

CRANFIELD UNIVERSITY

DREWAN S. SANDERS

Boundary Layer Ingestion Performance Assessments with
Application to Business Jets

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING

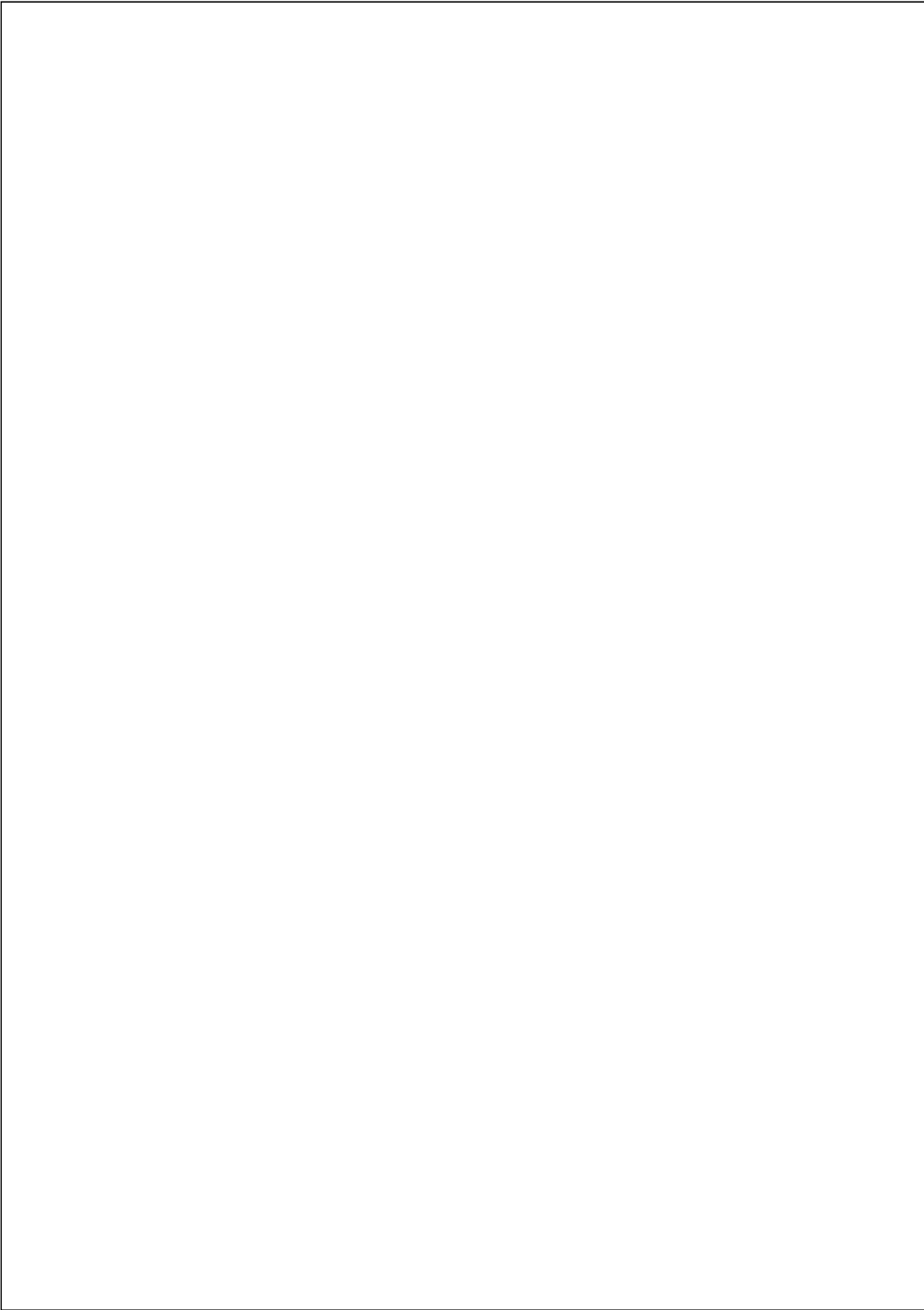
CENTRE FOR PROPULSION ENGINEERING

PhD

Academic Year: 2014-2018

Supervisor: Dr. P. Laskaridis

July 2018



CRANFIELD UNIVERSITY

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING

CENTRE FOR PROPULSION ENGINEERING

PhD

Academic Year: 2014-2018

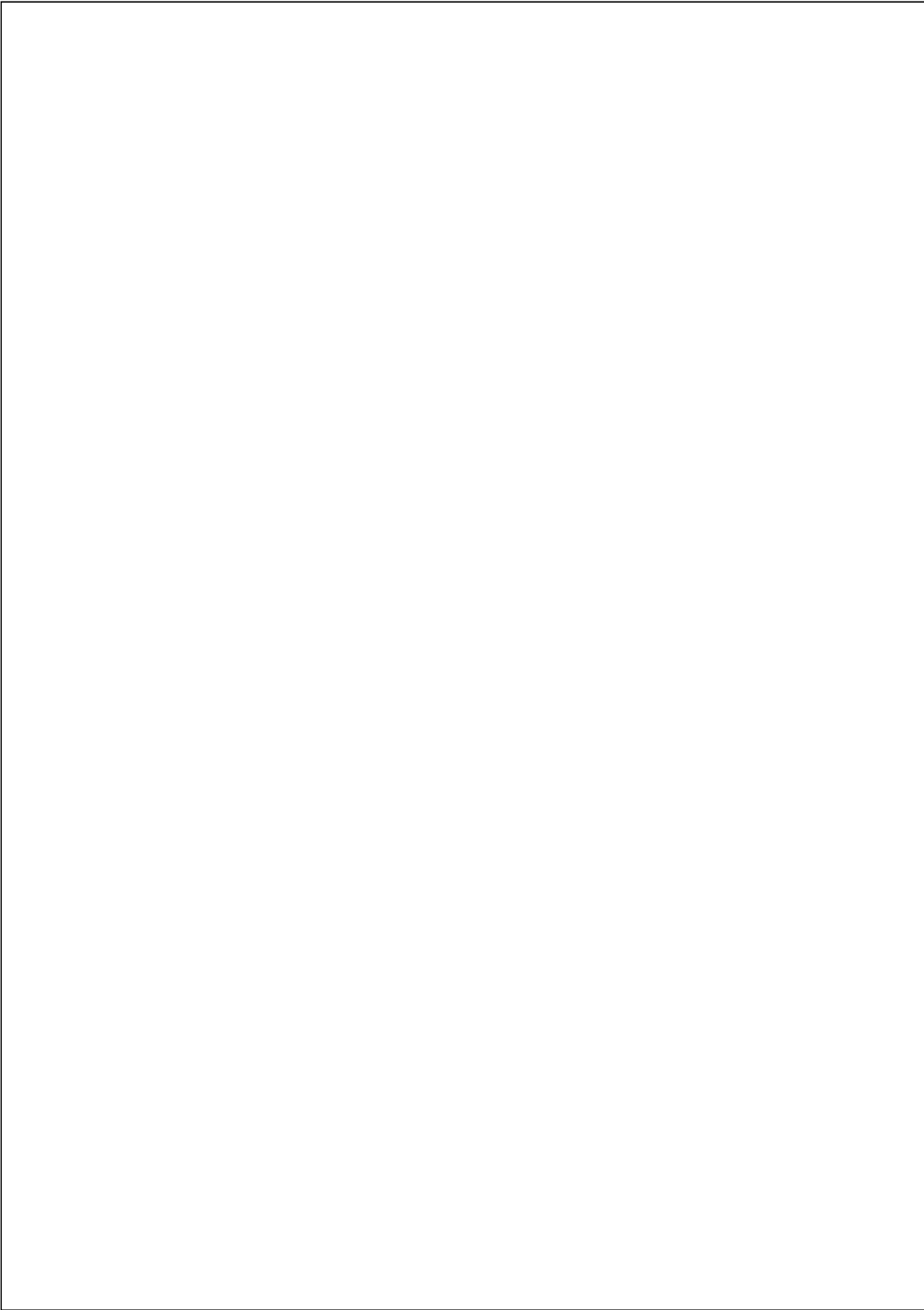
DREWAN S. SANDERS

Boundary Layer Ingestion Performance Assessments with
Application to Business Jets

Supervisor: Dr. P. Laskaridis

July 2018

©Cranfield University 2018. All rights reserved. No part of this publication may
be reproduced without the written permission of the copyright owner.



Executive Summary

Advancements in propulsion system performance are reliant on improvements in propulsive efficiency, through increases in turbofan bypass ratio. This requires larger nacelle diameters, which incur external aerodynamic penalties. Business jets cruise at high subsonic Mach numbers, and are therefore normally propelled by high specific thrust turbofans. The business jet may benefit from a BLI propulsion system, whereby the specific thrust may be reduced without incurring such heavy penalties in external drag rise. The aim of the research is to perform a design exploration study on BLI applied to a business jet, with emphasis on external aerodynamics. Methods are developed to thoroughly analyse aerodynamic coupling between propulsor and airframe. A multi-physics, control-volume based approach led to the development of near-field momentum-based, far-field momentum-based and energy-based net-vehicle-force formulations. The former two, allowed for a set of thrust-force accounting systems to be defined. Energy-based methods, allowed for flow-field decompositions into different physical mechanisms. These include flow phenomena internal and external to the jet plume. The practical implications associated with applying these methods to RANS CFD solutions, is examined. This hinges around viscous stress tensor field continuity in the flow domain. It was found that the $k - \omega$ SST turbulence model, along with a Green-Gauss Cell-Based gradient scheme, produced a continuous viscous stress tensor field. Having resolved this, the assessment methods were applied to solutions of non-propelled and propelled bodies. These methods were applied to control volumes having varying extents, which showed the far-field momentum-based method to be sensitive to spurious affects. The energy-based formulation, on the other hand, was observed to be relatively insensitive spurious affects. Good agreement (within 4%) was found between the forces predicted by all three methods over a non-propelled body. A very close agreement was observed between far-field momentum-based and energy-based results (within 1%) over the propelled body. However, much larger discrepancies were observed when compared against the near-field results. This was attributed to the increase in flow-field complexity, which now contained BL, shock and jet interaction regions. A design exploration study was performed by retrofitting a business jet with a fuselage concentric propulsor, powered by the baseline podded engines. A preliminary parametric study was first performed to gauge conditions favourable to BLI benefit. A ram drag approach to modelling BLI benefit was based on a flat plate analogy to obtain boundary layer profiles. Thrust-split, BLR, fan efficiency and intake pressure recoveries, were varied parametrically to asses

potential benefits. An optimum SFC benefit between 5-7.5% was achieved at thrust-splits between 30-35%, when ingesting 65-90% of the BL thickness. This guided the parametric CFD studies, where two tail-cone positions were examined. The first was placed at the top of the tail-cone, and the second positioned midway along the tail-cone. Benefits were only realised for the latter, where a 3-4% improvement in SFC was realised for a thrust-split around 20%, by ingesting 40% of the BL thickness. Energy breakdowns and decompositions were performed on all of the cases. One of the significant outcomes of this research was revealing that a significant proportion of the thrust force may be attributed to the isentropic expansion region within the jet plume's core.

Acknowledgements

I would firstly like to thank Dr. Panagiotis Laskaridis for his supervision. The support he so readily gave during academically and personally tough times is truly appreciated. In hindsight, the patience he exhibited during my excursions into risky academic areas, allowed the work to evolve into the thesis I wanted to produce. For that I am very grateful. I have been most fortunate to be apart of the Rolls-Royce UTC here at Cranfield University, and I would like to thank the various members of staff, research fellows and other PhD candidates, with whom I shared this experience.

I was exceptionally fortunate to interact directly with various engineering teams within Rolls-Royce. In particular I would like to thank Rory Stieger, who initiated this research project. During the PhD, Rory exhibited great enthusiasm for my work, was always encouraging and gave so willingly of his time. I also had the great privilege to work closely with Martin Goodhand, who always made me feel so welcome during my visits to Derby. I am truly grateful for his hospitality and invaluable guidance. I would also like to thank Robert Hill, for his help on engine performance during my visit to Dahlewitz, and for making the trip so enjoyable. There are numerous other Rolls-Royce employees who have influenced this work, in one way or the other, and I express my sincerest gratitude to them all.

At Cranfield, I was very fortunate to work with so many talented individuals. I would particularly like to thank Salvatore Ippedico for his help with formulating the energy-based assessments presented in this thesis. I would also like to thank David John for his advice and guidance on all things CFD, and to John Otter for being my Fluent and ICEM user manual. Thank you for all your help, but mostly, thank you for your friendship. I would also like to mention Davood Sabaei and Joana Dias, who have been very good friends to me, outside of the office.

I am so fortunate to have met a life long friend in Lucas Pawsey, who helped see me through the toughest obstacles I have faced so far in both my career and personal life. From sharing our first office together, and many a beer after work, his friendship has been one that I will cherish for years to come. Apart from our friendship, Lucas walked me through my first CFD simulations, and helped me to figure out some of the most challenging aspects of this thesis. Without him, this thesis would not have been possible.

Katie McCallum and I met in the last six months of my PhD. From day one she has been a constant source of support and encouragement, and has taken such amazing care

of me, at a time when I was unable to take care of myself. Furthermore, I can't thank her enough for proof reading this document, and for the many weekends she spent beside me working. I am so truly grateful for everything she has done to help me reach this finish line.

My sister Megan is my best friend, and has been a voice of reason helping me through all the difficult times I have faced during my PhD. She is a constant source of inspiration, and I aspire to be more like her everyday. Although we live far apart, she has always been there when I have needed her.

This thesis is dedicated to my incredible parents. I am forever grateful to them for their unconditional love and unwavering support. They have so selflessly sacrificed so much to ensure my sister and I achieve our dreams. I am so extremely fortunate to have such loving, caring, and, above all, patient parents.

Drewan S. Sanders
July, 2018

Contents

Executive Summary	i
Acknowledgements	iii
List of Figures	xi
List of Tables	xix
Nomenclature	xxi
1 Introduction	1
1.1 Research Rationale	1
1.2 Aim and Objectives	3
1.3 Knowledge Contribution	3
1.4 Thesis Outline	5
2 Literature Review	7
2.1 Background	7
2.2 Principles of BLI Benefit	7
2.2.1 Propulsive Efficiency Improvement from Reduced Ram Drag	8
2.2.2 Minimising Energy Lost to the Atmosphere	10
2.3 Why the Business Jet?	11
2.4 Energy-Based Performance Assessments	12
2.4.1 Background and Rationale	13
2.4.2 Implementation	16
3 Momentum-Based Approach to Evaluating Performance	19
3.1 Reference Frames and Properties	20
3.2 Continuity and Momentum Equations	21
3.3 Application to Streamtubes	22
3.4 Application to Propulsor Control Volumes	23
3.5 Application to Propelled Bodies	24
3.6 Thrust-Force Accounting Systems	25
3.6.1 Control Volumes	26

3.6.2	Propulsor Net Thrust	28
3.6.3	Intrinsic Net Thrust	29
3.6.4	Standard Net Thrust	29
3.6.5	Overall Net Thrust	30
3.6.6	Pre-Diffusion Net Thrust	31
3.6.7	Pre-Diffusion Overall Thrust	32
3.6.8	Reference Net Vehicle Force	32
3.6.9	A Note on Thrust-Drag Accounting	33
3.7	Propulsor Power and BLI Performance Evaluation	33
3.7.1	Propulsor Shaft Power	33
3.7.2	Propulsion Efficacy	35
3.8	Summary	35
3.8.1	Concepts and Formulations	36
3.8.2	Contributions	37
3.8.3	Future Work	37
4	Energy-Based Approach to Evaluating Performance	39
4.1	Generalised Energy Equations Applicable to Arbitrary Control Volumes	41
4.1.1	Kinetic Energy Equation	41
4.1.2	Total and Internal Energy Equations	45
4.2	Application to Streamtubes	47
4.3	Application to Non-Propelled Bodies	51
4.4	Application to Propulsor Control Volumes	54
4.5	Application to Propelled Bodies	55
4.5.1	Net Vehicle Force Derivation	56
4.5.2	Net-Vehicle-Force Imbalance	57
4.5.3	Net Fuselage Force	58
4.5.4	Flow Phenomena Decompositions	58
4.6	BLI Performance Evaluation	59
4.7	Summary	59
4.7.1	Concepts and Formulations	60
4.7.2	Contributions	61
4.7.3	Future Work	62
5	Practical Implementation in RANS CFD	65
5.1	Spatial Regions of Flow Mechanisms	67
5.1.1	Boundary Layer and Viscous Regions	68
5.1.2	Shock-Wave Regions	73
5.1.3	Jet Region Partitioning	75
5.1.4	Combined Partitioning Procedure	76
5.1.5	A Note on Spurious Drag Removal	82
5.2	Viscous Stress Tensor and Near-Field Surface Forces	82
5.3	Drag Prediction and Spurious Contributions	85
5.3.1	Near-Field Momentum-Based Force Coefficient	93

5.3.2	Far-Field Momentum-Based Force Coefficient	94
5.3.3	Energy-Based Force Coefficient	96
5.3.4	Methods Comparison and Uncertainties	104
5.4	NVF Calculations	105
5.4.1	Near-Field Boundary Conditions and Propulsor Forces	106
5.4.2	Thrust-Force Accounting Approach	107
5.4.3	Far-Field Momentum-Based NVF Coefficient	112
5.4.4	Energy-Based NVF Coefficient	112
5.4.5	Methods Comparison and Uncertainties	121
5.5	Propulsor Power	122
5.6	Summary	124
5.6.1	Concepts and Formulations	126
5.6.2	Contributions	127
5.6.3	Future Work	128
6	Fuselage Concentric BLI Propulsion Performance	129
6.1	Outcomes of Preliminary Parametric Studies	131
6.2	CFD Assessment Tools and Methodology	133
6.2.1	Geometry Definitions	133
6.2.2	Pre-Processing Tool	138
6.2.3	Meshing Tool	142
6.2.4	In-Solution and Post-Processing Tools	142
6.2.5	System Level Performance at Cruise	143
6.2.6	Design-Space Exploration Parametric Cases	144
6.3	Propulsor Performance	145
6.3.1	FPR and Isentropic Efficiency	145
6.3.2	Pre-Diffusion and Fan Inlet Mach Number	146
6.3.3	Propulsor Pressure Losses	146
6.3.4	Distortion Descriptors and Intensity	150
6.3.5	Shaft Power	152
6.3.6	Summary	152
6.4	Near-Field Momentum-Based Assessment	154
6.4.1	Surface forces	154
6.4.2	Thrust-Force Accounting Assessments	168
6.4.3	System Level Performance	172
6.5	Far-Field Momentum-Based Assessments	177
6.5.1	Propulsor Power	179
6.5.2	Decoupled BLI Propulsor Performance	179
6.5.3	SFC Benefits	180
6.6	Energy-Based Assessments	180
6.6.1	Decoupled BLI Propulsor Performance	183
6.6.2	SFC Benefits	183
6.6.3	Trefz Plane Postion Breakdowns	186

6.6.4	Flow Field Decompositions	189
6.7	Comparisons	198
6.8	Summary	198
6.8.1	Contributions	201
6.8.2	Future Work	202
7	Summary, Conclusions and Recommendations	205
7.1	Summary and Conclusions	205
7.1.1	Research Rationale	205
7.1.2	Assessment Methods	205
7.1.3	Practical Implementation in RANS CFD	206
7.1.4	Design Exploration Study	207
7.2	Knowledge Contributions	207
7.3	Recommendations for Future Work	209
7.3.1	Momentum-Based Assessment Methods	209
7.3.2	Energy-Based Assessment Methods	210
7.3.3	Practical Implementation in RANS CFD	211
7.3.4	Fuselage Concentric BLI Propulsion Performance	211
A	Viscous Terms	213
A.1	Tensor Notation, Operations and Identities	213
A.1.1	Gradient of a Vector	213
A.1.2	Divergence of a Vector	214
A.1.3	Tensor Products and Identities	214
A.2	Rate-of-Strain	215
A.3	Deviatoric Stress	216
A.4	Boussinesq Eddy Viscosity Assumption	217
A.5	Viscous Forces, Work and Dissipation	218
A.5.1	Viscous Stress Forces	218
A.5.2	Viscous Boundary Work	219
A.5.3	Viscous Dissipation	219
B	Drag Estimation	221
B.1	Lift Independent Drag	222
B.1.1	Fuselage	223
B.1.2	Wing and Tail Surfaces	223
B.1.3	Nacelle	223
B.1.4	Secondary Items	224
B.2	Lift Dependent Drag	224
B.3	Wave Drag Estimation	224
B.4	Drag Polar Matching	226

