Mill Valley, Calif University Science Books, 1997.
- ¹⁸³Paul G. Constantine, Eric Dow, and Qiqi Wang. Active subspace methods in theory and practice: Applications to kriging surfaces. *SIAM Journal on Scientific Computing*, 36(4):A1500–A1524, 2014, doi:10.1137/130916138.
- ¹⁸⁴Antonio Martínez Fernández and Howard Smith. Effect of a fuselage boundary layer ingesting propulsor on airframe forces and moments. *Aerospace Science and Technology*, 100:105808, may 2020, doi:10.1016/j.ast.2020.105808.
- ¹⁸⁵Kyungjae Lee, Bohwa Lee, Sanghun Kang, Sooseok Yang, and Daesung Lee. Inlet Distortion Test with Gas Turbine Engine in the Altitude Engine Test Facility. In *27th AIAA Aerodynamic Measurement Technology and Ground Testing Conference*, Chicago, Illinois, jun 2010. American Institute of Aeronautics and Astronautics, doi:10.2514/6.2010-4337.
- ¹⁸⁶Y. Colin, B. Aupoix, J.F. Boussuge, and P. Chanez. Numerical Simulation of the Distortion Generated by Crosswind Inlet Flows. In *18th ISABE 2007 ISABE-2007-1210*, pages 1–13, Beijing, China, 2007.
- ¹⁸⁷D. K. Hall, E. M. Greitzer, and C. S. Tan. Analysis of fan stage conceptual design attributes for boundary layer ingestion. *Journal of Turbomachinery*, 139(7):1–10, 2017, doi:10.1115/1.4035631.
- ¹⁸⁸DAVID C. Wilcox. *Turbulence Modelling for CFD*. DCW Industries, Inc, San Diego, California, 3rd edition, 2006.
- ¹⁸⁹B.E. Launder and D.B. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, 3(2):269–289, mar 1974, doi:10.1016/0045-7825(74)90029-2.
- ¹⁹⁰J. O. Hirschfelder, Charles F. Curtiss, and R. Byron Bird. *Molecular Theory Of Gases And Liquids*. 1954.
- ¹⁹¹W. Sutherland. The Viscosity of Gases and Molecular Forces. *Philosophical Magazine*, 5((36)):507–531, 1893.
- ¹⁹²John D. Anderson. *Modern compressible flow: with historical perspective*. McGraw-Hill Education, Singapore, third edit edition, 2004.
- ¹⁹³David Lovely and Robert Haimes. Shock detection from computational fluid dynamics results. In *14th Computational Fluid Dynamics Conference*, Norfolk, VA, U.S.A., nov 1999. American Institute of Aeronautics and Astronautics, doi:10.2514/6.1999-3285.
- ¹⁹⁴S. L. Dixon and Cesare Hall. *Fluid Mechanics and Thermodynamics of Turbomachinery*. Elsevier, 7th edition, 2013.
- ¹⁹⁵Arne Seitz, Fabian Peter, Julian Bijewitz, Anaïs Habermann, Zdobyslaw Goraj, Mariusz Kowalski, Alejandro Castillo Pardo, Cesare Hall, Frank Meller, Rasmus Merkle, Olivier Petit, Sebastian Samuelsson, Biagio Della Corte, Martijn Van Sluis, Guido Wortmann, and Martin Dietz. Concept Validation Study for Fuselage Wake-Filling Propulsion Integration. *31st Congress of the International Council of the Aeronautical Sciences, ICAS 2018*, 2018.
- ¹⁹⁶Dean R Eklund, Wright-patterson Air Force Base, Vicente J Romero, Jeffrey A Pearce, and Nicholas S Keim. Simulation Credibility. Advances in Verification, Validation and Uncertainty Quantification. Technical Report November 2016, NASA and JANNAF, 2016.
- ¹⁹⁷Tomasz P Stańkowski, David G MacManus, Christopher TJ Sheaf, and Robert Christie. Aerodynamics of aero-engine installation. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 230(14):2673–2692, dec 2016, doi:10.1177/0954410016630332.

¹⁹⁸John J. Otter, Ioannis Goulos, David G. MacManus, and Michal Slaby. Aerodynamic Analysis of Civil Aeroengine Exhaust Systems Using Computational Fluid Dynamics. *Journal of Propulsion and Power*, 34(5):1152–1165, sep 2018, doi:10.2514/1.B36867.

¹⁹⁹Amaury Awes, Renaud Daon, Guillaume Dufour, Xavier Carbonneau, Julien Marty, and Raphaël Barrier. Unsteady body force methodology for fan operability assessment under clean and distorted inflow conditions. *AIAA Scitech 2021 Forum*, (January):1–13, 2021, doi:10.2514/6.2021-0388.

APPENDIX A

Governing equations

This Appendix summarises the main equations behind the Computational Fluid Dynamics (CFD) approach. A Favre-Averaged Navier Stokes approach is used for all the CFD evaluations of this work. The definition of the different reference frames is also included in this appendix.

A.1 Definition of the reference frame

A.1.1 2D axisymmetric reference frame

For the 2D axisymmetric cases a conventional cylindrical coordinate system (Fig. A.1) is applied (Eqs. A.1-A.3). The axial coordinate of the system is referred as x , the radial as r , and the azimuthal as θ . The reference operation plane of the 2D axisymmetric model is the x-r plane. All the off-plane or azimuthal components are constant under this model. This means that any radial and axial position will be the same independently on the θ component. This also affects the fluid properties (ρ, P, T) and the components of the velocity vector ($\mathbf{V} = (V_x, V_r, V_\theta)$). The origin of the 2D axisymmetric coordinate system is placed axially at the fan face location of the aft-mounted propulsor (Fig. A.2). The radial component r starts from the rotation axis.

$$x_c = x \tag{A.1}$$

$$y_c = r \cdot \sin(\theta) \tag{A.2}$$

$$z_c = r \cdot \cos(\theta) \tag{A.3}$$

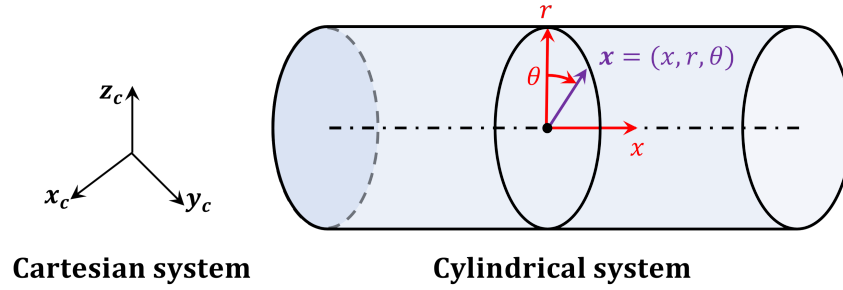


FIGURE A.1: Basic definition of the cylindrical coordinates system applied in the 2D axisymmetric models

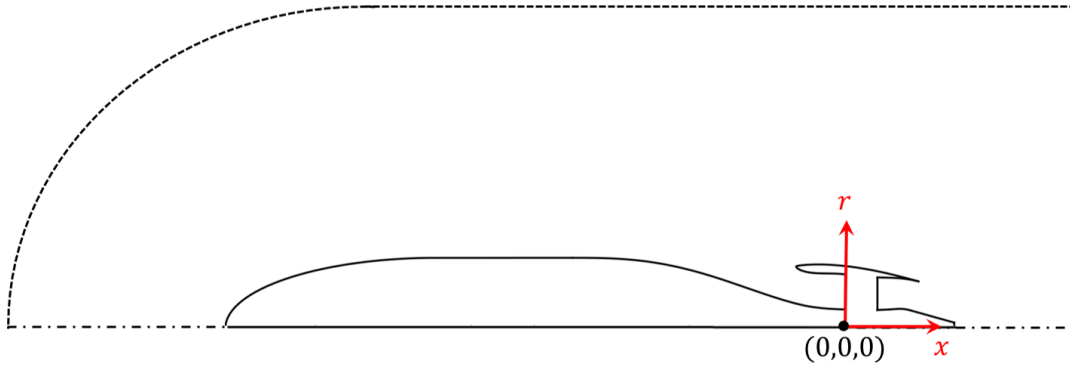


FIGURE A.2: Schematic of the origin of the 2D axisymmetric reference frame

A.1.2 3D reference frame

Three reference systems are used for the 3D models (Fig. A.3): the aircraft reference frame, the fan reference frame, and the wind reference frame.

The aircraft reference frame is the absolute reference frame of the CFD model. The domain is centred on the origin of this coordinate system. The origin of the system is based on the center of mass of the reference aircraft²⁵. The x_{ac} component of the reference system is oriented by the central axis of the aircraft and set positive opposite to the aircraft axial motion. The z_{az} component is defined as the vertical component. This vector is normal to the aircraft belly at the symmetry plane and defined positive with the direction towards the suction side of the wing. The off-plane y_{ac} component of the system is defined to be normal to the other two components to close the Cartesian system ($\mathbf{y}_{ac} = \mathbf{x}_{ac} \times \mathbf{z}_{ac}$).

The fan reference frame has its center at the fan face interface axial position at the center of the circumference defined by the fan blade tip. The x_{fan} axis is oriented normal to the plane containing the foremost component of the blade leading edge and

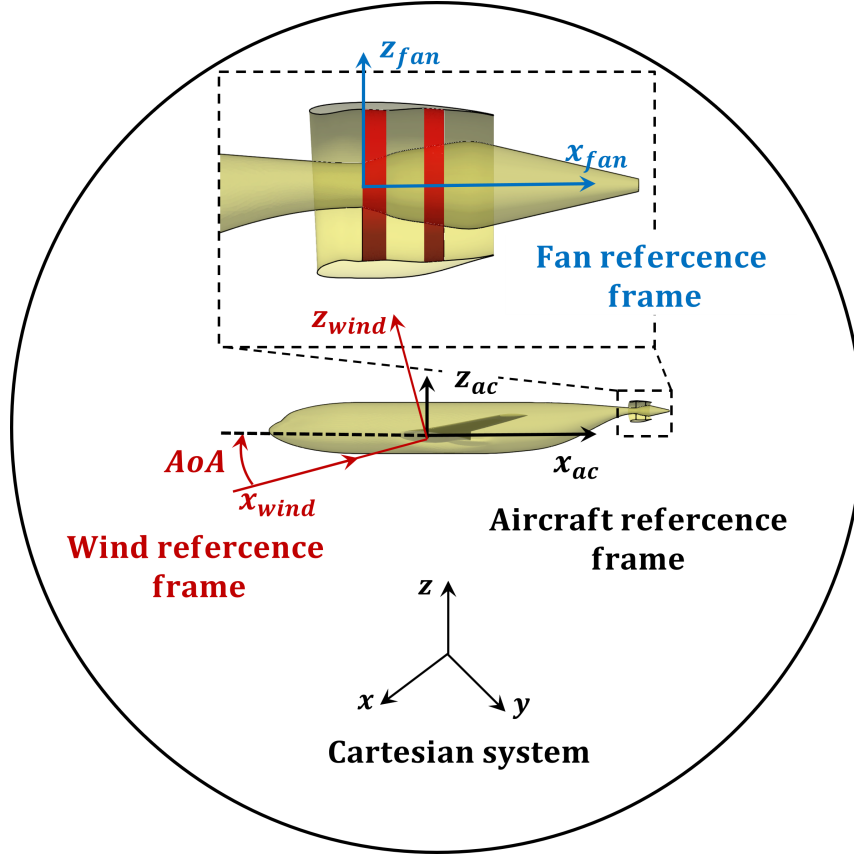


FIGURE A.3: Schematic of the 3D reference frames for the aircraft and the fan

defined positive towards the exhaust system. The z_{fan} component of the reference frame is defined normal to x_{fan} on the symmetry plane of the aircraft and positive oriented towards the crown of the nacelle. The off-plane y_{fan} component of the system is defined to be normal to the other two components to close the Cartesian system ($\mathbf{y}_{fan} = \mathbf{x}_{fan} \times \mathbf{z}_{fan}$). Internally, for the low order fan model source term computation the Cartesian system is transformed to the local Cylindrical equivalent (Eqs. A.1-A.3).

The wind reference frame is based on the relative orientation of the freestream flow to the aircraft. The reference frame is based on the aircraft center of mass defined for the reference aircraft²⁵. The x_{wind} axis is oriented and defined positive in the wind direction. For the present work, the only relative angle of the wind reference frame is the angle of attack (AoA). While the introduction of crosswind component is supported, it is not included in any of the models of this work. The z_{wind} axis is defined as normal to x_{wind} in the aircraft symmetry plane and positive normal to the ground. The off-plane y_{wind} component of the system is defined to be normal to the other two components to close the Cartesian system ($\mathbf{y}_{wind} = \mathbf{x}_{wind} \times \mathbf{z}_{wind}$).

A.2 Favre-averaged Navier-stokes equations

To model the compressible turbulent flow of the experiments, the Favre-Averaged Navier-Stokes equations (Eqs. A.4-A.7) are applied^{122,188}. This approach has three main equations: the conservation of mass (Eqs. A.4), the conservation of momentum (Eqs. A.5), and the conservation of energy A.6. When these equations are resolved the main fluid properties that can be extracted are the density (ρ), the velocity ($\mathbf{V}$), and the internal energy (e). To close the system¹⁸⁸ it is also necessary to define a gas model and its thermodynamic properties (Section A.3).

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \tilde{V}_i)}{\partial x_i} = 0 \quad (\text{A.4})$$

$$\frac{\partial \bar{\rho} \tilde{V}_i}{\partial t} + \frac{\partial \bar{\rho} \tilde{V}_i \tilde{V}_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_j} + \frac{\partial (\bar{t}_{ij} + \bar{\rho} \tau_{ij})}{\partial x_j} \quad (\text{A.5})$$

$$\begin{aligned} \frac{\partial \bar{\rho} ((\tilde{e} + \frac{1}{2} \tilde{V}_i \tilde{V}_i) + k)}{\partial t} + \frac{\partial \bar{\rho} \tilde{V}_j (\tilde{h} + \frac{1}{2} \tilde{V}_i \tilde{V}_i) + k}{\partial x_j} &= \frac{\partial}{\partial x_j} [\tilde{V}_i (\bar{t}_{ij} + \bar{\rho} \tau_{ij})] + \\ &\frac{\partial}{\partial x_j} \left[\left(\frac{\mu}{Pr_L} + \frac{\mu_T}{Pr_T} \right) \frac{\partial \tilde{h}}{\partial x_j} + (\mu + \sigma_k \mu_T) \frac{\partial k}{\partial x_j} \right] \end{aligned} \quad (\text{A.6})$$

$$\bar{t}_{ij} = 2\mu \bar{S}_{ij}, \bar{\rho} \tau_{ij} = 2\mu_T - \frac{2}{3} \bar{\rho} k \delta_{ij}, \bar{S}_{ij} = S_{ij} - \frac{1}{3} \frac{\partial \tilde{V}_k}{\partial x_k} \delta_{ij} \quad (\text{A.7})$$

In these equations (Eqs. A.4-A.7), the over tilde symbol in a variable (e.g. $\tilde{x}$) denotes a mass averaged value. The overline symbol (e.g. $\bar{x}$) is used to define a mean value^{122,188}. The δ_{ij} stands for the Kronecker delta.

The Favre-Averaged Navier-Stokes equations depend on the static pressure (p), the specific enthalpy (h), the molecular and eddy viscosities (μ and μ_T), the turbulent kinetic energy (k), and the laminar and turbulent Prandtl numbers (Pr_L and Pr_T). Additionally, the Reynolds stress tensor ($\bar{\rho} \tau_{ij}$), the instantaneous stress tensor (t_{ij}), and the mean strain rate tensor (S_{ij}) are required for the compressible RANS equations (Eq. A.7). In this case, the Reynolds stress tensor is defined by the Boussinesq approximation.

A.2.1 k - ω SST turbulence model

A turbulence closure model is also required to model the compressible viscous flow. In the present work, the two-equation Menter $k - \omega$ Shear Stress Transport (SST)¹⁵⁴

is applied (Eqs. A.8-A.13). This turbulence model defines the transport of the turbulent kinetic energy (k) and the specific dissipation rate (ω). The model constraints (σ_{k_l} , σ_{ω_l} , γ_l , β_l , β^* , κ , and a_1) are applied to control the model settings. The values were set constant for all the experiments of the present work. The majority of these constants, generalised as ψ_l , are obtained by solving the equation A.11. This is achieved by imposing to ψ_1 and ψ_2 the values of the constants of the first (Eq. A.12) and second (Eq. A.13) sets of equations. In addition, to control the turbulence model the norm of the vorticity (Ω), the kinematic turbulent viscosity (ν_t), and the dimensional wall distance (y) are required^{122,154}

$$\frac{\partial \bar{\rho} k}{\partial t} + \frac{\partial \bar{\rho} k \tilde{V}_i}{\partial x_i} = \tau_{ij} \frac{\partial \tilde{V}_i}{\partial x_j} - \beta^* \bar{\rho} k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_T) \frac{\partial k}{\partial x_j} \right] \quad (\text{A.8})$$

$$\begin{aligned} \frac{\partial(\bar{\rho} \omega)}{\partial t} + \frac{\partial(\bar{\rho} \omega \tilde{V}_i)}{\partial x_i} = & \frac{\gamma}{\nu_T} \tau_{ij} \frac{\partial \tilde{V}_i}{\partial x_j} - \beta \bar{\rho} \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + \\ & 2(1 - F_1) \bar{\rho} \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (\text{A.9})$$

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, \Omega \tanh(\arg_2^2))}, \arg_2 = \max\left(2 \frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \nu}{y^2 \omega}\right), a_1 = 0.31 \quad (\text{A.10})$$

$$\begin{aligned} \psi &= F_1 \psi_1 + (1 - F_1) \psi_2, F_1 = \tanh(\arg_1^4), \\ \arg_1 &= \min \left[\max\left(\frac{\sqrt{k}}{0.09 \omega y}, \frac{500 y}{y^2 \omega}\right), \frac{4 \bar{\rho} \sigma_{\omega 2} k}{C D_{k\omega} y^2} \right], \\ C D_{k\omega} &= \max\left(2 \bar{\rho} \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\omega}{\partial x_j}, 10^{-20}\right) \end{aligned} \quad (\text{A.11})$$

$$\begin{aligned} \sigma_{k1} &= 0.85, \sigma_{\omega 1} = 0.5, \beta_1 = 0.075, \beta^* = 0.09, \kappa = 0.41, \\ \gamma_1 &= \beta_1 / \beta^* - \sigma_{\omega 1} \kappa^2 / \sqrt{\beta^*} \end{aligned} \quad (\text{A.12})$$

$$\begin{aligned} \sigma_{k2} &= 1.0, \sigma_{\omega 2} = 0.856, \beta_2 = 0.0828, \beta^* = 0.09, \kappa = 0.41, \\ \gamma_2 &= \beta_2 / \beta^* - \sigma_{\omega 2} \kappa^2 / \sqrt{\beta^*} \end{aligned} \quad (\text{A.13})$$

A.2.2 Wall functions in k- ω SST turbulence model

Wall functions are applied when the thickness of the first cell normal to the surface (near wall node) exceeds a given threshold. This threshold is controlled through the dimensionless velocity, y^* (Eq. A.14); where κ is the von Kármán constant ($\kappa = 0.4187$), C_μ is a constant of value 0.09, μ is the fluid local dynamic viscosity, k_P is the near wall turbulent kinetic energy at the near-wall node P , and y_P is the

distance from the point P to the wall. The threshold for the application of the wall function on the present work is $y^* > 11.225$.

$$y^* = \frac{\rho C_\mu^{\frac{1}{4}} k_P^{\frac{1}{2}} y_P}{\mu} \quad (\text{A.14})$$

When the y^* is higher than the given threshold, a logarithmic law of the wall is applied (Eq. A.15). In this case, E is an empirical constant for the logarithmic correction and the value is set to $E = 9.793$. The value of the near wall velocity V_P is then obtained from the definition of V^* (Eq. A.16)¹⁸⁹.

$$V^* = \frac{1}{\kappa} \ln(Ey^*) \quad (\text{A.15})$$

$$V^* = \frac{V_P C_\mu^{\frac{1}{4}} k_P^{\frac{1}{2}}}{\tau_w / \rho} \quad (\text{A.16})$$

The turbulent kinetic energy (k) and the specific dissipation rate (ω) are computed in all the domain according to the $k - \omega$ SST model (Eqs. A.8-A.13). The boundary condition on the near wall regions for both metrics is set to be non-permeable (Eqs. A.17-A.18). With that boundary condition, for the wall functions the production of kinetic energy, G_k , can be obtained as defined in the function Eq. A.19 based on the logarithmic law of the wall function¹⁸⁹. Thus, the k_p can be obtained. Finally, the specific dissipation rate at the point P is obtained with the Eq. A.20.

$$\frac{\partial k}{\partial n} = 0 \quad (\text{A.17})$$

$$\frac{\partial \omega}{\partial n} = 0 \quad (\text{A.18})$$

$$G_k \approx \tau_w \frac{\partial V}{\partial y} = \tau_w \frac{\tau_w}{\kappa \rho k_P^{1/2} y_P} \quad (\text{A.19})$$

$$\omega_P = \frac{V_\tau}{\sqrt{C_\mu \kappa y_P}} = \frac{\sqrt{k_P}}{C_\mu^{\frac{1}{4}} \kappa y_P} \quad (\text{A.20})$$

A.3 Equations for the properties of the air

The Favre-Averaged Navier Stokes equations need to be coupled with a gas model to close the system of equations and obtain the static pressure value, P . For the present work, the ideal gas law (Eq. A.21) is applied to obtain the pressure from

the fluid properties, temperature, and density. Since there is no mix of gasses on the present work, the air specific gas constant ($R = 287 J/(kgK)$) is applied.

$$p = \rho R t \quad (A.21)$$

To obtain the specific heat capacity at constant pressure (C_p) the kinetic theory¹⁹⁰ is applied (Eq. A.22). M_w denotes the molecular weight of air ($M_w = 28.96 g/mol_{air}$) and f is equivalent to the number of degrees of freedom of the model that was set to 5. The heat capacity at constant volume (C_v) is derived from C_p (Eq. A.23).

$$C_p = \frac{1}{2} \frac{R}{M_w} (f + 2) \quad (A.22)$$

$$C_v = C_p - R \quad (A.23)$$

The thermal conductivity of the air (Π) is also modelled based on the molecular kinetic theory (Eq. A.24^{122,190}). The specific internal energy and enthalpy are defined by equations A.25 and A.26, respectively.

$$\Pi = \frac{15}{4} \frac{R}{M_w} \mu \left(\frac{4}{15} \frac{C_p}{R} + \frac{1}{3} \right) \quad (A.24)$$

$$e = C_v(t) T \quad (A.25)$$

$$h = C_p(t) T \quad (A.26)$$

The dynamic viscosity of the air is defined as a function of the static temperature according to Sutherland's law¹⁹¹ (Eq. A.27). The model's constants (μ_{ref} , T_{ref} and S) are defined in Eq. A.28.

$$\mu = \mu_{ref} \left(\frac{T}{T_{ref}} \right)^{\frac{3}{2}} \frac{T_{ref} + S}{T + S} \quad (A.27)$$

$$\mu_{ref} = 1.716 \times 10^{-5} \frac{kg}{m \cdot s}, T_{ref} = 273.11 K, S = 110.56 K \quad (A.28)$$

APPENDIX B

Propulsive system sizing

In this Appendix, the approach to size the aft mounted boundary layer propulsor is summarised. The main assumptions taken and the rationale of the decisions are also included. The sizing of the configuration is based on the reference aircraft or on the equivalent axisymmetric fuselage. The boundary layer properties at a fixed stage of the fuselage are evaluated for this purpose. The methods and theory behind the sizing approach are summarised in Sections B.1-B.3. The methodology to scale the turbomachinery stage is also summarised in Section B.4. The sizing process is preformed for the 2D axisymmetric (Section B.5) approaches (coupled and uncoupled propulsive system), and the 3D propulsor model (Section B.6). As a complement to the sizing methods, a preliminary evaluation of the performance of low order thrust and drag accounting methods is included (Section B.7).

B.1 Study of the boundary layer properties

The determination of the boundary layer thickness and the amount to be ingested becomes a fundamental problem for the analysis of boundary layer ingestion systems⁶⁴. Different approaches can be found in the literature. This includes approaches such as the commonly applied 99% of freestream velocity criteria¹⁹², various models based on the inviscid-viscous flow^{150,192} separation, and other approaches based on viscous region selectors^{64,84,193}. While the application of any of these methods is valid within the requirements of the different problems, there is still some debate about which is the most adequate approach. Since the present work is not focused on the study of the physics of the boundary layer ingestion systems, a simplified approach has been taken for the estimation of the boundary layer thickness.