C	Isolated Fuselage Aerodynamic Matching	231
C.1	Model Setup	231
C.2	Grid Independence Study	232
C.3	Drag Matching Procedure and Results	235
D	Baseline Engine Model	241
E	Aircraft Performance	245
E.1	The Vehicle Performance Deck	246
E.2	Mission Profile and Performance Calculations	246
E.3	Description of Main and Reserve Mission Segments	247
E.3.1	A-B: Taxi Out	247
E.3.2	B-C: Take-Off	250
E.3.3	C-D: Climb to First Cruise Altitude	250
E.3.4	D-I: Stepped Cruise	253
E.3.5	I-J: Main Mission Descent	254
E.3.6	J-K: Standard Instrument Approach	254
E.3.7	K-L: Climb to Hold Altitude Following Missed Approach	255
E.3.8	L-L: Hold for Clearance	255
E.3.9	L-M: Climb from Hold to Diversion Cruise Altitude	255
E.3.10	M-N: Diversion Cruise	256
E.3.11	N-O: Diversion Descent to Land	256
E.4	Verification	256
E.5	Baseline Aircraft Payload Range Diagram	256
F	Preliminary Parametric Assessments	259
F.1	Baseline Aircraft	260
F.2	Methodology	260
F.3	Decoupled BLI Propulsor Maps	264
F.4	Decoupled Turbofan Performance Map	266
F.5	System-Level Performance Results	266
F.6	Mission Level Performance	267
	References	271

List of Figures

2.1	Propulsive Efficiency Improvement from Reduced Ram Drag	8
2.2	BLI Benefit from Minimising Wake Kinetic Energy	11
2.3	Aircraft type distribution across the civil aviation market.	12
3.1	Streamtube encapsulating an arbitrary body	22
3.2	Control volume encapsulating arbitrary aropulsor	23
3.3	NVF control volume	24
3.4	Control volume divisions for thrust-force accounting definitions	26
3.5	Propulsor Net Thrust Control Volumes	28
3.6	Intrinsic Net Thrust Control Volumes	29
3.7	Standard Net Thrust Control Volumes	29
3.8	Overall Net Thrust Control Volumes	30
3.9	Pre-Diffusion Net Thrust Control Volumes	31
3.10	Pre-Diffusion Overall Thrust Control Volumes	32
4.1	Isolated force propelling an arbitrary body	45
4.2	Kinetic energy in absolute and relative reference frames	45
4.3	Control volume encapsulating a non-propelled body	51
4.4	Control volume decomposition for non-propelled body	54
4.5	Control volume decomposition for propelled body	59
5.1	An example of the ξ viscous region for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$	73
5.2	Contours of ζ showing regions where $ p\nabla \cdot \mathbf{V} $ becomes very small, for an example fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$	74
5.3	An example of the ζ shock region for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$	75
5.4	An example of the Υ jet region for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$	76

5.5	ζ contours for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	77
5.6	ζ jet contours for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	78
5.7	Flow structures of an under-expanded jet	78
5.8	ζ contours within the jet's shock regions inside its core. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	79
5.9	ξ contours within what is assumed to be predominantly the jet transition region. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	80
5.10	Υ contours within what is assumed to be the non-dissipating jet core. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	80
5.11	ζ contours within the external shock region. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	81
5.12	ξ contours within the external viscous regions. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	81
5.13	y^+ over isolated fuselage for different gradient schemes and first cell heights on the Non-Propelled Fuselage at FL450 M0.85	86
5.14	Friction coefficient corresponding to Figure 5.13 on the Non-Propelled Fuselage at FL450 M0.85 as reported by ANSYS Fluent	87
5.15	Friction coefficient obtained from velocity derivatives using Boussinesq's approximation on the Non-Propelled Fuselage at FL450 M0.85	88
5.16	Rate-of-strain tensor second invariant corresponding to Figure 5.13 on the Non-Propelled Fuselage at FL450 M0.85	89
5.17	Friction coefficient difference between Figures 5.14 and 5.15 on the Non-Propelled Fuselage at FL450 M0.85	90
5.18	Pressure coefficient to Figure 5.13 on the Non-Propelled Fuselage at FL450 M0.85	91
5.19	Turbulent viscosity error at sharp trailing edge on the Non-Propelled Fuselage at FL450 M0.85	92
5.20	Control volumes with varying size and trefftz plane locations.	94
5.21	Momentum-based far-field extraction of the non-propelled fuselage force coefficient at FL450 M 0.85, for control volumes varying in size.	96
5.22	Momentum-based far-field extraction of the non-propelled fuselage force coefficient at FL450 M 0.85, for control volumes varying in size (extended to include more trefftz plane locations).	97
5.23	Comparison between the near and far-field momentum-based force coefficient extractions of the non-propelled fuselage force coefficient at FL450 M 0.85	97

5.24	Variation in kinetic energy flux $C_{\dot{E}_k}$ corresponding to different treftz plane locations, for the non-propelled fuselage at FL450 M 0.85	99
5.25	Variation in pressure boundary work $C_{\dot{E}_p}$ corresponding to different treftz plane locations	100
5.26	Variation in viscous boundary work $C_{\dot{E}_\tau}$ corresponding to different treftz plane locations	101
5.27	Variation in volumetric compression work $C_{\dot{E}_{pv}}$ corresponding to different treftz plane locations	102
5.28	Variation in volumetric viscous dissipation $C_{\dot{E}_{\tau v}}$ corresponding to different treftz plane locations	103
5.29	Net forces coefficient $C_{\dot{E}}$ extracted using the volumetric dissipation method	103
5.30	Difference in net forces coefficient $C_{\dot{E}}$ extracted using the volumetric dissipation method, using near-field as reference	104
5.31	Breakdown of different $C_{\dot{E}}$ contributions corresponding to different treftz plane locations, for $\frac{\Delta s}{l_{fuse}} = 1/3$	105
5.32	Comparison of the difference in net forces coefficient $C_{\dot{E}}$ extracted using near-field, far-field momentum and energy-based methods	106
5.33	Control volume divisions for a BLI propulsor represented as a set of outlet (S_i) and inlet (S_j) boundary conditions.	107
5.34	Example of streamlines through the real and assumed highlight surfaces S_f , shown within contours of pressure coefficient. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	109
5.35	Example of streamlines through the real and assumed nozzle exit surfaces S_l , shown within contours of pressure coefficient. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	111
5.36	Momentum-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	113
5.37	Control volume uncertainties of the momentum-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	113
5.38	Kinetic energy flux force coefficient $C_{\dot{E}_k}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	114
5.39	Pressure boundary work force coefficient $C_{\dot{E}_p}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	115

5.40	Viscous boundary work force coefficient $C_{\dot{E}_\tau}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	116
5.41	Propulsor $C_{\dot{E}_{prop}}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	117
5.42	Volumetric compression work $C_{\dot{E}_{pv}}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	118
5.43	Volumetric dissipation $C_{\dot{E}_{\tau v}}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	118
5.44	Energy-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	119
5.45	Control volume uncertainties of the energy-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	120
5.46	Energy-based far-field force coefficient breakdown, for different trefftz plane locations and $\Delta s/l_{fuse} = 6/3$. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	121
5.47	Uncertainties in force coefficient methods, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	123
5.48	Comparison of near-field versus far-field predictions of propulsor power, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.	125
6.1	Schematic of a rear-fuselage BLI propelled business jet, with thrust-split.	132
6.2	Preliminary parametric summary of optimum SFC benefits as a function of thrust-split, BLR, fan isentropic efficiency and intake pressure recovery.	133
6.3	Overall fuselage concentric BLI propulsor geometry definitions.	134
6.4	Fuselage concentric BLI propulsor intake and fore-body geometry definitions.	137

6.5	Fuselage concentric BLI propulsor exhaust nozzle and after-body geometry definitions.	138
6.6	Fuselage concentric BLI propulsor station numbers.	139
6.7	Flowchart of pre-processing tool station calculations.	140
6.8	Domain blocking and boundary conditions.	142
6.9	Propulsor mesh example for a fuselage concentric propulsor placed mid-way along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	143
6.10	Tail-cone propulsor positions examined	145
6.11	Fan pressure ratio and isentropic efficiency achieved to match polytropic efficiency of 89% for a geometry estimated to produce a specified input FPR	147
6.12	Pressure coefficient surrounding the nacelle for different BLRs and therefore nozzle exit positions	148
6.13	The mass-weighted-average fan inlet (surface S_i) Mach number achieved by the geometry (design target set to $\bar{M}_{S_i} = 0.65$), with corresponding MFR	149
6.14	Pressure recovery within the pre-diffusion, intake and exhaust internal stream-tubes	150
6.15	Total pressure radial distortion intensity for each BLR, for non-dimensional hub-to-tip fan face radial locations	151
6.16	Fan shaft power in explicit and mass specific forms, to illustrate the purpose of a constant polytropic efficiency.	153
6.17	Pressure coefficients along the fuselage upstream of the fan, for Position A.156	
6.18	Pressure coefficients along the fuselage upstream of the fan, for Position B.157	
6.19	Pressure coefficients along the fuselage downstream of the fan, for Position A.	158
6.20	Pressure coefficients along the fuselage downstream of the fan, for Position B.	159
6.21	Pressure coefficients over nacelle surfaces	161
6.22	Axial pressure force contributions for the nacelle, fuselage and combined airframe surfaces (with comparison against the baseline).	163
6.23	Position A friction coefficients upstream of fan inlet along fuselage surface. Comparison against non-propelled baseline fuselage.	164
6.24	Position B friction coefficients upstream of fan inlet along fuselage surface. Comparison against non-propelled baseline fuselage.	165
6.25	Position A friction coefficients downstream of fan exit along fuselage surface166	
6.26	Position B friction coefficients downstream of fan exit along fuselage surface167	
6.27	Design exploration thrust-force accounting breakdown at Position A . . .	169
6.28	Design exploration thrust-force accounting breakdown at Position B . . .	170
6.29	Reference Net Vehicle Thrust and Thrust Split for both Positions A and B173	
6.30	Fan shaft power and Thrust Split for both Positions A and B	174
6.31	TSPC and Thrust Split for both Positions A and B	174

6.32	Propulsive efficiency and Thrust Split for both Positions A and B	175
6.33	Decoupled turbofan engine performance map corresponding to BLI propulsive efficiency and thrust-split combinations	177
6.34	Near-field momentum-based SFC benefits for propulsors at Position B. . .	178
6.35	Far-field versus near-field propulsor power, for propulsors located at Position B along the tail-cone.	179
6.36	Decoupled BLI propelled fuselage performance, using far-field momentum-based methods (compared against near-field), for propulsors at Position B.	181
6.37	Far-field momentum-based SFC benefits for propulsors at Position B, with reference to near-field momentum-based values.	182
6.38	Decoupled BLI propelled fuselage performance, using far-field energy-based methods (compared against near-field), for propulsors at Position B.	184
6.39	Decoupled BLI propelled fuselage performance, using far-field energy-based methods (compared against far-field momentum-based values), for propulsors at Position B.	185
6.40	Far-field energy-based SFC benefits for propulsors at Position B, with reference to near and far-field momentum-based values.	187
6.41	Breakdown of kinetic energy flux $\dot{E}_k$ and pressure boundary work $\dot{E}_p$ at different trefftz locations, for propulsors at Position B.	188
6.42	Breakdown of volumetric dissipation $\dot{E}_{\tau\nu}$ and volumetric compression work $\dot{E}_{p\nu}$ at different trefftz locations, for propulsors at Position B.	190
6.43	Volumetric viscous dissipation within the jet flow regions, for propulsors at Position B	192
6.44	Volumetric viscous dissipation in flow regions outside of the jet flow, for propulsors at Position B	194
6.45	Volumetric compression work within the jet flow regions, for propulsors at Position B	196
6.46	Volumetric compression work in flow regions outside of the jet flow, for propulsors at Position B	197
6.47	Uncertainty in SFC benefits for propulsors at Position B, based on near-field momentum-based, far-field momentum-based and far-field energy-based methods.	199
B.1	Baseline Aerodynamics Matching at Low Speeds	228
B.2	Baseline Aerodynamics Matching at High Speeds	229
B.3	Baseline Aerodynamics Matching at All Speeds	229
C.1	Isolated fuselage geometry	232
C.2	Isolated fuselage domain	232
C.3	Isolated fuselage structured mesh example	233
C.4	Isolated fuselage GCI study: trailing edge wake properties with Richardson extrapolation	236

C.5	Isolated fuselage GCI study: GCI uncertainties of trailing edge wake properties	237
C.6	Comparison between empirical analytical formula and CFD drag estimations for the final fuselage geometry	238
C.7	Final fuselage geometry superimposed over a picture of a Gulfstream G650 for reference as to how representative the geometry is.	239
D.1	TURBOMATCH flow chart of baseline engine	242
D.2	Baseline engine model off-design performance versus data supplied. Showing SFC as a function of intrinsic net thrust at FL450 for M 0.6 - 0.925.	243
D.3	Baseline Engine Match for All Flight Conditions	244
E.1	NBAA Jet Aircraft Range Format	248
E.2	NBAA IFR Reserves Profile Calculation Procedure	249
E.3	Aircraft Mission Performance Code Verification	257
E.4	Baseline aircraft payload range diagram	258
F.1	Flat plate analogy applied to a cylinder, to approximate the BL profile over the fuselage.	262
F.2	Summary of methodology used to calculate SFC benefits for the preliminary parametric study.	263
F.3	Diagram of decoupled propulsor performance trends for an arbitrary thrust-split and fan efficiency, but no intake pressure recovery losses.	264
F.4	Change in trends from Figure F.3 after introducing penalties in fan efficiency (to account for distortion) and losses in intake pressure recovery.	265
F.5	Decoupled turbofan engine performance map corresponding to BLI propulsive efficiency and thrust-split combinations	266
F.6	Preliminary parametric study SFC benefits, with corresponding BLR values.	267
F.7	Preliminary parametric study SFC benefits, with corresponding BLR values.	268
F.8	Preliminary parametric study SFC benefits, with corresponding BLR values.	268
F.9	Block Fuel Burn (and fuel savings) vs. Range diagrams, for a BLI propelled business jet flying at different cruise Mach numbers.	269
F.10	Global map of increased range due to BLI.	270

List of Tables

3.1	Summary of Thrust-Force Accounting Definitions	36
5.1	Summary of integrated viscous, pressure and total stream-wise forces on the non-propelled fuselage at FL450 M0.85	93
5.2	Propulsor boundary conditions integration error	108
5.3	Thrust-force accounting system table example. Demonstrated on a fuse- lage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$	111
6.1	Summary of propulsor station calculation inputs	139
B.1	Error's in drag contributions of the matched geometry at the given drag break-down point of FL450 M0.85	230
C.1	Isolated fuselage GCI study: number of cells N	234
C.2	Isolated fuselage GCI study: domain 2D volume $\sum_{i=i}^N \nu_{i,2D}$	234
C.3	Isolated fuselage GCI study: mesh density $\left(\sum_{i=i}^N \nu_{i,2D}\right)^{1/2}$ with refinement ratio $(\cdot)$	234

Nomenclature

Please note that symbols represented by Greek letters are listed first, and the remaining symbols listed in alphabetical order.

Symbols

β	Geometric angle	(deg)
δ	Boundary layer thickness	(m)
δ^*	Boundary displacement thickness	(m)
η	Efficiency	
κ_A	Wave drag technology factor	
Λ	Wing/stabilizer sweepback angle	(deg)
λ	$(\nabla \cdot \mathbf{V})$	
λ	Taper ratio	
$\Lambda_{\#c}$	Wing/stabilizer sweepback angle at # fraction of the chord	(deg)
μ	Dynamic viscosity	(Pa · s)
μ_t	Turbulent viscosity	(Pa · s)
ν	Volume	(m ³)
ω	Vorticity	
Ω	Flux surface force vector $\iint_{S_G} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS$	
Φ	Unbounded streamtube surface force vector $-\iint_S [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS$	
Π	Geometry ratio	
Ψ	Physical surface force vector $\iint_S [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS$	

ρ	Density	$\left(\frac{kg}{m^3}\right)$
$\bar{\bar{\tau}}$	Viscous stress tensor	
$\vec{\tau}$	Viscous stress vector on a surface	
Υ	Jet region selector	
ξ	Modified BL and viscous region selector	
ζ	Shock region selector	
A	Area	(m^2)
AR	Aspect ratio	
BLR	Boundary Layer Ratio $\left(\frac{\delta_{ing}}{\delta_{BL}}\right)_{npf}$	(%)
BPR	Bypass Ratio	
c	Chord length	(m)
$C_{\dot{E}}$	Force coefficient from energy contributions Equation (5.10)	
C_F	Force coefficient from momentum contributions Equation (5.9)	
C_f	Net friction coefficient	
c_f	Friction coefficient	
C_p	Pressure coefficient	
C_{D_0}	Lift independent drag coefficient	
C_{D_I}	Lift dependent drag coefficient	
C_{D_T}	Trim drag coefficient	
C_{D_W}	Compressibility/wave drag coefficient	
C_{NVF}	Net Vehicle Force coefficient	
CR	Lip Contraction Ratio	
$\bar{\bar{d}}$	Deviatoric stress tensor	
D	Drag	(N)
DS	Diffusion split ratio between pre-diffusion and intake	
$\dot{E}_{\tau\nu}$	Volumetric energy dissipation rate to fluid	(W)

$\dot{E}_\tau$	Rate of viscous stress boundary work on fluid CV	(W)
e	Internal energy	$\left(\frac{J}{kg}\right)$
e	Oswald efficiency factor	
$\dot{E}$	Rate of energy transfer to fluid	(W)
$\bar{\bar{e}}$	Rate-of-strain tensor	
$\dot{E}_{\text{prop}}$	Energy transferred to fluid via propulsor flux surfaces	(W)
$\dot{E}_B$	Rate of energy transfer to fluid by surface forces	(W)
EFR	Ellipse Fineness Ratio	
$\dot{E}_k$	Rate of kinetic energy transfer to fluid	(W)
$\dot{E}_{p\nu}$	Volumetric compression/expansion work rate	(W)
$\dot{E}_p$	Rate of pressure boundary work on fluid CV	(W)
$\dot{E}_W$	Rate of energy transfer to fluid by due to shocks	(W)
F	Profile drag form factor	
$\mathbf{F}$	Force vector	(N)
$\mathbf{f}$	Fluid mass-specific body force vector	
F_N	Net thrust force	(N)
F_{BL}	BL and viscous region selector by Paparone and Tognaccini	
FPR	Fan Pressure Ratio	
g	Gravitational acceleration	$\left(\frac{m}{s^2}\right)$
H	Stagnation enthalpy	$\left(\frac{J}{kg}\right)$
h	Conical frustum hypotenuse length	(m)
h	Static enthalpy	$\left(\frac{J}{kg}\right)$
H_p	Geopotential altitude	(m) or (ft)
$\bar{\bar{I}}$	Second order identity tensor	
II	Second tensor invariant	
III	Third tensor invariant	