The possibility of applying a viscous region selection based on the separation between rotational and irrotational flows was contemplated. Similarly to the approach proposed by Lovely¹⁹³, the limit of the viscous region of the boundary layer can be found in the regions where the vorticity (ω) becomes zero. This approach is based on a simple study of the Helmholtz equation of hydrodynamics (Eq. B.1) as defined in White¹⁵⁰. This definition is for incompressible flows, but the main ideas behind the assumptions still holds true. The rate of change of vorticity is driven by two terms, one convective and one diffusive (Eq. B.1). The first convective term is referred as the vortex-stretching term¹⁵⁰. The second term defines the diffusion of the vorticity derived by the fluid's viscosity. However, in viscous flows this term is always present derived from the no-slip condition in walls where it does not vanish¹⁵⁰. This means that the vorticity production near the wall is driven by the relative motion of the surface and the fluid. Thus, there is always a vorticity generation on the near wall region. For high Reynolds cases, this high vorticity region is bounded to the boundary layer and in theory the freestream value of the vorticity tends to zero outside the boundary layer region. Nevertheless, the vorticity is rarely zero outside the boundary layer. This is due to the presence of the freestream turbulence propagation by the convective term and in CFD the inherent error of approximation on the dissipation effects. For this reasons, it is required to define a threshold of the vorticity to define the boundary layer thickness region. Hence, to avoid the definition of an arbitrary criteria for the boundary layer thickness, this approach was discarded.

$$\frac{D(\omega)}{Dt} = (\omega \cdot \nabla) \mathbf{V} + \nu \nabla^2 \omega \quad (\text{B.1})$$

Since alternative viscous region selection methods^{64,84,193} found in the literature depend on an arbitrary offset for a metric of interest, a simpler approach based on a combination of the 99% of the velocity (δ_{99}) definition¹⁹² and the separation between inviscid and viscous flow^{150,192}. For the 2D axisymmetric cases, the evaluation of the boundary layer shape and properties at a given axial station is a trivial problem. Hence, a fixed reference station of the fuselage was selected (Fig. B.1) as reference point for the fuselage boundary layer properties. At this station, δ_{99} can be easily obtained as well as the mass flow and momentum distributions across the sample plane. This allows the definition of the Boundary Layer Ingestion rate, BLR , (Eq. B.2) as a function of the thickness.

$$BLR = \frac{h_{BLI}}{\delta_{99}} \quad (\text{B.2})$$

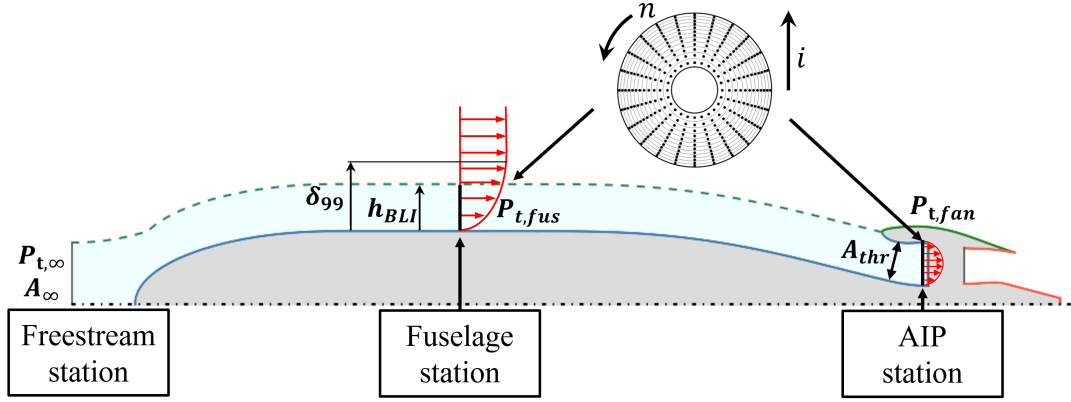


FIGURE B.1: Intake performance reference measure planes and rake distribution

Alternatively, the thickness can be estimated based on an inviscid-viscous flow separation. For this purpose, the displacement (δ^*) and the momentum thickness (Θ) are defined (Eqs. B.3-B.4)¹⁵⁰. An approximate definition of the displacement thickness (Eq. B.3) is that it is the thickness corresponding to the deficit in mass flow rate produced in the boundary layer region with respect to the mass flow rate that would flow through the same region without the presence of the boundary layer. Analogously, the momentum thickness (Eq. B.4) is defined as the thickness corresponding to the deficit in momentum.

$$\delta^* = \int_0^\infty \left(1 - \frac{\rho(y)}{\rho_\infty} \frac{V(y)}{V_\infty} \right) dy \quad (\text{B.3})$$

$$\Theta = \int_0^\infty \frac{\rho(y)}{\rho_\infty} \frac{V(y)}{V_\infty} \left(1 - \frac{V(y)}{V_\infty} \right) dy \quad (\text{B.4})$$

An simple analysis of the effect of the boundary layer on the local flow properties can be performed with the comparison of the main production terms of the displacement and momentum thickness in incompressible flows (Fig. B.2). The mass flow deficit ($1 - V/V_\infty$) decreases with the increase on the freestream velocity. This focusses the region of the highest mass flow rate deficit with respect to the freestream on the near wall sections of the boundary layer. This is interesting to highlight the non-linear trade between the *BLR* and the mass flow of the system. Additionally, the contribution of the momentum deficit terms ($\frac{V(y)}{V_\infty} \left(1 - \frac{V(y)}{V_\infty} \right)$) goes to zero on the near wall region based on the no-slip condition and rises again with the increase of the boundary layer velocity until a maximum at $\frac{V(y)}{V_\infty} = 0.5$ is found, then decreases monotonically. This indicates that depending on the boundary layer shape most of the deficit of the momentum flux is localized on the regions with near the wall, i.e. for lower *BLR* values. This is of particular importance, since the aim of the

boundary layer ingestion system is to recover this momentum flux deficit generated by the boundary layer on these regions. Hence, the amount of momentum flux deficit to be recovered should be one of the main criteria to size the BLI propulsor. For this purpose, the momentum thickness is transformed to the momentum flux deficit term (Eq. B.5)¹⁵⁰ to provide a metric to account for the recoverable momentum during the propulsor sizing.

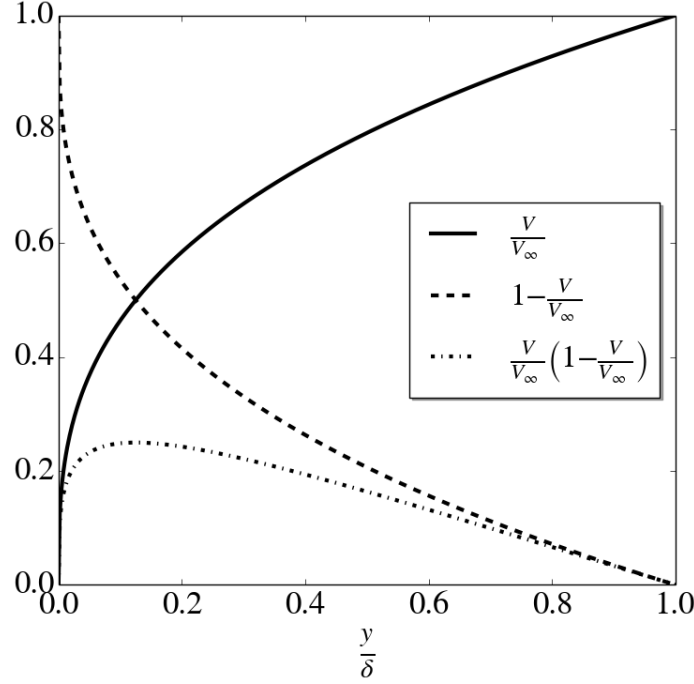


FIGURE B.2: Schematic of the velocity, displacement and momentum distributions in an incompressible boundary layer

$$\dot{q} = \rho_{\infty} V_{\infty}^2 \Theta = \int_0^{\infty} \rho(y) V(y) (V_{\infty} - V(y)) dy \quad (\text{B.5})$$

For 3D non-homogeneous flows the definition of the 99% thickness at a given station is smeared with the local variation of the thickness and the disruption produced by additional elements that can generate additional flow distortion, such as the wing wakes. Thus, an alternative approach based on the trade between mass flow rate and momentum flux deficit (Eq. B.5) is proposed. On this approach the momentum flux deficit and momentum thickness are computed based on the area (A) instead on the component normal to the surface (y). The area integration of the momentum flux deficit provides a metric to evaluate the potential momentum recovery of the three-dimensional BLI propulsor. However, this metric does not indicate the limits

of the boundary layer domain. Hence, it is necessary to find a methodology to bound the boundary layer effects. For this purpose, the normalised momentum flux deficit field is defined (Eq. B.6). Note this metric does not provide an integral value but a field value and it is normalised as pressure with the freestream dynamic pressure value.

$$\hat{q} = \frac{\frac{\partial \dot{q}}{\partial A}}{\frac{1}{2}\rho_{\infty}V_{\infty}^2} = \frac{V(y)(V_{\infty} - V(y))}{\frac{1}{2}\rho_{\infty}V_{\infty}^2} \quad (\text{B.6})$$

The normalised momentum flux deficit field can be applied to identify the boundary layer region of influence. The momentum flux deficit contribution has to dissipate outside the boundary layer region of influence (Fig. B.2) and converges to a maximum value. That value is used to define a iso-surface at the reference plane (Fig. B.3). The boundary layer effects are assumed to be held within the isoline of maximum normalised momentum flux deficit. Hence, the internal isolines as a function of the maximum momentum flux deficit can be used as reference areas for the integration of the momentum flux deficit value and the local mass flow. With the mass flow, and the mass weighted averages of the total pressure and total temperature¹⁵⁶ the sizing of the propulsor can be carried out. To the author's knowledge, this methodology has not been applied before and it is an additional contribution to the novelty of the present work

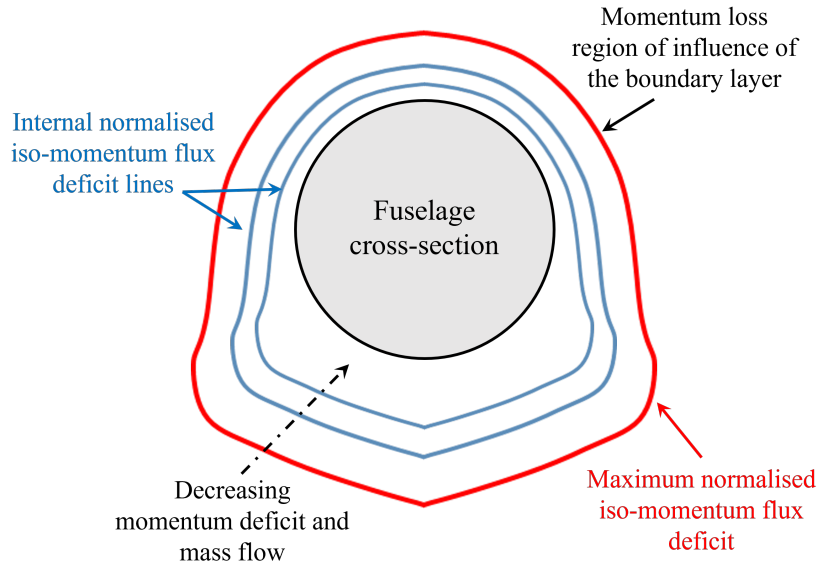


FIGURE B.3: Schematic of the momentum flux deficit potential integration method over a section of the fuselage

B.2 Fan dimensions and capacity matching

Once the criteria to define the boundary layer dimensions have been defined, the process to size the boundary layer ingestion propulsor is started. To guide the analysis, between the fuselage sampling plane and the exhaust of the propulsor, five reference stations are defined (Fig. B.4). The main criterion applied on the present work to define the fan radial dimensions is based on the capacity, Q , (Eq. B.8) at the fan face, Station 3 (Fig. B.4). The value of Q is defined based on the design Mach number at the fan face. This number is normally set based on the expected loads of the fan and the design operation cycle¹⁷². A common range for this value is between Mach 0.4 and 0.6 at the fan face. A Mach number value of 0.5 at the fan face is selected for the 2D axisymmetric cases of the present work.

$$Q = \frac{\dot{m}\sqrt{P_t}}{AT_t} = \sqrt{\frac{\gamma}{R}} M \left(1 + \frac{\gamma-1}{2} M^2\right)^{-\frac{\gamma+1}{(\gamma-1)}} \quad (\text{B.7})$$

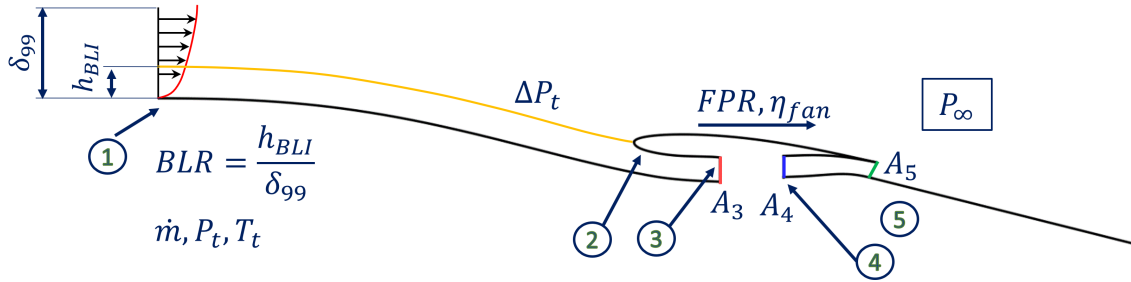


FIGURE B.4: Schematic of the main stations of the flow on the aft mounted boundary layer propulsor

The area for the target capacity can simply be extracted based on the mass flow rate, $\dot{m}$, total pressure, P_t , and total temperature, T_t at the given stage. The mass flow rate can be obtained from the boundary layer properties at the reference station of the fuselage, Station 1 (Fig. B.4), based on the selected Boundary Layer Ingestion Rate (BLR). Likewise, the total pressure and total temperature mass weighted average values of the Station 1 (Fig. B.4) can be obtained. If no heat transfer or flow discontinuities (shock waves) between the Stage 1 and the Stage 2 are assumed, the total temperature between the reference stage of the fuselage and the fan highlight remains constant. This value is also conserved at the fan face (Station 3). However, the total pressure value is subject to viscous dissipation effects at the aft section of the fuselage and the intake. Hence, to correctly provide an estimation of the fan

face area required to match the target capacity at the fan face a total pressure drop (ΔP_t) has to be assumed. This value was adjusted based on the CFD results. With all the previous considerations, the area of the fan face can be obtained (Eq. B.7). Finally, based on the design hub-to-tip ratio ($httr$) the radial dimension of the fan is obtained (Eq. B.8).

$$A_{fan} = \pi r_{fan}^2 (1 - httr^2) \quad (B.8)$$

B.3 Exhaust sizing under boundary layer effects

After the fan stage, Station 4 (Fig. B.4), an increase of the total pressure and the total pressure value has occurred due to the addition of work by the fan. These changes can be quantified in terms of the Fan Pressure Ratio (FPR) and the definition of the fan efficiency (η_{fan}). The total magnitudes at the fan are assumed to remain constant across the duct based with an isentropic expansion process¹⁹². If the area at Station 4 is not constrained by the fan geometry, the dimensions can be obtained based on a target Mach number and capacity (Eq. B.7), analogous to the fan face sizing process.

The determination of the exhaust duct discharge area (Station 5) requires further analysis. The ideal mass flow of the isentropic duct (Eq. B.9) can be simply obtained based on the fan nozzle pressure ratio, λ (Eq. B.10). With the conservation of mass, the mass flow rate at the fan face and the total pressure at the duct should suffice to obtain the discharge area value. However, this definition of the ideal area and mass flow ratio is highly dependent on the static pressure value at the discharge plane. Initially the static freestream value could be used as a reference, but the presence of the boundary layer and the diffusion of the flow at the nacelle can lead to significant changes of this value. Hence, it is convenient to apply the effective value of the static pressure at the discharge plane to obtain a more accurate value of the required area in terms of mass flow. Nonetheless, this is not always achievable. For example, in three dimensional flows the heterogeneous static pressure distribution derived from the boundary layer makes impossible to fix a value for the freestream reference pressure. Also, given the nature of the optimisation and design processes the nacelle geometry will change moving the effective pressure value. Thus, in the initial sizing problem, the freestream value was first obtained and then corrected with the CFD results. A mass flow matching approach is defined in Chapter 3.4.2.1 for the uncoupled fan approach. For the coupled fan, the change on the exhaust area

will displace the operation point of the fan map for the selected rotational speed.

$$\left(\frac{\dot{m}}{A}\right)_{ideal} = P_t \left(\frac{1}{\lambda}\right) \sqrt{\frac{2\gamma}{(\gamma-1)RT_t} \left(1 - \left(\frac{1}{\lambda}\right)^{\frac{\gamma-1}{\gamma}}\right)} \quad (\text{B.9})$$

$$\lambda = \begin{cases} \frac{P_t}{p} & \text{if } \lambda < \lambda_{crit} \\ \lambda_{crit} & \text{if } \lambda \geq \lambda_{crit} \end{cases} \quad (\text{B.10})$$

$$\lambda_{crit} = \left(\frac{\gamma+1}{2}\right)^{\frac{\gamma}{\gamma-1}} \quad (\text{B.11})$$

B.4 Fan stage scaling

To scale the dimensions of the NASA R4²³ for the different operation points of the present work, dimensional analysis is applied based on Pi-Buckingham theorem¹⁹⁴. For an axial compressor the dependency of the pressure ratio and the efficiency can be narrowed to four non dimensional groups (Eq. B.12). The first term of the groups is effectively a non dimensional mass flow, equivalent to the capacity definition (Eq. B.7). The second term is the normalisation of the maximum tangential velocity based on the speed of sound. The next two terms are effectively the Reynolds number and the specific heat relation of the working fluid, γ .

$$\frac{p_{t2}}{p_{t1}}, \eta, \Delta T_t/T_{t1} = f\left(\frac{\dot{m}\sqrt{\gamma RT_{t1}}}{D^2 p_{t1}}, \frac{\Omega D}{\sqrt{\gamma RT_{t1}}}, Re, \gamma\right) \quad (\text{B.12})$$

On the present work the gas properties are considered to be constant, so the γ dependency can be removed¹⁹⁴. Also, the operation of the machine is assumed to be at high Reynolds numbers so, according to Dixon and Hall¹⁹⁴, the Re dependency can be also removed. Thus, the operation of the fan is only dependent on the non-dimensional mass flow and rotational speeds (Eq. B.13). The fan diameter and rotational speeds can be adjusted to match a particular point of the performance map based on the operation conditions based on this relation.

$$\frac{p_{t2}}{p_{t1}}, \eta, \Delta T_t/T_{t1} = f\left(\frac{\dot{m}\sqrt{\gamma RT_{t1}}}{D^2 p_{t1}}, \frac{\Omega D}{\sqrt{\gamma RT_{t1}}}\right) \quad (\text{B.13})$$

B.5 Sizing of 2D axisymmetric configurations

Based on the aforementioned sizing criteria and methodology, the 2D axisymmetric propulsor can be sized based on the reference cruise conditions (Tab. 5.1). The maximum momentum flux deficit at the reference stage of the boundary layer was compared with the boundary layer ingestion rate (Fig. B.5). The momentum deficit is concentrated on the regions of lower *BLR*, with lower mass flow rates. This indicates that to recover most of the momentum loss by the presence of the fuselage boundary layer it is not required to ingest a considerable amount of the boundary layer. To match a given capacity the area of the fan scales almost linearly with the *BLR* on this region (Fig. B.6). The drag of the nacelle scales approximately proportional to the radial component. Also, the increase of the nacelle radial position exposes the cowl to higher velocity regions of the boundary layer. This also increases the overall drag. Thus, it is of interest to reduce the overall fan area to minimize the installation drag penalties. Hence, the momentum flux deficit and the *BLR* must be balanced to minimize the drag contribution of the nacelle.

In addition to the drag considerations, a more detailed preliminary sizing analysis should include the impact of the embedded propulsive system on the overall vehicle weight^{1,26,179}. The weight then should be balanced with the propulsive and aerodynamic performance of the overall system to optimise for a desired range^{26,115}. However, the implementation of a weight model requires a further knowledge of the power plant requirements for the aft mounted propulsor. While the weight of the materials for the cowl and propulsor blades can be estimated, the challenge relies on the definition of the power source and the actual space and weight requirements. To this date, multiple power sources have been proposed for this particular novel aircraft configuration^{1,26,195}. The selection of one particular power distribution will have a significant effect on the design requirements. Thus, the application of the weight model will require to commit to a particular configuration. However, the aerodynamic criteria should be consistent between the different configurations. Hence, since the aim of the present work is focused on the aerodynamic performance of such novel systems, the impact of the weight on the performance is not included. In future work, corrections on this methodology could be applied to evaluate the impact of the weight of the system.

To base the sizing only in aerodynamic criteria, the analysis is focused on balancing the amount of momentum flux deficit ingested with the mass flow requirements given by the *BLR*. The 75% of the momentum flux deficit is concentrated around the

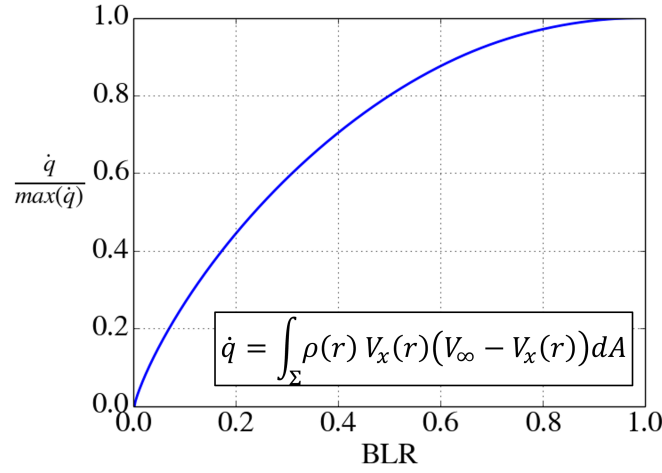


FIGURE B.5: Momentum flux deficit variation with the boundary layer ingestion rate for the reference axisymmetric fuselage in mid-cruise conditions

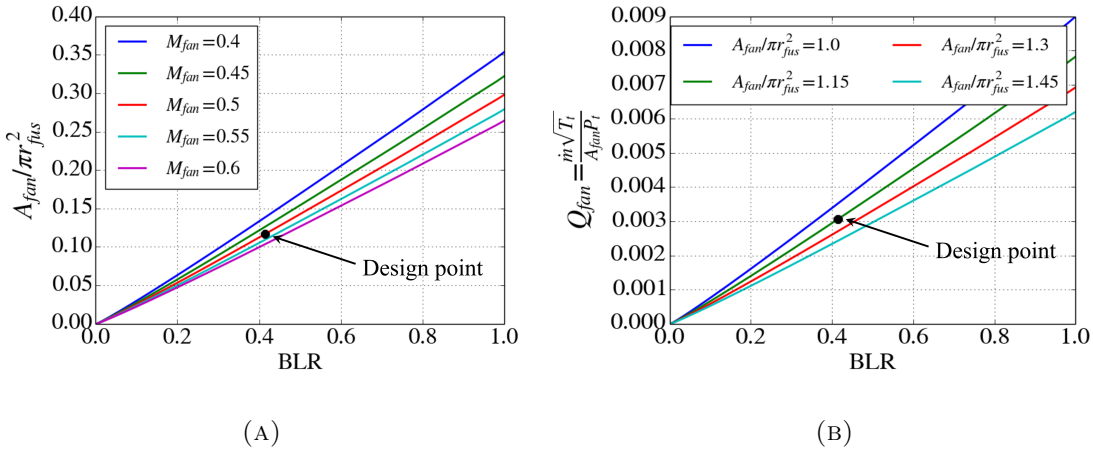


FIGURE B.6: Effect of the BLR on the fan dimensions for the 2D axisymmetric cases: (A) Fan face area variation with the boundary layer ingestion rate for the reference axisymmetric fuselage in mid-cruise conditions, and (B) fan face capacity variation with the boundary layer ingestion rate for the reference axisymmetric fuselage in mid-cruise conditions

41.5% of the boundary layer (Fig. B.5). This is in agreement with the values defined by Welstead et al.¹ that suggested that the optimal boundary layer to be ingested for the NASA STARC-ABL configuration was around the 40% of the thickness. Thus, a BLR of 0.415 is selected for the configuration of the present work. With this assumption, a target Mach number of 0.5 was selected for the fan face based on common fan operating criteria¹⁹⁴. Based on the estimated values of total pressure at the intake and the boundary layer conditions, the required area of the fan was

obtained. The radius of the fan at different locations can then be obtained based on the desired radius of the fan hub.

B.6 Sizing of 3D configurations

With the defined methodology to sample the momentum flux deficit regions of the boundary layer, the momentum flux deficit can be compared against the required mass flow rate ingested by the fan at a given fuselage station (Fig. B.7). The scaled DLR-F6²⁵ is sampled at a 60% of the fuselage length. This corresponds to the last point of the airframe geometry that remains unaltered. The aft-fuselage section further downstream of this point is modified based on the geometry design methodology. The mass flow rate of the different momentum flux deficit regions evolves as a parabola with the increase of the momentum flux deficit (Fig. B.7). Similarly to the *BLR* for the 2D axisymmetric case, it is not necessary to ingest a significant amount of the boundary layer to recover the majority of the momentum flux deficit. It was found that an 80% of the momentum flux deficit at this stage was containing approximately a 50% of the mass flow rate of the altered momentum region (Fig. B.7). This point was selected to evaluate the dimensions of the 3D propulsor.

However, it is important to note that the heterogeneous boundary layer and its momentum flux deficit distribution does not maintain a consistent shape across different length stations of the fuselage (Fig. B.8). The momentum flux deficit regions increase on size towards the aft most section of the fuselage. This is believed to be partially due to the relative importance of convective effect of the vorticity and distortion propagation (Eq. B.1) when compared with the effect of the diffusion. Another source for the increase of the distortion is the aircraft geometry itself. The final shape of the boundary layer distribution it is directly affected by the fuselage afterbody geometry and the elements that introduce additional momentum loss, such as the wing and the fairings. Thus, the selection of the station to sample the boundary layer properties has notable consequences on the sizing methodology. The 60% of the aircraft length criteria applied in this case is used to mitigate the effect of the arbitrary location of the sample boundary layer. The current design methodology uses this point as the reference location to change the afterbody geometry. Since the fuselage afterbody design has a notable effect on the boundary layer distribution at the different length stations (Fig. B.8), this point can be considered to be the last section (S2) that is not contaminated by the changes on the installation design.

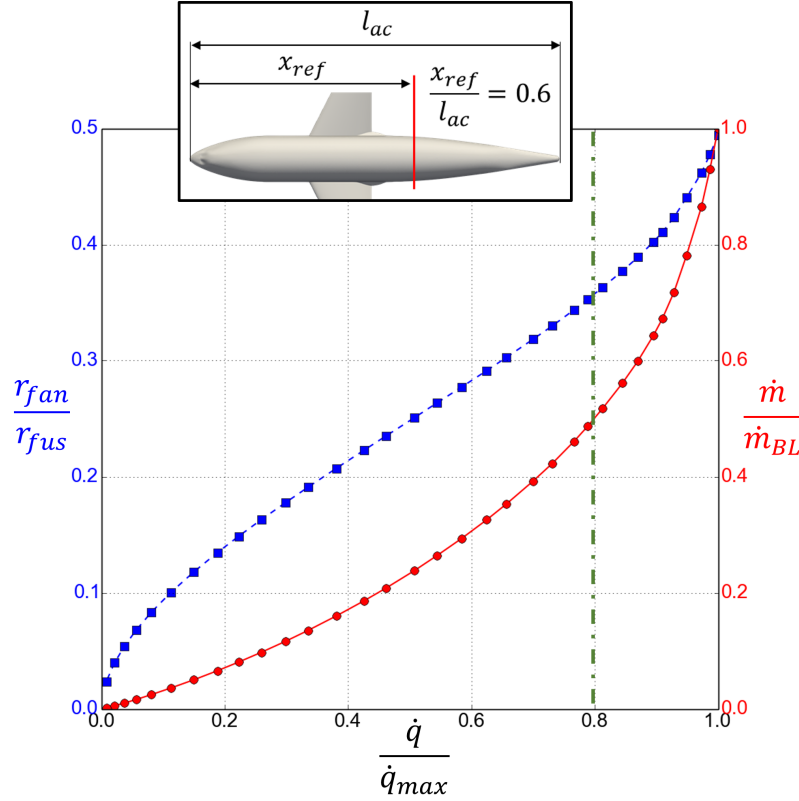


FIGURE B.7: Momentum flux deficit variation vs fan mass flow and radial requirements for a target Mach number of 0.5 at the fan face. Sampled from the DLR-F6 fuselage²⁵ at cruise conditions. Values at reference stage at 60% of the fuselage length

B.7 Preliminary performance accounting based on low order models

On previous sections, a method to size the propulsor based purely on aerodynamic criteria was presented. However, this did not provide a representation of the system aerodynamic performance. Across the different chapters of this work, the aerodynamic performance is evaluated numerically with different levels of models of CFD fidelity. Nevertheless, from a preliminary design point of view, it is of interest to evaluate the aerodynamic performance prior to CFD computations. Different low order performance accounting methods were presented in the Chapter 3.3.7 to provide an estimation of the thrust contribution of the propulsor. In this section, a comparative assessment of these methods is presented.

As a test configuration, an initial axisymmetric fuselage configuration with the dimensions of the reference aircraft. The propulsor fan hub coordinate is located

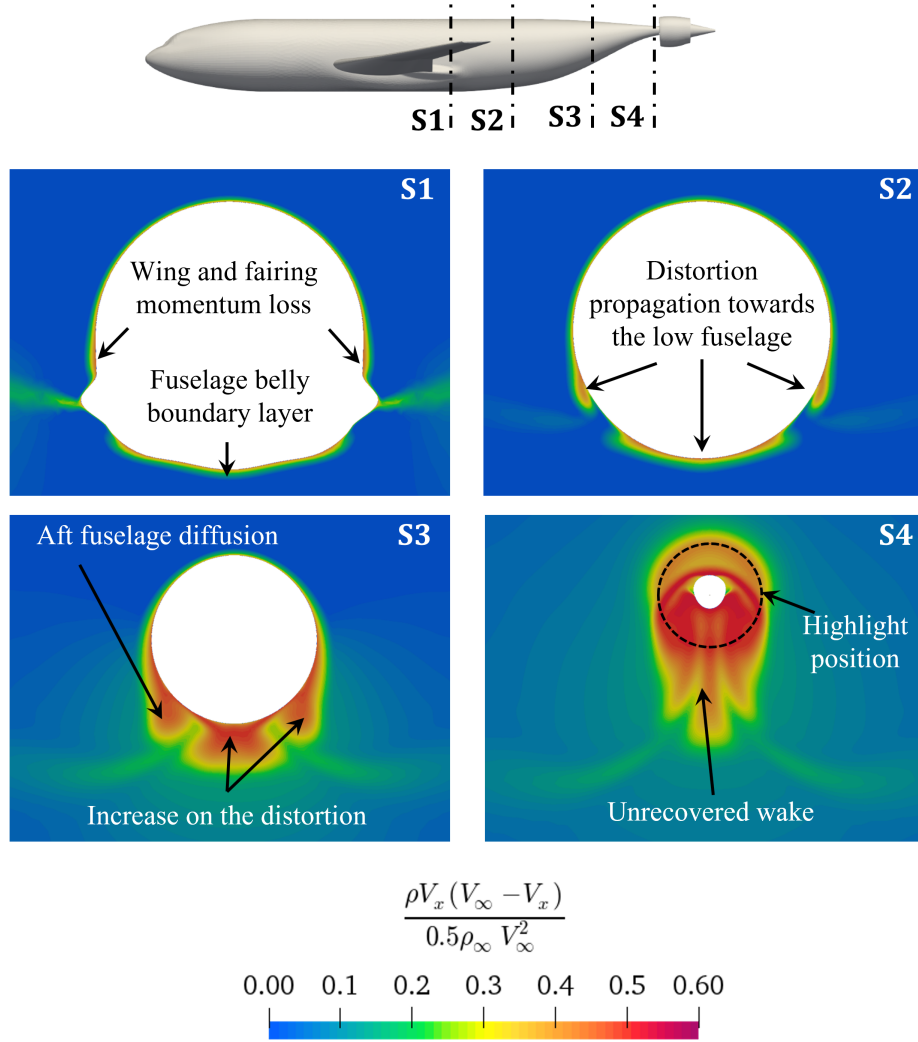


FIGURE B.8: Momentum flux deficit propagation at different lengths for a sample 3D aft-mounted boundary layer ingestion propulsor

axially at 0.96% of the total axial length and radially at 0.25% of the fuselage radius. The fan radial dimensions are corrected for the required fan face area based on the required capacity to match the target Mach number at the fan face (Fig. B.6). The sample cruise conditions (Tab. 5.1) are applied. The reference boundary layer to evaluate the momentum deficit ingested is taken from the CFD simulation of the fuselage in isolation (Fig. B.5). The viscosity values to obtain the Reynolds number are corrected for the altitude based on Sutherland's law (Eq. A.27). Also, from the total pressure distributions at the reference plane, a constant IPR_{ing} value of 0.97 is applied. To evaluate the turbomachinery efficiency for different fan pressure ratios, a fixed polytropic efficiency of 0.91 is used. Also, fixed fan axial dimensions (l_{fan}) are applied. To maintain the cowl design under different fan radial dimensions, constant ratios for the cowl main parameters are defined in Table B.1.

TABLE B.1: Cowl dimensions for the low order performance evaluation

Main cowl dimensions for the reference designs		
Metric	Preliminary	Modified
$\frac{r_{hi}}{r_{fan}}$	1.08	1.14
$\frac{r_{te}}{r_{hi}}$	0.9	0.74
Ω_{int}	0.0°	11.5°
$r_{max} - r_{hi}$ [m]	0.15	0.067
f_{max}	0.35	0.25
AR_{int}	0.45	0.35
$\frac{l_{nac}}{r_{fan}}$	3.26	2.28

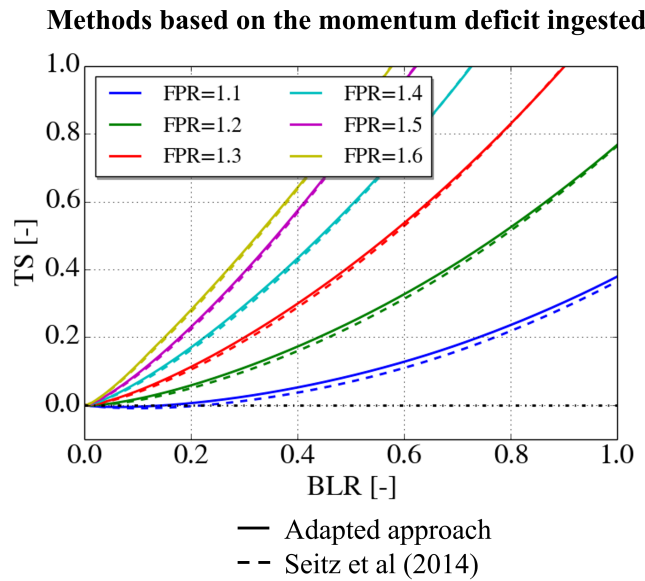


FIGURE B.9: Evolution of the thrust split with the boundary layer ingestion rate for different fan pressure ratios using the momentum deficit ingested approaches. Comparison between the adapted approach and the approach proposed by Seitz et al²⁶

Three methods were evaluated to understand the differences between the preliminary performance estimation methods. The first two are the methods based on the estimation of the amount of momentum ingested. These are the method defined by Seitz et al²⁶ (S-MFD) and the adapted method for the proposed accounting methodology (MFD)(Chapter 3.3.7.1). These formulations estimate the integration effects based on an integration factor (f_{int}), but the drag values are exclusively obtained from the surface drag values of the bodies in isolation. The third method is the approach based on the modified near-field method (MNFM)(Chapter 3.3.7.2). This method includes some effect of the aerodynamic coupling between the propulsor and

the airframe. The final integration effects still require of an integration factor, but the contribution of the post-exit and pre-entry streamtube forces on the system are accounted.

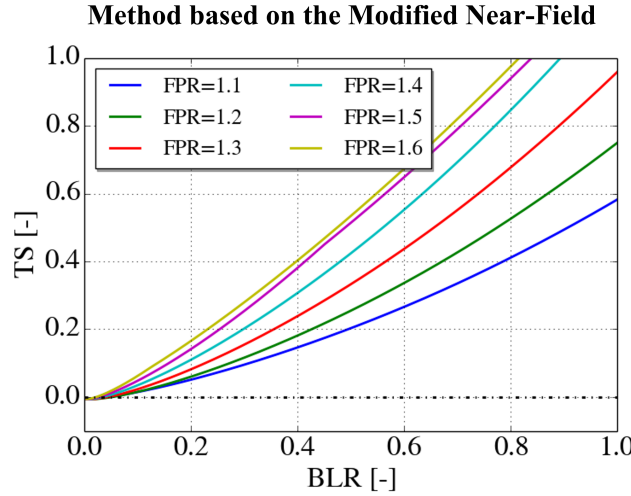


FIGURE B.10: Evolution of the thrust split with the boundary layer ingestion rate for different fan pressure ratios using the modified near-field method approach

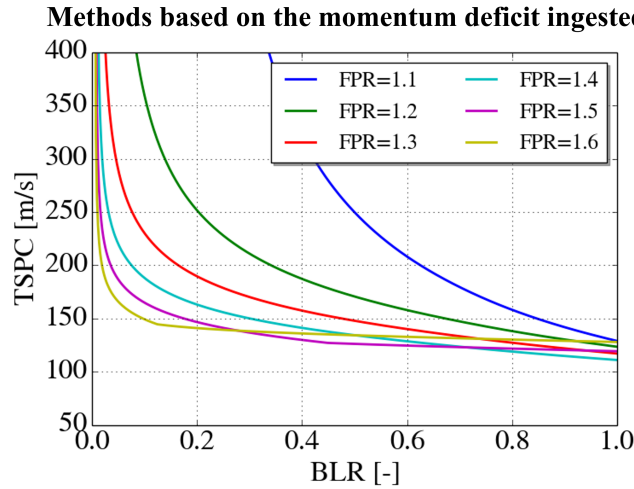
The thrust split, TS , prediction with three approaches discussed present similar general trends, increasing with the increase of the boundary layer ingestion rate, BLR , and the increase on fan pressure ration, FPR (Figs. B.9-B.10). The two methods based on the momentum deficit ingested (S-MFD and MFD) present similar distributions of the TS . In these models, the evolution of the TS is governed by the trade between the momentum deficit and the mass flow on the reference boundary layer. Hence, it is effectively the shape factor of the boundary layer¹⁵⁰ what governs the possible benefits. The momentum deficit distribution will then determine the drag reduction while the mass flow will provide the thrust contribution of the propulsor for a given fan pressure ratio. However, the methodology proposed by Seitz et al²⁶ and the adapted approach based on the momentum deficit ingested (MFD) present some discrepancies with the prediction of the TS . These discrepancies increase with the reduction of the MFCR and they are reduced slightly with the increase of the BLR (Fig. B.9). These discrepancies can be attributed to the difference between both TS formulations (Chapter 3.3.7). On the S-MFD formulation, the thrust benefit of the BLI propulsor is accounted on the denominator of the TS (Eq. 3.55), while on the MFD method proposed the changes are accounted on the numerator (Eq. 3.29). This is achieved by balancing the net vehicle force of the vehicle with the new propulsor with the reference unpropelled configuration to obtain the force benefit. Additionally, the MFD method includes the effect of the nacelle surface drag.

However, this value has been observed to have a lower impact on the preliminary drag estimation through this method in comparison with the fuselage contribution and the momentum deficit ingested (f_β).

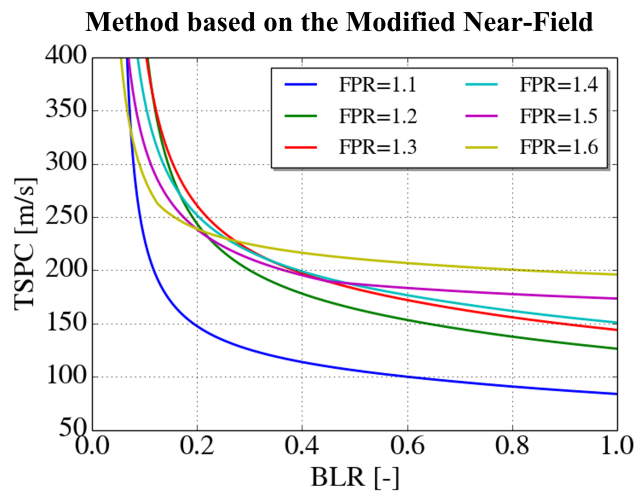
The approach based on the modified near-field method (MNFM) also defines an increase of the TS with the BLR and the BLR (Fig. B.10). However, significant differences with the previous methods are observed. The FPR seems to have a lower impact than the other two methods. Thus, the TS prediction is concentrated on a more narrow region on the design space. This produces a lower estimation of the maximum thrust produced for the higher fan pressure ratios and a higher thrust contribution at lower pressure ratios. This is also a consequence of the definition of the preliminary accounting method. In this case, the drag contribution of the integrated BLI propulsor is accounted in such way that the contribution of the pre-entry terms to drag spillage are included. For this the mass flow distribution of the boundary layer is not only accounted in the thrust contribution but also in the drag values through the value of ϕ_{pre} (Eq. 3.64). The momentum contribution of the boundary layer is also accounted in this term by the prediction of the gauge forces at the highlight plane. Thus, the form factor of the reference boundary layer still has influence on the values predicted, but the contribution of the mass flow and the momentum distribution in the boundary layer are coupled to account for the drag effects.

The differences on the TS prediction between both approaches lead to two completely different Thrust Specific Power Consumption, $TSPC$, predictions (Fig. B.11). The method based on the momentum deficit ingested present a continuous improvement of the propulsive efficiency by the reduction of the $TSPC$ with the increase of fan pressure ratio B.11a. With this method the lower FPR values such as 1.1 and 1.2 require of large amounts of boundary layer ingested (BLR) to maximise the system efficiency. Hence, to obtain efficient propulsors for a given thrust split higher fan pressure ratios are required. The higher FPR values the lower the mass flow rate is required to improve the propulsive efficiency.

The MNFM approach presents a completely different view about the the effects of the FPR and the BLR on the propulsive performance (Fig. B.11b). The lower variation of TS with the FPR and BLR leads to higher efficiencies at lower FPR . This is a direct consequence of the higher predicted thrust contribution with the lower fan pressure ratios (Fig B.10). This may indicate that lower FPR aft mounted propulsors could be a suitable option to maintain a higher propulsive efficiency. This would imply also a reduction of the required power for the aft mounted propulsor.



(A)



(B)

FIGURE B.11: Evolution of the Thrust Specific Power Consumption with the boundary layer ingestion rate for different fan pressure ratios using: the adapted momentum deficit ingested approach (A), and the modified near-field method approach (B)

This could have a positive impact on the overall system power saving depending on the type of power configuration applied for the aft-mounted propulsor. Nevertheless, higher BLR values will be required to increase the thrust contribution and maximise the power savings. This would require an increase of the cowl and turbomachinery dimensions. This could present some penalties on the weight of the propulsive system. However, a dedicated weight model will be required to properly understand the impact of the propulsion integration.

Before final conclusions are reached about the preliminary performance prediction

methods, it is necessary to acknowledge the limitations of both methods. This is necessary to understand the uncertainty of the application of low order preliminary accounting. The main assumption behind both approaches were described in Chapter 3.3.7. From the assumptions four clear limitations must be mentioned:

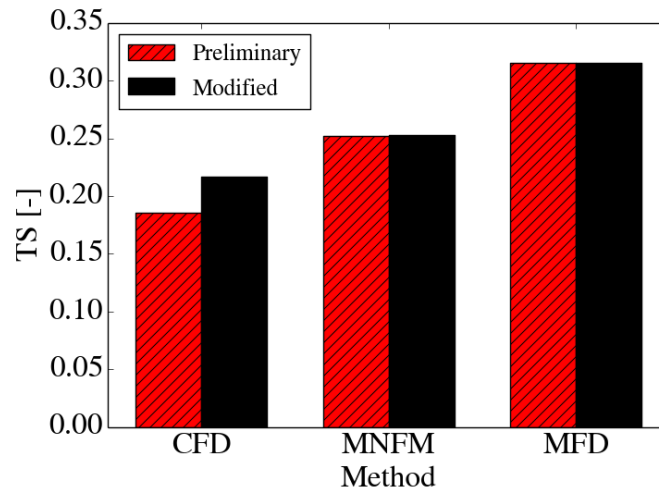
- The application of Jones velocity for the net uninstalled thrust definition relies on the assumption of isentropic expansion at the exhaust. Also, the transfer of momentum and energy from the post-exit streamtube to the freestream is neglected. This presents a limitation for the prediction of the thrust values at higher fan nozzle pressure ratios, where the exhaust operates under choked conditions. For under-expanded flows a shock structure will appear on the exhaust cone with a non-isentropic expansion of the flow^{36,122}. Thus, the uncertainty on the predicted uninstalled thrust is expected to increase with the *FPR*
- The drag accounting of both methods relies only on the prediction of viscous and form drag. The wake and induced drag are then neglected to simplify the model. For an axisymmetric prediction of the performance this approach could have some validity in subcritical conditions where the wake drag does not have a significant effect. However, while the flow incidence might now have a notable effect on the axisymmetric fuselage, the flow relative to the nacelle will have a relative incidence angle defined by the aft fuselage geometry. There is not a clear way to quantify this effect on the preliminary sizing phase. Thus, some uncertainty is expected on the predicted nacelle drag due to this assumption. This could be more significant on designs where incidence angle of the aft-fuselage geometry is higher
- Both approaches do not include the effect of the tail cone on the performance. This allowed to simplify the performance prediction but it is also expected to affect the predicted thrust values for both models and the drag value for the MNFM approach
- The integration effects are accounted in both models with a simplified integration factor f_{int} . In this approach the value has been considered constant. However, as was noted by Bijewitz et al¹⁵¹ this value will be affected by the flight conditions and the propulsion integration geometry. This could be calibrated with higher order solutions of the system performance. In a preliminary design activity this information is not available for the designer and requires further effort. For this reason, the constant value was maintained even though some additional uncertainty is expected

After the limitations of the models have been introduced, it becomes evident that it is not possible to directly determine the best approach for the preliminary performance prediction. Both approaches present some uncertainties that will affect the predicted value of the performance on the conceptual design stage. To explore the discrepancy between both approaches and a higher order solution, the predicted results were compared with a 2D axisymmetric CFD approach with semi-coupled propulsor model (Fig. B.12). Two designs were evaluated (Tab. B.1). First, the preliminary design (Preliminary) geometry (Tab. B.1) under reference mid-cruise conditions (Tab. 5.1). Second, a compact cowl geometry designed based on the local flow conditions and with the intention of improve the overall thrust performance (Modified).

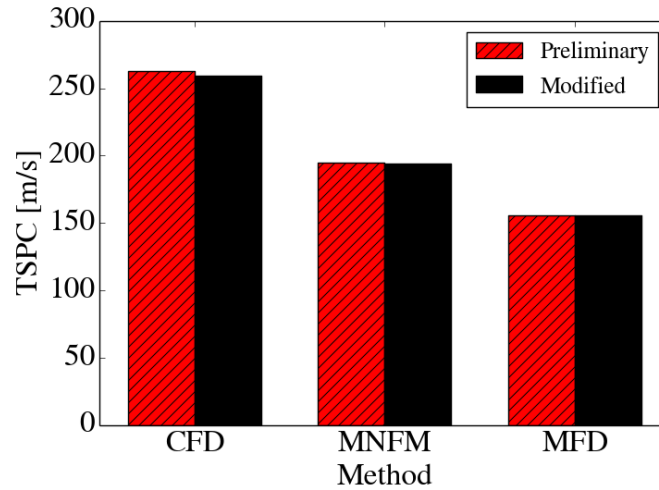
It was observed that for a given low order performance prediction method (MNFM or MFD) the change on the nacelle geometry does not have a significant impact on the predicted TS and $TSPC$ values (Fig. B.12). In both preliminary design methods the surface drag (viscous and form) was predicted applying the models defined by Raymer¹⁴⁶. In this approach the nacelle drag always have a much lower value relative to the fuselage, given the dimensions of the system. Thus, while some differences are predicted between both geometries with 0.18% and 0.15% difference in the predicted TS relative to the preliminary design for the MNFM and the MFD, respectively. However, when the 2D axisymmetric CFD is applied as a higher order model, significant differences on the TS value predicted can be observed between both propulsion integration designs (Fig. B.12b) with an increase of approximately 16.8% with the modified design respective to the preliminary baseline design. This is translated in more than a 3% of increase on the total thrust contribution just by the improvement of the propulsion integration aerodynamic design.

The low order approaches for the preliminary aerodynamic performance accounting have failed to predict notable changes on the performance based on the propulsion integration aerodynamic design. In addition to that, the prediction of the aerodynamic performance with the different models varies enormously. The MFD approach overpredicted the TS by 0.13 (~70%) for the Preliminary design and 0.098 (~45%) for the Modified design. The MNFM approach has better prediction of the values, with 0.066 (~36%) for the Preliminary design and 0.034 (~16.5%) for the Modified design. However, in any of the cases these differences are insufficient to capture the importance of the propulsion integration on the preliminary design stage.

The increase on the aerodynamic performance from the Preliminary design is attributed to a combination of the the aft-fuselage geometry, fan cowl compactness,



(A)



(B)

FIGURE B.12: Comparison of the predicted values for the Thrust Split (A) and the Thrust specific Power consumption (B) for two propulsion integration geometries: a baseline design based on the preliminary design geometry (Tab. B.1), and a manually designed compact cowl

and the cowl orientation. The change on the aft-fuselage geometry allows to control the orientation of the flow towards the propulsive system (Fig. B.13). At the same time, the flow diffusion in the afterbody can be controlled to increase the pressure force contribution in the thrust direction. These modifications of the geometry could not be captured with the low order performance prediction methods. The reduction of the cowl length did not provide significant changes on the predicted TS values with the preliminary performance accounting methods. However, the

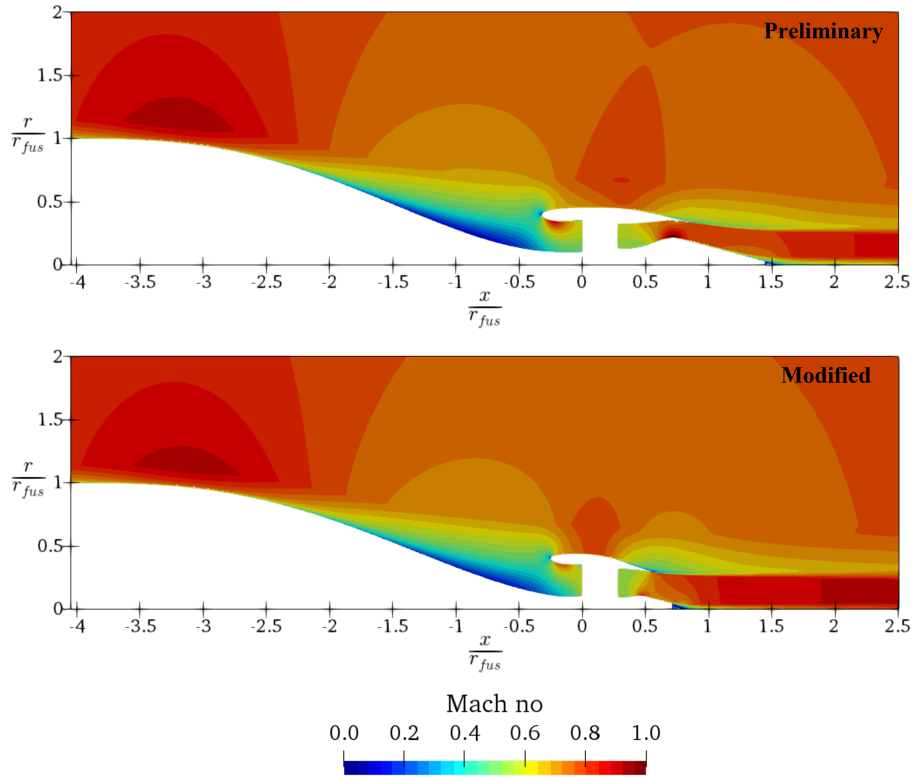


FIGURE B.13: Mach number contours of the Preliminary and Modified propulsion integration designs

Modified design applies a shorter intake and exhaust, while the fuselage geometry can be used to increase the pressure force contribution towards the thrust direction. This is combined with a reorientation of the propulsive system towards the flow direction dictated by the aft-fuselage geometry. The changes on the cowl dimensions and orientation allows the increase of the diffusion in the nacelle afterbody. This changes the effective pressure at the discharge plane, which increases the required fan pressure ratio required for a given mass flow. This increases the TS contribution. However, the increase of the input power required by the fan produces a less significant improvement on the TS_{PC} (Fig. B.12b). This can be used to exemplify the importance of the detailed propulsion integration design, since the low order models cannot capture the real benefits and the impact of the aerodynamic design on the performance.

B.8 Conclusions

On this appendix, the methods applied to determine the dimensions of the aft mounted annular boundary layer ingestion propulsor. An analysis of the boundary layer definition and properties was performed to evaluate the most suitable sizing criteria for the selected configuration. The analysis was performed separately for 2D axisymmetric and 3D configurations based on the reference cylindrical fuselage and reference aircraft, respectively. The main conclusions of this chapter can be summarised as the following:

- The thickness of reference boundary layer at a fixed fuselage station for the 2D axisymmetric configurations was determined with a 99% of the freestream velocity criterion. This was selected because of its simplicity and after the evaluation of other criteria as the inviscid-viscous flow separation or viscous region selection methods
- The 99% of the freestream velocity criterion was determined unsuitable for the heterogeneous boundary layer distributions of the 3D aircraft configuration. For this reason, a methodology based on the momentum flux deficit was selected to define the regions of interest for the boundary layer ingestion analysis. To the author's knowledge, this methodology has not been applied before and it is an additional contribution to the novelty of the present work
- A methodology for the propulsive system sizing has been proposed based on the design requirements at the fan face and the amount of boundary layer distribution. The main assumptions and considerations for this approach have been defined. The process to determine the main dimensions of the propulsive system includes the fan dimensional requirements, the operation of the fan, and the exhaust design requirements for embedded configurations
- The fan stage has been scaled based on the similitude principle to maintain the fan operation point under different operation conditions
- The amount of boundary layer ingested was compared with the momentum flux deficit that can be recovered from the boundary layer to determine the appropriate dimensions of the propulsive system for the 2D axisymmetric cases. It was found that it is not necessary to ingest a big amount of the boundary layer thickness to recover the majority of the momentum flux deficit. To maintain an adequate compromise between the wetted areas of the propulsor and

the amount of boundary layer ingestion approximately a 40% of the boundary layer requires to be ingested. This was found to contain approximately the 75% of the momentum flux deficit of the system

- For the 3D configurations, the momentum flux deficit at a given fuselage stage was compared with the mass flow contained in the same region. It was found that the largest amount of momentum flux deficit was concentrated on regions with lower mass flow rate ingested. Similarly to the 2D axisymmetric configurations, it is not necessary to ingest a major part of the boundary layer to ingest large amounts of momentum loss. A 50% of the mass flow of the boundary layer domain was selected to size the propulsive system. This accounts for approximately 80% of the momentum deficit
- The selection of the reference stage to sample the momentum flux deficit requirements was challenged to remove any arbitrariness on the methodology. For the 3D aircraft configuration a very high dependence on the stage selection was found. This was linked to the momentum deficit generated by the change on the aft fuselage geometry and the effect of the downstream convection of the vorticity generated in the boundary layer. However, under the system constraints the most consistent approach to select the reference sampling plane was based on the geometry design method. For the present work, the sample plane is based at the end of the non-modified section of the reference aircraft
- Different low order performance accounting methods have been tested to identify their accuracy for the preliminary design of embedded propulsive systems. The first method (MFD) was adapted from the current state of the art. The other method (MNFM) was developed as an improvement for the existing methods in the literature
- The low order performance accounting methods tested shown similar trends on the prediction of the TS variation with the fan pressure ratio and the amount of boundary layer ingested. However, the MNFM predicted a lower variation of the thrust contribution of the system with the fan pressure ratio, with higher predicted gains at lower fan pressure ratios than the MFD method. This affected the predicted propulsive efficiency of the configuration for a given method, with an opposite evolution with the fan pressure ratio for each method
- Two-dimensional axisymmetric CFD was applied to compare the low order performance prediction methods. The CFD was used to highlight the limitations of the methods on the prediction of the changes on the aerodynamic

performance with the propulsion integration aerodynamic design. Both methods also overestimated the TS value for the configurations tested, with the modified approach (MNFM) producing closer values to the ones provided by the CFD. Nevertheless, the uncertainty for these models is still too high to be applied even on the preliminary design stages

Summary of uncertainty accounting

In this appendix, the methods for the uncertainty accounting are summarised. Also, an evaluation of the numerical uncertainty of the computational models applied is provided. This includes the different propulsor approaches, the different dimensional cases (2D and 3D), and additional CFD models applied for the validation of the methodology.