K	Induced drag coefficient in $C_{D_I} = KC_L^2$	
k	Turbulent kinetic energy	$\left(\frac{m^2}{s^2}\right)$
K_{BL}	BL and viscous region selector threshold coefficient by Paparone and Tognaccini	
L	Length	(m)
L	Lift	(N)
M	Mach Number	
$\dot{m}$	Mass flow rate	$\left(\frac{kg}{s}\right)$
m	Mass	(kg)
MFR	Mass Flow Ratio	
$\hat{n}$	Unit normal direction vector	
NVF	Net-Vehicle-Force vector	
P	Stagnation pressure	(Pa)
p	Static pressure	(Pa)
p_G	Gauge static pressure	(Pa)
$\dot{q}$	Volumetric heating	$\left(\frac{J}{kg}\right)$
$\mathbf{q}_c$	Heat conduction vector due to heat transfer coefficient	
Q	Profile drag interference factor	
R	Radius	(m)
S	Surface (and area)	(m^2)
SFC	See TSFC	
T	Stagnation temperature	(K)
t	Static temperature	(K)
t	Time	(sec)
t/c	Thickness-to-chord ratio	
TET	Turbine Entry Temperature	
TS	Thrust-Split	

$TSFC$	Thrust Specific Fuel Consumption	$\left(\frac{g/s}{kN}\right)$
$TSPC$	Thrust Specific Power Consumption	$\left(\frac{W}{N}\right)$
u	Cartesian velocity component in $\hat{x}$ direction	$\left(\frac{m}{s}\right)$
v	Cartesian velocity component in $\hat{y}$ direction	$\left(\frac{m}{s}\right)$
$\mathbf{V}_A$	Velocity vector in the absolute frame of reference	$\left(\frac{m}{s}\right)$
$\mathbf{V}_R$	Velocity vector in the relative frame of reference	$\left(\frac{m}{s}\right)$
$\mathbf{V}\mathbf{V}$	Velocity dyadic product	
$\mathbf{V}$	Velocity vector (relative frame of reference if no subscript)	$\left(\frac{m}{s}\right)$
$\mathbf{V}^2$	$\mathbf{V} \cdot \mathbf{V}$	
w	Cartesian velocity component in $\hat{z}$ direction	$\left(\frac{m}{s}\right)$
$\dot{W}$	Power / work rate	(W)

Subscripts

∞	Free-stream condition
A	Absolute
BL	Boundary Layer
cr	Critical
CV	Control Volume
DD	Drag Divergence
e	BL edge for BL discussions
eq	Equivalent uniform flow
G	Gauge
h	Fan hub
$hade$	Fan casing Hade angle
lam	Laminar
noz	Nozzle
$poly$	Polytropic

<i>R</i>	Relative
<i>s</i>	Isentropic
<i>t</i>	Fan tip
<i>turb</i>	Turbulent
<i>wet</i>	For wetted areas
<i>i</i>	Highlight area
ing	Ingested
int	Intrinsic
mech	Mechanical
npf	Non-Propelled Fuselage (i.e. clean fuselage with no BLI propulsor)
pre-diff	Pre-diffusion
prop	Propulsor/propulsive
ref	Reference
std	Standard
th	Throat

Abbreviations

ATM	Air Traffic Management
BL	Boundary Layer
BLI	Boundary Layer Ingestion
BLR	Boundary Layer Ratio (see list of symbols)
BPR	Bypass Ratio
BWB	Blended Wing Body
CAS	Calibrated Airspeed
CFD	Computational Fluid Dynamics
kCAS	Calibrated Airspeed (knots)
CR	Lip Contraction Ratio

CV	Control Volume
EAS	Equivalent Airspeed
EFR	Ellipse Fineness Ratio
EIF	Equivalent Inviscid Flow
kEAS	Equivalent Airspeed (knots)
FC	Flight Conditions
FPR	Fan Pressure Ratio
FSAR	Fuel Specific Air Range
GCI	Grid Convergence Index
GGCB	Green-Gauss Cell-Based
GGNB	Green-Gauss Node-Based
HPC	High Pressure Compressor
HPT	High Pressure Turbine
IFR	Instrument Flight Rules
LHS	Left Hand Side
LPC	Low Pressure Compressor
LPT	Low Pressure Turbine
LSCB	Least-Squares Cell-Based
MFR	Mass Flow Ratio
NACA	National Advisory Committee for Aeronautics
NBAA	National Business Aviation Association
NFF	Net Fuselage Force
NVF	Net Vehicle Force
OPR	Overall Pressure Ratio
RANS	Reynolds Averaged Navier-Stokes
RHS	Right Hand Side
ROC/D	Rate-of-Climb/Descent

SFC	See TSFC
TAS	True Airspeed
TET	Turbine Entry Temperature
kTAS	True Airspeed (knots)
TOC/D	Top of Climb/Descent
TS	Thrust-Split
TSFC	Thrust Specific Fuel Consumption
TSPC	Thrust Specific Power Consumption
VFR	Visual Flight Rules
VPD	Vehicle Performance Deck

Chapter 1

Introduction

1.1 Research Rationale

In recent years, improvements in jet propulsion performance has largely relied on gains in propulsive efficiency. This is due to diminishing returns in cycle performance, and is evident by the shift toward geared turbofans [1]. Propulsive efficiency improvements are realised through reductions in specific thrust, by ingesting larger amounts of mass-flow. As turbofan engine core sizes approach their lower limits, the only way to achieve higher propulsive efficiencies, is to increase the bypass ratio (BPR) with larger fan diameters. However, larger nacelle diameters incur external aerodynamic penalties, particularly for aircraft with high subsonic cruise speeds.

Following the above discussion, it is clear why high specific thrust engines are typically used to propel business jets. This is due to a trade-off between the benefits of improved propulsive efficiency, versus penalties in drag and weight associated with the larger engine frontal areas. The implications of drag associated to large frontal areas, are amplified when considering the high operational cruise Mach numbers of these aircraft (up to $M 0.925$ for the Gulfstream G650 [2]), where transonic wave drag becomes dominant. At very high speeds, wave drag is largely dependent on the aircraft's area distribution, to which the engine's frontal area contributes directly. Therefore, area-ruling places further constraints on engine diameter, and subsequently the propulsive efficiency attainable. The tight propulsion-airframe integration of Boundary Layer Ingestion (BLI), offers the opportunity to reduce specific thrust whilst circumventing corresponding enlargements in area distributions, and therefore drag. The primary concept behind BLI, is to restore the aircraft's wake to free-stream conditions, as closely as possible. This means that less fuel energy is lost to the atmosphere, and a larger proportion is usefully applied in propelling the airframe.

Improving Thrust Specific Fuel Consumption (TSFC), is generally an objective for reducing airliner operational costs and emissions. Therefore at first glance, BLI may appear irrelevant in its application to business jets, where these factors are perceived to be of little concern. However, range is a highly valued attribute for business jet customers, particularly at high speeds, thus emphasising the importance of investigating

non-conventional ways to improve fuel consumption. In this context, the behaviour of BLI at high subsonic speeds is of interest, as it is important to determine whether BLI will hinder performance and operational limits, rather than improving upon them. Usually, as in the case of military fighter jets, diverters (or alternative methods) are used to avoid the ingestion of BLs at high speeds, because it compromises engine performance. This is mainly due to the implications associated with flow distortion on turbomachinery operability. However, unlike business jets, these aircraft are expected to perform extreme manoeuvres in a much larger flight envelope. Instead, business jets are required to fly in much more stable operation, and thus the application of BLI may be more appropriate.

Apart from the above, the business jet application of BLI offers the opportunity to transfer power mechanically instead of electrically, due to its relatively smaller size in comparison to commercial aircraft. This avoids the current technological limitations of electrical transmission systems [3], thus improving BLI near-term feasibility. The development time and cost of producing a new business jet is much lower than that of a large commercial airliner, whilst the market is large enough to yield sufficient gains from the benefits of a new technology. Additionally, the business jet market may be more receptive to the introduction of a disruptive technology. A new disruptive technology represents high risk for airline companies in a competitive environment, as the aircraft is a primary asset around which their business is hinged. This is not the case for customers of the business jet, where the aircraft asset is valued differently. Therefore, in this context, the business jet may also serve as an appropriate platform for BLI development and transfer onto commercial aircraft. Having identified the business jet as potential candidate for successful BLI application, the next debate is how to best assess those potential benefits.

A large number of studies have focused on assessing BLI benefits from a top-level systems approach, relying on assumptions to avoid high fidelity aerodynamic assessments. Although such studies give ball-park figures on expected benefits, they do not provide insight into the physical mechanisms governing propulsion-airframe aerodynamic interactions. Additionally, many still attempt to use conventional thrust-drag accounting methods (to estimate propulsion and airframe performance in isolation) as well as performance metrics (such as propulsive efficiency), which are no longer valid when considering highly integrated airframe-propulsion configurations. More advanced methods follow a control volume approach, incorporating the airframe and propulsion system, to better assess the external aerodynamic interactions. Moreover energy-based control volume approaches, are under development because of the potential to avoid conventional thrust-drag accounting definitions, as well as to allow for decomposition and tracking of energy loss mechanisms. This allows for better understanding of the physics driving BLI benefit, thereby aiding more strategic design space explorations. Although growing in popularity, these methods have not yet been applied extensively or rigorously in literature. The work presented in literature has focused on large passenger airliners and unconventional airframes, which tend to fly at lower transonic speeds. Therefore, little is known about BLI performance at the high subsonic speeds at which business jets fly. This work attempts to bridge the gap between top-level and higher fidelity studies. Thus,

a multi-physics based approach is taken under the premise that each method brings with it different insights into the source of BLI benefit.

1.2 Aim and Objectives

The aim of this research is to explore the design-space of rear-fuselage boundary layer ingestion, with application to business jets. The scope is focussed on external aerodynamics of BLI propulsion-airframe coupling, as opposed to the performance of the propulsion system's internal components. In addition, the analysis is constrained to turbo-mechanically driven propulsion of tube-and-wing airframes. The objectives of the research are to:

- Develop generalised multi-physics based methods for assessing aircraft performance, from RANS CFD solutions.
- Implement the above methods within a design exploration study on the application of BLI to business jets.
- Identify and assess additional possible performance synergies which may exist, and are enabled, through the application of BLI.
- Perform a uncertainty analysis of the methods developed, along with a technology risk assessment of BLI applied to business jets

1.3 Knowledge Contribution

There are numerous contributions to knowledge contained within this thesis. These are listed in the introduction and summary sections of each chapter within the main body of this document. A few of the prominent contributions are summarised here:

1. A fully customisable, momentum-based, control volume approach, has been developed to produce a unique set of thrust-force accounting definitions, which are applicable to tightly-integrated airframe-propulsion systems. This work includes a step-by-step procedure for extracting these definitions from RANS CFD solutions. In addition, it is demonstrated that a comparison between the definitions allows for improved insight into the mechanisms behind BLI propulsion.
2. A method of BLI performance quantification, based on a modified propulsive efficiency for retrofit design approaches. This avoids the flaws associated with the conventional definition, when applying it to BLI assessments (the conventional propulsive efficiency may theoretically exceed 100%, but this is not possible with the modified definition used in this work).
3. Formulation of the kinetic and total energy equations in the absolute reference frame. The derivation is unique to this work, and having the energy equation

in the absolute reference frame provides an intuitive perspective into the physics behind the mathematical terms. This work directly addresses concerns of Renard and Deck [4] about Drela's [5] formulation in the aircraft's body axes system. In so doing it accounts for pressure and friction forces on solid surfaces, as well as the kinetic energy imparted to the fluid (see Section 4.1.1 for more details). This leads to an energy-based control volume formulation with *NVF* as the subject, which sheds light on the contributions of various energy terms. Other energy formulations [5, 6] take a *energy source = energy sink* format, which makes it difficult to understand the contributions of some terms.

4. A procedure of spatial decomposition by Paparone and Tognaccini [7] into different flow mechanisms has been adopted, modified and improved from formulations on entropy drag, and incorporated into the energy approach of this work. This decomposition allows for volume integrals describing energy dissipation and volumetric compression work, to be quantified for different flow regions (e.g. shock, jet and BLs). A BL and viscous wake region selector, modified from Paparone and Tognaccini's work, is a particularly significant contribution (see Section 5.1). In addition, the precedence procedure developed, allows for further decomposition of the jet into core expansion, core compression and viscous transition regions, and is unique to this work.
5. This thesis provides clear evidence that the volumetric compression work term of energy-based formulations, has a significant thrust contribution, which cannot be assumed small or negligible. This advances the work by Drela [5], who speculated its contribution to only be significant where heat is added at pressure within the control volume. This is now known not to be the case, and making *NVF* the subject of the energy equation gives insight as to why not. The majority of the volumetric compression work term's contribution is found to be located within the isentropic expansion regions of the jet core. This discovery was made possible by the flow decomposition procedure developed in this thesis. More importantly, it shows that there is a natural thrust-force definition that avoids ambiguity. This definition attributes all rearward forces to volumetric viscous dissipation in the flow, whereas thrust is attributed to the surface forces on fan blades, along with the volumetric compression (or rather expansion) work in the jet plume.
6. Design exploration of BLI applied to a business jet which, to the author's knowledge, has not yet been published in public literature. This includes trends which show how the benefit of a retrofit design is optimised through managing thrust-split. This is based on a unique turbofan performance map, developed specifically to analyse thrust-split retrofit designs. In addition, the author is unaware of any other publicly available studies which have considered the effect of different BLI propulsor tail-cone positioning. The final comparison of BLI benefit has been performed using various methods, based on different physical laws. This multi-physics based approach to BLI performance is unique to this work, and is opposed to prescribing a single method for future use. Instead, improved physics based

and numerical understanding of CFD solutions is gained through comparisons between methods. In addition, these comparisons allow for uncertainties in predicted benefits to be quantified.

7. To the author's knowledge, no other study has yet performed the full energy breakdowns and decompositions, from RANS CFD solutions, like those presented in this thesis. This work includes comprehensive analyses of energy progression and transfer within the wake. It also performs a full decomposition of physical mechanisms in the jet internal and external flow-fields.

1.4 Thesis Outline

Following a focussed review of the public literature, this thesis begins with the theoretical development of control volume based assessment methods. It then shifts focus to the practical implementation of these methods to RANS CFD solutions, and culminates in a design exploration of a boundary layer ingesting business jet. The chapters of the main body are summarised as follows:

- Chapter 3 develops near- and far-field momentum-based formulations of the net-vehicle-force (NVF), as well as a series of thrust-force accounting definitions. A method for evaluating BLI performance is introduced, which defines a modified propulsive efficiency appropriate for application to retrofit design approaches.
- Chapter 4 develops formulations of the NVF derived from the energy equation, which is applied to a control volume encapsulating the aircraft. The derivations are formulated in the absolute frame of reference, which aids the physical interpretation of mathematical terms.
- Chapter 5 takes a comprehensive look at how the theory in Chapters 3 and 4 may be implemented in post-processing RANS CFD solutions. This chapter also develops procedures for spatially decomposing the flow into different flow phenomena.
- Chapter 6 is the concluding results chapter, which addresses the aim of this research by documenting a design exploration study into fuselage concentric BLI propulsor performance, applied to a business jet.

Chapter 2

Literature Review

2.1 Background

Aircraft applications of BLI have been investigated as early as 1947 by Smith and Roberts [8], who considered the benefits of BL suction over the wing by the aircraft's engines. Their results suggested fuel burn benefits in excess of 30%, as well as higher optimum cruise speeds, resulting predominantly from BL stability control. Subsequent studies at the Douglas Aircraft Company [9, 10], looked at the more modest approach of rear-fuselage BLI, and claimed potential propulsive efficiency benefits of up to 10%. Substantially later, in 1993, Smith [11] published a paper detailing BLI of an axisymmetric body through an un-ducted propeller placed hypothetically far downstream of the body. The study used incompressible actuator disk theory to account for propulsion losses due to the ingestion of non-uniform flow, and the calculations yielded a 7% improvement in propulsive efficiency. More recently, aviation industry targets for reduced emissions have sparked further interest into BLI concepts. The majority of studies have focused on the popular idea of turbo-electrically driven BLI applications for Blended Wing Body (BWB) concepts. However, other BLI concepts apart from the BWB have been investigated, with Bauhaus Luftfahrt claiming fuselage concentric BLI as the most advantageous configuration for tube-and-wing airframes [12]. Among other tube-and-wing like concepts, is the Double-Bubble, or D8, with claimed fuel burn benefits totalling in excess of 70% [13], and wind tunnel experiments indicating a 6% benefit in having BLI over podded installations [14].

2.2 Principles of BLI Benefit

The benefit of BLI has typically been explained from two points of view. The first is an over simplification, which considers improvements in propulsive efficiency due to a reduction in perceived ram drag through ingesting the lower momentum flow contained in the boundary layer. A second, more correct viewpoint, explains BLI benefit as a means for reducing the amount of energy lost to the atmosphere by minimising kinetic energy deviation in the wake from free-stream conditions. The following subsections describe

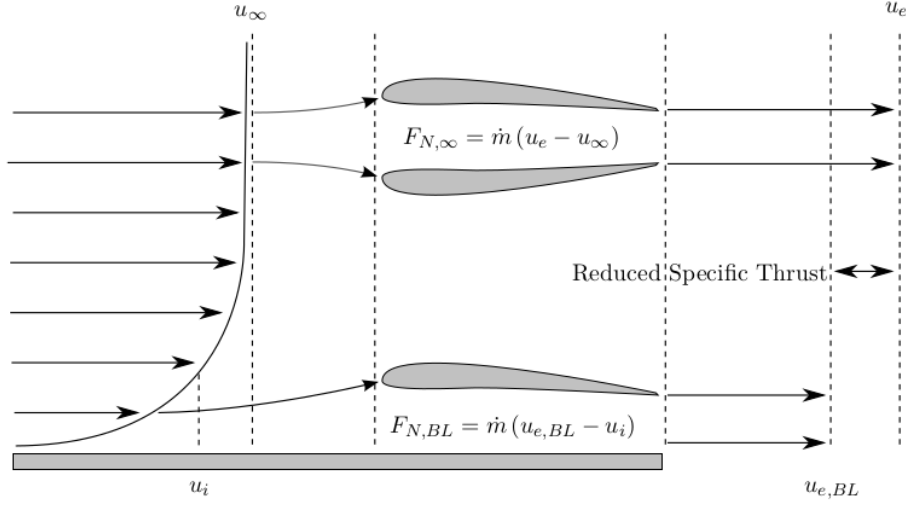


Figure 2.1: Propulsive Efficiency Improvement from Reduced Ram Drag

these two perspectives, and it becomes clear that assessing aircraft performance from an energy point-of-view can be hugely advantageous.

2.2.1 Propulsive Efficiency Improvement from Reduced Ram Drag

One of the simplest ways to visualise BLI benefit is from a propulsive efficiency standpoint, and the approach from Reference [15] will be used here. Consider a hypothetical propulsor that always produces the same amount of thrust for a given ingested mass flow rate, as depicted by Figure 2.1. Assuming a static pressure differential of unity across the propulsor (i.e. a perfectly expanded exhaust), the thrust generated may be approximated as the amount of momentum imparted to the ingested flow,

$$F_N = \dot{m} (u_e - u_i) \quad (2.1)$$

where the subscripts i and e denote the propulsor inlet and exit flows respectively. Note, that these equations assume the net thrust F_N is equal to the intrinsic net thrust. In order to maintain a constant thrust value for a given mass flow rate, the difference between inlet and exit flow velocity must also remain constant for changes in ingested flow velocity. Therefore, a change in ingested flow velocity requires an equivalent change in exit flow velocity,

$$u_e = u_i + \Delta u = u_i + \text{constant} \quad (2.2)$$

Subsequently, the power transferred to the ingested flow to produce the required thrust may be expressed as the rate of kinetic energy transferred to the ingested flow,

$$\begin{aligned}
 P &= \frac{\dot{m}}{2} (u_e^2 - u_i^2) \\
 &= \frac{\dot{m}}{2} \Delta u (u_e + u_i) \\
 &= \frac{\dot{m}}{2} \Delta u (\Delta u + 2u_i)
 \end{aligned} \tag{2.3}$$

It now becomes evident from Equations 2.2 and 2.3, that the power required to produce the same amount of specific thrust increases with ingested flow velocity, and vice versa. However, this conclusion is misleading as the corollary would seemingly be to fly at a slower airspeed, thereby allowing podded engines (intended to ingest free-stream flow) to ingest lower velocity flow thus reducing the power required. Although it would not necessarily be false statement, the power reduction benefit is rather a simple inherency of reduced airspeed¹ and furthermore, it doesn't take into account the power of vehicle at its flight speed. In-fact, as it will be shown, the efficacy by which this power is transferred to flight power is enhanced at higher flight speeds. This becomes evident when considering propulsive efficiency, which is a ratio of the propulsive power (a product of airspeed and thrust) over the rate of kinetic energy transferred to the flow in producing that thrust (Equation 2.3),

$$\begin{aligned}
 \eta_{prop} &= \frac{\dot{m} \Delta u u_\infty}{\frac{\dot{m}}{2} \Delta u (u_e + u_i)} \\
 &= \frac{2u_\infty}{(u_e + u_i)} \\
 &= \frac{2u_\infty}{(\Delta u + 2u_i)}
 \end{aligned} \tag{2.4}$$

which may be expressed more conveniently in non-dimensional term, represented as the free-stream relative inlet and exit velocities,

$$\eta_{prop} = \frac{2}{\left(\frac{\Delta u}{u_\infty} + 2\frac{u_i}{u_\infty}\right)} \tag{2.5}$$

In the case of a podded installations, assuming purely free-stream flow ingestion, the above equation reduces to,

$$\eta_{prop,\infty} = \frac{2}{(\Delta u/u_\infty + 2)} \tag{2.6}$$

which shows more clearly how the propulsive efficiency may be improved by simply increasing flight airspeed. However, comparing Equations 2.5 and 2.6, it becomes evident that for a given flight speed and specific thrust (represented here as Δu), the propulsive efficiency improves if the engine intake velocity is less than the flight speed (i.e.

¹Additionally the aircraft's drag is dependent, roughly parabolically, on airspeed

$u_i < u_\infty$). In fact, a combined effect of increasing flight speed and reducing propulsion intake velocity will result in further improvements in propulsive efficiency. Furthermore, taking into account variations of specific thrust, it may be observed that there are three parameter changes that influence the improvement in propulsive efficiency; a reduction in specific thrust, increase in flight velocity, and reduction in intake velocity.

Nonetheless, the airframe's boundary layer, or wake, is a possible "source" of reduced flow velocity for ingestion. Therefore for a given airspeed, thrust and ingestion mass flow rate, the propulsive efficiency may be improved by ingesting the airframe's boundary layer. This means that through the implementation of BLI, less mechanical power is needed to power the propulsors, and hence a reduction in fuel burn may be realised. Other descriptions of this benefit (different to the above, but ultimately equivalent) also from a propulsion efficiency perspective, may be found in Reference [16]. It is noted that the above description, and those of Reference [16], serve only as an illustration as to how BLI may be of benefit to the propulsion system, and does not capture the necessary details to describe the true implications of BLI. To do so, one would need to account for changes in aircraft drag, airframe and propulsion system weights as well as propulsion system transfer and component efficiencies under such operation.

2.2.2 Minimising Energy Lost to the Atmosphere

An alternative perspective for understanding BLI benefit, is to consider energy transfer from the aircraft to the atmosphere [5, 6]. Figure 2.2 depicts the stream-wise velocity profile of wakes trailing behind free-stream and boundary layer ingesting configurations. In both cases, the net force on the aircraft is balanced such that it is in steady level flight. The stream-wise velocity profiles can be seen to deviate from the free-stream velocity both positively (in jets) and negatively (behind airframe wakes). To maintain steady level flight, these positive and negative regions should roughly cancel one another out. The deviations will diminish almost entirely far down-stream of the aircraft, as the wake returns to free-stream conditions. This attenuation is a result of viscous dissipation in the flow, which is energy lost to the atmosphere. It is therefore irrelevant if the profile deviations are positive or negative, but what does matter is the magnitude of the deviations. The larger the deviation in velocity profile from the free-stream condition, the more energy is dissipated to the atmosphere. In comparing the BLI configuration against the free-stream ingesting configuration, it becomes evident that by re-energising the airframe boundary layers (or a portion thereof) the jet stream velocity has to compensate less to achieve steady level flight. As such, the net deviations in wake profile velocity from the free-stream is reduced and less energy is imparted to the atmosphere. Theoretically, an ideal propulsor would return all the wake of the aircraft to exactly the free-stream velocity, resulting in no energy transfer to the atmosphere in the form of kinetic energy. This is hypothetical of course, but it is worthwhile considering re-energising a portion of the aircraft's wake where practical. In essence, an aircraft's overall efficiency is improved by minimising its disturbance on the atmosphere. This is a more holistic approach to understanding BLI rather than a system level perspective of reduced ram drag forces on a propulsor.

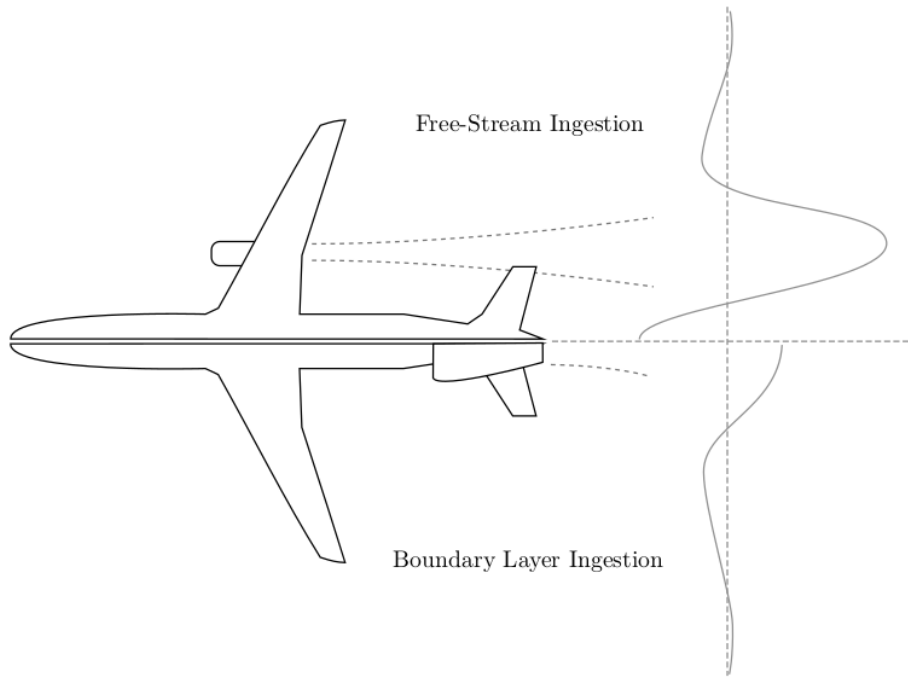


Figure 2.2: BLI Benefit from Minimising Wake Kinetic Energy

2.3 Why the Business Jet?

For the purposes of this work it is necessary to select an appropriate aircraft type for the application of near-term distributed propulsion technologies. Figure 2.3 illustrates the distribution of aircraft type across the civil aviation market in terms of the current and future fleets between 2014 and 2033. The top graph illustrates the distribution in numbers of units, whereas the lower two graphs depict the associated costs of the new deliverables required for the future fleet projections. From this figure it is evident, as it is intuitive, that the larger the aircraft the higher its unit cost. However there are, also intuitively, fewer of these larger aircraft in the current and future projected fleets. Single-aisle jets appear to dominate the market with respect to numbers and market value, whereas regional jets seem to be of the lowest market value and decreasing in number. Business jets, on the other hand, are projected to have the largest fleet in 2033, but are associated with a much lower cost per unit and overall market value.

For introducing a novel technology, it seems sensible to select a market where the development costs and times are relatively low, but with a high enough number of units such that the market share is sufficiently significant for the new technology to have a

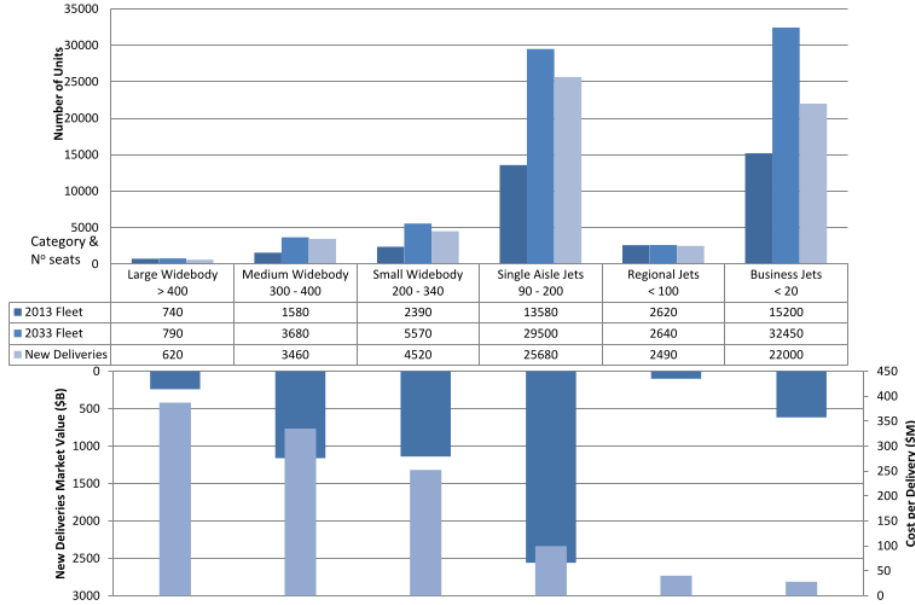


Figure 2.3: Aircraft type distribution across the civil aviation market. Data collected from References [17] and [18]

relevant impact. This is because, as seems logical to the author, a lower development cost and time will have reduced risks for investors, but enough incentive because of the high number of units (and hence significant market impact). Based on the information in this figure, it seems reasonable to discount the regional jet sector, as the market value it holds is comparatively small and there is little sense in developing a new technology for a market on the decline. It also seems reasonable to discount wide-body aircraft due to the high unit cost which correlates to high development cost and time, and hence risk. The two remaining aircraft types are single aisle and business jets, with the former and latter holding the largest market value and number of units, respectively. Both of these are attractive prospects for introducing a new technology, however the author proposes a favoured direction of technology transfer beginning with the business jet (lowest development cost and time, and therefore risk), following up with the single-aisle jet and subsequently moving to wide-body aircraft in order of ascending size.

2.4 Energy-Based Performance Assessments

In this section, various literature on energy-based formulations of performance assessment are considered. In studying the rationale behind energy-based methods, it is concluded that they exhibit great potential for improved identification and analysis of var-

ious flow mechanisms and how they contribute (either to the benefit or detriment) to vehicle overall efficiency. This is because energy-based methods allow for decomposition of these various flow mechanisms in the context of an overall energy accounting system. The energy methods also allow for more physically intuitive mathematical forms of such decompositions, adding to flow field mechanism insight, which is particularly valuable when examining the inherently coupled flow phenomena of BLI. In the pursuit of identifying the most beneficial way to make use of such methods, examples of the applications of these methods in literature were examined and the following knowledge gaps identified:

- Few studies implemented these methods in compressible RANS CFD solution, and those that did used simplifying assumptions to approximate dissipative phenomena instead of calculating them directly.
- Based on the above point, it is unclear these methods may be used practically with RANS CFD, or if it is a limitation of the methods. No studies, to the authors knowledge, have comprehensively examined this practical applicability and discussed the possible limitations
- Power balance formulations have been based on the kinetic energy equation, and dissipation volume integrals may be presented by internal energy fluxes through the total energy equation. This substitution of internal energy flux surface integrals may be used to avoid the approximations used in literature.
- None of the literature has yet explored how to spatially partition the CFD domain (in the context of the energy methods) so that the energy-based methods may be applied to decompose out power consumption contributions of different flow mechanisms
- The majority of CFD studies making use of these energy methods, use actuator disks to represent propulsors. This fundamentally contradicts the energy-based formulation principle, as it violates thermodynamic laws in compressible flows. Furthermore Gauss' Theorem, upon which these formulations are based, no longer holds true for flows containing discontinuities. Therefore, it is necessary to model the propulsor in an alternative way using spatially separated inlet and outlet boundary conditions which are thermodynamically valid.

2.4.1 Background and Rationale

An alternative way to analyse aircraft performance is from an energy point-of-view, instead of momentum. Two such approaches found in literature are the *Power Balance Method* [5] and *Exergy-Based Method* [6]. Instead of directly analysing the forces acting on an aircraft (be it using near or far-field methods) these methods look at the flow rates of energy (or exergy, in the latter) passing through a control volume encompassing the aircraft. To describe the advantage of the Power Balance Method, Sato [19] draws an analogy to a ball rolling along a surface with variations in vertical displacement along its

path. A momentum-based analysis, must account for all the forces acting on the ball (i.e. normal and friction forces) and how they vary along the path, in order to calculate the ball's final velocity at some destination. An energy-based analysis only requires details of frictional forces along the path (i.e. the energy lost by the ball to its surroundings) in order to determine its final velocity from energy conservation. Based on this analogy, Sato attempts to claim the simplicity of an analysis using energy conservation as opposed to momentum conservation. Therefore, the Power Balance formulation aims to examine aircraft performance by tracking the amounts of power in-flow (i.e. from a propulsor) usefully used in propelling the aircraft and the amounts lost to the surroundings through various physical mechanisms (such as energy dissipation in boundary layers, wakes, jets and shock waves). The Exergy-Based approach amounts to the same idea, except that exergy is the accounting metric and the formulation from the total energy equation (as

fore, in theory, and given a perfectly resolved flow-field, the momentum equation will give an accurate result of the net-vehicle-force even with interfering bodies. In actuality, these results are more accurate than that of the dissipation coefficient, as they are not reliant on a boundary layer edge velocity, which requires calculation of boundary layer displacement thickness from from inviscid flow solutions [21, 22].

A significant potential advantage of energy-based methods is not in the determination of a net-vehicle-force, but rather in the decomposition of power sources and sinks. Such decomposition allow for better insight into the impact of various flow mechanisms on the efficiency of the aircraft as a whole, and subsequently allow for improved strategic design space explorations. The importance of flow-field decompositions is emphasised by Spalart [23] :

"it would be very valuable if a Navier-Stokes solution could be processed to rigorously separate induced drag, parasite drag, compressible drag, and finally thrust (in fact, even the division into drag and thrust is not exact)"

The above quotation highlights limitations of current methods to decompose flow phenomena satisfactorily, and this is where energy-based methods may give performance analysis the edge, particularly when it comes to assessments of tightly integrated airframe-propulsion systems. Both the power balance and exergy formulations include flow-field decompositions, which relate power consumption to different physical mechanisms. Power decompositions include contributions due to [19]:

- **Vortex Region** - energy loss due 3D effects of vortical flow typically associated with lift induced drag
- **Wave Region** - the energy deposited in the flow due to the presence of shock waves
- **Surface Region** - energy dissipated in the boundary layers along airframe surfaces
- **Wake Region** - the energy associated with the wake produced by the airframe surfaces
- **Jet Region** - the energy introduced to the flow by the propulsor in the form of a jet plume

The exergy analysis incorporates additional decomposed contributions due to heat, such as:

- heat transfer by the propulsor
- heat transfer by surface conduction
- thermal mixing losses

The latter terms do indeed provide a more comprehensive energy assessment, but introduce numerous modelling complications, which may be initially unnecessary for BLI assessments. As such, it seems more appropriate to focus on the power balance approach, while the methods and their implementation reach maturity for simplifying assumptions about heat transfer. Nonetheless, it must be noted that exergy does appear to offer a beneficial global accounting system for full aircraft performance analysis, a review of which is provided by Hayes *et. al.* [24].

In light of the above, it seems worthwhile to pursue the decompositions made theoretically possible by the power balance method, because the mechanisms behind BLI performance have not yet been fully identified or understood. It is noted that flow field decompositions are not entirely new, as numerous approaches have been developed for extracting various contributions to overall drag through manipulation of the momentum equation [7, 23, 25–28]. However, these decomposition formulations result in complex mathematical terms (often in the form of entropy and enthalpy drag [7]) which lack physical intuitiveness and introduce theoretical uncertainty. A prime example has been highlighted by Drela [5], who was able to give physical interpretation of the negative stream-wise velocity term found in the classical induced drag integrand $\rho(v^2 + w^2 - u^2)/2$, attributing it to pressure work across the trefftz plane. A lengthy discussion about this term is provided by Spalart [23], who discusses how it is often neglected due to its counter-intuitive energy contribution, despite its omission violating conservation of energy. This example illustrates how decompositions from both momentum and energy-based perspectives can be used to compliment one another by deciphering physical attributions of mathematical terms. Tighter airframe-propulsion integration, leads to flow fields which become more complex to decompose. Therefore, in the context of this work, energy methods are of interest because of the additional physical insight they bring. In addition, they decompose physical phenomena within the overall power accounting of the entire aircraft, which gives immediate insight as to the impact of various flow phenomena on vehicle efficiency.

2.4.2 Implementation

In putting into practice the power balance method, Drela [5] reduces the explicit formulations of energy surface and volume integrals to simplified forms. In determining induced drag power, it is suggested to place a trefftz plane sufficiently close to the aircraft such that approximately all of the induced drag power has not yet been dissipated, and may be evaluated as a surface integral of transverse kinetic energy flux. This assumption avoids the volume integral of the explicit formulation, but results in the same momentum-based induced drag wake extraction formulation. These surface integral approaches fail to correctly account for vortex roll-up, as discussed by Spalart [23]. In accounting for power consumption of the propulsive jets, Drela recommends using a Froude propulsive efficiency in place of the jet dissipation volume integral. This is counter intuitive, because it is desirable to rather determine the jet efficiency from the power consumption, instead of specifying the power consumption through a pre-defined efficiency. Drela also suggests approximating shock-wave dissipation volume integrals using relations to convert



cluded, to the author's knowledge, that no attempt has been made to apply the power balance method explicitly to viscous, compressible RANS CFD solutions. "Explicitly", in this case, refers to applying the formulations as they are, without any simplifying relations to approximate surface and volume integral terms. This immediately brings into question the applicability and practicality of energy-based assessments using RANS CFD. In light of this observation, an objective of this research is to explore the practical implementation of energy-based methods, in explicit form, to compressible RANS CFD solutions.

In addition, the CFD studies reviewed in this work, tend mostly to rely on actuator disk models to represent the propulsor. As already mentioned, compressible flow CFD is required (even if run at incompressible flight speeds) to capture the energy conservation and dissipative sinks within the flow field. This makes inappropriate use of the actuator disk, as source terms are required to allow for the flow to be compressed (but at the same time maintaining a constant area across it) as a discontinuity. In ANSYS Fluent, this occurs as a violation in the second law of thermodynamics, via a discontinuous decrease in entropy across the disk. In light of this argument, and the previous objective, it seems necessary to find an alternative means for modelling the propulsor using two separate inlet and outlet boundary conditions. A further argument opposing the use of actuator disks, is that Gauss' Theorem (used in both derivations of the power balance and exergy methods) is not correct for flows containing discontinuities. Ordinarily, in RANS CFD, natural discontinuities (i.e. shocks) are actually made continuous due to smearing from discretization [25].

As a final note of this section (because no literature has been found where the energy-based methods have been applied directly to compressible CFD solutions in their original form) there is the additional knowledge gap of how to spatially partition CFD flow domains in the context of energy-based methods, for quantification of surface and volume integral terms which capture different flow phenomena (e.g. shocks, boundary layer, wakes and jets). Furthermore, should volume integrals prove to be impractical, is it possible to reformulate these terms into other surface integral flux parameters. It is shown in a following chapter, that this is indeed the case, by considering the total energy equation and how the volume integrals of dissipation relate to fluxes of internal energy.