C.1 Uncertainty accounting methods

To evaluate the numerical uncertainty of the computational methods the sources of uncertainty were divided in three groups¹⁹⁶:

- **Discretisation errors.** These errors originate by the uncertainty derived by the discretisation of the computational domain. Hence, they are inherent to any computational process. In the CFD analysis this kind of errors derive from the mesh definition
- **Iterative convergence errors.** These errors have their origin on the iterative process behind the computational approach. While the convergence of the solution can be assured under some margin, the nature of the iterative process still can produce some uncertainty from the final solution
- **Physical approximation errors.** These errors derive from the model selected to define the real physical phenomena. The model always have a degree of uncertainty from the real process. Thus, for some applications it is of interest to quantify this error to understand the accuracy of the selected approach

The first two sources of error can be accounted in all the process that require numerical simulation. The physical approximation errors require a further knowledge of the exact or target solution, and they will be included in the uncertainty analysis while possible.

For the present work, the mesh independence is evaluated based in the procedure described by Celik et al.¹⁵⁸ to obtain the discretization errors. The method consist in the calculation of the Grid Convergence Index (GCI) based in a set of three meshes with three levels of refinement. A representative cell size, h , is defined for each of the meshes applied in the study (Equation C.1). The refinement factor, r , is defined as the ratio between the h of two consecutively finer meshes, with the coarser mesh in the denominator, e.g. $r_{coarse-medium} = h_{medium}/h_{coarse}$. The apparent order of the system, p , is computed (Equation C.2) for a metric of interest, ϕ_i . Finally, the value of the Richardson extrapolation of the metric of interest (Equation C.3), the different errors (Equations C.4 and C.5) and the GCI (Equation C.6).

$$\begin{aligned} h_{(3D)} &= \left[\frac{1}{N} \sum_{i=1}^N \Delta V_i \right]^{\frac{1}{3}} \\ h_{(2D)} &= \left[\frac{1}{N} \sum_{i=1}^N \Delta A_i \right]^{\frac{1}{2}} \end{aligned} \quad (C.1)$$

$$\begin{cases} p &= \frac{1}{\ln(r_{medium-fine})} \left| \ln \left(\frac{\epsilon_{coarse-medium}}{\epsilon_{medium-fine}} \right) + q(p) \right| \\ q(p) &= \ln \left(\frac{r_{medium-fine}^p - s}{r_{coarse-medium}^p - s} \right) \\ s &= 1 \cdot \operatorname{sgn} \left(\frac{\epsilon_{coarse-medium}}{\epsilon_{medium-fine}} \right) \\ \epsilon_{coarse-medium} &= \phi_{medium} - \phi_{coarse} \\ \epsilon_{medium-fine} &= \phi_{fine} - \phi_{medium} \end{cases} \quad (C.2)$$

$$\begin{aligned} \phi_{ext}^{coarse-medium} &= \frac{r_{coarse-medium}^p \phi_{medium} - \phi_{coarse}}{r_{coarse-medium}^p - 1} \\ \phi_{ext}^{medium-fine} &= \frac{r_{medium-fine}^p \phi_{fine} - \phi_{medium}}{r_{medium-fine}^p - 1} \end{aligned} \quad (C.3)$$

$$\begin{aligned} e_a^{medium-fine} &= \left| \frac{\phi_{fine} - \phi_{medium}}{\phi_{fine}} \right| \\ e_a^{coarse-medium} &= \left| \frac{\phi_{medium} - \phi_{coarse}}{\phi_{medium}} \right| \end{aligned} \quad (C.4)$$

$$\begin{aligned} e_{ext}^{fine} &= \left| \frac{\phi_{ext}^{medium-fine} - \phi_{fine}}{\phi_{ext}^{medium-fine}} \right| \\ e_{ext}^{medium} &= \left| \frac{\phi_{ext}^{coarse-medium} - \phi_{fine}}{\phi_{ext}^{coarse-medium}} \right| \end{aligned} \quad (C.5)$$

$$\begin{aligned} GCI^{fine} &= \frac{1.25 e_a^{medium-fine}}{r_{medium-fine}^p} \\ GCI^{medium} &= \frac{1.25 e_a^{coarse-medium}}{r_{coarse-medium}^p} \end{aligned} \quad (C.6)$$

The GCI of the selected mesh methodology is used as the relative uncertainty of the model for a given mesh refinement. In addition to that uncertainty, the numerical

oscillation of the metric of interest (generically ψ) is added to error estimation as the iterative error. This uncertainty is estimated by finding the maximum absolute deviation from the mean value of ψ . The errors ($\delta\psi$) are added using quadrature error additions as defined by Taylor¹⁸² (Eq. C.7).

$$\delta\psi = \sqrt{\delta\psi_{grid}^2 + \delta\psi_{ite}^2} \quad (C.7)$$

Since in the present work there are many metrics that require the combination of two or more measured metrics of interest, the error addition approach defined by Taylor¹⁸² is also used. When a metric ψ is a result of the sum or difference between two metrics (ξ and λ) the errors are added as defined in Eq. C.8. When a metric ψ is a result of the product or quotient between two metrics (ξ and λ) the errors are added as defined in Eq. C.9.

$$\delta\psi(\xi, \lambda) = \sqrt{\delta\xi^2 + \delta\lambda^2} \quad (C.8)$$

$$\frac{\delta\psi(\xi, \lambda)}{\psi} = \sqrt{\left(\frac{\delta\xi}{\xi}\right)^2 + \left(\frac{\delta\lambda}{\lambda}\right)^2} \quad (C.9)$$

C.2 2D axisymmetric aircraft models

The uncertainty of the 2D axisymmetric approaches has been obtained for the reference cruise conditions of the reference aircraft (Tab. 5.1). This analysis is repeated for the 2D axisymmetric reference cylindrical fuselage, the uncoupled propulsor model 2D axisymmetric model, and the coupled 2D axisymmetric model.

C.2.1 Reference axisymmetric fuselage

To determine F_{RN} (Eq. 3.28) it is necessary to obtain a value for the reference aircraft Net-Vehicle-Force, NVF_{ref-ac} . This metric is defined as the drag of the reference aircraft. For the 2D axisymmetric computations the reference aircraft is an equivalent axisymmetric cylindrical fuselage. The summary of the grid independence study can be found in Table C.1. Three progressively refined meshes are applied in these study. Since the calculation of the NVF_{ref-ac} is only performed once, the finest mesh is selected as the reference for the values.

TABLE C.1: Grid sensitivity study for the 2D axisymmetric reference cylindrical fuselage

Decoupled propulsor 2D axisymmetric approach			
Mesh	3	2	1
N. of elements	1.34×10^5	4.46×10^5	1.481×10^6
Metric	GCI_{23} (%)	GCI_{12} (%)	
NVF_{ref-ac}	0.034	0.011	

C.2.2 Decoupled propulsor 2D axisymmetric structured mesh

The main metrics for the propulsive, intake and exhaust performance quantification of the 2D axisymmetric model with the uncoupled propulsor approach are applied in this study. Three meshes with progressive refinements are used for the uncertainty quantification (Tab. C.2). An adequate level of uncertainty was obtained with the intermediate mesh. Hence, it was selected as the reference mesh resolution for the uncoupled propulsor cases of the present work to reduce the computational cost.

TABLE C.2: Grid sensitivity study for the 2D axisymmetric decoupled boundary layer ingestion approach

Decoupled propulsor 2D axisymmetric approach			
Mesh	3	2	1
N. of elements	2.79×10^5	5.09×10^5	9.28×10^5
Metric	GCI_{23} (%)	GCI_{12} (%)	
NVF	1.442	0.513	
W_{prop}	0.0088	0.00061	
RDI	0.399	0.209	
IPR_{ing}	0.008	0.003	
C_V	0.003	0.001	
C_V^*	0.0043	0.00001	
C_D	0.014	0.002	

$TSPC$ and TS require to operate with the metrics extracted by the CFD simulation (Tab. C.2). The error of each of the metrics is coupled with the iterative error extracted. The method defined by Taylor¹⁸² to add the cumulative error of the individual metrics is applied to obtain an uncertainty of $\pm 1.84 \times 10^{-4}$ units for the TS , and ± 2.3 m/s for $TSPC$. On the intake performance side, the cumulative error of discretisation and iterative convergence is added to obtain an error of $\pm 5 \times 10^{-4}$

units and $\pm 7.7 \times 10^{-5}$ units for the RDI and IPR_{ing} , respectively. Similarly, for the exhaust performance the cumulative error is estimated to be $\pm 3.0968 \times 10^{-5}$ units, $\pm 4.305 \times 10^{-5}$ units, and $\pm 1.403 \times 10^{-4}$ units for the C_V , C_V^* , and the C_D , respectively.

C.2.3 Coupled propulsor 2D axisymmetric structured mesh

The main metrics for the propulsive, intake and exhaust performance quantification of the 2D axisymmetric model with the coupled propulsor approach are applied in this study. Three meshes with progressive refinements are used for the uncertainty quantification (Tab. C.2). An adequate level of uncertainty was obtained with the intermediate mesh. Hence, it was selected as the reference mesh resolution for the uncoupled propulsor cases of the present work to reduce the computational cost.

TABLE C.3: Grid sensitivity study for the 2D axisymmetric coupled boundary layer ingestion propulsor approach

Coupled propulsor 2D axisymmetric approach			
Mesh	3	2	1
N. of elements	3.91×10^5	7.08×10^5	1.286×10^6
Metric	GCI ₂₃ (%)	GCI ₁₂ (%)	
NVF	0.524	0.282	
W_{prop}	0.354	0.0965	
RDI	0.057	0.001	
IPR_{ing}	0.02	0.002	
C_V	0.004	0.001	
C_V^*	0.0146	0.00734	
C_D	0.051	0.001	

$TSPC$ and TS require to operate with the metrics extracted by the CFD simulation (Tab. C.2). The error of each of the metrics is coupled with the iterative error extracted. The method defined by Taylor¹⁸² to add the cumulative error of the individual metrics is applied to obtain an uncertainty of $\pm 1.925 \times 10^{-5}$ units for the TS , and ± 1.953 m/s for $TSPC$. On the intake performance side, the cumulative error of discretisation and iterative convergence is added to obtain an error of $\pm 9.98 \times 10^{-5}$ units and $\pm 1.956 \times 10^{-4}$ units for the RDI and IPR_{ing} , respectively. Similarly, for the exhaust performance the cumulative error is estimated to be $\pm 4.03 \times 10^{-5}$

units, $\pm 1.53 \times 10^{-5}$ units, and $\pm 5.05 \times 10^{-5}$ units for the C_V , C_V^* , and the C_D , respectively.

C.3 3D aircraft models

C.3.1 DLR-F6 aircraft grid sensitivity study

To determine F_{RN} (Eq. 3.28) it is necessary to obtain a value for the reference aircraft Net-Vehicle-Force, NVF_{ref-ac} . This metric is defined as the drag of the reference aircraft. For the 3D configurations the reference aircraft is the DLR-F6²⁵ scaled to the reference aircraft length. The summary of the grid independence study can be found in Table C.4. Three progressively refined meshes are applied in these study. Since the calculation of the NVF_{ref-ac} is only performed once, the finest mesh is selected as the reference for the values.

TABLE C.4: Grid sensitivity study for the DL-F6 configuration scaled to reference length under cruise conditions

DLR-F6 scaled fuselage			
Mesh	3	2	1
N. of elements	11.51 x 10 ⁶	22.99 x10 ⁶	59.33 x10 ⁶
Metric	GCI ₂₃ (%)	GCI ₁₂ (%)	
NVF_{ref-ac}	1.67	0.68	

C.3.2 Coupled propulsor 3D non-axisymmetric hybrid mesh

The main metrics for the propulsive, intake and exhaust performance quantification of the 3D model with the coupled throughflow propulsor approach are applied in this study. Three meshes with progressive refinements are used for the uncertainty quantification (Tab. C.7). An adequate level of uncertainty was obtained with the intermediate mesh. Hence, it was selected as the reference mesh resolution for the uncoupled propulsor cases of the present work to reduce the computational cost.

$TSPC$ and TS require to operate with the metrics extracted by the CFD simulation (Tab. C.7). The error of each of the metrics is coupled with the iterative error

TABLE C.5: Grid sensitivity study for the 3D configuration with the through-flow fan model

Through-flow 3D propulsion integration			
Mesh	3	2	1
N. of elements	50.11 x 10 ⁶	88.5 x10 ⁶	168.18 x10 ⁶
Metric	GCI ₂₃ (%)	GCI ₁₂ (%)	
W_{prop}	1.311	0.0618	
NVF	1.365	0.0645	
RDI	1.372	0.061	
CDI	0.791	0.118	
DC_{60}	3.46	1.996	
IPR	0.395	0.115	
C_V^*	0.1549	0.00218	
C_D	0.037	0.136	

extracted. The method defined by Taylor¹⁸² to add the cumulative error of the individual metrics is applied to obtain an uncertainty of $\pm 3.51 \times 10^{-3}$ units for the TS , and ± 5.37 m/s for $TSPC$. On the intake performance side, the cumulative error of discretisation and iterative convergence is added. This results in an uncertainty of $\pm 3.5 \times 10^{-3}$ units of IPR , $\pm 1.03 \times 10^{-3}$ units of RDI , $\pm 1.0269 \times 10^{-3}$ units of CDI , and $\pm 9.745 \times 10^{-3}$ units of DC_{60} . Similarly, for the exhaust performance the cumulative error is estimated to be $\pm 1.58 \times 10^{-3}$ units, and $\pm 3.49 \times 10^{-4}$ units for the C_V^* , and the C_D , respectively.

C.4 Turbomachinery models

Thee CFD approaches were used (3D sector BFM, 2D axisymmetric BFM, and Mixing Plane) on the validation of the BFM against the NASA rotor 4 geometry^{23,24}. To determine the mesh requirement for the three approaches, a grid independence study is performed based on the method proposed by Celik et al.¹⁵⁸. As the reference, for a fixed capacity at the $\Omega/\Omega_{max} = 70\%$ fan speed one operation point defined with a fixed static pressure at the outlet is applied. Through the comparison of three refinement levels for each approach, the finest meshes were selected for the 3D-sector BFM and the 2D-axisymmetric approaches with a GCI of the 0.027% and the 0.09% for the temperature ratio and a 0.197% and a 0.12% for the stage pressure ratio (Tabs. C.6-C.7). The second finest mesh was selected for the Mixing Plane Model after obtaining of satisfactory GCI values with a 0.025% and a 0.154%

for the stage pressure and temperature ratio respectively (Tab. C.8). Oscillatory convergence was observed on the 2D-axisymmetric and the Mixing Plane models for the temperature ratio. The oscillation on the value is considered sufficiently small to be neglected in both cases, and the GCI values are calculated with these values.

TABLE C.6: Grid sensitivity study for the 2D axisymmetric BFM approach

Grid sensitivity for the 2D axisymmetric BFM			
Mesh	3	2	1
N. of elements	2.34×10^4	4.22×10^4	7.56×10^4
Metric	GCI_{23} (%)	GCI_{12} (%)	
PR	0.274	0.197	
TR	0.078	0.090	

TABLE C.7: Grid sensitivity study for the 3D sector BFM approach

Grid sensitivity for the 3D sector BFM			
Mesh	3	2	1
N. of elements	2.135×10^5	4.84×10^5	1.22×10^6
Metric	GCI_{23} (%)	GCI_{12} (%)	
PR	0.228	0.12	
TR	0.048	0.027	

TABLE C.8: Grid sensitivity study for the Mixing plane approach

Grid sensitivity for the Mixing Plane model						
Mesh	3		2		1	
Domain	Blade	Vane	Blade	Vane	Blade	Vane
N. of elements	9.4×10^5	1.73×10^5	1.93×10^6	4.46×10^6	4.15×10^6	1.05×10^6
Metric	GCI_{23} (%)			GCI_{12} (%)		
PR	0.025			0.001		
TR	0.154			0.249		

APPENDIX D

Validation of the computational fluid dynamics methodology

To this date, there is a notable absence of open source benchmark cases for the validation of embedded propulsion systems in representative cruise conditions. Therefore, to validate the CFD methodology of this work, different representative cases of propulsion integration aerodynamics have been selected. This appendix is focused on the validation of the CFD software and the methodology applied. The implementation of the low order turbomachinery model on the aerodynamics solver was considered of higher scientific relevance, so it was included on the main body of this work (Chapter. 6).

As the representative cases for transonic aerodynamics, two validation cases were selected. First, the validation of transonic nacelle aerodynamics performed by Robinson et al²⁸ was replicated for two representative nacelle cases. These cases were validated for 2D and 3D flows applying structured grids. Second, to validate the hybrid mesh methodology with wall functions, the drag-polar of the medium range model aircraft DLR-F6^{18,25} was used.

Additionally, important steps for the validation of the aerodynamic performance of the different housing components have been performed for analogous CFD methodologies by other authors^{28,29,36,93,197}. A summary of these approaches can be also found in this appendix.

D.1 Summary of previous validation cases

Important steps for the validation of the propulsion integration aerodynamics have been taken on recent years at Cranfield University for analogous CFD methodologies. These processes have not been replicated in the present work. However, the author considers of interest to include a summary of these cases to justify the CFD methodology selected. The most representative cases to date are:

- **Intake aerodynamics.** The CFD methods and requirements to model the aerodynamics of short intakes at different angles of attack were evaluated by Boscagli et al⁹³
- **Nacelle aerodynamics.** The aerodynamics of transonic nacelles were validated for benchmark cases by Robinson et al²⁸. This validation was replicated for an updated CFD methodology by Tejero et al⁵⁰ with the contribution of the author of this work. These cases are later included in this appendix
- **Exhaust aerodynamics.** The CFD methodology was evaluated for both structured and hybrid unstructured meshes against reference exhaust configurations by Otter et al¹⁹⁸ and Goulos et al¹⁵, respectively
- **2D axisymmetric flows.** The aerodynamics of 2D axisymmetric transonic flows were validated by Milea²⁹ for a set of axisymmetric bodies of revolution
- **Aircraft aerodynamics and installation aerodynamics.** The aerodynamics of reference aircrafts and propulsion installation aerodynamics have been validated for different cases. Milea²⁹ performed a structured mesh validation of the methodology for the DLR-F6 configuration. A structured mesh validation of the NASA's Common Research Model (CRM) was performed by Stankowski et al¹⁹⁷. The CRM was also validated for hybrid structured meshes by Goulos et al.¹⁵

D.2 Nacelle aerodynamics validation

D.2.1 Validation cases

The first configuration applied in this validation process is the open source ARA cowl 1 (ARA1) design²⁷. This configuration was originally conceived as an alternative to the conventional NACA series 1 nacelle configurations. It is based on an axisymmetric geometry based on a cylindrical centerbody, a simple circular arc for the forebody, and straight line for the afterbody. The second configuration applied (PVD) is an axisymmetric design based in a more modern approach with no centerbody and²⁸ a continuously curved geometry from leading to trailing edge. The main dimensions of both nacelles can be found in the Table D.1. Both configurations required of a mounted sting (Fig. D.1) for the experimental measurements. Both configuration accounted the effect of the sting on the nacelle drag. However, there are differences on the accounting methodology for the sting contribution. The differences in the account methodology are discussed in the next section. The CFD methodology applied for the validation is analogous to the methodology applied in RINGO. The operation conditions were adapted to model the reference wind tunnel values. The sting was modelled in all cases using no-slip adiabatic surface boundaries. The mass flow rate requirement was derived from the mass flow capture ratio reported by the experimental references and the tunnel operation conditions. The presence of the intrusive measure pressure probes (Fig. D.1) was removed from the CFD models.

TABLE D.1: Key geometry parameters of the nacelles tested in the validation

	ARA1	PVD
$\frac{r_{max}}{r_{hi}}$	1.176	1.237
$\frac{L_{nac}}{r_{hi}}$	6.722	3.68
$\frac{r_{if}}{r_{hi}}$	0.1034	0.484

D.2.2 Reference metrics and accounting

The presence of the sting attached to the the fan cowl requires of a definition of the nacelle drag that accounts for the sting effects. This is accounted in two different

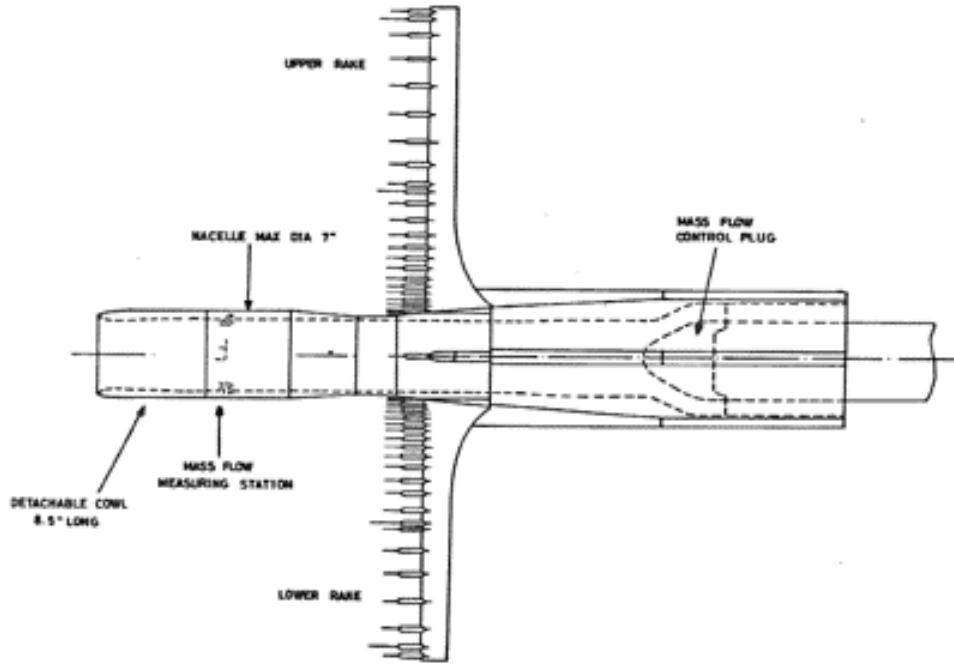


FIGURE D.1: Definition of the experimental rake by Langley (1979)²⁷

ways for the two configurations of the study based on the experimental reference^{27, 28} :

- **ARA1:** The ARA cowl 1 configuration includes the drag of the cylindrical sting (Eq. D.1) from the nacelle trailing edge to the location of the measurement rakes (Fig. D.2). This portion of the sting length equates to a 12% of the nacelle length.
- **PVD:** On the PVD case, the forces of the sting are subtracted from the nacelle drag with the application of a flat plate drag correlation based on the local Reynolds and Mach numbers (Eq. D.2) and weighted based on the wetted areas of the nacelle and the sting (Eq. D.4). Finally, the corrected nacelle drag (Eq. D.5) can be obtained as the different between the drag coefficient (Eq. D.1).

All the drag results on this validation are normalised based on the front area of the maximum radius of each nacelle (Fig. D.2). To evaluate the effect of the drag rise with the increase of the Mach number, the drag rise Mach number, M_{DR} , (Eq. D.6) is defined. To obtain the M_{DR} value a second order interpolation between the sampled drag coefficient values is obtained. Then, a second order finite differences

scheme is applied to obtain the derivatives of the drag coefficient.

$$C'_D = \frac{\phi_{pre} + \phi_{nac} + \phi_{sting}}{0.5\rho_\infty V_\infty^2 A_m} \quad (D.1)$$

$$C_f = \frac{0.455}{(\log_{10} Re)^{2.58} (1 + 0.144 M^2)^{0.65}} \quad (D.2)$$

$$Re_{nac} = \frac{\rho_\infty V_\infty L_{nac}}{\mu} \quad ; \quad Re_{comb} = \frac{\rho_\infty V_\infty (L_{nac} + L_{sting})}{\mu} \quad (D.3)$$

$$C_{D,sting} = \left[\frac{A_{wc} + A_{ws}}{A_m} \right] C_{f,combined} - \frac{A_{wc}}{A_m} C_{f,nac} \quad (D.4)$$

$$C_{D_c} = C'_D - C_{D,sting} \quad (D.5)$$

$$\left. \frac{\partial C_D}{\partial M_\infty} \right|_{M=M_{DR}} = 0.1 \quad (D.6)$$

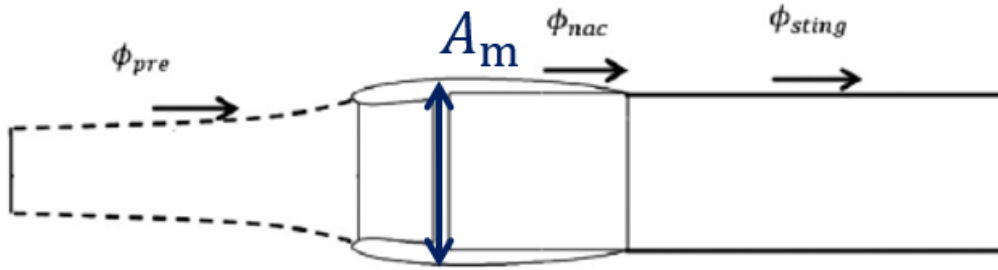


FIGURE D.2: Definition of the force components on the validation process²⁸

D.2.3 2D axisymmetric validation

ARA cowl 1

The ARA1 configuration was evaluated for different flight conditions (Tab. D.2). A Mach number sweep is performed for three different $MFCR$ values : 0.6, 0.7, and 0.76. Also, a $MFCR$ sweep is performed for a constant Mach number of 0.85. The operation total pressure and total temperature were adjusted to match sea level values as reported by Langley²⁷. A 2D axisymmetric approach is applied with Fluent v19.1¹⁵⁵ with the k- ω SST turbulence closure model. The near wall mesh size is adapted to provide a value of y^+ of unity. A second order upwind scheme is utilized for the space discretisation. The gradient computations is based on a Green-Gauss cell-based method. The gas thermodynamic properties are defined

applying the kinetic theory and a real gas model. The flow conditions with altitude are defined by the International Standard Atmosphere (ISA) model¹⁴⁵. The fluid viscosity is adjusted with altitude based on Sutherland's law. The numerical results reported by Robinson et al.²⁸ are used as numerical reference.

The Fluent v19.1 2D axisymmetric model of the drag variation with the MFCR and the Mach number presents an adequate agreement with the experimental reference²⁷ (Fig. D.3). A difference of 11 drag counts (2.85%) at the reference design point (MFCR=0.7, M=0.85) is observed (Tab. D.3). The 2D axisymmetric approach with Fluent v19.1 also provides a sufficient agreement on the Drag Rise Mach number (Eq. D.6). The difference on the M_{DR} between Fluent v19.1 and the experimental reference varies between the 0.25% for MFCR=0.6 and 0.35% at MFCR equal to 0.7 and 0.76 (Tab. D.3). In addition, the CFD evaluation of the ARA1 cowl presents a very close matching with the results reported by Robinson et al.²⁸. In summary, the Fluent v19.1 2D axisymmetric approach can be applied for both cases where the drag rise with the Mach number and the drag absolute value are required.