Chapter 3

Momentum-Based Approach to Evaluating Performance

The forces acting on all the surfaces of an integrated airframe-propulsion system, may be calculated from near or far-field properties. In the context of the derivations carried out in this chapter, near-field refers to the integration of net pressure and shear stress forces acting on control volume surfaces which lie directly on the airframe and propulsion system surfaces. Alternatively, the net force acting on the airframe-propulsion system may be obtained from the conservation of momentum, by integrating momentum terms along flux surfaces bounding the configuration. In theory, this is useful as it is not always practical or convenient to perform near-field integrations. There is also an argument as to which gives more accurate drag predictions, which is discussed in Reference [26]. Nonetheless, this concept allows for a control volume encapsulating the configuration to be subdivided into any desirable combination of streamtubes. This in turn allows for convenient definitions of various thrust-force accounting systems.

This chapter begins with a brief description of appropriate reference frames to which generalised forms of the continuity and momentum equation may be applied given an arbitrary closed control volume. The resulting form of the momentum equation is applied to a streamtube which is partially bound by a physical wall. This allows for different definitions of thrust-force accounting systems to be explored, which are discussed in detail with reference to BLI applications. Finally, a new definition for propulsive efficiency is adopted for application in this work.

The contributions in this chapter are as follows:

- A standardised method for deriving customizable thrust-force accounting systems from first principles, and generalised applicability to any aircraft-propulsion integration.
- A definition of net-vehicle-force and a set of thrust-force accounting definitions options, applicable to tightly integrated airframe-propulsion systems. To the authors knowledge, this approach to thrust-force accounting is unique to this work and has not been properly studied in the available literature.

- A practical definition of propulsive efficiency in the context of this work, which allows for decoupled analysis of main engine cycle and BLI propulsor performance for retrofit thrust-split BLI configurations.
- A control volume analysis which allows for the inclusion of a propulsor in the CFD fluid domain using a combination of inlet and outlet boundary conditions.

3.1 Reference Frames and Properties

External aerodynamic problems are mostly modelled in the aircraft's reference frame, where the atmosphere moves over a still aircraft. An alternative reference frame considers a moving aircraft through a stationary atmosphere. The former and latter are referred to as relative and absolute reference frames, following the definition of Renard and Deck [4]. The relative reference frame is more useful since it allows for a constant control volume around the aircraft to be defined, which is easier to model in CFD and to simulate experimentally in a wind tunnel. In terms of momentum, the forces obtained in each reference frame are equally intuitive. However, when considering the energy equation, power sources/sinks become less intuitive in the relative reference frame, which is the topic of Chapter 4. For example, the velocity on airframe surfaces is zero following the no-slip condition, and this gives the impression that no power is introduced to the flow due to surface forces, which is counter-intuitive. From this perspective, power is supplied to the flow to drive it over the stationary aircraft, rather than power being supplied to the aircraft in driving it through the stationary atmosphere (as is the case in an absolute reference frame). Although total powers are equivalent between the two reference frames, these alternate perspectives result in power source decompositions which have different mathematical expressions and thus physical interpretations. As will be seen later in Chapter 4, the absolute reference frame allows for more physically intuitive power decompositions and therefore a transformation from the relative to absolute reference frame is required. Following this argument, both momentum and energy-based formulations will be constructed in the absolute reference frame. This work uses a velocity relation between reference frames as given by Equation (3.1).

$$\mathbf{V}_R = \mathbf{V}_\infty + \mathbf{V}_A \quad (3.1)$$

In the derivations that follow it becomes more convenient to make use of gauge pressure instead of absolute pressure, which filters out the buoyancy related forces and powers. The relation between absolute and gauge pressures is:

$$p_A = p_\infty + p_G \quad (3.2)$$

It is noted that for the remainder of the work, that subscripts have been dropped for relative velocity and absolute static pressure (i.e. $\mathbf{V}_R \rightarrow \mathbf{V}$ and $p_A \rightarrow p$).

3.2 Continuity and Momentum Equations

The differential conservation form of the continuity equation is given by Equation (3.3). Assuming steady flow conditions and taking the volume integral of Equation (3.3) applied to an arbitrary closed control volume, Equation (3.4) is obtained from divergence theorem¹.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \quad (3.3)$$

$$\oint_S (\rho \mathbf{V}) \cdot d\mathbf{S} = 0 \quad (3.4)$$

Similarly, the momentum equation in integral form, Equation (3.6), is obtained from the differential conservation form, Equation (3.5). A more convenient form, Equation (3.7), is obtained by substituting Equations (3.1) and (3.2) into Equation (3.6) and using the continuity relation Equation (3.4).

$$\frac{\partial}{\partial t} (\rho \mathbf{V}) + \nabla \cdot (\rho \mathbf{V} \mathbf{V}) = -\nabla p + \nabla \cdot \bar{\bar{\tau}} + \rho \mathbf{f} \quad (3.5)$$

$$\oint_S (\rho \mathbf{V} \cdot d\mathbf{S}) \mathbf{V} = \oint_S (-p \hat{n} + \bar{\bar{\tau}} \cdot \hat{n}) dS + \iiint_V \rho \mathbf{f} dV \quad (3.6)$$

$$\oint_S [(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n}] dS = \oint_S [-(p - p_\infty) \hat{n} + \bar{\bar{\tau}} \cdot \hat{n}] dS + \iiint_V \rho \mathbf{f} dV \quad (3.7)$$

At this point, it is noted that the divergence theorem (or Gauss' theorem) does not hold true in regions containing discontinuities, such as the shock waves present in transonic flow regions. However, Desterac and van der Vooren [25] point out that in real viscous flow there are no actual discontinuities present, and therefore use of the divergence theorem is mathematically correct. Furthermore, in numerical CFD RANS solutions, artificial viscosity sources are introduced and smear out any discontinuities. The result is that any formulations based on the divergence theorem throughout this work, are mathematically correct, both in real flow and numerical RANS solutions.

¹ $\iiint_V \nabla \cdot () dV = \oint_S () \cdot d\mathbf{S}$

3.3 Application to Streamtubes

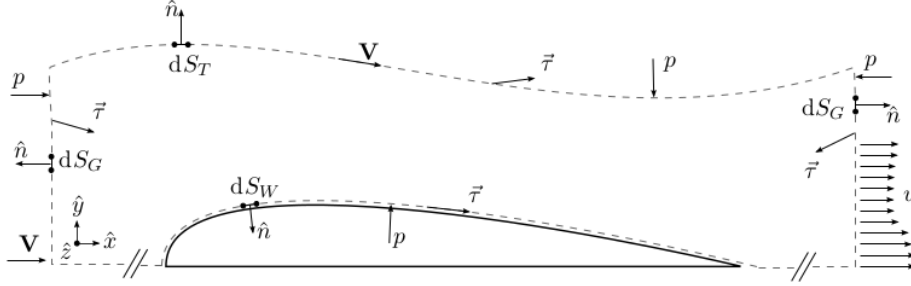


Figure 3.1: Streamtube encapsulating an arbitrary body

Figure 3.1 depicts an axisymmetric streamtube encapsulating an arbitrary body. The streamtube is comprised of three types of surfaces, 1) S_W are surfaces draping physical walls, 2) S_T are unbounded surfaces parallel to flow streamlines, and 3) S_G are flux surfaces. Assuming no body forces act on the flow within the streamtube, Equation (3.7) may be rearranged as:

$$\iint_{S_W} [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS = \iint_{S_G} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS - \iint_{S_T} [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \quad (3.8)$$

The LHS of Equation (3.8) is the *sum of forces acting on physical surfaces by the fluid*. The first RHS term is the *sum of forces acting on the flux surfaces*. The second RHS term is the *sum of forces acting on the unbounded streamtube walls*. The second RHS term is just a simplification of the first RHS term, as no mass flow crosses it. It is noted that shear stress terms exist on all surfaces. These are typically small values for flux surfaces and unbounded streamtube walls, and are often neglected. However, they are retained here in the context of BLI, as the streamtubes being considered may be submersed within the boundary layer. A more convenient notation for Equation (3.8) may be adopted:

$$\Psi = \Omega - \Phi \quad (3.9)$$

3.4 Application to Propulsor Control Volumes

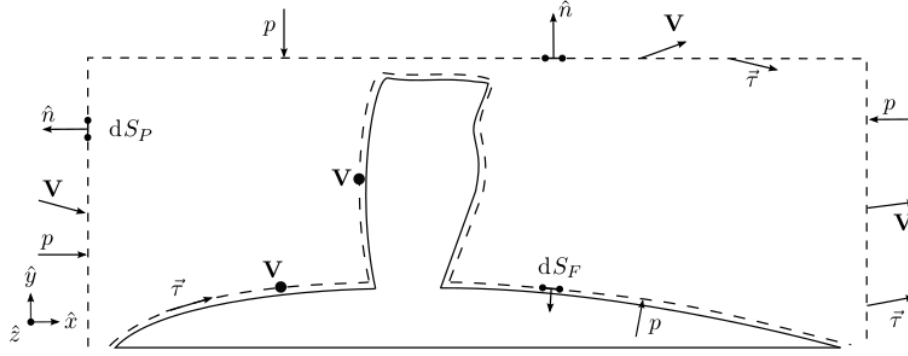


Figure 3.2: Control volume encapsulating arbitrary propulsor

It is convenient to consider the propulsion system in isolation, as it generally has a multitude of complex parts and geometry, making it difficult to visualise how the control volume may wrap around its surfaces. A control volume encapsulating an arbitrary rotating fan is depicted in Figure 3.2. The rotation is depicted by the tangential velocity vector pointing out of the page. The control volume surfaces are split into an inner fan blade surface S_F and outer encapsulating surface S_P . All the surfaces of S_P are considered to be flux surfaces. The net force acting on the propulsor is due to the sum of static pressure and shear stresses acting on the surface draped by S_F . This is obtained by simply applying Equation (3.9) to yield Equations (3.10) and (3.11). The propulsor may now be represented by the encapsulating control volume surface, S_P , when incorporating it within the aircraft's overall control volume. This simplifies the derivation of forces acting on the aircraft as a whole.

$$\Psi_{S_F} = \Omega_{S_P} \quad (3.10)$$

$$\iint_{S_F} [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS = \iint_{S_P} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \quad (3.11)$$

3.5 Application to Propelled Bodies

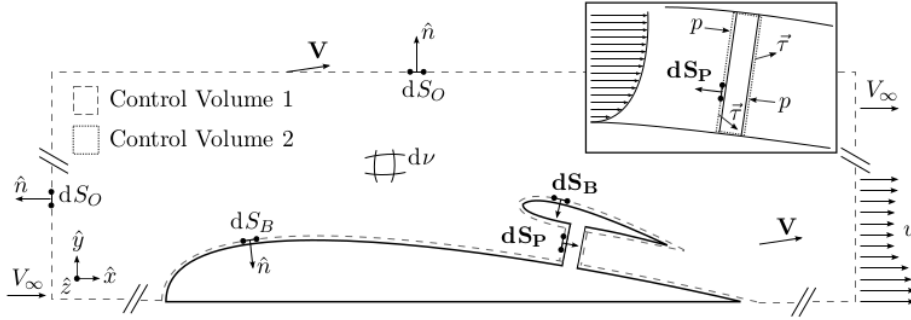


Figure 3.3: NVF control volume

In the absence of a commonly accepted thrust-drag accounting system for highly integrated airframe-propulsion systems, it is convenient to work with a Net Vehicle Force (*NVF*), so as to avoid any ambiguity. The *NVF* allows for indirect determination of forces on portions of streamtubes that would normally be impractical to calculate directly (as will be discussed in Section 3.6). The *NVF* is the total force acting on the aircraft as a combined system i.e. incorporating contributions from both propulsion system and airframe. It is a rigorous approach which avoids double accounting or neglecting forces. The generalised form of the momentum equation, Equation (3.9), is applied to an airframe-propulsion system in order to derive the *NVF*. Referring to Figure 3.3, the control volume encapsulating the airframe-propulsion system may be split in two. The first control volume has an outer surface S_O , drapes all airframe surfaces S_B and the surfaces of a second control volume S_P encapsulating the propulsor. The sum of the two control volumes equates to a single control volume encapsulating the entire system and wrapping around all physical surfaces. In defining these two control volumes, it is important to make note the normal vector $\hat{n}$, which by convention points outwards, is in opposite directions for adjacent control volumes. In the approach taken here, all unbounded surfaces are considered to be flux surfaces for generalisation. The forces acting on the airframe surfaces are obtained by applying Equation (3.9) to the first control volume:

$$\Psi_{S_B} = \Omega_{S_O} + \Omega_{S_P, CV_1} = \Omega_{S_O} - \Omega_{S_P, CV_2} \quad (3.12)$$

$$\begin{aligned} \mathbf{F}_B = \iint_{S_B} [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS &= \iint_{S_O} - [(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS + \\ &\quad \iint_{S_P, CV_1} - [(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \end{aligned}$$

Similarly, the forces acting on the propulsor are obtained by applying Equation (3.9) to the second control volume:

$$\Psi_{S_P} = \Omega_{S_P, CV_2} = -\Omega_{S_P, CV_1} \quad (3.13)$$

$$\begin{aligned} \mathbf{F}_P &= + \iint_{S_P, CV_2} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \\ &= - \iint_{S_P, CV_1} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \end{aligned}$$

Subsequently, the *NVF* is calculated as a sum of the forces acting on the airframe and propulsor surfaces:

$$\mathbf{NVF} = \Psi_{S_B} + \Psi_{S_P} = \Omega_{S_O} \quad (3.14)$$

$$\begin{aligned} \mathbf{NVF} &= \iint_{S_B} [(p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS - \\ &\quad \iint_{S_P, CV_1} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \\ &= \iint_{S_O} -[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n} + (p - p_\infty) \hat{n} - \bar{\tau} \cdot \hat{n}] dS \end{aligned} \quad (3.15)$$

At this point it is noted that this generalised *NVF* concept can be applied to portions of airframe-propulsion systems in isolation. For instance, when considering rear-fuselage BLI it is convenient to model the fuselage-propulsion system in isolation to the rest of the airframe (i.e. wings and empennage), yielding a Net Fuselage Force (*NFF*). This is done under the crude assumption that the isolated fuselage-propulsion system is aerodynamically decoupled from the rest of the aircraft. This in turn significantly reduces the complexity of the aircraft modelling in CFD and allows for faster design space explorations.

3.6 Thrust-Force Accounting Systems

Appropriate thrust-force accounting systems for highly integrated airframe-propulsion systems are difficult to define. Airframers and engine manufacturers may choose to adopt different definitions to gain an advantage as to the perceived performance of their product. Some studies [5] try to avoid such definitions by analysing aircraft directly from a power perspective, and claim to mitigate any ambiguity. However, such energy based methods contain inherently a *NVF* term, which is, in effect, a form of thrust-force accounting, and can be obtained using momentum. Nonetheless, both momentum and energy methods have their advantages and disadvantages. In this section, the advantage of the momentum based method is that it allows for thrust-force definitions formulated on more conventional and intuitive concepts. This section is aimed at developing formulations for an array of different accounting options, a summary of which is provided in Table 3.1.

3.6.1 Control Volumes

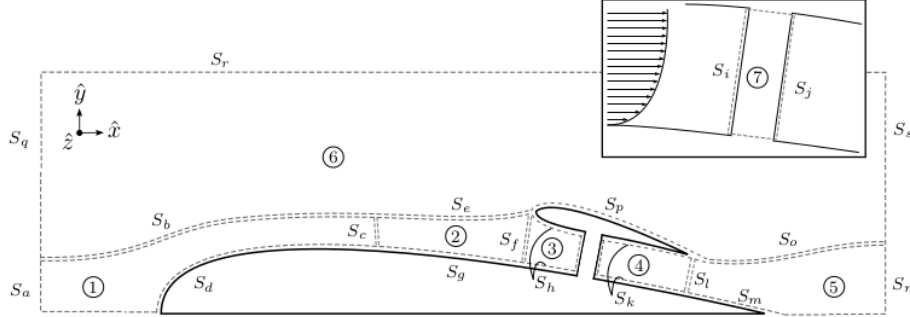


Figure 3.4: Control volume divisions for thrust-force accounting definitions

Figure 3.4 depicts how a control volume encapsulating an aircraft may be split so as to easily define various possible thrust-force accounting systems applicable to BLI aircraft. The approach is based on an axisymmetric body, to which it may be most easily applied. However, it is noted that the approach may be expanded to allow for more complex 3D vehicle designs. The main split occurs along the streamlines separating the *internal* (CVs 1-5,7) versus *external* (CV 6) flow. The internal flow streamtube is considered to be that which enters and exits the propulsor, and the external is that which bypasses over the nacelle. The internal flow is split into upstream (CVs 1-3), propulsor (CV 7) and downstream (CVs 4-5) portions, respectively. The internal streamtube is split according to definitions of the following flux surfaces:

- S_a : positioned sufficiently upstream where the atmosphere flow is not influenced by the aircraft
- S_c : the start of propulsor induced diffusion (the plane where flow conditions begin to change from those of the same geometry without a propulsor present)
- S_f : intake highlight plane (located at the intake lip stagnation point)
- S_i : propulsor inlet plane
- S_j : propulsor exit plane
- S_l : nozzle exit plane
- S_n : jet-wake exit plane where flow has fully expanded to free-stream static pressure

Unbounded streamtube surfaces $S_{b,e,o}$ divide the internal and external flow. Forces acting on these surfaces are applicable when applying different thrust-force accounting definitions. Surfaces $S_{d,g,h,k,m,p}$ are the physical boundaries of the airframe and the bounded

walls of their corresponding streamtubes. The net force acting on these surfaces, combined with that of the propulsor, gives the *NVF*. In relating back as close to convention as possible [36,37], Φ_{S_e} and Φ_{S_o} are the pre-entry and post-exit streamtube forces, respectively. The remaining surfaces of the external flow stream tube are $S_{q,r,s}$. S_q and S_s are situated in a streamwise position correlating to S_a and S_n respectively. S_r is positioned far enough away such that the static pressure is at free-stream conditions, and is defined here as a flux surface, as the velocity may not have returned to free-stream.

The forces on each of the bounded surfaces may be obtained by applying equation Equation (3.9) to each control volume, as given in Equations (3.16a) to (3.16g). The number in the subscript of each term refers to the control volume it is associated with. This is important as it specifies the direction of the normal vector where, by convention, the normal vector points out of a control volume. Therefore adjacent surfaces are equal and opposite (e.g. $\Phi_{S_{b1}} = -\Phi_{S_{b1}}$). The above control volumes, and corresponding momentum equations may be used to define a number of thrust-force accounting systems. These will be introduced and discussed in the sections that follow.

$$\Psi_{S_d} = \Omega_{S_{a1}+S_{c1}} - \Phi_{S_{b1}} \quad (3.16a)$$

$$\Psi_{S_g} = \Omega_{S_{c2}+S_{f2}} - \Phi_{S_{e2}} \quad (3.16b)$$

$$\Psi_{S_h} = \Omega_{S_{f3}+S_{i3}} \quad (3.16c)$$

$$\Psi_{S_k} = \Omega_{S_{j4}+S_{l4}} \quad (3.16d)$$

$$\Psi_{S_m} = \Omega_{S_{l5}+S_{n5}} - \Phi_{S_{o5}} \quad (3.16e)$$

$$\Psi_{S_p} = \Omega_{S_{q6}+S_{r6}+S_{s6}} - \Phi_{S_{b6}+S_{e6}+S_{o6}} \quad (3.16f)$$

$$\Psi_{\text{prop}} = \Omega_{S_{i7}+S_{j7}} \quad (3.16g)$$

Subsequently, the *NVF* is obtained by summing together all of the LHS terms of Equations (3.16a) to (3.16g):

$$\mathbf{NVF} = \Psi_{S_d} + \Psi_{S_g} + \Psi_{S_h} + \Psi_{S_k} + \Psi_{S_m} + \Psi_{S_p} + \Psi_{\text{prop}} \quad (3.17)$$

Re-arranging Equation (3.17), yields the *NVF* in terms of far-field flux surfaces:

$$\mathbf{NVF} = \Omega_{S_a} + \Omega_{S_n} + \Omega_{S_q} + \Omega_{S_r} + \Omega_{S_s} \quad (3.18)$$

In summary the *NVF* may be determined from near-field or far-field control volumes. This is useful, as the combination of the two allows for different definitions of thrust-force accounting to be formulated and calculated from CFD solutions. In addition it provides a means for verifying the *NVF* calculated by checking the consistency between the two integral approaches.

3.6.2 Propulsor Net Thrust

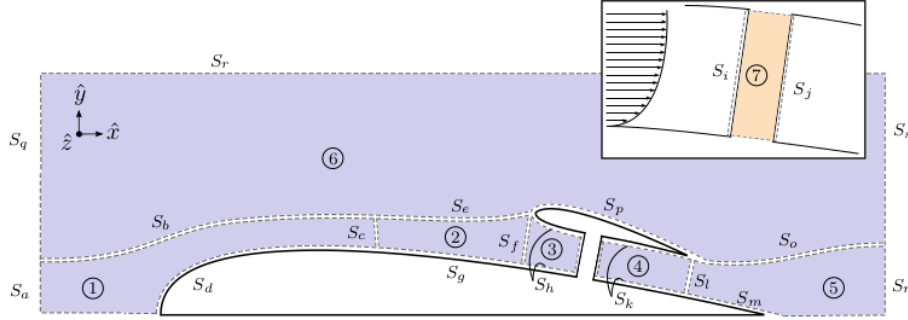


Figure 3.5: Propulsor Net Thrust Control Volumes

The thrust here is defined as the net force in the streamwise direction, x , on all surfaces between the propulsor inlet and exit planes, S_i and S_j , Equation (3.19). The corresponding streamwise NVF , Equation (3.20), is then obtained from Equations (3.16a) to (3.16g), (3.17) and (3.19).

$$F_{N,\text{prop}} = \Omega_{x,S_{i7}+S_{j7}} \quad (3.19)$$

$$NVF_x = F_{N,\text{prop}} + \Psi_{x,S_d} + \Psi_{x,S_g} + \Psi_{x,S_h} + \Psi_{x,S_k} + \Psi_{x,S_m} + \Psi_{x,S_p} \quad (3.20)$$

In this definition all surface forces, apart from the internal workings of the propulsor (i.e. turbomachinery blading etc.) are attributed to the airframe aerodynamics. It may be useful for engine manufacturers in assessing the efficacy of the internal components given a specific installation design. However it may be less favourable from an airframer's perspective as the losses associated with the intake and exhaust ducts detract from the airframe efficiency. From an overall perspective, this thrust-force definition is helpful in examining the pressure force distribution around a tightly integrated airframe-propulsion system.

3.6.3 Intrinsic Net Thrust

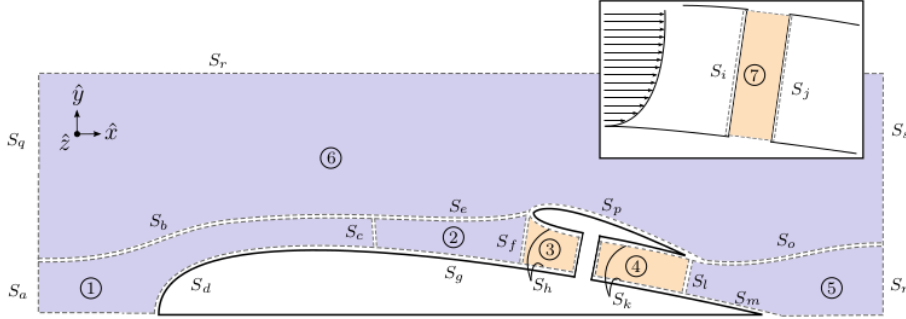


Figure 3.6: Intrinsic Net Thrust Control Volumes

The intrinsic net thrust definition follows the convention outlined in References [36,37], and is the net streamwise force between the intake highlight plane S_f and nozzle exit plane S_l , Equation (3.21). Unlike the previous definition, the losses associated with intake and exhaust ducts are included in the propulsor performance. The corresponding streamwise NVF , Equation (3.22), is then obtained from Equations (3.16a) to (3.16g), (3.17) and (3.21). This definition is useful, in eliminating the substantial pressure force distributions inside the propulsor ducts, thereby allowing for pressure distributions on the airframe and external nacelle surfaces to be examined.

$$F_{N,\text{int}} = \Omega_{x,S_f3+S_l4} \quad (3.21)$$

$$NVF_x = F_{N,\text{int}} + \Psi_{x,S_d} + \Psi_{x,S_g} + \Psi_{x,S_m} + \Psi_{x,S_p} \quad (3.22)$$

3.6.4 Standard Net Thrust

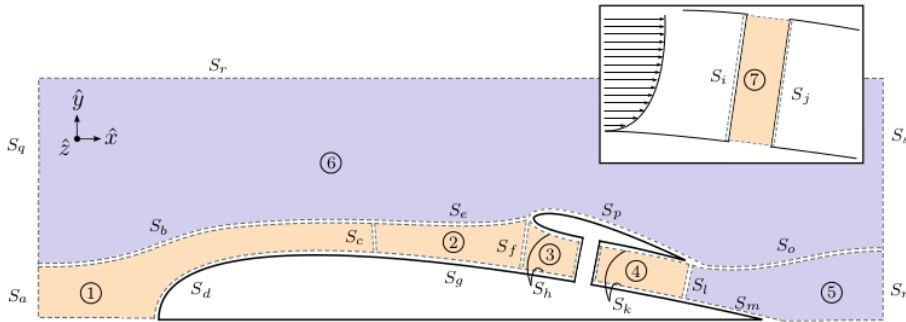


Figure 3.7: Standard Net Thrust Control Volumes

In trying to follow the conventions of References [36,37], the standard net thrust is the net streamwise force between the flow ingested infinitely upstream S_a and nozzle exit plane S_l , Equation (3.23). The corresponding streamwise NVF , Equation (3.24), is then obtained from Equations (3.16a) to (3.16g), (3.17) and (3.23).

$$F_{N,\text{std}} = \Omega_{x,S_{a1}+S_{l4}} \quad (3.23)$$

$$NVF_x = F_{N,\text{std}} + \Psi_{x,S_m} + \Psi_{x,S_p} - \Phi_{x,S_{b1}} - \Phi_{x,S_{e2}} \quad (3.24)$$

As a result of this definition, the pre-airframe streamtube force $\Phi_{x,S_{b1}}$ and pre-entry streamtube force $\Phi_{x,S_{e2}}$, are required. This definition is unfavourable to the engine manufacturer, as all the losses associated with the airframe scrubbed by the upstream internal flow are accounted for on the propulsion performance. In addition, the increased static pressure of the jet acting on surface S_m belongs to the airframe and, having a rearward facing area, has a large contribution in the upstream direction. This gives a negative perceived engine performance and a positive airframe performance. This gives illustration as to why this conventional thrust-force accounting definition is inappropriate for tightly integrated systems.

3.6.5 Overall Net Thrust

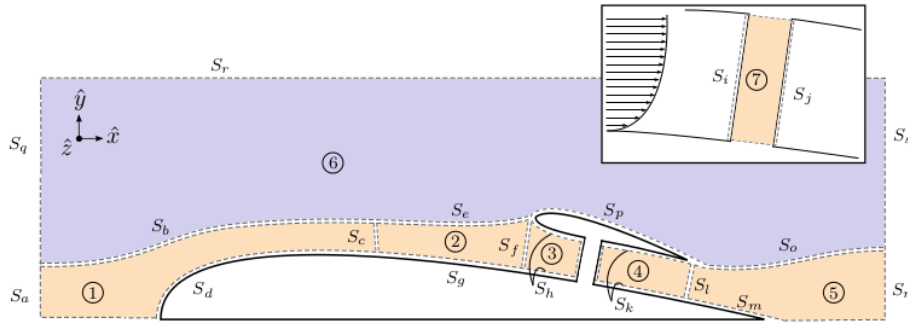


Figure 3.8: Overall Net Thrust Control Volumes

Still following the convention of Reference [36,37], the overall net thrust is the net streamwise force between the internal flow infinitely upstream S_A to a position downstream (S_n) where the static pressure has returned to free-stream conditions, Equation (3.25). The corresponding streamwise NVF , Equation (3.26), is then obtained from Equations (3.16a) to (3.16g), (3.17) and (3.25).

$$F_{N,\text{overall}} = \Omega_{x,S_{a1}+S_{n5}} \quad (3.25)$$

$$NVF_x = F_{N,\text{overall}} + \Psi_{x,S_p} - \Phi_{x,S_{b1}} - \Phi_{x,S_{e2}} - \Phi_{x,S_{o5}} \quad (3.26)$$

This definition must now also incorporate the post-exit streamtube force $x_{S_{o5}}$ and passes further downstream airframe pressure losses onto the engine performance accounting. However, the forward acting pressure forces over S_m are now included on the engine accounting.

3.6.6 Pre-Diffusion Net Thrust

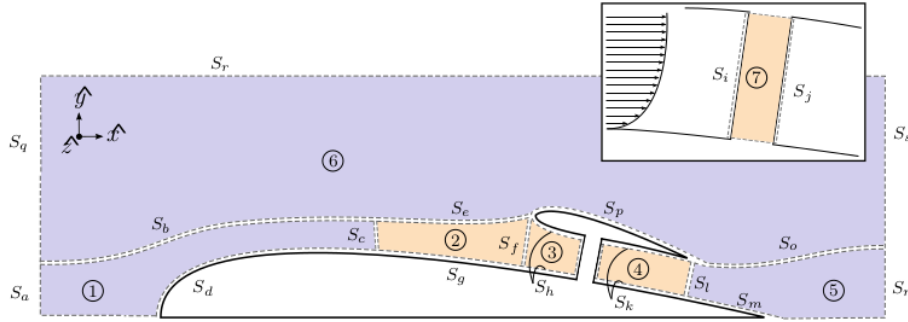


Figure 3.9: Pre-Diffusion Net Thrust Control Volumes

Perhaps a more fair division of force would be that based on a pre-diffusion net thrust, as it divides the upstream internal flow into a portion only influenced by the airframe (between S_a and S_c) from the portion influenced by both the airframe and propulsor (between S_c and S_f). The pre-diffusion net thrust is defined between flux surfaces S_c and S_l , Equation (3.27). The corresponding streamwise NVF , Equation (3.28), is then obtained from Equations (3.16a) to (3.16g), (3.17) and (3.27).

$$F_{N, \text{pre-diff net}} = x_{S_{c2}+S_{l4}} \quad (3.27)$$

$$NVF_x = F_{N, \text{pre-diff net}} + x_{S_d} + x_{S_m} + x_{S_p} - x_{S_{e2}} \quad (3.28)$$

The difficulty associated with this definition is that the location of S_C will continuously shift position significantly throughout the mission as thrust requirements change along with mass flow requirements. Despite the moving location of S_c , this definition is a more fair division as the propulsion system incurs the losses associated with MFR, which is a result of its sizing and off-design operation.

3.6.7 Pre-Di usion Overall Thrust

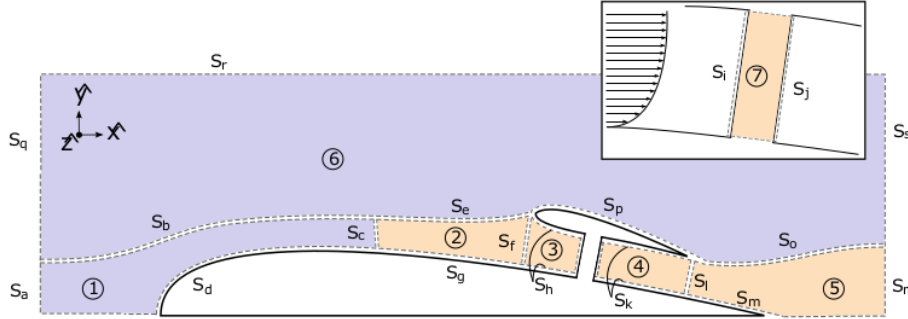


Figure 3.10: Pre-Di usion Overall Thrust Control Volumes

Another argument may be to incorporate the downstream airframe surfaces scrubbed by the propulsor jet, within the engine performance accounting. Depending on the configuration, these forces may have a positive or negative effect on the perceived performance of engine and airframe surfaces. The pre-di usion overall net thrust is defined between flux surfaces S_c and S_n , Equation (3.29) and incorporates the post-exit streamtube force $x_{;S_o}$. The corresponding streamwise NVF, Equation (3.30), is then obtained from Equations (3.16a) to (3.16g), (3.17) and (3.29).

$$F_{N,pre-di \text{ overall}} = x_{;S_c} + x_{;S_n} \quad (3.29)$$

$$NVF_x = F_{N,pre-di \text{ overall}} + x_{;S_d} + x_{;S_p} - x_{;S_e} - x_{;S_o} \quad (3.30)$$

This definition is more natural division using the pre-di usion plane S_c , because it included the surface S_m which will in all likelihood be a thrust surface scrubbed by the jet. However this does depend on how far back the propulsor is placed along the tail-cone. Too far forward, and it may be argued that the tail-cone is more of an airframe surface. If placed right at the back S_m may be viewed as the propulsor's plug and inherent of the thrust produced.

3.6.8 Reference Net Vehicle Force

Another useful perspective of thrust-force accounting is based on a comparison between the NVF of a baseline airframe in isolation, versus one within an integrated propulsion system. This definition is based on the idea that the force on the airframe remains constant and that any changes in this force, due to propulsion system integration, are attributed to thrust. This includes any reshaping of the airframe to accommodate the propulsor, as well as additional propulsor surfaces (such as a nacelle). The thrust is therefore defined in reference to a baseline NVF, with a corresponding force equal to the baseline NVF.

$$F_{N,ref} = NVF - NVF_{baseline} \quad (3.31)$$

The advantage here is that there is a constant basis for comparison of different integration configurations. However, it has limitations in that it may be difficult to define an airframe shape without the propulsor for unconventional airframe designs. Nevertheless, for rear-fuselage integration it works well as the baseline fuselage shape is easily defined.

3.6.9 A Note on Thrust-Drag Accounting

Based on Prandtl's extension to d'Alembert's Paradox, drag may be defined as the difference between the force in real viscous flow versus the force in potential flow [36,37]. This is because the sum of the forces, over the components comprising a body, equals zero in potential flow. The potential flow force on each component is referred to as its buoyancy force. Thus, each component's contribution to drag is given by its force in real viscous flow minus its buoyancy force obtained from potential flow. However, calculating the potential flow forces in a way in which streamtube wall shapes and fluxes are equivalent, is not possible. Assumption therefore need to be made, and the actual buoyancy force can only be approximated. For this reason force, rather than drag, is used in this work as it is more rigorous and avoids unnecessary assumptions, complication and ambiguity.

3.7 Propulsor Power and BLI Performance Evaluation

The previous sections have focused around force evaluations of the vehicle. However, the interest behind BLI configurations is to establish potential improvements in how efficiently the NVF is achieved. To do so, it is first necessary to look at how the shaft power of the BLI propulsor may be obtained from a control volume analysis. Secondly, it is helpful to establish a parameter of comparison to evaluate how effectively the shaft power is used in propelling the aircraft.

3.7.1 Propulsor Shaft Power

In order to assess how effectively forces are generated by a propulsor in CFD and experimental studies, the power required to drive the propulsor must be determined from control volume fluxes. The energy equation in differential conservation form is presented in Equation (3.32). The integral form for an arbitrary closed control volume in steady flow, Equation (3.33), is obtained by taking the volume integral and applying the divergence theorem where appropriate.

$$\frac{\partial}{\partial t} \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \right] + \nabla \cdot \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} \right] = \rho \dot{q} - \nabla \cdot \mathbf{q}_c - \nabla \cdot (p \mathbf{V}) + \nabla \cdot (\bar{\tau} \cdot \mathbf{V}) + \rho \mathbf{f} \cdot \mathbf{V} \quad (3.32)$$

$$\oint_S \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} \right] \cdot d\mathbf{S} = \iiint_V [\rho (\dot{q} + \mathbf{f} \cdot \mathbf{V})] dV + \oint_S [-\mathbf{q}_c - (p \mathbf{V}) + (\bar{\tau} \cdot \mathbf{V})] \cdot d\mathbf{S} \quad (3.33)$$

Continuity, Equation (3.4), allows for the convenient use of reference energy terms on the LHS of Equation (3.33), where the following substitutions may be made².

$$e \rightarrow e - e_\infty, \quad \mathbf{V}^2 \rightarrow \mathbf{V}^2 - \mathbf{V}_\infty^2$$

Incorporating the guage pressure substitution, yields Equation (3.34).

$$\begin{aligned} \oint_S \left\{ \rho \left[(e - e_\infty) + \frac{(\mathbf{V}^2 - \mathbf{V}_\infty^2)}{2} \right] \mathbf{V} \right\} \cdot d\mathbf{S} = \oint_S [-\mathbf{q}_c - (p - p_\infty) \mathbf{V} + \bar{\tau} \cdot \mathbf{V}] \cdot d\mathbf{S} + \\ \iiint_V [\rho(\dot{q} + \mathbf{f} \cdot \mathbf{V})] dV - p_\infty \oint_S \mathbf{V} \cdot d\mathbf{S} \end{aligned} \quad (3.34)$$

Referring to Figure 3.2, the work done on the propulsor may be determined assuming the absence of body force, volumetric heating and heat conduction (i.e. $\mathbf{f} = 0$, $\dot{q} = 0$, $\mathbf{q}_c = 0$). Equations (3.33) and (3.34) may be applied to the surfaces S_F and S_P of Figure 3.2. The propulsor blades move relative to the control volume's surface S_F , and the no-slip condition dictates a non-zero velocity vector tangential to S_F . Flux values are zero through S_F and so is pressure work, because $\mathbf{V} \cdot \hat{n} = 0$, but shear stress work is not as the tensor product $(\bar{\tau} \cdot \mathbf{V}) \cdot \hat{n} \neq 0$. The work done on the control volume by the propulsor is given by Equation (3.35), where it is noted that the RHS is the net total enthalpy flux across the propulsor control volume (assuming shear stress work on S_P to be negligible).

$$\begin{aligned} \dot{W}_{CV} = \iint_{S_F} (\bar{\tau} \cdot \mathbf{V}) \cdot d\mathbf{S} = \iint_{S_P} \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} + p\mathbf{V} - \bar{\tau} \cdot \mathbf{V} \right] \cdot d\mathbf{S} \\ = \iint_{S_P} \left\{ \rho \left[(e - e_\infty) + \frac{(\mathbf{V}^2 - \mathbf{V}_\infty^2)}{2} \right] \mathbf{V} + p\mathbf{V} - \bar{\tau} \cdot \mathbf{V} \right\} \cdot d\mathbf{S} \end{aligned} \quad (3.35)$$

The above is the total power seen by the flow via the control volume. In determining the shaft power, the mechanical efficiency of the system must be taken into account. Incorporating a mechanical efficiency, the shaft power required to drive the BLI propulsor becomes:

$$\dot{W}_{\text{shaft}} = \frac{\dot{W}_{CV}}{\eta_{\text{mech}}} \quad (3.36)$$

In the context of this work, η_{mech} incorporates losses in the BLI propulsor (i.e. form bearings and friction etc.) as well as the transmission efficiency. This is because this work assumes the use of a mechanical transmission system to power BLI fans from podded turbofan main engines. Therefore, the shaft work is of primary interest, as it accounts for all the losses involved in the BLI propulsor's ability to propel the configuration.