TABLE D.2: Test conditions for the ARA cowl 1 configuration

ARA Cowl 1 test conditions			
Condition	Mach number	MFCR	AoA [°]
Mach sweep 1	0.65-0.95	0.6	Axi
Mach sweep 2	0.65-0.95	0.7	Axi
Mach sweep 3	0.65-0.95	0.76	Axi
MFCR sweep	0.85	0.35-0.8	Axi

TABLE D.3: Evaluation of the net performance values at selected design points for ARA cowl 1²⁷

ARA cowl 1		
$\Delta = \frac{\varphi - \varphi_{exp}}{\varphi_{exp}}$	Exp ²⁷	Fluent v19 k- ω SST
C_D (M=0.85, MFCR=0.7)	0.0385	0.0396 (2.85%)
M_{DR} (M=0.85, MFCR=0.6)	0.825	0.827 (0.24%)
M_{DR} (M=0.85, MFCR=0.7)	0.848	0.845 (-0.35%)
M_{DR} (M=0.85, MFCR=0.76)	0.852	0.855 (0.35%)

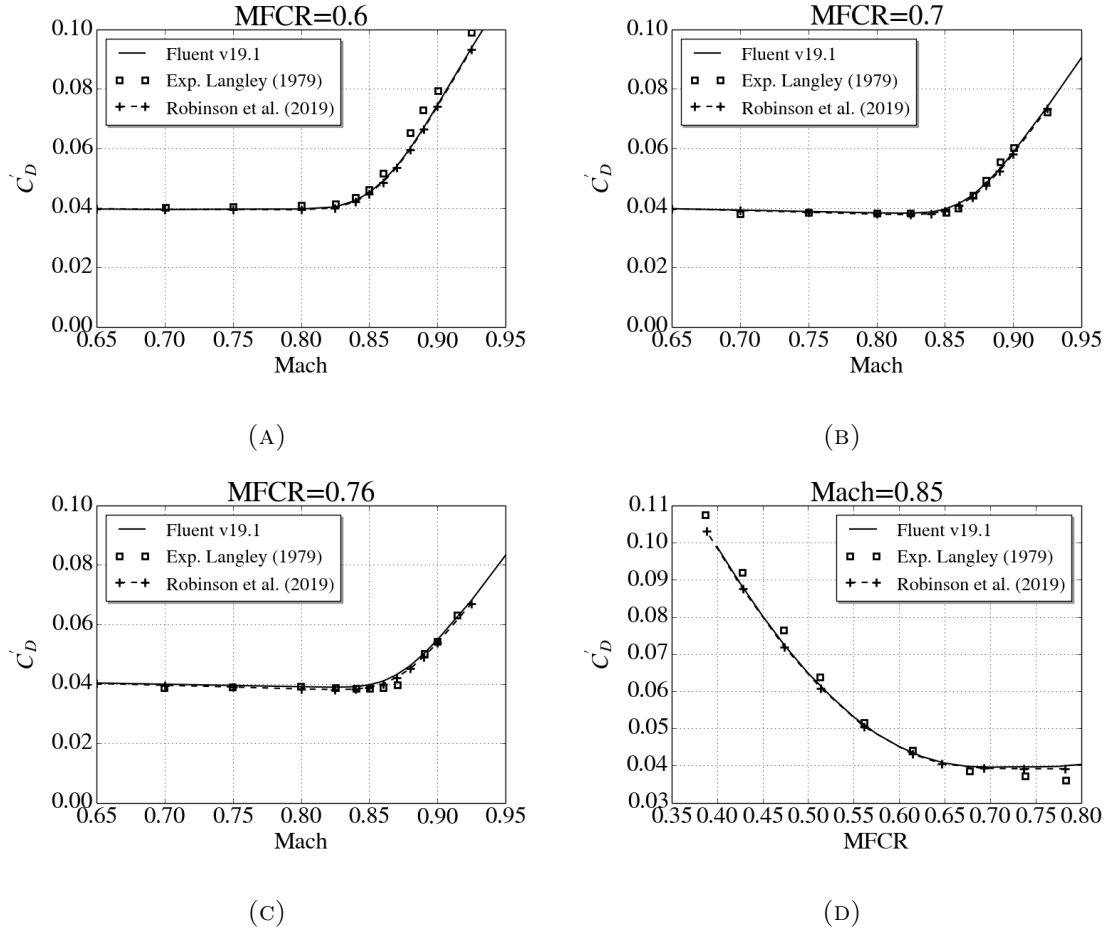


FIGURE D.3: Drag coefficient of nacelle and sting section as a function of freestream Mach number and MFCR for the ARA cowl 1. Experimental results from Langley²⁷. Numerical reference from Robinson et al.²⁸

PVD cowl

The 2D axisymmetric approach was also evaluated with the PVD cowl configuration²⁸ at zero degrees of incidence. Additionally, a 3D extruded mesh of the axisymmetric geometry was evaluated at zero angle of attack to compare both Fluent v19.1 approaches. A Mach number sweep is performed for $MFCR = 0.7$ (Tab. D.4).

An adequate level of agreement was observed in the drag values between both Fluent approaches and the experimental results reported by Robinson et al.²⁸ (Fig. D.4a). The 3D geometry at zero angle of attack has a better agreement with the experimental values when compared to the 2D axisymmetric approach. In both approaches, the discrepancy with the experimental reference increases with the Mach number after $M = 0.88$. This increase on the discrepancy was considered tolerable, since the general trend of the drag rise with the Mach number is maintained and only a

TABLE D.4: Test conditions for the PVD configuration

PVD test conditions			
Condition	Mach number	MFCR	AoA [°]
Mach sweep 1	0.8-0.9	0.7	0
Mach sweep 2	0.8-0.9	0.7	2
Mach sweep 3	0.8-0.9	0.7	4

TABLE D.5: Evaluation of the net performance values at selected design points for PVD²⁸

	PVD		
	Exp. ²⁸	Fluent v19	
		k- ω SST	
		2D axi	3D
$C_D/C_{D,ref}$ (M=0.85, MFCR=0.7)	1.01289	1.00081 (-1.2%)	1.0031 (-0.97%)
M_{DR} (0°) (M=0.85, MFCR=0.7)	0.878	0.88 (0.23%)	0.88 (0.23%)
M_{DR} (2°) (M=0.85, MFCR=0.7)	0.876	-	0.878 (0.228%)
M_{DR} (4°) (M=0.85, MFCR=0.7)	0.873	-	0.874 (0.115%)

small difference was observed on the drag rise Mach number (Tab. D.5). Also, the trend presented by both approaches is consistent with the numerical values reported by Robinson et al.²⁸.

D.2.4 3D flow validation

To validate the 3D CFD approaches, the PVD configuration was evaluated for three angles of attack: 0, 2 and 4 degrees (Tab. D.4). A Mach number sweep was performed for each angle of attack for a fixed $MFCR = 0.7$. The operation total pressure and total temperature were adjusted to match sea level values based on the experimental reference²⁸. A 3D mesh methodology with symmetry plane is applied using Fluent v19.1 with the k- ω SST turbulence closure model. The y^+ value is set to unity. The numerical results reported by Robinson et al.²⁸ are used as numerical reference.

Following the approach defined by Robinson²⁸ the drag rise with the Mach number is reported scaled with the drag value at $M = 0.8$ (Fig. D.4). An adequate level of agreement on the drag rise can be observed for all the evaluated angles of attack.

An increase on the discrepancy with the experimental drag rise values is observed for Fluent v19.1 at the angle of attack of 4 degrees. These discrepancies are reflected on the drag rise Mach number (Tab. D.5) where a consistent difference with the experimental reference is observed for the Fluent v19.1 cases. However, these discrepancies can be considered within tolerable values. Hence, the CFD methodology can be used to perform 3D nacelle drag analysis under flow incidence.

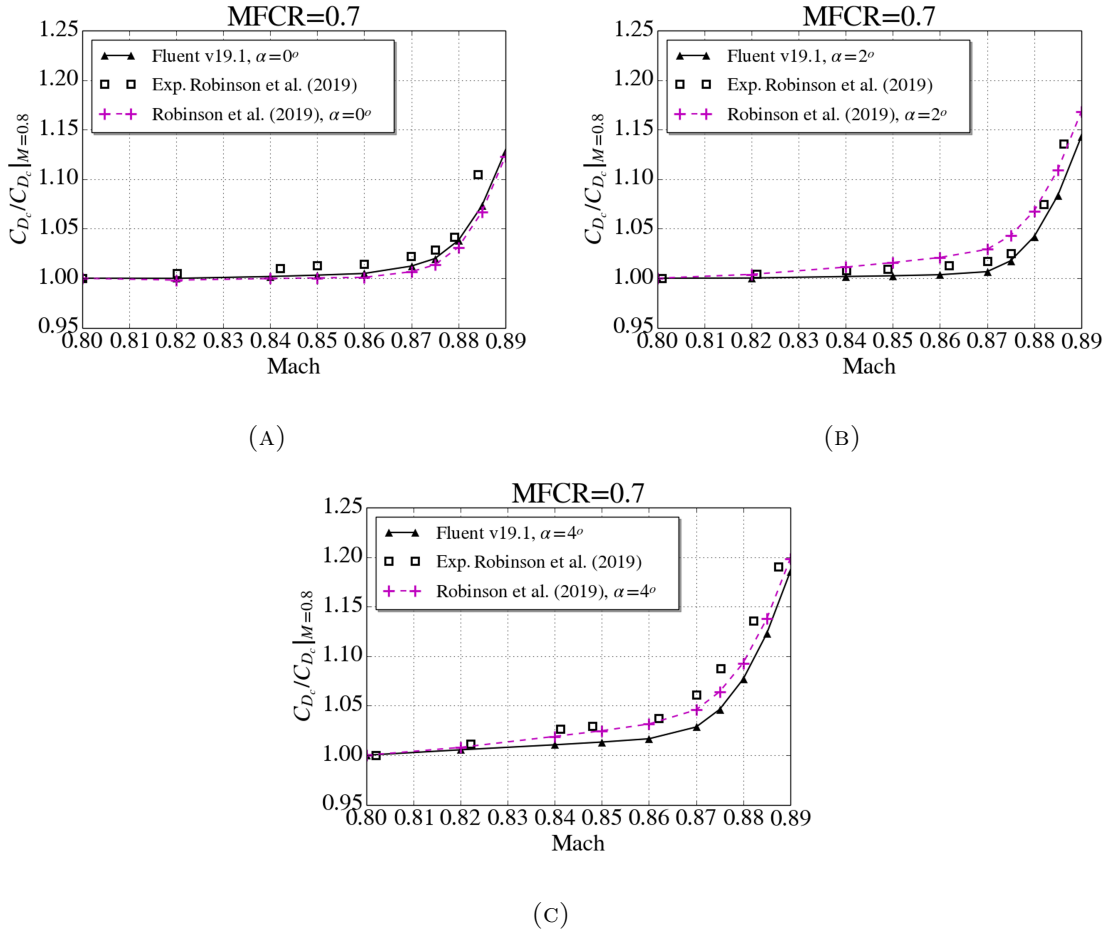


FIGURE D.4: Effect of incidence on drag rise for the PVD geometry, normalised against drag at $M = 0.8$ at $MFCR = 0.7$. Experimental results obtained from Robinson et al.²⁸. Numerical reference from Robinson et al.²⁸

D.3 DLR F6 validation

For the validation of the transonic aerodynamics with hybrid unstructured meshes and wall functions, the DLR-F6^{18,19} aircraft model without nacelle and pylon was selected (Fig. D.5). This configuration is representative of a medium range aircraft with a cruise Mach number of 0.75. The operating lift coefficient is 0.5, achieved

approximately at 0.25 degrees of angle of attack. The coordinate system applied for this model was described in Appendix A. The reference values for the validation are obtained from the Onera wind tunnel experiments referred in the Second AIAA Drag Prediction Workshop¹⁸. In addition to the DLR-F6 aircraft, the DLR-F6 with the wing body fairing FX2B configuration of the Third AIAA Drag Prediction Workshop¹⁹ is also studied in this section (Fig. D.5). However, to the author's knowledge there are not experimental results to compare for this last configuration.

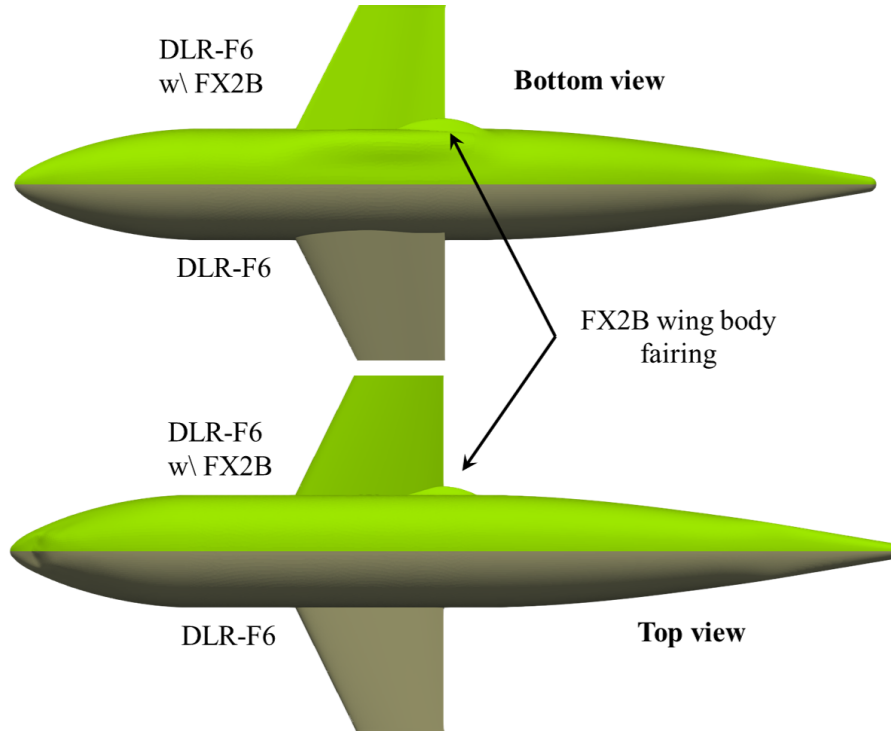


FIGURE D.5: Comparison between the original DLR-F6¹⁸ and the DLR-F6 with the FX2B fairing¹⁹

A tetrahedral hybrid unstructured mesh methodology was used for this validation case. The aim of this validation was to evaluate the accuracy in the drag prediction of the $k-\omega$ SST model with standard wall functions. Thus, an average value of 50 was targeted for the y^+ , and the maximum value was controlled to not exceed a hundred (Fig. D.6). A mesh of approximately 9 million elements was created for the DLF F6 and a mesh of approximately 11 million elements was used for the DLR-F6 with the FX2B fairing. Freestream boundary conditions were applied to perform an angle of attack (AoA) sweep at the cruise Mach number conditions. The total pressure and temperature conditions were adapted to match the experimental conditions. The lift and drag of the aircraft were projected to the wind axis to compare them with the experimental reference.

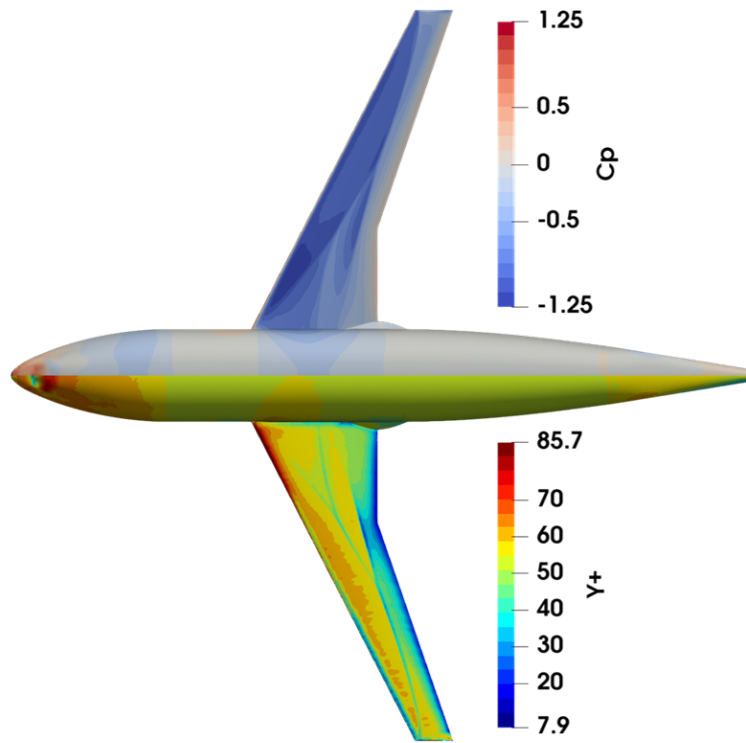


FIGURE D.6: Sample of C_p and y^+ values for the angle of attack 0.0 case with the DLR-F6 with FX2B body fairing for the 3rd Drag Prediction Workshop¹⁹

For further context, the validation studies for the structured grids and y^+ value of unity performed by Milea²⁹ are also included on the result analysis. A more consistent agreement with the experimental values was found for the hybrid mesh with wall functions, when compared with the structured mesh of Milea²⁹ (Fig. D.7). The agreement is consistent across all the drag polar with a difference between the experiments and the CFD model of between 6.5 and 4 drag counts for the 9 CFD points. The drag value at the target lift coefficient ($C_L = 0.5$) was obtained with cubic interpolation. A discrepancy of 4.96 drag counts (-1.68%) was found. Milea's results show a higher agreement on this region of interest (Fig. D.7). However, the hybrid mesh with wall functions provides a more consistent representation of the drag polar. Thus, the mesh methodology with wall functions can be considered appropriate for studies that involve aircraft drag despite some uncertainty on the global drag value is expected. Nevertheless, for comparative studies the change of the drag values between designs is expected to be consistent.

For the case with the FX2B faring, it was observed how the overall drag polar of the

configuration maintains the general distribution. The results are offset to lower drag and lift values overall ($\sim 1.5\%$). This is due to the correction of the flow separation near the wing fuselage joint¹⁹. The fairing was included to mitigate those effects and reduce the drag. Overall, the hybrid mesh methodology for both configurations provides adequate results for the test of the present work.

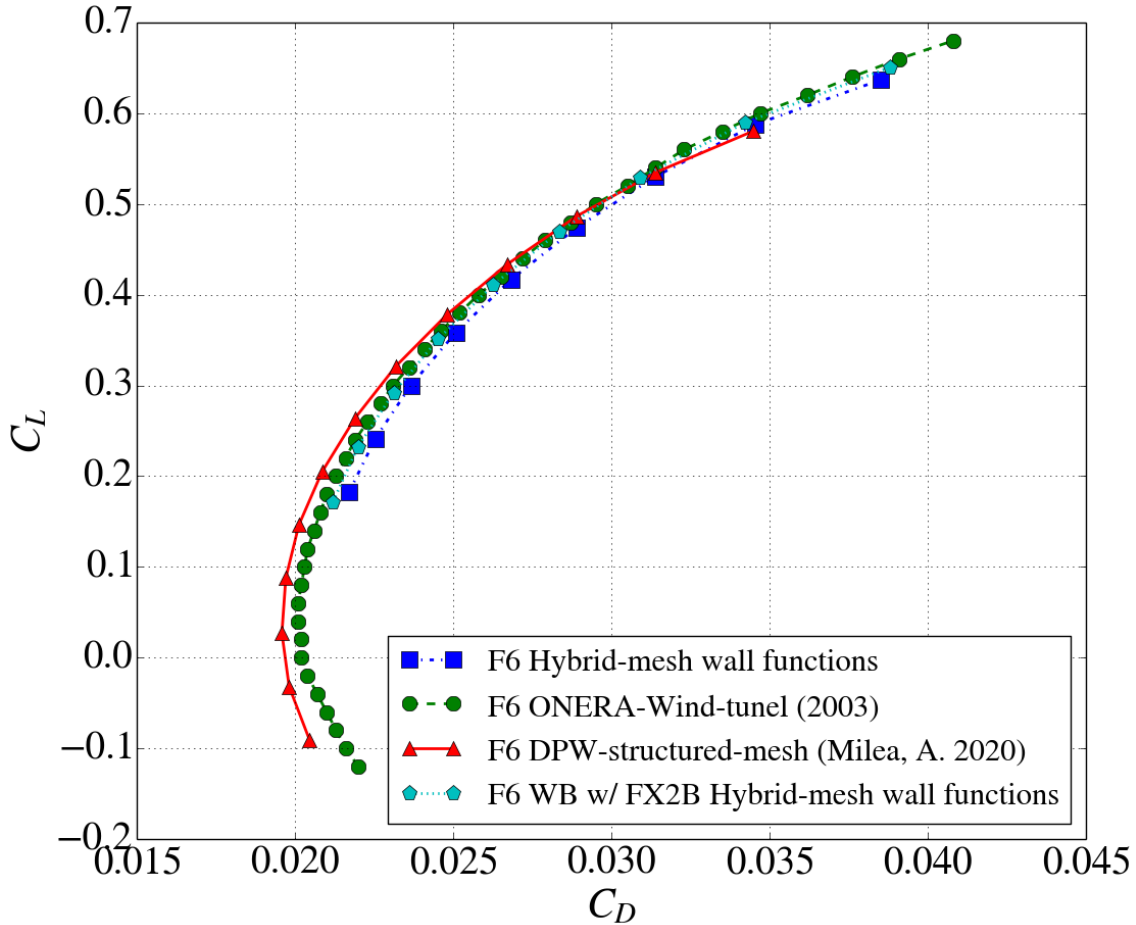


FIGURE D.7: Validation of the DLR-F6 polar and based on the experimental results of ONERA wind tunnel for the Second Drag Prediction Workshop¹⁸. The analysis includes the structured mesh validation performed by Milea²⁹ for comparison. The DLR-F6 with FX2B body fairing for the 3rd Drag Prediction Workshop¹⁹ are also included for additional context

D.4 Conclusions

This appendix was focused on the validation of the general CFD methodology applied for the present work. The main conclusions of this validation chapter are:

- Important steps have been taken in the past for the validation of the CFD approach for the aerodynamics of the propulsion integration. The present work is build partly from these cases
- The validation of the low order turbomachinery model was included on the main body of the thesis
- Two additional cases have been considered to validate the CFD methodology for transonic aerodynamics. First, the structured mesh methodology was validated for benchmark cases for nacelle aerodynamics. Second, the hybrid mesh methodology with wall functions was validated for a reference aircraft configuration under transonic conditions
- It was found that the CFD methodology of the nacelle cases for both 2D axisymmetric and 3D structured meshes provided an adequate prediction of the Drag Rise Mach number. The net drag value prediction was also considered adequate for both approaches. Thus, it was concluded that the methodology can be used to predict transonic aerodynamics for nacelles
- The DLR-F6¹⁸ experimental drag polar was compared with the polar obtained with the same CFD methodology. Overall, an adequate agreement was found with a discrepancy of between 4 and 6.5 drag counts. At the cruise design lift requirement the error was 4.96 drag counts. Thus, it was concluded that the hybrid mesh methodology with wall functions is adequate to represent the aircraft drag characteristics

APPENDIX E

Additional considerations for the low order turbomachinery model

In this appendix the additional considerations on the implementation of the low order turbomachinery model are described. This appendix includes the blade and vane geometry extraction, the detailed explanation of the calibration process, and finally some additional results on the verification of the CFD methodology.

E.1 Geometry feature extraction

The blade (Fig. E.1) and vane (Fig. E.2) geometry must be simplified to provide an input for the BFM. Two main elements of the geometry are required as inputs for the model: the element geometry normal vectors, and the metal blockage terms. The main methodology to define the surface normals and blade angles is based on Thollets work¹⁷. To obtain the geometry normal vectors the blade and vane, the mean curve (θ_m) is defined from the azimuthal component of the element at the suction (θ_{ss}) and pressure (θ_{ps}) sides at a given axial and radial location (Eq. E.1). Then, the mean camber angle (Eq. E.2), β_m , and the mean lean angle (Eq. E.3), λ_m , are defined using a second order centred finite difference scheme for the derivatives. The mean camber and lean angles are used then to define the normal vector components in polar coordinates for each blade or vane location (Eqs. E.4-E.6).

The metal blockage factor (Eq. E.7), b , is defined as the fraction of the space between consecutive blades referred to two consecutive mean curves (Fig. 3.29). For the Hall-Thollet model¹⁷, the gradient of the blockage factor is also required. This could be achieved at internal level in the solver. However, it was observed to be

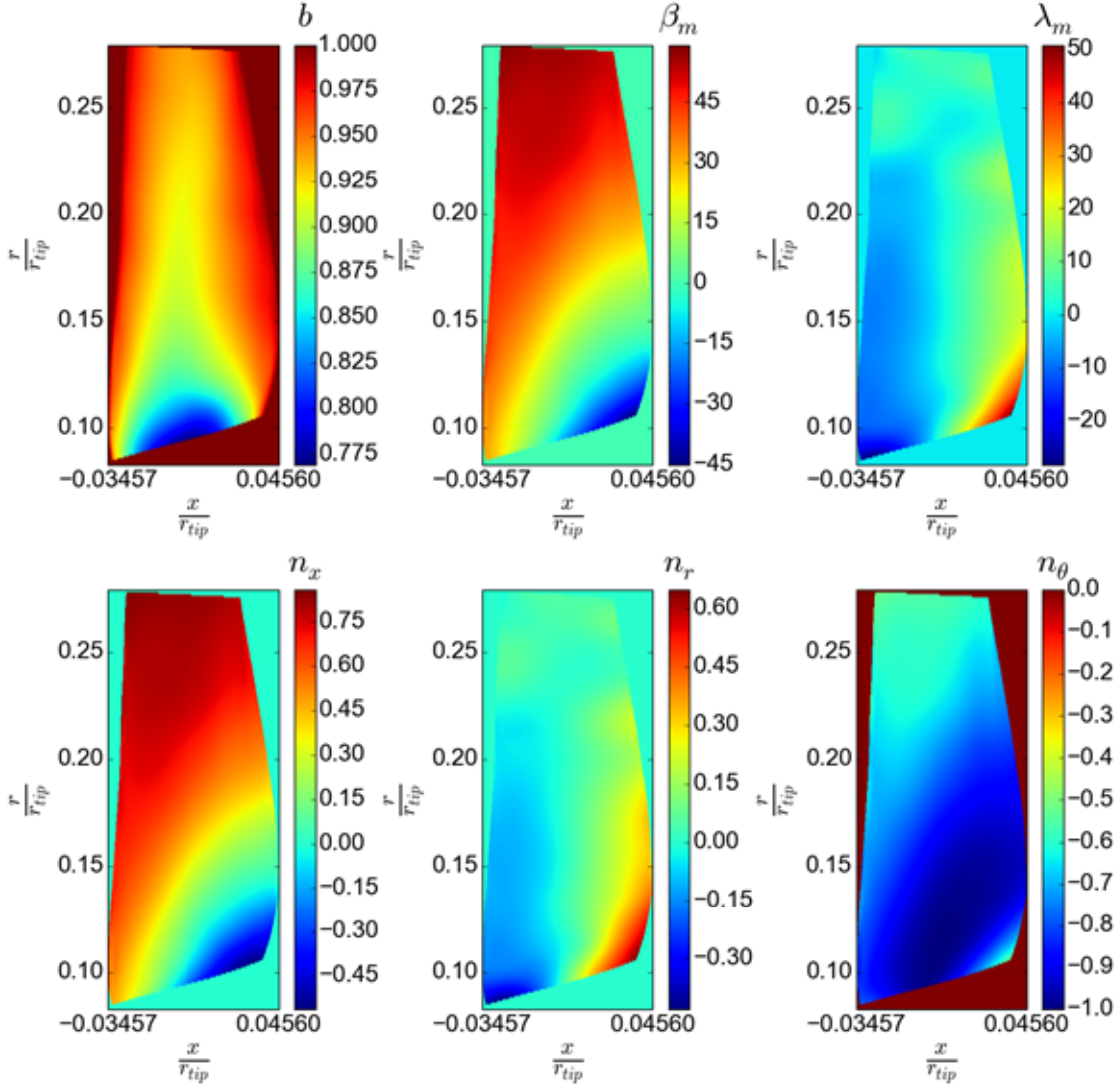


FIGURE E.1: Blade geometry characteristics to be input on the BFM Fluent implementation. Length elements normalized with the reference blade tip radius

repeated for any iteration. Hence, to reduce the computational cost of the model the gradient is computed externally using a second order centred finite differences scheme.

$$\theta_m = \frac{(\theta_{ps} + \theta_{ss})}{2} \quad (E.1)$$

$$\beta_m = \arctan\left(r \frac{\partial \theta_m}{\partial x}\right) \quad (E.2)$$

$$\lambda_m = \arctan\left(r \frac{\partial \theta_m}{\partial r}\right) \quad (E.3)$$

$$n_x = \frac{\beta_m}{|\beta_m|} \sqrt{\frac{\tan(\beta_m)^2}{1 + \tan(\beta_m)^2 + \tan(\lambda_m)^2}} \quad (E.4)$$

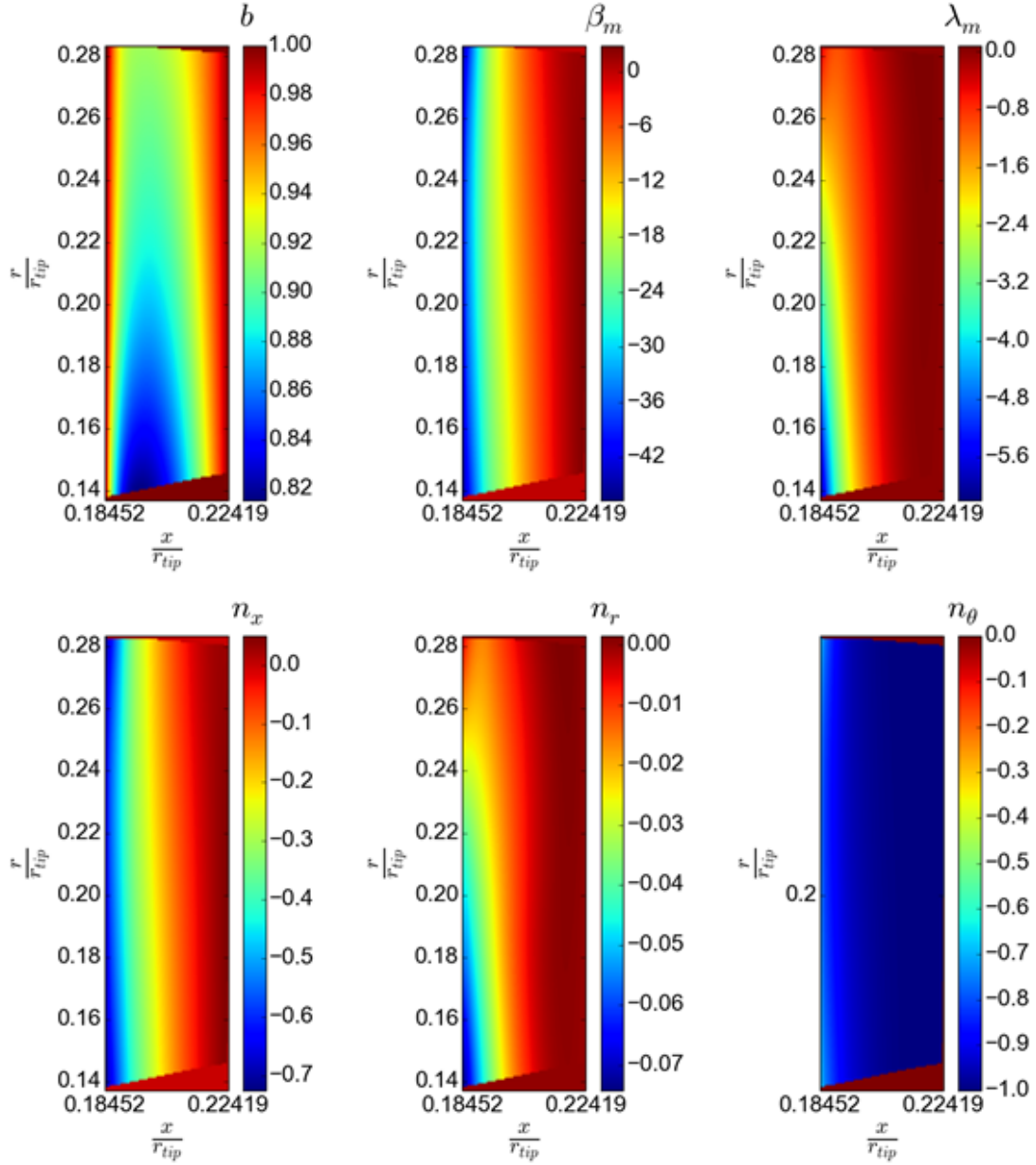


FIGURE E.2: Outlet Guide Vane geometry characteristics to be input on the BFM Fluent implementation. Length elements normalized with the reference blade tip radius

$$n_\theta = -\frac{n_x}{\tan(\beta_m)} \quad (\text{E.5})$$

$$n_r = -\frac{n_\theta}{\tan(\lambda_m)} \quad (\text{E.6})$$

$$b = N_{blades} \frac{\theta_{ps} - \theta_{ss}}{2\pi} \quad (\text{E.7})$$

E.2 Hall-Thollet model calibration process

The Hall-Thollet model¹⁷ requires calibration to adjust the isentropic efficiency distributions to match the higher fidelity reference. These reference results can be higher fidelity numerical approaches or experimental data. The calibration is performed with the activation of the second term of the parallel force (Eq. E.8) with the definition of the reference deviation angle $\delta^{\eta_{cal}}$. This calibration of the η_{is} has a limited impact on the FPR (Fig. E.3). Some, applications of the Hall-Thollet model¹⁹⁹ have suggested to apply the maximum isentropic efficiency point as reference for the local deviation angle, and to correct the differences with the higher order reference with an calibration constant on the C_f definition. However, the selection of a different reference point for the local deviation angle eliminates the necessity to include additional calibration constants (Fig. E.3).

$$f_p = \frac{0.5W^2}{sb|n_\theta|} \left(2C_f + 2\pi K_{Mach} (\delta - \delta^{\eta_{cal}})^2 \right) \quad (E.8)$$

The process is carried in two steps. First, a range of capacity, Q , points are sampled

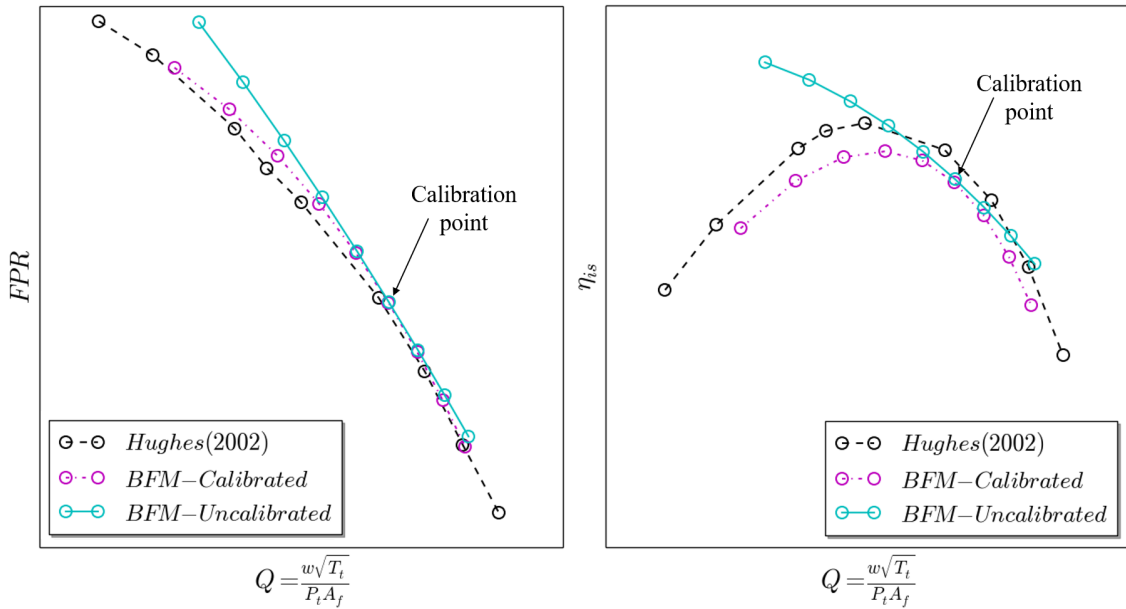


FIGURE E.3: Calibration process schematic, effects on the FPR and η_{is} . Experimental data from Hughes et al.²³

for a given fan rotational speed considering only the friction coefficient, C_f , effect on the parallel force (Eq. E.8). Second, a point of the operation line is selected as the calibration point (Fig. E.3) and the local deviation angle (Eq. E.9), δ , is

selected as reference. For this work, results are adjusted with the experimental values where possible and with the higher order numerical reference when the uncalibrated reference line does not allow a better experimental match.

$$\delta = \arcsin \left(\frac{\mathbf{W} \cdot \mathbf{n}}{|\mathbf{W}|} \right) \quad (\text{E.9})$$

E.3 Radial distributions

Experimental results were only available for one operation point at two of the calibrated fan rotational speeds. To assess the performance of the BFM approaches for different operating points, the radial distributions of the pressure ratio, isentropic efficiency, and swirl angle at near-peak-efficiency point for all the calibrated rotational speeds are compared with the Mixing Plane model as a higher accuracy numerical reference. The results are sampled at the Fan Exit Rake and the OGV Exit Rake stations (Fig. 6.1).

The Hall-Thollet model provides a fair representation of the fan throughflow across different operation conditions compared with a higher fidelity numerical reference. The radial distribution of PR_{fan} of the two BFM approaches (2D axisymmetric and 3D sector) provides an adequate match with the Mixing Plane model at both measurement stations and for all the fan operation lines (Figs. E.4-E.7). The pressure ratio differences between the two models increases in across the region from the 30% to the 75% of the span. The isentropic efficiency radial distributions of the BFM approaches have a good level of agreement with the Mixing Plane model (Figs. E.4-E.7), with discrepancies of $< 1\%$ for the 3D sector BFM and 1.5% for the 2D axisymmetric BFM approach. For all the fan rotational speeds, the 2D axisymmetric approach has an increase in the discrepancy with the Mixing Plane model from the hub to the 40% of the span at the Fan Exit Rake station. The swirl angle radial distributions of the two BFM approaches have a good level of agreement with the Mixing Plane reference (Figs. E.4-E.7). The discrepancy on the swirl angle decreases with the increase of the fan rotational speed, with a similar reduction on the differences in the pressure ratio radial distributions. The maximum differences between the BFM approaches and the reference are located between the 40% and the 90% of the span, with differences between 2 to 5 degrees for the different rotational speeds.

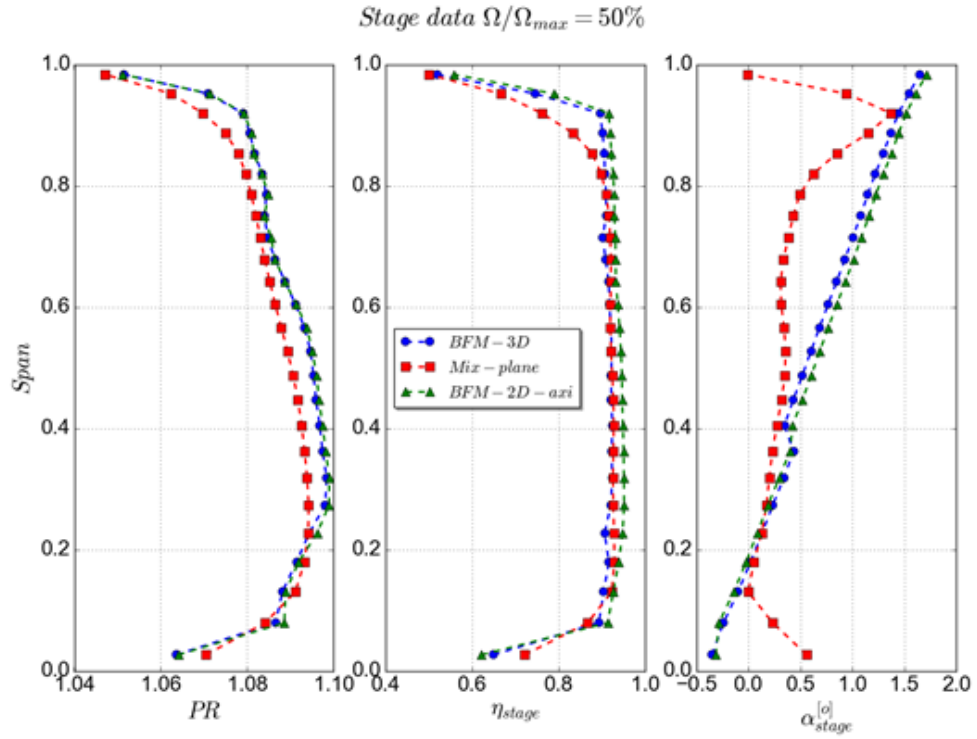
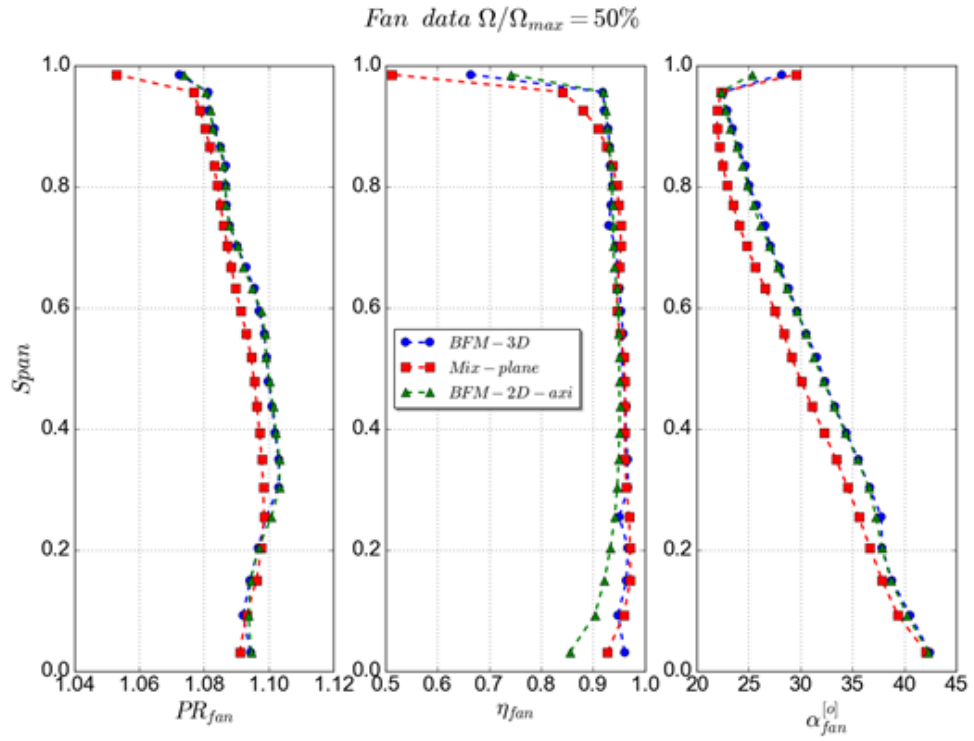


FIGURE E.4: Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 50% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit

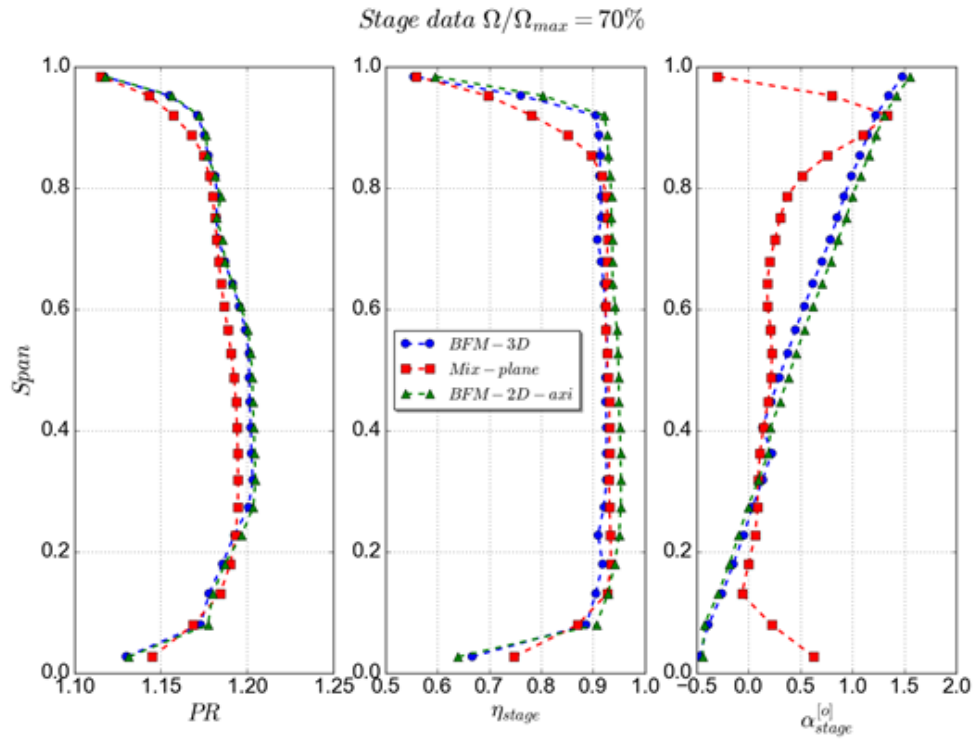
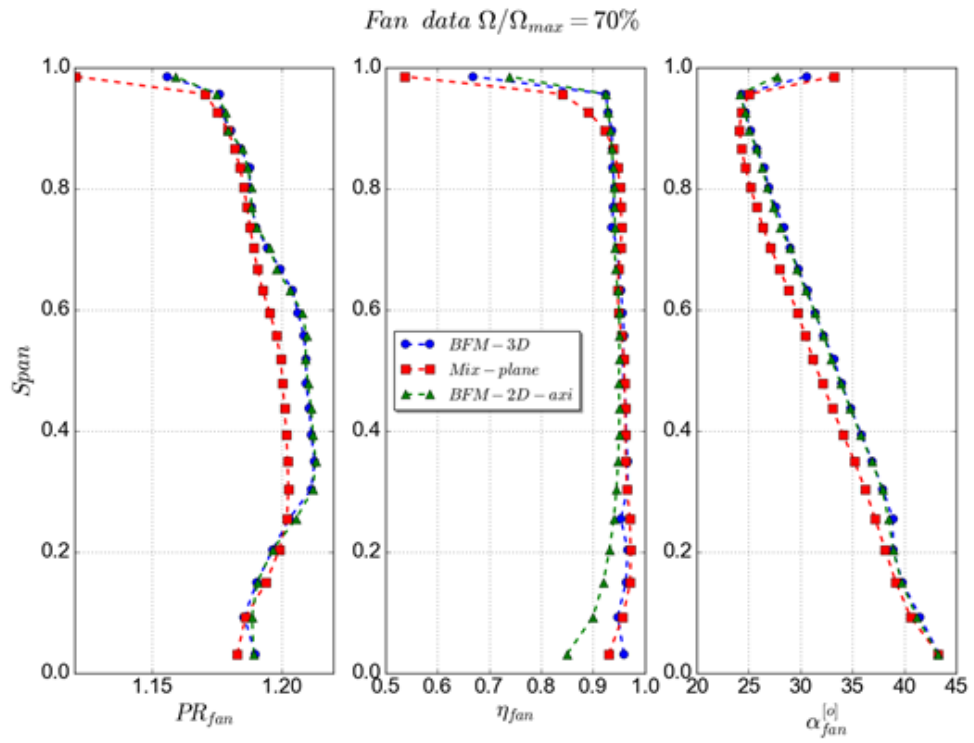
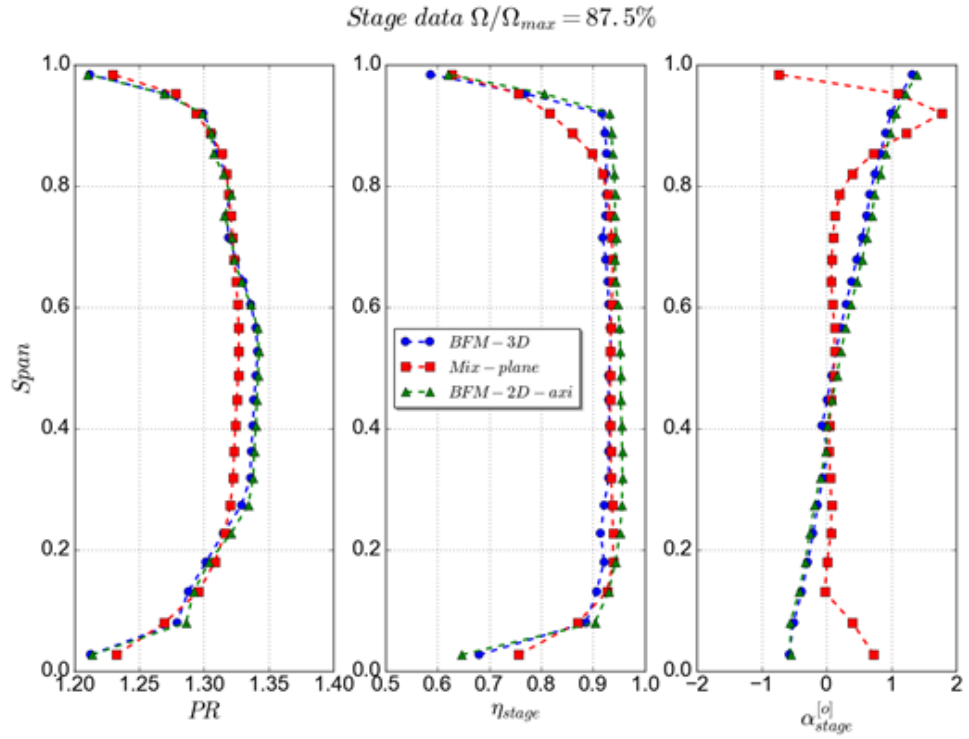
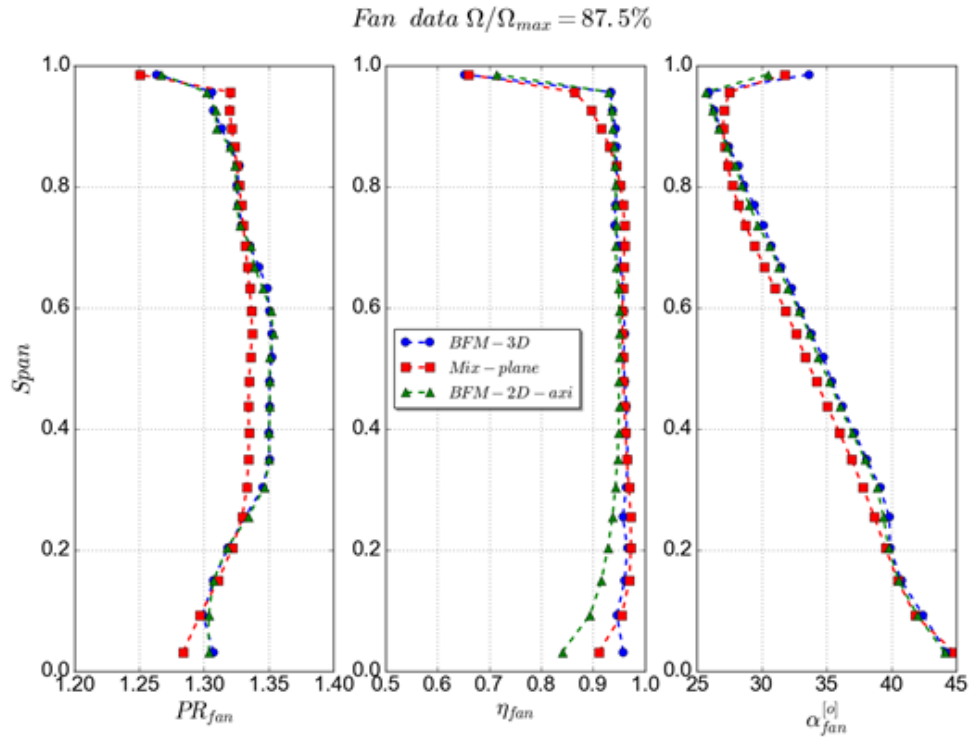


FIGURE E.5: : Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 70% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit



(A)



(B)

FIGURE E.6: Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 87.5% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit

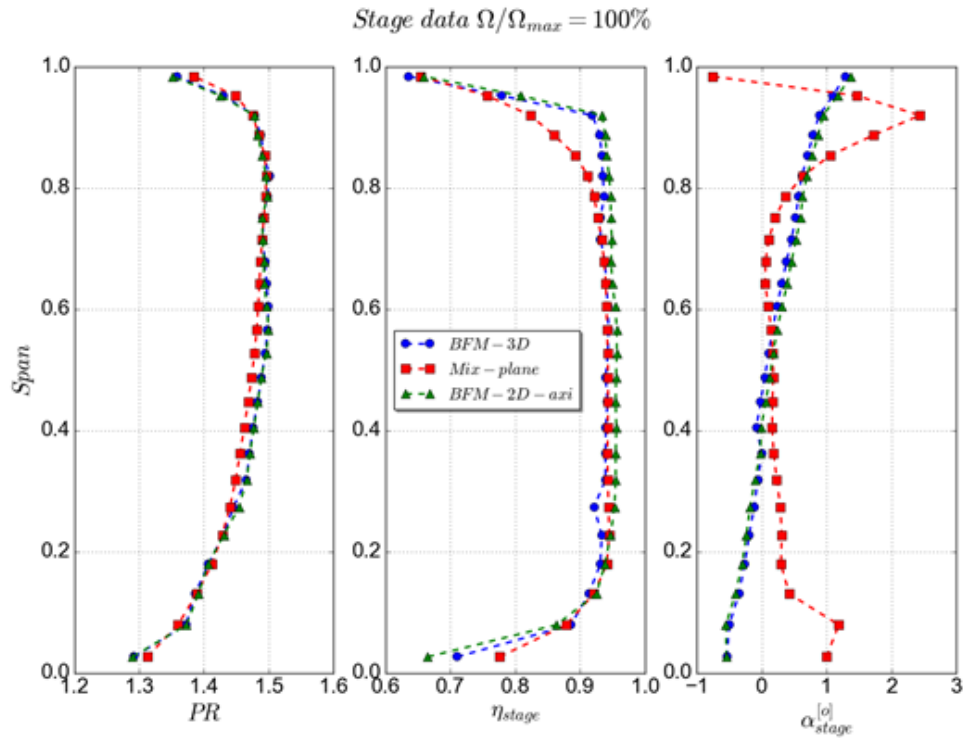
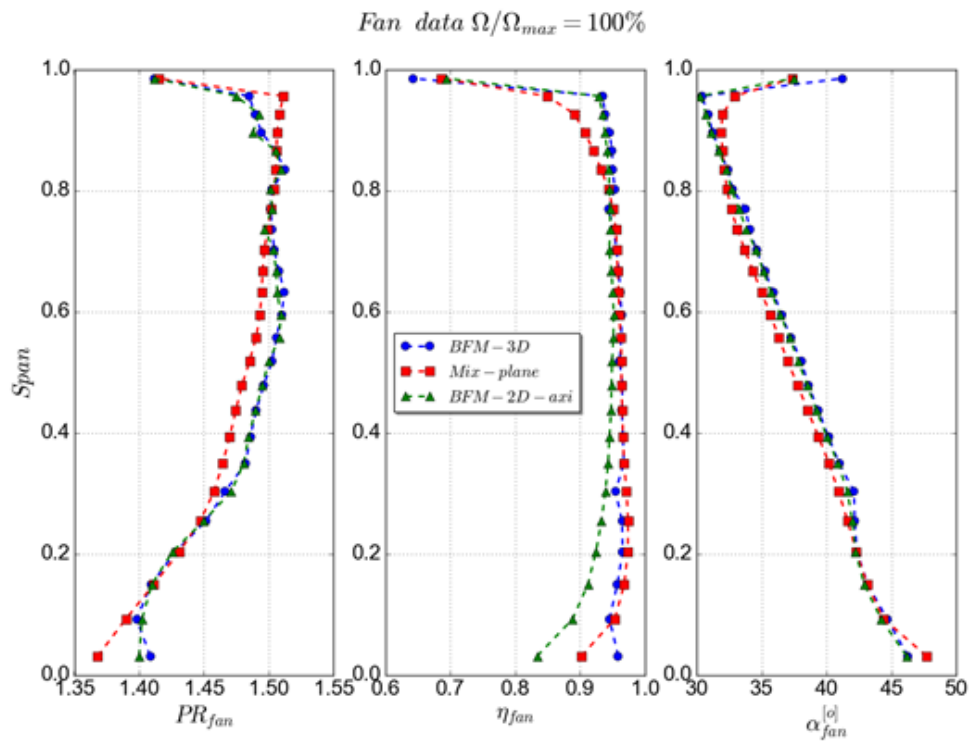


FIGURE E.7: Near-peak efficiency pressure ratio, isentropic efficiency, and swirl angle radial profiles, taken at the OGV and Fan Exit Rake locations for the 100% maximum rpm line: (a) Measured at the OGV exit, and (b) measured at the fan exit

APPENDIX F

Multi-objective optimization benchmark problems and algorithm selection

In this appendix, the multi-objective optimisation algorithms applied on the present work (NSGA-II, IBEA, and OMOPSO) are tested against a set of optimisation benchmark problems. Two of the proposed Zitzler-Deb-Thiele's test functions (ZDT)³⁰ are selected for this purpose. Each of the test problems have an analytical solution to the Global Pareto-set ($\bar{Y}$) and present a different Pareto topology. The algorithm performance is evaluated by the comparison of the convergence rate via the Hypervolume Index (Eq. 3.92) and the consistency of the Local Pareto-optimal ($\bar{Y}'$) set after a fixed number of generations.