² $_{const} \times \oint_S (\rho \mathbf{V}) \cdot d\mathbf{S} = 0$

3.7.2 Propulsion Efficacy

Propulsive efficiency is used in evaluating the efficacy of conventional engines. Definitions of conventional propulsive efficiency have already been covered in Section 2.2, and how the definition breaks down for BLI configurations because the efficiency may exceed unity. To circumvent this problem, this work modifies the definition of propulsive efficiency and bases it on the Reference Net Vehicle Force. Unfortunately NVF cannot be used as it is impractical, because its associated flight power ($NVF \cdot \mathbf{V}_\infty$) may be zero (for steady level flight). Therefore in establishing an alternative definition, a performance metric akin to TSFC may be defined:

$$TSPC = \frac{\dot{W}_{CV}}{-F_{N,ref}} \quad (3.37)$$

This metric gives an indication of how efficiently the shaft power (after mechanical efficiency losses) has been used in producing a forward force relative to a reference. The negative sign in the denominator is included, because a forward force acts in the opposite direction to the coordinate system definition, and it is more intuitive to have a metric with positive sign. This is a particularly useful definition for evaluating the efficacy of a retrofit type approach to incorporating BLI propulsion on an aircraft. In the context of this work, the retrofit type approach takes an existing fuselage geometry and modifies it to incorporate a BLI propulsor. In that sense, $F_{N,ref}$ describes how the force of the configuration (or fuselage in the case of this work) has been altered through the introduction of a BLI propulsor. $F_{N,ref}$ will always be in the thrust direction for a reasonable propulsor, and if negative it is one of no interest and may be discarded from the analysis. Additionally, the TSPC becomes useful in decoupling the performance evaluations of BLI propulsors from podded main engine performance, in thrust split configurations. An alternative way of expressing TSPC, is to non-dimensionalise it based on the flight velocity. This forms a definition for propulsive efficiency:

$$\eta_{prop} = \frac{-F_{N,ref} \cdot \mathbf{V}_\infty}{\dot{W}_{CV}} \quad (3.38)$$

This definition does not exceed unity, as it is highly unlikely that the propulsor will effect a change in force (and hence change in flight power) from the reference that exceeds the work put into the flow. This would only occur in the instance where the reference is poorly defined.

3.8 Summary

This chapter has described in detail a control volume approach for examining forces on tightly integrated airframe-propulsion systems. The concept of NVF has been introduced, which is the total net force vector acting on a propulsion airframe configuration. Near-field and far-field derivations of the NVF were derived from first principles, using continuity and momentum equations applied to an encapsulating control volume. A

generalised application of momentum to arbitrary streamtubes and propulsor control volumes, allowed for the development of a standard method for deriving customisable thrust-force accounting systems. This method was used to develop a number of options applicable to BLI configurations, which were discussed with reference to conventional standards. Finally, a generalised formula for the power of a BLI propulsor was derived, as well as a modified propulsive efficiency, to allow for performance evaluations of different BLI configurations. This power based evaluation is reliant on a reference body to which a BLI propulsor may be fitted in a retrofit type approach.

3.8.1 Concepts and Formulations

The near-field, far-field and thrust-force accounting definitions discussed in this chapter are summarised in Table 3.1, with reference to Figure 3.4 and the following equation:

$$NVF = Thrust + Force$$

Table 3.1: Summary of Thrust-Force Accounting Definitions

Definition	Thrust	Force
$NVF(\text{near-field})$	0	$\Psi_{x,S_d} + \Psi_{x,S_g} + \Psi_{x,S_h} + \Psi_{x,S_k} + \Psi_{x,S_m} + \Psi_{x,S_p} + \Psi_{x,\text{prop}}$
$NVF(\text{far-field})$	0	$\Omega_{x,S_a} + \Omega_{x,S_n} + \Omega_{x,S_q} + \Omega_{x,S_r} + \Omega_{x,S_s}$
Propulsor Net Thrust	$\Omega_{x,S_{i7}+S_{j7}}$	$\Psi_{x,S_d} + \Psi_{x,S_g} + \Psi_{x,S_h} + \Psi_{x,S_k} + \Psi_{x,S_m} + \Psi_{x,S_p}$
Intrinsic Net Thrust	$\Omega_{x,S_{f3}+S_{l4}}$	$\Psi_{x,S_d} + \Psi_{x,S_g} + \Psi_{x,S_m} + \Psi_{x,S_p}$
Standard Net Thrust	$\Omega_{x,S_{a1}+S_{l4}}$	$\Psi_{x,S_m} + \Psi_{x,S_p} - \Phi_{x,S_{b1}} - \Phi_{x,S_{e2}}$
Overall Net Thrust	$\Omega_{x,S_{a1}+S_{n5}}$	$\Psi_{x,S_p} - \Phi_{x,S_{b1}} - \Phi_{x,S_{e2}} - \Phi_{x,S_{o5}}$
Pre-Diffusion Net Thrust	$\Omega_{x,S_{c2}+S_{l4}}$	$\Psi_{x,S_d} + \Psi_{x,S_m} + \Psi_{x,S_p} - \Phi_{x,S_{e2}}$
Pre-Diffusion Overall Thrust	$\Omega_{x,S_{c2}+S_{n5}}$	$\Psi_{x,S_d} + \Psi_{x,S_p} - \Phi_{x,S_{e2}} - \Phi_{x,S_{o5}}$
Ref. Net Vehicle Thrust	$NVF - NVF_{\text{baseline}}$	NVF_{baseline}

The shaft power driving the BLI propulsor takes into account a mechanical efficiency (inclusive of a transmission efficiency), and is given as:

$$\dot{W}_{\text{shaft}} = \frac{\dot{W}_{CV}}{\eta_{\text{mech}}} \quad (3.36)$$

which is based on the power seen by the control volume flow:

$$\begin{aligned}\dot{W}_{CV} &= \iint_{S_F} (\bar{\tau} \cdot \mathbf{V}) \cdot d\mathbf{S} = \iint_{S_P} \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} + p\mathbf{V} - \bar{\tau} \cdot \mathbf{V} \right] \cdot d\mathbf{S} \\ &= \iint_{S_P} \left\{ \rho \left[(e - e_\infty) + \frac{(\mathbf{V}^2 - \mathbf{V}_\infty^2)}{2} \right] \mathbf{V} + p\mathbf{V} - \bar{\tau} \cdot \mathbf{V} \right\} \cdot d\mathbf{S}\end{aligned}\quad (3.35)$$

The efficacy of the BLI propulsion is evaluated using a Thrust Specific Power Consumption:

$$TSPC = \frac{\dot{W}_{CV}}{-F_{N,ref}} \quad (3.37)$$

or alternatively the performance may be expressed as a non-dimensional metric, in the form of a modified propulsive efficiency

$$\eta_{prop} = \frac{-F_{N,ref} \cdot \mathbf{V}_\infty}{\dot{W}_{CV}} \quad (3.38)$$

3.8.2 Contributions

The contributions in this chapter are summarised as the following:

- A standardised method for deriving customizable thrust-force accounting systems from first principles, and generalised applicability to any aircraft-propulsion integration.
- A definition of net-vehicle-force and a set of thrust-force accounting definitions options, applicable to tightly integrated airframe-propulsion systems. To the authors knowledge, this approach to thrust-force accounting is unique to this work and has not been properly addressed in the available literature.
- A practical definition of propulsive efficiency in the context of this work, which allows for decoupled analysis of main engine cycle and BLI propulsor performance for retrofit thrust-split BLI configurations.
- A control volume analysis which allows for the inclusion of a propulsor in the CFD fluid domain using a combination of inlet and outlet boundary conditions.

3.8.3 Future Work

From Thrust-Force Accounting to Thrust-Drag Accounting

At this stage of the work, the concept of "drag" was neglected. Definitions of drag are based on Prandtl's extension of D'Alembert's paradox [36, 37]. This definition requires the removal of the potential flow pressure forces from the real viscous force, when calculating drag contributions over a partial body. The author was unable to find a way

to do this, that is physically and mathematically rigorous, or indeed practical. This is because (at least how it seems to the author) the elimination of terms is reliant on the assumption that streamtube geometries are equivalent in potential and viscous flow. It also relies on the additional assumption that unbounded streamtube surfaces do not experience any viscous force due to shear. It would be interesting to see if there is a possible way to formulate drag, according to the aforementioned definition, without having to make these assumptions. This would lead to better definitions of thrust-drag accounting, which allow for improved evaluation of drag contributions by different airframe and propulsion system surfaces.

Performance Metrics

One of the primary difficulties in BLI performance evaluation, is finding an alternative definition for propulsive efficiency from the conventional definition. The conventional propulsive efficiency is dependent on a thrust-drag accounting system which is no longer applicable to BLI configurations. This work presented a modified propulsive efficiency reliant on the reference net thrust, which works well in the context of retrofitting an existing airframe design. However, more important for future analysis is an efficiency metric which can be used when starting a new vehicle design which does not have a baseline for reference. The author was unable to create such a definition, but ideally one is required which describes a vehicles overall propulsive efficiency. This is challenging because flight power of the NVF is zero in steady level flight. In summary, a vehicle efficiency metric is required.

Chapter 4

Energy-Based Approach to Evaluating Performance

The previous chapter looked at evaluating an aircraft's performance from momentum-based net-vehicle-force formulations and thrust-force accounting divisions. One of the reasons for looking into different thrust-force accounting systems is to breakdown the contributions to NVF , and attribute these contributions to specific components of the airframe and nacelle. This helps to identify areas which may be improved through design. However, the thrust-force accounting definitions are limiting in the insight they provide into sources of losses and inefficiencies associated with the aircraft's aerodynamics. To better understand contributions to NVF , it is supplementary to decompose the flow-field into different types of flow mechanisms and examine how they influence the NVF . By analysing the energy content and dissipative properties of different flow mechanisms, the aerodynamic behaviour of tightly integrated systems may be better understood and more strategic design approaches taken.

Some approaches [7, 25–28] make use of entropy and enthalpy terms, initially derived from the momentum equation, to analyse aerodynamic energy sinks and sources, respectively. Alternative approaches [5, 6] have looked at more direct means of applying the energy equation to the flow-fields surrounding the aircraft, and formulate an energy balance of sources and sinks to evaluate aircraft efficacy. In this chapter, a unique spin on the latter approach is taken to derive the NVF directly from the energy equation, making it the subject of the formulation (i.e. the sole term on the LHS of the equation). This allows for full decomposition of the NVF into energy contributions, which may be spatially applied to various flow mechanisms to better understand BLI propulsion.

This chapter begins with generalised formulations of the kinetic energy equation, which is transformed into the absolute, instead of the relative, reference frame. This approach and method of derivation is unique to this work, and provides improved insight into energy terms which may sometimes seem counter-intuitive. The kinetic energy equation is then placed in the context of the total energy equation, to further generalise formulations to include terms associated with heating and internal energy. Following a similar procedure to the previous chapter, the energy formulations are applied to

arbitrary streamtube and propulsor control volumes, the outcomes of which are used to derive the NVF for non-propelled and propelled bodies. Finally, a discussion follows on spatial decomposition of the aircraft's flow-field to examine energy content of different flow mechanisms.

The contributions of this chapter are as follows:

- Formulation of the kinetic and total energy equations in the absolute reference frame. The derivation is unique to this work, and having the energy equation in the absolute reference frame provides an intuitive perspective into the physics behind the mathematical terms. This work directly addresses concerns of Renard and Deck [4] about Drela's [5] formulation in the aircraft's body axes system. In doing so, it accounts for pressure and friction forces on solid surfaces, as well as the kinetic energy imparted to the fluid (see Section 4.1.1 for more details).
- The derivation in this work is generalised, and interpretation of the physics behind mathematical terms is not control volume definition dependent, as is the case in References [5, 6]. Furthermore, the generalised formulation makes it more easily applicable to other orthogonal curvilinear coordinate systems, and not just limited to a Cartesian system.
- The equations are formulated with a free-stream velocity vector for further generalisation. This allows for different axes systems to be used, other than the aircraft's body axes (as is the case in References [5, 6]). This is also an important step taken in deriving the transformation to the absolute reference frame.
- The formulation is complete, and does not neglect any viscous terms.
- The formulation is arranged with NVF as the subject, which sheds light on the contributions of various energy terms. Other energy formulations [5, 6] take a *energy source = energy sink* format, which makes it difficult to understand the contributions of some terms.
- The internal energy is incorporated, unlike Drela's method [5]. This provides a convenient alternative form, which allows for volume integrals of dissipative terms to be calculated as surface integrals of net changes in internal energy. This is particularly advantageous when incorporating propulsors into numerical flow domains, without having to include its own flow domain too (i.e. representing it as a set of inlet and outlet boundary conditions). Another advantage is that it provides another form to calculate the NVF , and thereby is a further source of formulation verification.
- A procedure of spatial decomposition by Paparone and Tognaccini [7] into different flow mechanisms has been adopted, modified and improved from formulations on entropy drag, and incorporated into the energy approach of this work. This decomposition allows for volume integrals describing energy dissipation, to be quantified

for different flow regions (e.g. shock, jet and BLs). A BL and viscous wake region selector, modified from Paparone and Tognaccini's work [7], is a significant contribution presented in a later Chapter (see Section 5.1).

4.1 Generalised Energy Equations Applicable to Arbitrary Control Volumes

This section presents two concepts. First the kinetic energy equation is presented in generalised form, and transformed into the absolute reference frame. Subsequently, the total energy equation is incorporated to bring out an expression for internal energy, which allows for substitution of some volume integrals present in the kinetic energy equation form.

4.1.1 Kinetic Energy Equation

Assuming steady conditions, Drela [5] obtains an expression for kinetic energy from the dot product between the momentum equation (in differential conservation form) and the velocity vector. In doing so, the tensor identities Equations (4.2) and (4.3) are substituted into the dot product equation Equation (4.1) to yield Equation (4.4).

$$\nabla \cdot \left(\rho \frac{\mathbf{V}^2}{2} \mathbf{V} \right) = -\nabla p \cdot \mathbf{V} + (\nabla \cdot \bar{\tau}) \cdot \mathbf{V} + \rho \mathbf{f} \cdot \mathbf{V} \quad (4.1)$$

$$\nabla \cdot (p \mathbf{V}) = \nabla p \cdot \mathbf{V} + p \nabla \cdot \mathbf{V} \quad (4.2)$$

$$\nabla \cdot (\bar{\tau} \cdot \mathbf{V}) = (\nabla \cdot \bar{\tau}) \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V} \quad (4.3)$$

$$\nabla \cdot \left(\rho \frac{\mathbf{V}^2}{2} \mathbf{V} \right) = -\nabla \cdot (p \mathbf{V}) + p \nabla \cdot \mathbf{V} + \nabla \cdot (\bar{\tau} \cdot \mathbf{V}) - (\bar{\tau} \cdot \nabla) \cdot \mathbf{V} + \rho \mathbf{f} \cdot \mathbf{V} \quad (4.4)$$

The integral form of the kinetic energy equation, Equation (4.5), is obtained from taking the volume integral of Equation (4.4), applied to an arbitrary closed control volume, and using the divergence theorem where appropriate.

$$\oint_S \left(\rho \frac{\mathbf{V}^2}{2} \mathbf{V} \right) \cdot d\mathbf{S} = \oint_S (-p \mathbf{V} + \bar{\tau} \cdot \mathbf{V}) \cdot d\mathbf{S} + \iiint_V [(p \nabla \cdot \mathbf{V}) - (\bar{\tau} \cdot \nabla) \cdot \mathbf{V} + (\rho \mathbf{f}) \cdot \mathbf{V}] dV \quad (4.5)$$

Following on from the discussion on relative versus absolute reference frames (see Section 3.1), Equation (4.5) has been formulated in the relative reference frame, where the control volume is stationary and the flow passes through it. Remaining in the relative reference frame, a substitution for the difference in local mass specific kinetic energy (from the free-stream reference can be made), as well as a substitution for the static pressure to account for buoyancy¹, to give Equation (4.6)

$$\frac{\mathbf{V}^2}{2} \rightarrow \frac{\mathbf{V}^2}{2} - \frac{V_\infty^2}{2}, \quad p \rightarrow p - p_\infty$$

¹because $\left(\frac{V_\infty^2}{2} \oint_S (\rho \mathbf{V}) \cdot d\mathbf{S} = 0 \right)$ and $(p_\infty \iiint_V (\nabla \cdot \mathbf{V}) dV = p_\infty \oint_S \mathbf{V} \cdot d\mathbf{S})$

$$\oint_S \left[\frac{\rho}{2} (\mathbf{V}^2 - \mathbf{V}_\infty^2) \mathbf{V} \right] \cdot d\mathbf{S} = \oint_S [-(p - p_\infty) \mathbf{V} + \bar{\boldsymbol{\tau}} \cdot \mathbf{V}] \cdot d\mathbf{S} + \iiint_V [(p - p_\infty) \nabla \cdot \mathbf{V} - (\bar{\boldsymbol{\tau}} \cdot \nabla) \cdot \mathbf{V} + (\rho \mathbf{f}) \cdot \mathbf{V}] dV \quad (4.6)$$

Equation (4.6) is still in the relative reference frame when applying it to a control volume around an aircraft, and physical descriptions of its terms are not that intuitive. This is where the generalised formulation by Drela [5] stops and is applied to specific control volume definition to derive out a power balance. It is at this point that this work uniquely deviates from Drela's approach, and takes the direction of formulating a generalised kinetic energy equation in the absolute frame of reference. This is done because intuitive physical interpretations of each of the mathematical terms become more apparent when applying the above equation in different scenarios.

A transformation of Equation (4.6) may be found by considering the power associated with the momentum equation terms acting against the free-stream velocity vector. This is done by subtracting the dot product between Equation (3.7) and free-stream velocity vector $\mathbf{V}_\infty$ from Equation (4.6). In taking the dot product between Equation (3.7) and free-stream velocity vector, $\mathbf{V}_\infty$ may be brought within the surface and volume integrals as it is a constant vector field:

$$\oint_S [(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n}] \cdot \mathbf{V}_\infty dS = \oint_S [-(p - p_\infty) \hat{n} \cdot \mathbf{V}_\infty + (\bar{\boldsymbol{\tau}} \cdot \hat{n}) \cdot \mathbf{V}_\infty] dS + \iiint_V \rho \mathbf{f} \cdot \mathbf{V}_\infty dV \quad (4.7)$$

In finding a more convenient form for the LHS (i.e. $[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n}] \cdot \mathbf{V}_\infty = \frac{\rho}{2} [\text{scalar}] \mathbf{V} \cdot \hat{n}$), it is expanded in terms of the vector components in Cartesian coordinates (although it is noted that the same result will follow for any valid orthogonal curvilinear coordinate

system), which may then be arranged more conveniently:

$$\begin{aligned}
& [(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n}] \cdot \mathbf{V}_\infty \\
&= \rho \begin{bmatrix} (u - u_\infty) u u_\infty n_x + (u - u_\infty) v u_\infty n_y + (u - u_\infty) w u_\infty n_z & + \dots \\ (v - v_\infty) u v_\infty n_x + (v - v_\infty) v v_\infty n_y + (v - v_\infty) w v_\infty n_z & + \dots \\ (w - w_\infty) u w_\infty n_x + (w - w_\infty) v w_\infty n_y + (w - w_\infty) w w_\infty n_z & \end{bmatrix} \\
&= \rho \begin{bmatrix} (u - u_\infty) u u_\infty + (v - v_\infty) u v_\infty + (w - w_\infty) u w_\infty \\ (u - u_\infty) v u_\infty + (v - v_\infty) v v_\infty + (w - w_\infty) v w_\infty \\ (u - u_\infty) w u_\infty + (v - v_\infty) w v_\infty + (w - w_\infty) w w_\infty \end{bmatrix} \cdot \hat{n} \\
&= \rho [(u - u_\infty) u_\infty + (v - v_\infty) v_\infty + (w - w_\infty) w_\infty] \mathbf{V} \cdot \hat{n}
\end{aligned}$$

The above bracketed scalar can be rewritten in the form of generalised vector quantity magnitudes, by expanding the mass specific kinetic energy in the absolute reference frame:

$$\begin{aligned}
(\mathbf{V} - \mathbf{V}_\infty)^2 &= (u - u_\infty)^2 + (v - v_\infty)^2 + (w - w_\infty)^2 \\
&= (u^2 + v^2 + w^2) + (u_\infty^2 + v_\infty^2 + w_\infty^2) - 2(uu_\infty + vv_\infty + ww_\infty) \\
&= \mathbf{V}^2 + \mathbf{V}_\infty^2 - 2(uu_\infty + vv_\infty + ww_\infty)
\end{aligned}$$

Using the above, the expression for the bracketed scalar quantity becomes:

$$[(u - u_\infty) u_\infty + (v - v_\infty) v_\infty + (w - w_\infty) w_\infty] = \frac{\mathbf{V}^2 - \mathbf{V}_\infty^2 - (\mathbf{V} - \mathbf{V}_\infty)^2}{2}$$

And therefore the following substitution may be made into Equation (4.7):

$$[(\mathbf{V} - \mathbf{V}_\infty) \rho \mathbf{V} \cdot \hat{n}] \cdot \mathbf{V}_\infty = \frac{\rho}{2} [\mathbf{V}^2 - \mathbf{V}_\infty^2 - (\mathbf{V} - \mathbf{V}_\infty)^2] \mathbf{V} \cdot \hat{n} \quad (4.8)$$

In addition it can be shown that:

$$\oint_S (p - p_\infty) \hat{n} \cdot \mathbf{V}_\infty dS = \oint_S (p - p_\infty) \mathbf{V}_\infty \cdot d\mathbf{S} \quad (4.9)$$

and because the $\bar{\tau}$ is assumed to be symmetrical² second order tensor:

$$\oint_S (\bar{\tau} \cdot \hat{n}) \cdot \mathbf{V}_\infty dS = \oint_S (\bar{\tau} \cdot \mathbf{V}_\infty) \cdot d\mathbf{S} \quad (4.10)$$

²in reality it is not, but asymmetrical stress tensors are very difficult to model, and the hydrostatic assumption has to be made for RANS numerical calculations. Therefore this assumption is valid in the context of this work.