F.1 Reference functions

Zitler-Deb-Thiele's optimisation test functions³⁰ are a set of standard multi-objective optimisation test problems designed to compare the performance of different optimisation algorithms. The optimisation problem is based in two objective functions (f_1 and f_2) to minimise. At the same time, these objective functions are composed of two auxiliary functions (h and g). These auxiliary functions are designed to modify freely the Pareto front shape and requirements. An undetermined number of variables x_i can be applied with values between zero and one. This allows to test the performance of the algorithms with low and high dimensionality problems. The proposed test functions do not have constraints to the solution or the design space.

F.1.1 Zitzler-Deb-Thiele's function 1

The third Zitzler-Deb-Thiele's function (Eq. F.1) is designed for a convex Pareto-optimal front. The Pareto-optimal front is formed when $g(\mathbf{x}) = 1$. On the present test, the number of variables for the test is set to 13.

$$\text{Minimize} = \begin{cases} f_1(\mathbf{x}) = x_1 \\ f_2(\mathbf{x}) = f(\mathbf{x})h(f_1(\mathbf{x}), g(\mathbf{x})) & 0 \leq x_i \leq 1 \\ g(\mathbf{x}) = 1 \frac{9}{N-1} \sum_{i=2}^N x_i & 1 \leq i \leq N \\ h(f_1(\mathbf{x}), g(\mathbf{x})) = 1 - \sqrt{\frac{f_1(\mathbf{x})}{g(\mathbf{x})}} \end{cases} \quad (\text{F.1})$$

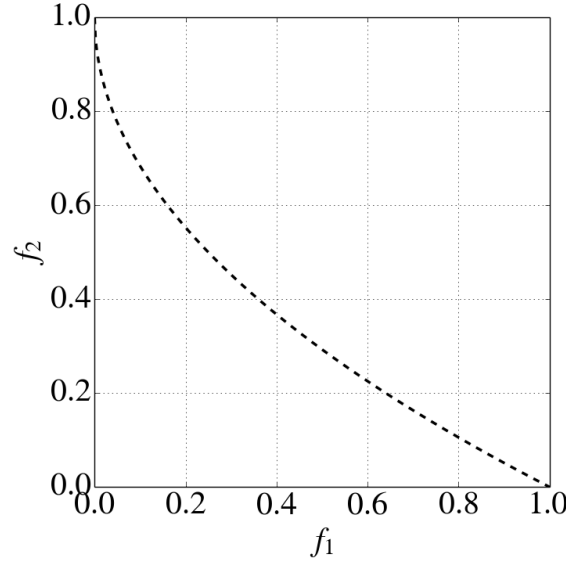


FIGURE F.1: Global Pareto-optimal set defined for the Zitzler-Deb-Thiele's function 1 ($g(\mathbf{x}) = 1$)

F.1.2 Zitzler-Deb-Thiele's function 3

The first Zitzler-Deb-Thiele's function (Eq. F.2) is designed to represent the discreteness feature. In this case, this means that the Pareto-Optimal solution is formed by a set non-contiguous convex parts. This is a convenient test to evaluate the performance of the algorithms for discontinuous design spaces. The Pareto-optimal front is formed when $g(\mathbf{x}) = 1$. On the present test, the number of variables for the

test is set to 13.

$$\text{Minimize} = \begin{cases} f_1(\mathbf{x}) = x_1 \\ f_2(\mathbf{x}) = f(\mathbf{x})h(f_1(\mathbf{x}, g(\mathbf{x}))) \\ g(\mathbf{x}) = 1 \frac{9}{N-1} \sum_{i=2}^N x_i \\ h(f_1(\mathbf{x}, g(\mathbf{x}))) = 1 - \frac{f_1(\mathbf{x})}{g(\mathbf{x})} \sin(10\pi f_1(\mathbf{x})) \end{cases} \quad \begin{matrix} 0 \leq x_i \leq 1 \\ 1 \leq i \leq N \end{matrix} \quad (\text{F.2})$$

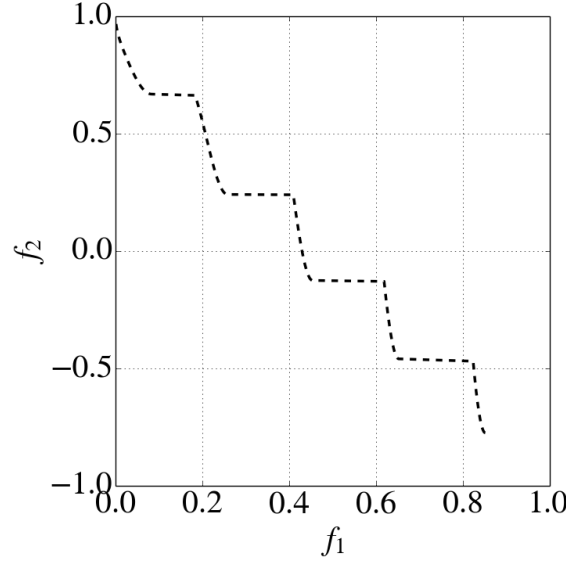


FIGURE F.2: Global Pareto-optimal set defined for the Zitzler-Deb-Thiele's function 3 ($g(\mathbf{x}) = 1$)

F.2 Selected algorithm configurations

The three algorithms selected for this test (Tab. F.1) are the evolutionary algorithms IBEA¹⁶², and NSGA-II¹⁶⁰; and the particle swarm algorithm OMOPSO¹⁶³. The implementation on the tool RINGO (Routine for Intake and Nacelle Genetic Optimisation) performed by Robinson²² is applied. OMOPSO and IBEA algorithms were obtained from the open source optimisation python library "Platypus"¹⁶¹. The mutation and crossover strategies for each algorithm are defined in Section 3.5.3. The algorithms are compared for two different seed dimensions with 65 and 365 individuals, respectively (Tab. F.2). A Latin-Hypercube Sampling (LHS)¹⁶⁴ is used to evenly populate the initial design space. The number of individuals per generation after the seed was fixed to 65 in both cases. The number of total generations was

TABLE F.1: Algorithm selected for the multi-objective methodology verification

Algorithm		
Particle Swarm	Evolutionary	
OMOPSO	IBEA	NSGA-II

adapted to have the same number of total evaluations for both seed dimensions. this results in 100 generations for the initial seed of 65 individuals, and 96 generations for the initial seed of 365 individuals. This test was repeated a total of 40 times per algorithm to evaluate the replicability of the Pareto-optimal set and its statistical variation.

TABLE F.2: Test array for the multi-objective optimization algorithm performance test

Objective function	DTZ1		DTZ3	
Degrees of freedom	13		13	
Number of generations	100	96	100	96
Seed size (LHS)	65	325	65	325
Generations size	65			
Repetitions	40			

F.3 Pareto consistency

Prior to the statistical analysis of the Pareto-optimal set convergence, the efficacy of the algorithms was verified against the reference Global Pareto-optimal set. For this verification, a single optimisation problem was run for both of the initial seed dimensions of the test (Tab. F.2). The verification was performed against both of the ZDT functions proposed.

For the ZDT function 1, both initial seed dimensions (65 and 365) converge closely to the reference Global Pareto front defined by Zitler et al.³⁰. However, a clear improvement on the population of the Pareto front space is observed for the evolutionary algorithms (IBEA and NSGA-II) with the increase of the initial population size (Figs. F.3-F.4). This implies a dependency on the design of experiments for the initial sample for the evolutionary algorithms. At the same time, the particle swarm algorithm (OMOPSO) has a higher consistency for both initial seed sizes. This indicates that for unconstrained convex Pareto front problems, the particle swarm algorithm could be applied to reduce the computational size. Also, the application of

the evolutionary algorithms will be conditioned by the possibility to perform larger initial samples.

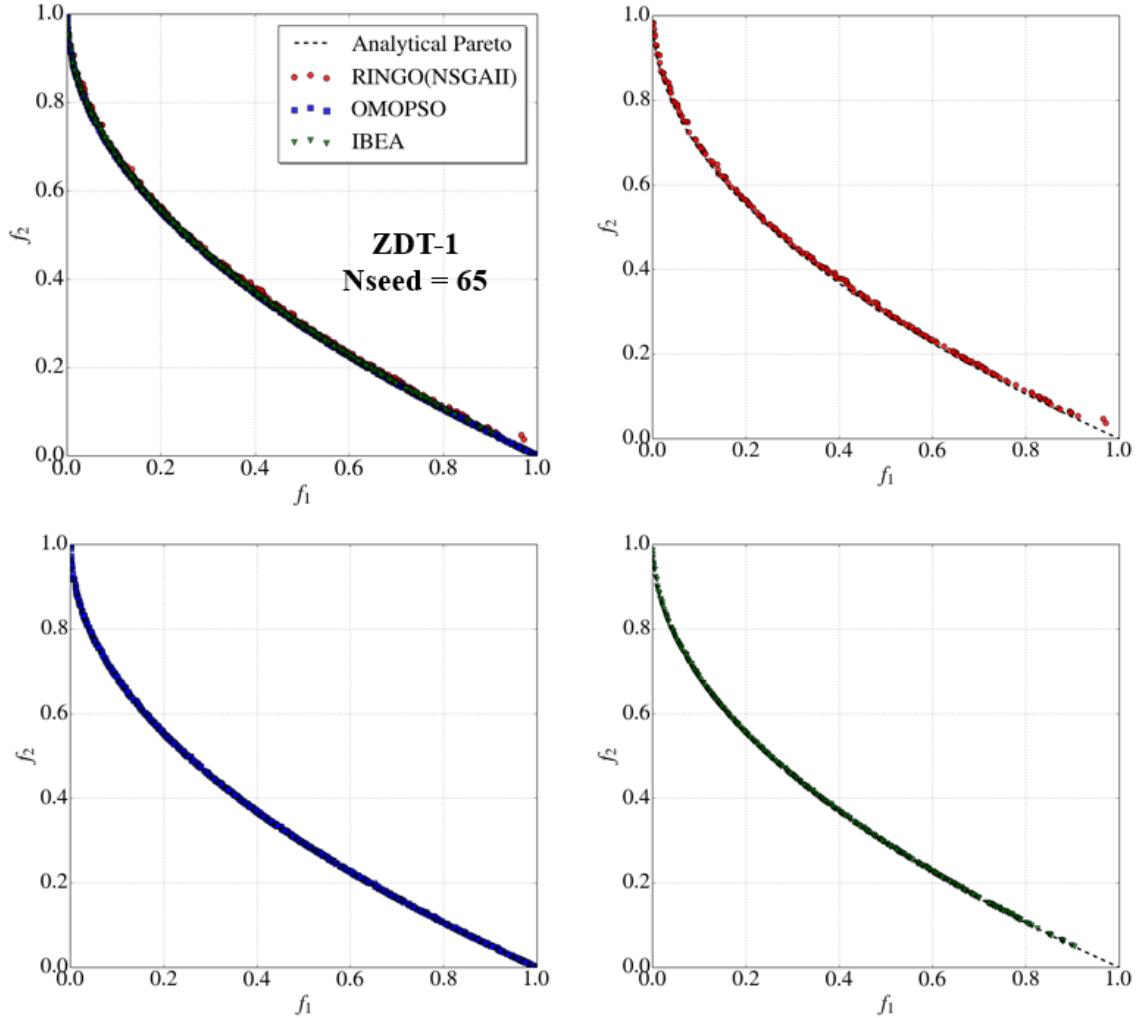


FIGURE F.3: ZDT function 1 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 65 individuals

For the ZDT function 3, the non-dominated set of solutions of IBEA and OMOPSO for both initial seed sizes (65 and 365) show an adequate agreement with the Global Pareto-optimal set by Zitzler et al.³⁰. However, the non-dominated solutions of NSGA-II do not fully approximate the Global Pareto-optimal set (Fig. F.5-F.6). The inaccuracy of the RINGO implementation of NSGA-II²² is more evident for the smaller seed size case (Fig. F.5), where the last discrete section of the Global Pareto front (minimum f_2 and maximum f_1) is not covered. The performance of NSGA-II improves with the size of the seed (Fig. F.10). However, the limits of the discrete Pareto clusters are not fully captured by the algorithm in the number of generations of the test. On the other hand, the consistency of IBEA and OMOPSO for different

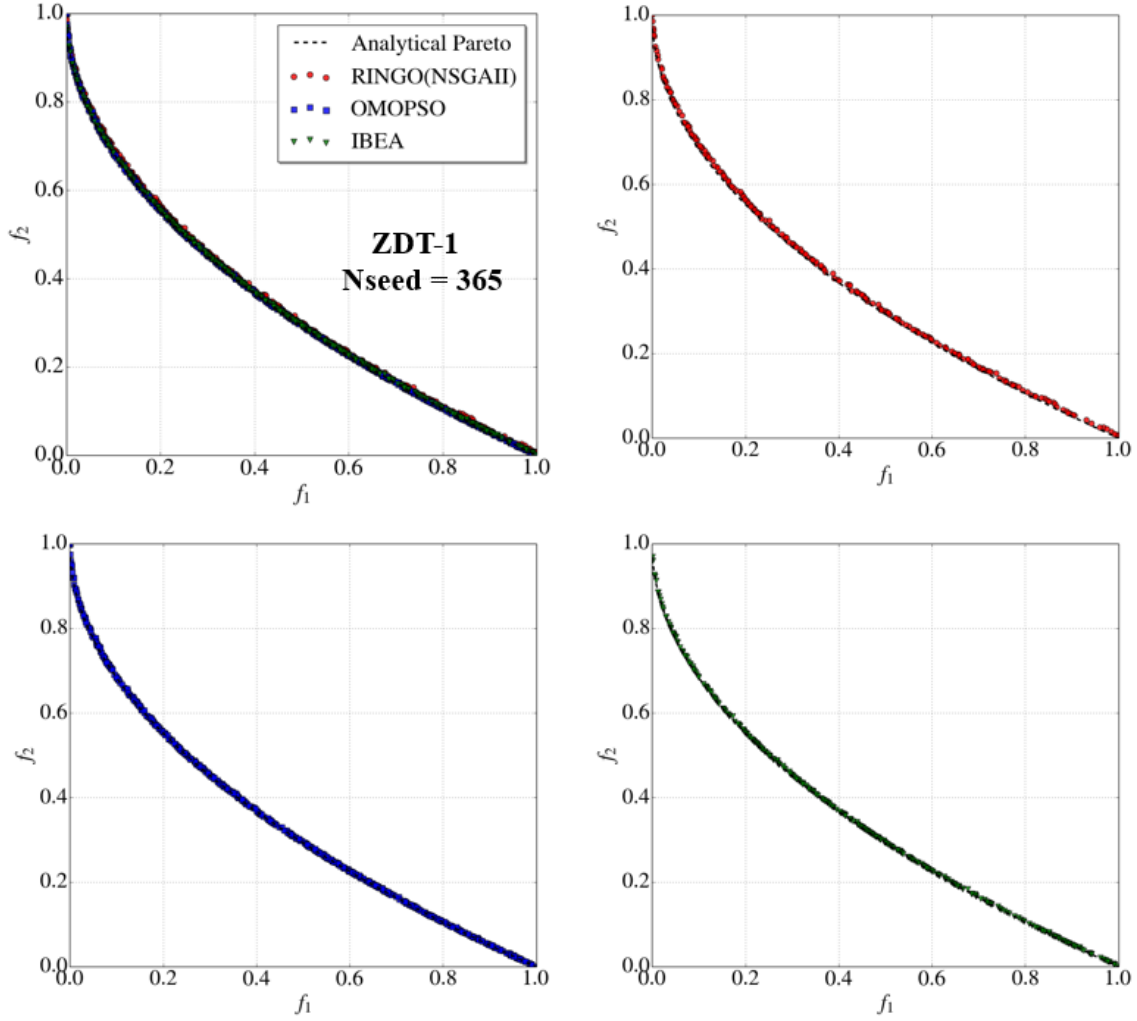


FIGURE F.4: ZDT function 1 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 365 individuals

seed sizes could indicate that both algorithms are suitable for unconstrained optimisation problems with different Pareto front clusters. Also, indicates that a possible reduction of the computational cost could be achieved with lower initial seeds.

F.4 Pareto convergence

To understand the convergence rate of the different algorithms (Tab. F.1) across the different generations, the Hypervolume Index, HV, (Eq. 3.92) is evaluated for the individual test case of the Pareto consistency case. The Hypervolume index is normalised by the Hypervolume value of the Global Pareto-optimal set defined by Zitzler et al.³⁰.

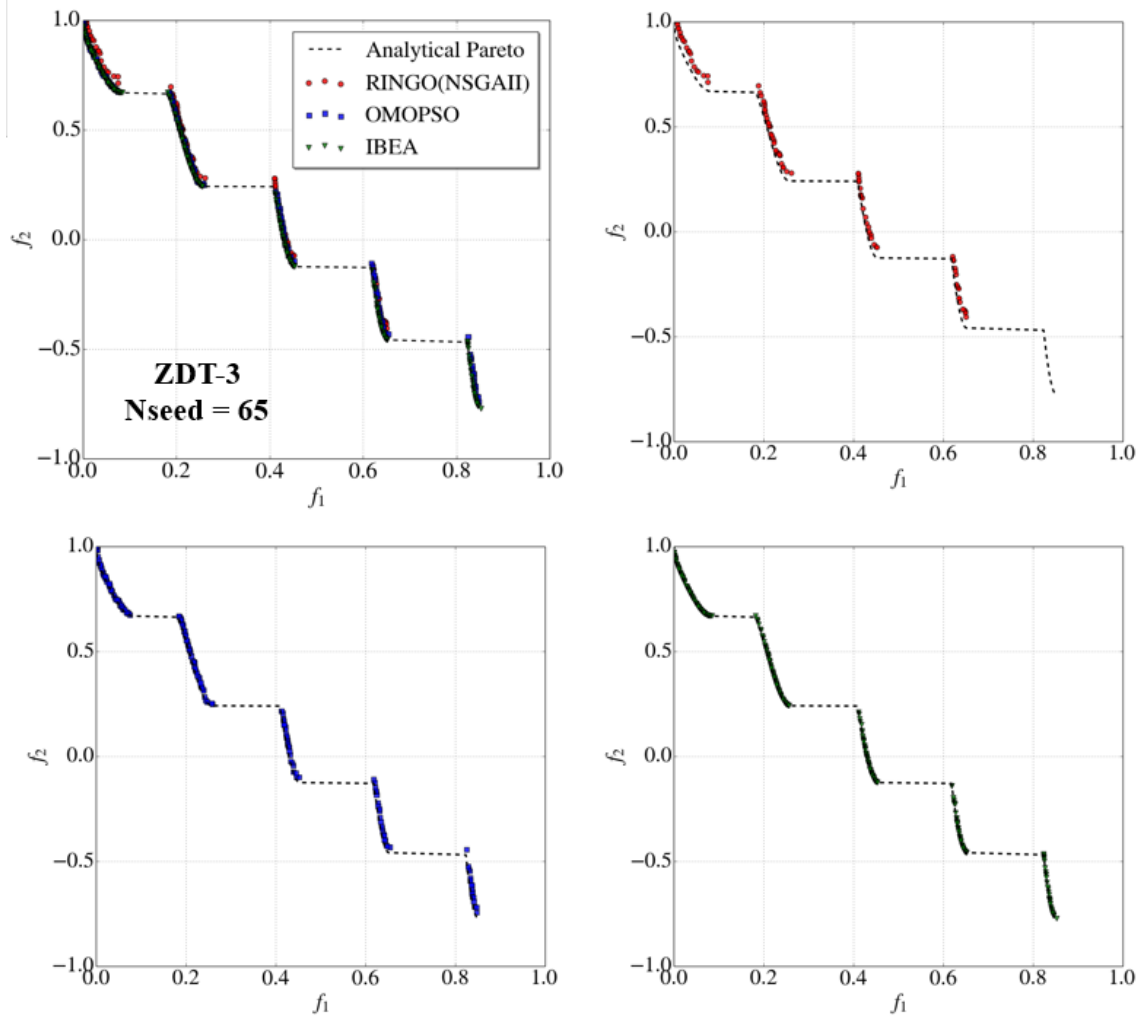


FIGURE F.5: ZDT function 3 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 65 individuals

For the convex Pareto front of the ZDT function 1, OMOPSO presents a faster convergence to the reference HV value than the evolutionary algorithms (Fig. F.7). The convergence rate of OMOPSO is better than the evolutionary algorithm for both of the tested seed sizes. The convergence rate of IBEA improves with the increase of the initial seed. This convergence rate is lower than the particle swarm algorithm, but a convergence before the generation 60 of 96 is achieved. RINGO's NSGA-II case shows a slow convergence rate for both initial seeds. Hence, there is a convergence advantage on the application of OMOPSO. Additionally, IBEA can be also applied to accelerate convergence while the seed size is adequately dimensioned.

For the discrete features of the Pareto front of the ZDT function 3, both IBEA and OMOPSO show a better convergence rate than NSGA-II (Fig. F.8). Both algorithms present very similar convergence rates. However, IBEA has a slightly more homogeneous convergence. Thus, both algorithms could be used almost indistinctly.

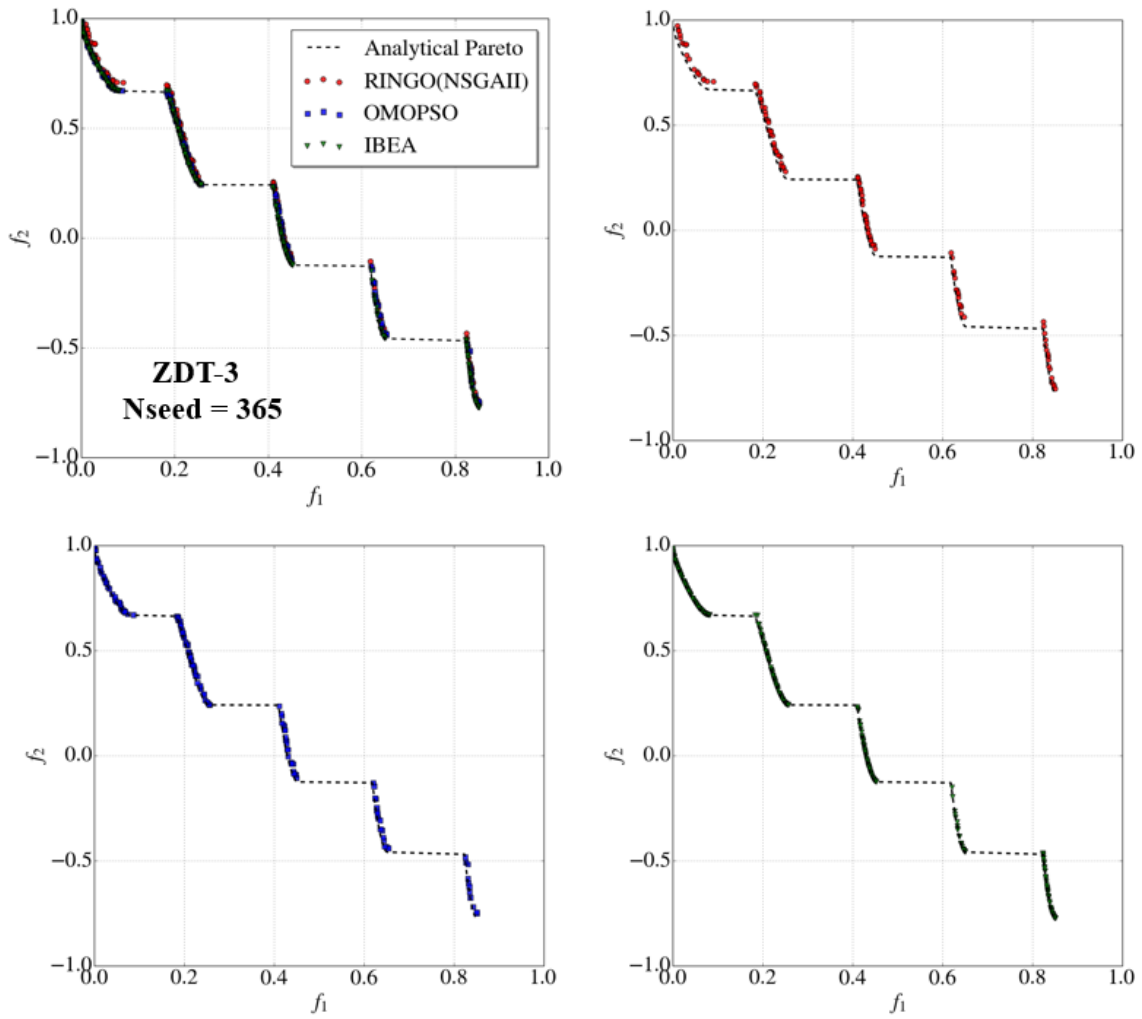


FIGURE F.6: ZDT function 3 Pareto-optimal set comparison for the three algorithms of the test (Tab. F.1) and a seed size of 365 individuals

for an unconstrained discretely clustered Pareto front optimisation problem. The convergence rate of RINGO's NSAGA-II makes the algorithm less suitable to achieve a fast algorithm convergence that will reduce the computational cost.

F.5 Convergence evaluations

While the analysis of the convergence rate of isolated cases can provide insightful information of the different algorithms performance, it is insufficient to determine the real consistency and repeatability of the Pareto solutions. Hence, in this section a statistical analysis of the Hypervolume index convergence is performed. The different algorithms of the test (Tab. F.1) are tested 40 times for the two ZDT functions and the two seed dimensions. Independent Latin Hypercube Samples are defined for

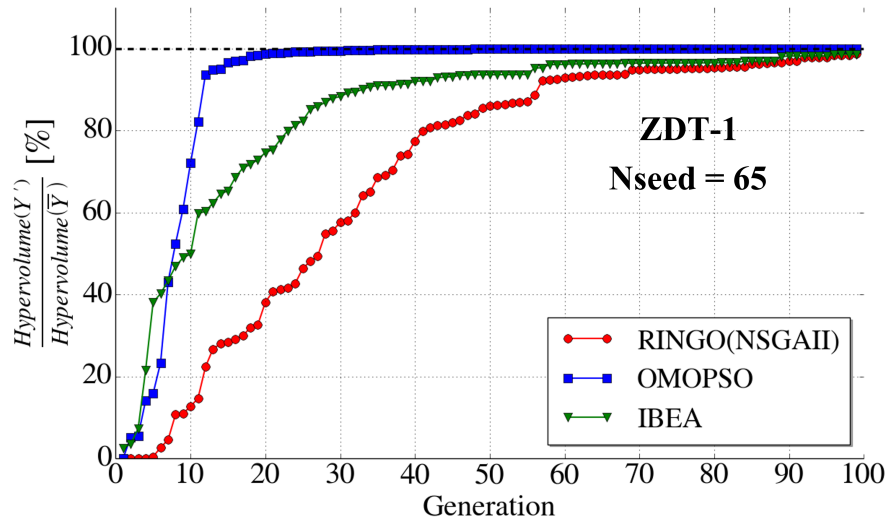
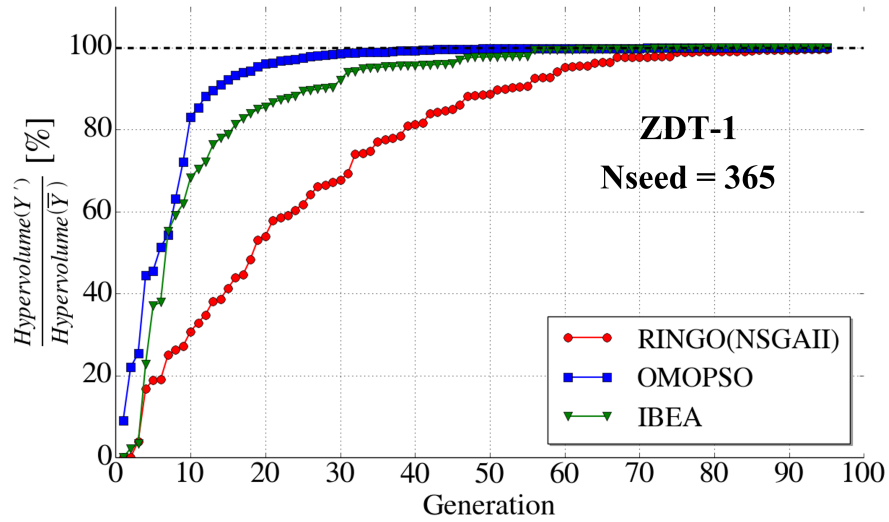
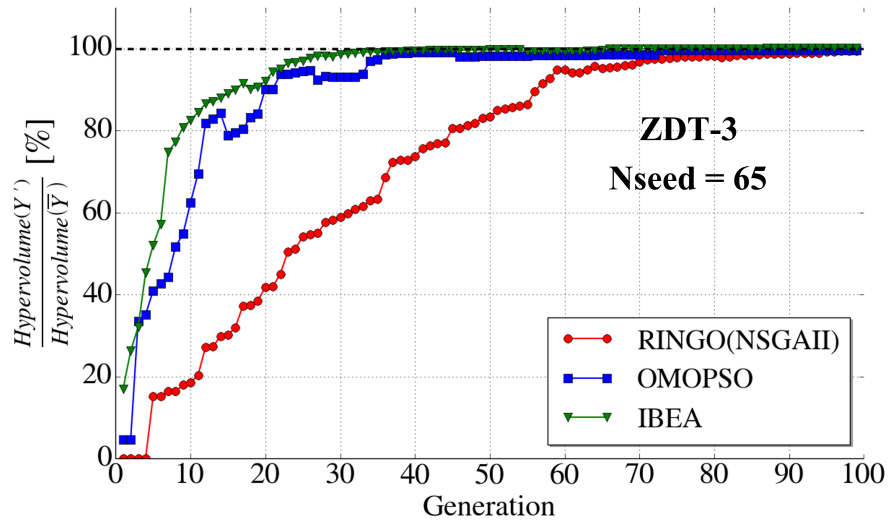
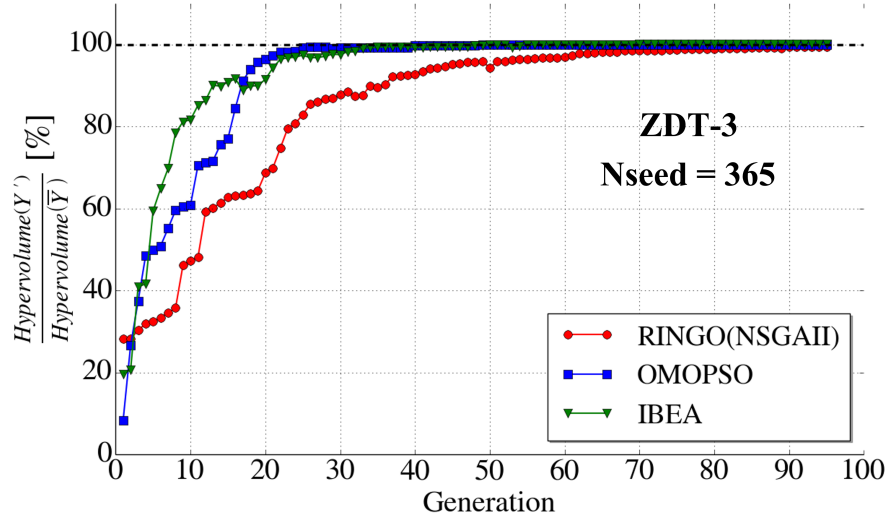


FIGURE F.7: Convergence of the selected algorithms for one test with the DTZ function 1³⁰. Evaluated for a seed size of 65 individuals (A), and a seed size of 365 individuals (B)

each of the tests. The convergence is addressed based on the Hypervolume index of the Local Pareto front at a given generation. The Hypervolume index of the Local Pareto-optimal set is normalised with the reference Hypervolume of the Global Pareto-optimal set.



(A)



(B)

FIGURE F.8: Convergence of the selected algorithms for one test with the DTZ function 3³⁰. Evaluated for a seed size of 65 individuals (A), and a seed size of 365 individuals (B)

Zitzler-Deb-Thiele's function 1

For ZDT function 1, OMOPSO can cover a higher percentage of the reference Hypervolume value on the earlier generations (Fig. F.9) than the evolutionary algorithms. On the first generation, the solutions that apply OMOPSO with both seed sizes (65 and 365) already predict a bigger Pareto front than the evolutionary algorithm counterparts. This coverage of the optimal design space of the few first generations

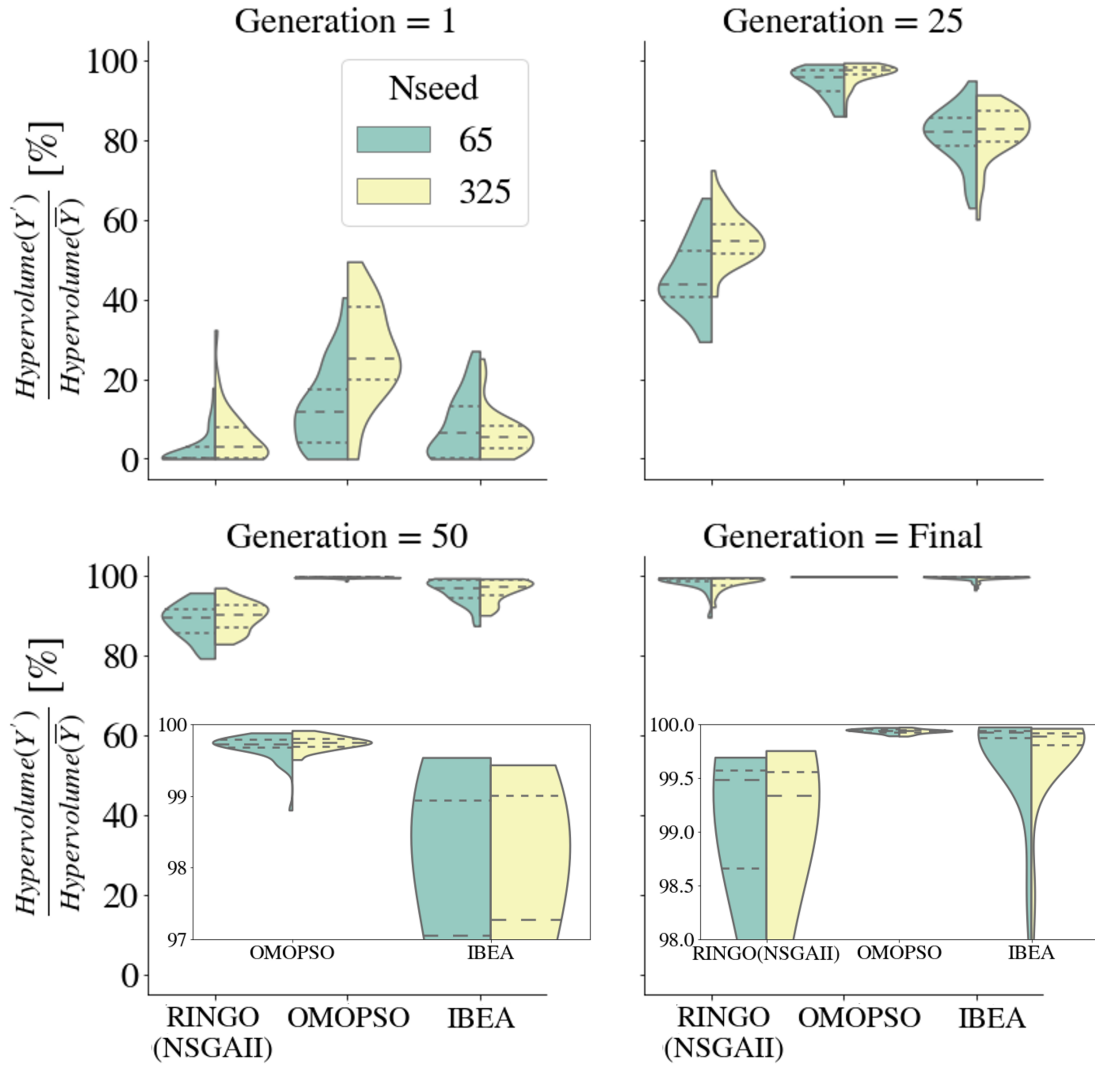


FIGURE F.9: Distribution of the Hypervolume index at different generations for the 40 tests with the DTZ function 1

improves with the size of the initial sample (Fig. F.9). At the same time, OMOPSO after the generation 25 converges to the reference value with a lower standard deviation than the other algorithms. Hence, a higher consistency on the Pareto front can be achieved with the particle swarm algorithm independent on the initial sample size. This trend continues until the final generation.

IBEA presents a more moderate convergence rate on the initial generations than OMOPSO (Fig. F.9) for the ZDT function 1 problem. However, an adequate coverage of the Hypervolume Global Pareto set is achieved on the initial generations of the problem. When compared with OMOPSO at the same generations, IBEA presents a much higher standard deviation of the Hypervolume value for the earlier and later generations. The increase on the standard deviation is independent on

the size of the seed. This may be a problem when this evolutionary algorithm is applied, since the replication and the consistency of the Global Pareto front is not assured. Hence, attention should be paid when this algorithm is applied for this type of unconstrained convex Pareto problems.

RINGO's NSGA-II is outperformed by any of the other two algorithms for the ZDT function 1. The overall convergence rate is slower than OMOPSO and IBEA for any of the given generations and independently on the seed dimensions. A higher variability of the Hypervolume index is observed. This could be a consequence of the settings of Gaussian Mutation process. Further studies should evaluate the sensitivity of the NSGA-II solutions to the mutation settings.

Zitzler-Deb-Thiele's function 3

For the ZDT function 3, the three algorithms of the test (Tab. F.1) present a similar convergence rate after the first generation (Fig. F.10). For the later generations, both IBEA and OMOPSO clearly outperforms the convergence rate of NSGA-II. Both OMOPSO and IBEA present a similar convergence rate for all the generations. A moderate improvement on the convergence can be achieved with the increase of the initial seed.

OMOPSO presents a lower standard deviation than the other algorithms after the first iterations. This indicates a higher consistency on the convergence after the initial stages of the algorithm. IBEA has a faster convergence rate than NSGA-II. However, the standard deviation of the final Hypervolume value is higher than any of the other two algorithms. This indicates a possible loss on the consistency of the Pareto shape for some of the experiments using IBEA. It was observed that not all the clusters of the discrete Pareto front were fully represented for the 40 IBEA solutions. This explains the higher variation on the final HV value. A similar situation was observed for NSGA-II. At the same time, NSGA-II failed to converge to the Global Pareto-optimal set reference on the maximum specified generations of the experiments. Thus, the particle swarm algorithm seems to be more accurate than the evolutionary algorithms to define the unconstrained discrete clustered Pareto of the ZDT function 3. Additionally, IBEA has a similar convergence rate than OMOPSO, but the higher standard deviation of the final solution could lead to non-optimal final solutions.

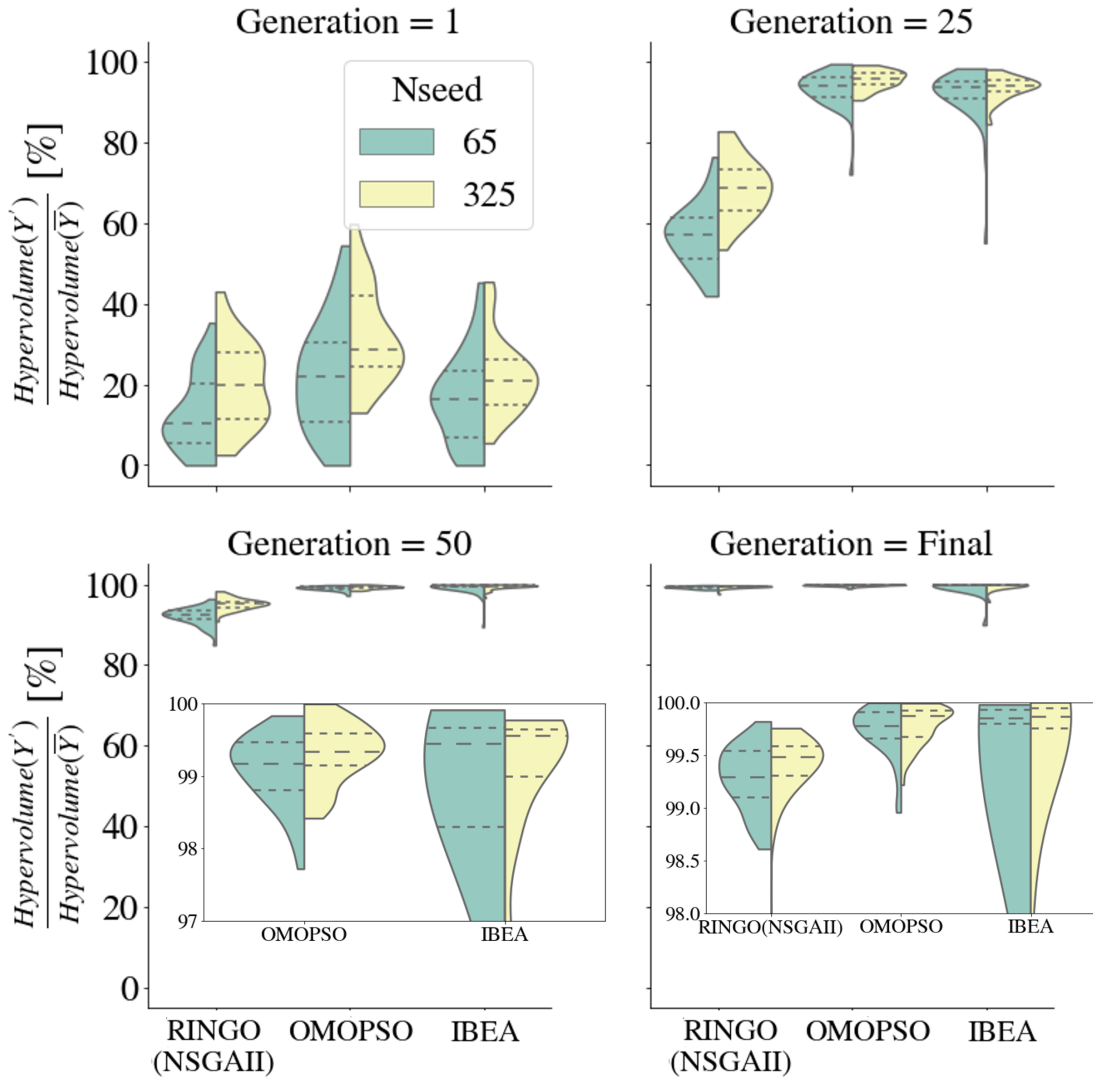


FIGURE F.10: Distribution of the Hypervolume index at different generations for the 40 tests with the DTZ function 3

F.6 Application of the algorithms to a real case

The performance of the algorithms was evaluated with continuous unconstrained design spaces for 13 degrees of freedom. However, the actual design space for these applications is constrained based on the geometry and the aerodynamic conditions (Chapter 3.5). To evaluate the performance of the algorithms under more realistic operation a reference test is proposed. The test consist on an aft-fuselage, intake and fan-cowl optimisation (13 degrees of freedom). Two objective functions are evaluated (Eq. F.3). The TS was selected to represent the aerodynamic performance of the full system and the IPR_{ing} was applied to evaluate the intake performance.

The aft mounted propulsor location was fixed at radially ($r_{hub}/r_{fus} = 0.59$) and axially ($l_{aft}/l_{ps} = 0.8$). The fan dimensions were fixed for the sample cruise conditions (Tab. 5.1) according to the sizing methodology defined in Appendix B. This resulted on a hub to tip ratio of ~ 0.3 . Constant isentropic efficiency ($\eta_{is} = 0.91$) is applied for this test. A sample fixed convergent exhaust geometry was applied for all the designs. The trailing edge position of the fan-cowl is fixed but the boat-tail angle (β_{nac}) was allowed to change. A Latin Hypercube Sampling¹⁶⁴ was applied to evenly populate the design space. The surface inflation constraints were applied for the intake and the fan cowl (Chapter 3.5). Aerodynamic filters were also applied to the designs. Maximum Mach number values at the fan face and the intake throat are imposed (Chapter 3.5). Also, the cases were discarded if the flow was not attached in any of the housing components.

The initial LHS consisted on 365 individuals, while the subsequent generations had 65 individuals each. The problem was constrained to a maximum of 50 generations to evaluate the Pareto convergence rate of each of the algorithms. To maintain consistency, the same initial seed was applied for the three algorithms of the test.

$$\begin{array}{l} \text{Objective} \\ \text{functions} \end{array} : \begin{cases} \text{maximise } (TS) \\ \text{maximise } (IPR_{ing}) \end{cases} \quad (\text{F.3})$$

After the maximum number of iterations, the evolution of the Hypervolume index was first addressed (Fig. F.11). OMOPSO almost converged to the final local Pareto at the early stages of the experiment (generation 10 upwards) with a negligible Hypervolume evolution after generation 30. NSGA-II presented a similar convergence rate to OMOPSO with an almost converged Hypervolume value after the generation 16 and a minor and slow convergence on the subsequent generations. Meanwhile, for the maximum number of generations of the test IBEA does not achieve the final Pareto convergence. However, while IBEA is not fully converged at the end of the test, the Hypervolume index obtained with this algorithm outweighs both OMOPSO and NSGA-II.

The difference on the final Hypervolume index between the configurations is more evident when the final Pareto fronts are compared (Fig. F.12). While OMOPSO achieved a quick convergence for the test conditions, the algorithm fails to cover the actual design space with a final Pareto that underestimates the maximum benefits of both objective functions. At the same time, while the convergence of the optimisation with IBEA was incomplete, a substantial improvement on the Pareto non-dominated solutions was obtained. This improvement was of around 0.1% of the

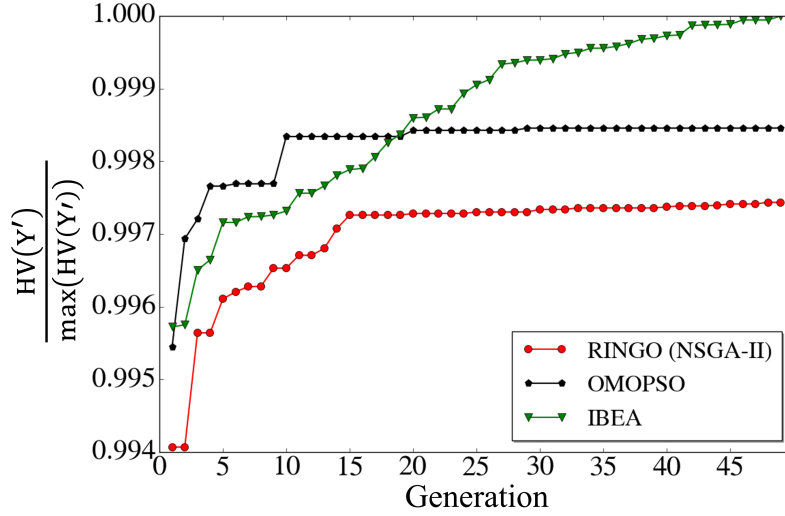


FIGURE F.11: Comparison of the Hypervolume Index convergence rate for the different multi-objective optimisation algorithms

total required thrust of the aircraft when compared with the best non-dominated design of OMOPSO. For the IPR_{ing} another substantial improvement was found with around a 1.5×10^{-3} increase with respect to OMOPSO. The Pareto front obtained with NSGA-II predicts lower maximum values for each of the objective functions than OMOPSO. However, the intermediate region of the Pareto front is better represented.

It has been shown that the selection of the algorithm can have a high impact on the final solution. The application of the test functions was used to support the algorithm selection, but when a real case with constraints was applied the actual performance of the algorithms varied. IBEA requires a higher convergence time than the other algorithms, but has demonstrated to operate better under constrained problems. OMOPSO operated better for unconstrained continuous design spaces. However, for the actual design space the algorithm gets stuck in a local Pareto front on the early generations. A similar behaviour was found for NSGA-II. Thus, the application of IBEA is recommended for the current optimisations. For this purpose, the maximum number of generations will have to be increased to achieve the desired convergence criteria.

F.7 Conclusions

The main conclusions of the this appendix can be summarized as the following:

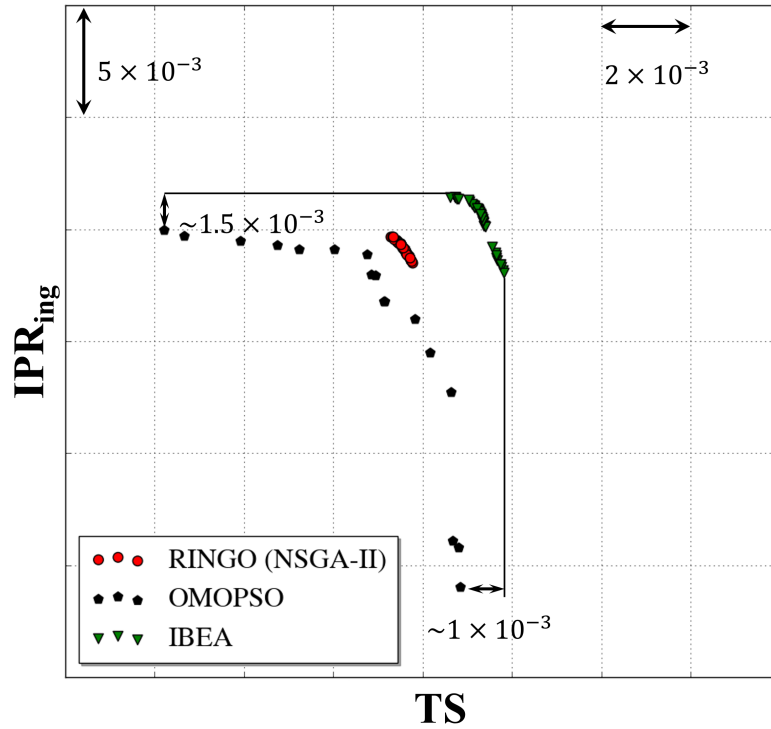


FIGURE F.12: Comparison of the Pareto fronts for the different multi-objective optimisation algorithms

- The implementation of two evolutionary algorithms (NSGA-II and IBEA), and one particle swarm algorithm (OMOPSO) has been verified for two benchmark multi-objective optimisation problems
- In addition to the verification of the algorithms, an statistical analysis of the convergence rate and the consistency of the Pareto solutions was performed
- The particle swarm optimisation algorithm OMOPSO performed better than the evolutionary algorithms with faster convergence rates and higher levels of consistency. This was observed for an unconstrained convex Pareto solution and a discrete cluster Pareto problem.
- The evolutionary algorithm IBEA presented a convergence rate similar to OMOPSO. This is more accused in the unconstrained discrete Pareto problem. However, a higher standard deviation at any given generation is observed for IBEA when compared with OMOPSO. This indicates a higher inconsistency on the final Pareto front shape
- The RINGO implementation of NSGA-II is consistently outperformed in terms of convergence by IBEA and OMOPSO. More consistent sets of solutions can

be achieved for the discrete clustered Pareto solutions when compared with IBEA. However, for the given number of generations, NSGA-II consistently failed on the coverage of the Global Pareto front

- A higher sensitivity to the initial seed was observed for the evolutionary algorithms with compared with the particle swarm. Hence, a possible reduction of the computational cost could be achieved by the application of this algorithm
- The algorithms were also tested against a representative case for the aerodynamic optimisation of an embedded propulsion system
- For a realistic case with design constraints, IBEA outperformed the rest of the algorithms with a higher Hypervolume index and substantial improvements on the predicted optima for each objective functions. Hence, this algorithm is recommended for the current applications
- OMOPSO and NSGA-II achieved fast convergence, but got trapped into local Pareto fronts. IBEA did not achieved convergence in the specified number of generations. Thus, for the application of this algorithm, the number of generations will have to be increased to achieved the required convergence criteria