Substituting Equations (4.8) to (4.10) into Equation (4.7) gives an expression, Equation (4.11), for the difference between the kinetic energy deficit in a relative reference frame versus the kinetic energy imparted to the flow in the absolute reference frame, respectively.

$$\begin{aligned} \oint_S \frac{\rho}{2} [(\mathbf{v}^2 - \mathbf{v}_\infty^2) - (\mathbf{V} - \mathbf{V}_\infty)^2] \mathbf{v} \cdot d\mathbf{S} = \\ \oint_S [-(p - p_\infty) \mathbf{V}_\infty + (\bar{\tau} \cdot \mathbf{V}_\infty)] \cdot d\mathbf{S} + \iiint_V \rho \mathbf{f} \cdot \mathbf{V}_\infty d\nu \end{aligned} \quad (4.11)$$

Equation (4.12) is obtained by subtracting Equation (4.11) from Equation (4.6), and is an expression for the kinetic energy imparted to the flow in the absolute reference frame where the control volume is moving through an initially stationary fluid.

$$\begin{aligned} \oint_S \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \right] \cdot d\mathbf{S} = \oint_S [-(p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) + \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot d\mathbf{S} + \\ \iiint_V [(p - p_\infty) \nabla \cdot \mathbf{V} - (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] d\nu + \\ \iiint_V [\rho \mathbf{f} \cdot (\mathbf{V} - \mathbf{V}_\infty)] d\nu \end{aligned} \quad (4.12)$$

Equation (4.12) is a significant contribution of this work, and to the author's knowledge it is a unique generalised formulation in the absolute reference frame. The advantage of this form is the immediate intuitiveness of the physical interpretation behind the mathematical terms, without having to apply it to a specific control volume definition. Renard and Deck [4] raise two main concerns with direct reference to Drela's Power Balance method [5], which is formulated in the aircraft's body reference frame. The first is that the power related to skin friction coefficient is zero in the relative reference frame, which is evident from the first RHS term of Equation (4.6). This is because the relative velocity vector is zero given the no-slip condition on walls. This argument can also be extended to the pressure coefficient (or rather forces) acting on physical surfaces. The second concern raised, is that regarding the concepts of kinetic energy excess and defect introduced by Drela in an attempt to interpret the LHS term of Equation (4.6). The LHS term of Equation (4.6) is not intuitive, and it is difficult to understand what this kinetic energy term represents.

Equation (4.12) directly addresses both of these concerns, and this is a significant contribution of this work. Firstly, both skin friction and pressure integrands of the first RHS integral in Equation (4.12) are clearly non-zero for a non-zero flight speed. At a wall, the relative velocity vector is zero and the integral reduces to:

$$\iint_{S_w} [(p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S}$$

This is evidently the work rate of the viscous and pressure forces acting on the body at the flight velocity. To better explain this, imagine a force isolated from the flow-field

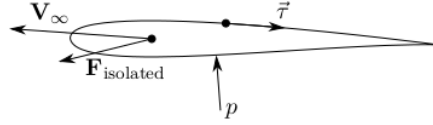


Figure 4.1: Isolated force propelling an arbitrary body

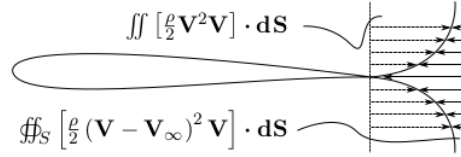


Figure 4.2: Kinetic energy in absolute and relative reference frames

$\mathbf{F}_{\text{isolated}}$, driving a body through the air at a velocity $\mathbf{V}_\infty$ as depicted in Figure 4.1. By "isolated from the flow-field" it is meant that the force does not interact with the flow directly (an example could be a magnetic force or gravity). This isolated force clearly has a work rate of $\dot{W} = \mathbf{F}_{\text{isolated}} \cdot \mathbf{V}_\infty$. The isolated force must have an equal and opposite response at the walls of the body, and this force manifests itself in the form of integrated net friction and pressure forces. Additionally, the isolated force work rate is the only source of power, and is therefore equal to the power transferred to the flow via the friction and pressure forces. This makes it clear that Equation (4.12) accounts for the work rate due to skin friction and pressure forces on airframe surfaces, unlike Equation (4.6) which is similar to Drela's generalised equation. The concern of kinetic energy is directly addressed in the LHS term of Equation (4.12) which is clearly the kinetic energy imparted to the flow. This can be seen with reference to Figure 4.2, which compares the kinetic energy content of a wake in relative (upper equation) versus absolute (lower equation) frames of reference. The former indicates how the wake results in a deficit of the free-stream kinetic energy. Subtracting the free-stream kinetic energy for the same amount of mass flow, leads to the confusing kinetic energy excess (i.e. LHS of Equation (4.6)). Instead, the amount of power imparted to the flow is more intuitive, and this is described by the lower equation in Figure 4.2, corresponding to the LHS of Equation (4.12). This is a more intuitive perspective, because it describes how the body has disturbed the fluid as it moves through it, and subsequently how much kinetic energy is lost/imparted to the atmosphere. The reader is referred to Renard and Deck [4] for a more in-depth discussion and explanation on these concerns. Although their paper is limited to a flat plate in incompressible flow, it provides great insight into the way of thinking necessary to best utilize energy-based assessments.

4.1.2 Total and Internal Energy Equations

The total energy equation in relative reference frame forms (Equations (3.33) and (3.34)) has been presented in Section 3.7, and are repeated below for convenience. The total

energy equation in the absolute reference frame (Equation (4.13)) is obtained in a similar manner to that of kinetic energy expression (Equation (4.12)) by substitution of Equation (4.11).

$$\oint_S \left[\rho \left(e + \frac{\mathbf{V}^2}{2} \right) \mathbf{V} \right] \cdot d\mathbf{S} = \iiint_V [\rho (\dot{q} + \mathbf{f} \cdot \mathbf{V})] dV + \oint_S [-\mathbf{q}_c - (p\mathbf{V}) + (\bar{\tau} \cdot \mathbf{V})] \cdot d\mathbf{S} \quad (3.33)$$

$$\begin{aligned} \oint_S \left\{ \rho \left[(e - e_\infty) + \frac{(\mathbf{V}^2 - \mathbf{V}_\infty^2)}{2} \right] \mathbf{V} \right\} \cdot d\mathbf{S} &= \oint_S [-\mathbf{q}_c - (p - p_\infty) \mathbf{V} + \bar{\tau} \cdot \mathbf{V}] \cdot d\mathbf{S} + \\ &\quad \iiint_V [\rho (\dot{q} + \mathbf{f} \cdot \mathbf{V})] dV - p_\infty \oint_S \mathbf{V} \cdot d\mathbf{S} \end{aligned} \quad (3.34)$$

$$\begin{aligned} \oint_S \left\{ \rho \left[(e - e_\infty) + \frac{(\mathbf{V} - \mathbf{V}_\infty)^2}{2} \right] \mathbf{V} \right\} \cdot d\mathbf{S} &= \\ \oint_S [-\mathbf{q}_c - (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) + \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot d\mathbf{S} + \\ \iiint_V [\rho (\dot{q} + \mathbf{f} \cdot (\mathbf{V} - \mathbf{V}_\infty))] dV - p_\infty \oint_S \mathbf{V} \cdot d\mathbf{S} \end{aligned} \quad (4.13)$$

Expressions for the internal energy, Equations (4.14) and (4.15), are obtained by subtracting Equation (4.5) from Equation (3.33) and Equation (4.6) from Equation (3.34) (or Equation (4.12) from Equation (4.13)), respectively.

$$\oint_S (\rho e \mathbf{V}) \cdot d\mathbf{S} = - \oint_S \mathbf{q}_c \cdot d\mathbf{S} + \iiint_V [\rho \dot{q} - p \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] dV \quad (4.14)$$

$$\begin{aligned} \oint_S \rho (e - e_\infty) \mathbf{V} \cdot d\mathbf{S} &= - \oint_S \mathbf{q}_c \cdot d\mathbf{S} + \\ &\quad \iiint_V [\rho \dot{q} - (p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] dV + \\ &\quad - p_\infty \oint_S \mathbf{V} \cdot d\mathbf{S} \end{aligned} \quad (4.15)$$

Equations (4.14) and (4.15) are convenient as the internal energy flux from CFD studies may be used to indirectly calculate the volume integral of expansion work and viscous dissipation, which represent losses of power to the previously undisturbed flow:

$$\iiint_V [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] dV = \oint_S [\rho (e - e_\infty) \mathbf{V} + \mathbf{q}_c + p_\infty \mathbf{V}] \cdot d\mathbf{S} - \iiint_V \rho \dot{q} dV \quad (4.16)$$

This becomes particularly useful in including portions of the control volume not included within the CFD domain, i.e. where the fluid field is not resolved using CFD and therefore where the volume integrals are not directly calculable. An example of such a scenario is where a propulsor is modelled in the CFD domain using an outlet and inlet boundary condition at the propulsor inlet and exit, respectively. This is discussed further in Section 4.4.

4.2 Application to Streamtubes

To further interpret the physical meaning behind each of the energy equation terms, they are applied to a streamtube control volume surrounding a simple body without any propulsors. First, Equation (4.12) is applied to the streamtube depicted in Figure 3.1 to give Equation (4.17), neglecting body forces and noting that $\mathbf{V} = 0$ on the body's surface following the no-slip condition and that $\mathbf{V} \cdot d\mathbf{S} = 0$ along a streamline such as the tube walls S_T .

$$\begin{aligned} & \iint_{S_G} \frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \cdot d\mathbf{S} + \iint_{S_G+S_T} [(p - p_\infty)(\mathbf{V} - \mathbf{V}_\infty) - \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot d\mathbf{S} + \\ & \iint_{S_W} [-(p - p_\infty)\mathbf{V}_\infty + \bar{\tau} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S} + \iiint_V [-(p - p_\infty)\nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] dV = 0 \end{aligned} \quad (4.17)$$

In considering the signs of the terms in Equation (4.17), energy sources and sinks may be identified. In it's above form, the terms on the LHS Equation (4.17) are energy sinks if positive, and energy sources if negative.

$$\iint_{S_G} \frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \cdot d\mathbf{S}$$

is the net kinetic energy imparted to the flow through which the streamtube is moving, and represents an energy sink. This is following the absolute reference frame definition where the control volume and body is moving through a stationary and initially undisturbed atmosphere. S_G is comprised of upstream and downstream flux surfaces. Assuming that the upstream surface is far enough ahead and thus $\mathbf{V} = \mathbf{V}_\infty$, the net kinetic energy imparted to the flow is equal to the kinetic energy flux across the downstream surface. This flux term is positive, as the mass flux across it is positive (by convention $\hat{n}$ is defined to point out of the CV), and is also intuitively a power sink in the system.

$$\iint_{S_G+S_T} [(p - p_\infty)(\mathbf{V} - \mathbf{V}_\infty) - \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot d\mathbf{S}$$

are pressure and shear stress work acting on the control volume via surfaces S_G and S_T . These work terms are likely to be less than zero (i.e. energy source term). This at

first appears perplexing, but it is an observed outcome that can be seen in Section 5.3.3. They are essentially temporary sources in a response to the work being done on the fluid by the wall. In moving through the fluid the wall displaces it and locally compresses it. The increased pressure acts on the streamtube walls and appears as a source. However, it is just a redirection of energy. This is discussed in the context of an example in Section 5.3.3.

$$\iint_{S_W} [-(p - p_\infty) \mathbf{V}_\infty + \bar{\boldsymbol{\tau}} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S}$$

is work done by on the fluid by the forces on the surface S_W acting at the free-stream velocity. Intuitively this an energy source, and the term is a negative one. This can be seen by taking $\mathbf{V}_\infty$ out of the integral, which yields the reciprocal to the LHS of Equation (3.8). In other words, it is the power behind the force driving the body through the atmosphere in the absolute reference frame. This force is required for steady flight and the use of the steady equations assumed in this work. In this context, it is the ideal power required to drive the body through the atmosphere. This demonstrates the intuitiveness of the absolute reference frame of Equation (4.12), which has a power associated to surface forces which would alternatively be zero in the relative reference frame.

$$\iiint_V [-(p - p_\infty) \nabla \cdot \mathbf{V}] dV$$

is related to the expansion or compression of the fluid within the control volume, because $\nabla \cdot \mathbf{V}$ is the *time rate of change of the volume of a moving fluid element, per unit volume* [21]. However, it is difficult to see intuitively if this term represents an energy source or sink. Instead, its physical interpretation is left until later in a discussion on internal energy flux.

$$\iiint_V [(\bar{\boldsymbol{\tau}} \cdot \nabla) \cdot \mathbf{V}] dV$$

is the viscous dissipation and, being a positive term, is also intuitively a power sink where energy is transferred to the flow. Ultimately, were the streamtube's control volume taken to be far enough away from the body, the majority of the power consumption would be contained within this term (apart from the pressure related volume integral power consumption). For example, the kinetic energy imparted to the flow will eventually dissipate to heat as the once disturbed flow returns to a stationary condition.

At this stage it is convenient to revisit the internal energy Equation (4.16). Applying it to this streamtube scenario, there is no volumetric heating and heat conduction across the body's surface is neglected. Noting that flux only occurs across surfaces S_G , the equation is written as:

$$\iiint_V [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\boldsymbol{\tau}} \cdot \nabla) \cdot \mathbf{V}] dV = \iint_{S_G} [\rho(e - e_\infty) \mathbf{V} + p_\infty \mathbf{V}] \cdot d\mathbf{S} \quad (4.18)$$

To interpret the physics behind the terms, Equation (4.18) is arranged, using the divergence theorem, into the following form:

$$\iint_{S_G} [\rho(e - e_\infty) \mathbf{V}] \cdot d\mathbf{S} = \iiint_V [-p \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] dV$$

The LHS is the net change in internal energy. It is positive and represents an energy sink. In accordance with the above, the net increase in internal energy is a result of two terms. The viscous dissipation term, which has already been discussed, raises the static temperature of the flow and contributes to the internal energy. The other term, $(-p \nabla \cdot \mathbf{V})$ is related to compression/expansion work. For expansion, $(\nabla \cdot \mathbf{V} > 0)$, the overall term is negative, and represents power extraction from the flow (as in the case of a turbine for instance), thereby lowering the internal energy. In the convention chosen, it is viewed as an energy source. This at first doesn't seem to be intuitive, but it may be seen as the flow's potential to do work on the body. In other scenarios, heat may be added (e.g. in a combustion scenario) to cause the expansion and, in this example, the energy source becomes more intuitive. On the other hand, compression $(\nabla \cdot \mathbf{V} < 0)$ represents work transfer to the flow and its internal energy is increased. This is intuitively an energy sink as work is transferred to the flow in compressing it. In compression, the flow has gained the potential to do work via expansion returning some of the energy back to the source (or all of it in an isentropic process).

It is convenient to rearrange Equation (4.17) in the format of Power Source = Power Sink, as in Equation (4.19). Shifting the S_W associated term to the LHS makes it positive. It is clear that it is the near-field extraction of the net force on the surface S_W by the fluid multiplied by the free-stream velocity (or flight velocity, in keeping in the context of an absolute reference frame). In steady level flight, the power associated with these forces may be interpreted as the power associated with a balancing force which is isolated from the flow (i.e. it does not directly influence the flow field). As a result, Equation (4.19) may be used to extract the forces acting on a body.

$$\begin{aligned} \mathbf{F}_B \cdot \mathbf{V}_\infty &= \iint_{S_W} [(p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S} \\ &= \iint_{S_G} \frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \cdot d\mathbf{S} + \\ &\quad \iint_{S_G + S_T} [(p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot d\mathbf{S} + \\ &\quad \iiint_V [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] dV \end{aligned} \quad (4.19)$$

Alternatively, the volume integrals of Equation (4.19), may be re-written as surface integrals of fluxes in internal energy using relation derived in Equation (4.16), and ignoring

any volumetric heating terms or heat conduction across surfaces:

$$\begin{aligned}
 \mathbf{F}_B \cdot \mathbf{V}_\infty &= \iint_{S_W} [(p - p_\infty) \mathbf{V}_\infty - \bar{\bar{\tau}} \cdot \mathbf{V}_\infty] \cdot \mathbf{dS} \\
 &= \iint_{S_G} \frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \cdot \mathbf{dS} + \\
 &\quad \iint_{S_G + S_T} [(p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\bar{\tau}} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot \mathbf{dS} + \\
 &\quad \iint_{S_G} [\rho (e - e_\infty) \mathbf{V}] \cdot \mathbf{dS} + p_\infty \iint_{S_G + S_T} \mathbf{V} \cdot \mathbf{dS} \quad (4.20)
 \end{aligned}$$

This form of the equation has potential practical advantages in post-processing CFD results, as it is easier to extract surface integrals rather than volume integrals. But more conveniently, the viscous stress tensor $\bar{\bar{\tau}}$ does not need to be acquired directly from the flow solution, which is difficult to do. Because it lumps together viscous dissipation and expansion/compression work terms, it is necessary to extract at least one of the volume integrals of Equation (4.19), and the other may be determined indirectly from Equation (4.16). As such, it is easier to directly extract the pressure expansion/compression work term, and obtain the viscous dissipation term indirectly from fluxes in internal energy. This is because of the added complexity in extracting the viscous stress tensor.

Continuing a discussion on the physical representation of the mathematical terms

$$\iint_{S_G} [\rho (e - e_\infty) \mathbf{V}] \cdot \mathbf{dS}$$

is positive (i.e. power sink) and is the net increase in internal energy imparted to the flow. This increase in internal energy correlates to an increase in static temperature mainly due to the viscous dissipation in the boundary layers and wake. It also has contributions from the compression/expansion work done on the flow.

$$p_\infty \iint_{S_G + S_T} \mathbf{V} \cdot \mathbf{dS} = p_\infty \iiint_V \nabla \cdot \mathbf{V} \, dV$$

is the work associated with the expansion/compression of the flow at free-stream atmospheric pressure. Depending on the flow scenario, this term may be positive or negative and therefore a power sink or source, respectively. This is seen by its volume integral form obtained from the divergence theorem, and an alternative surface integral form may be used to calculate this compression/expansion work.

4.3 Application to Non-Propelled Bodies

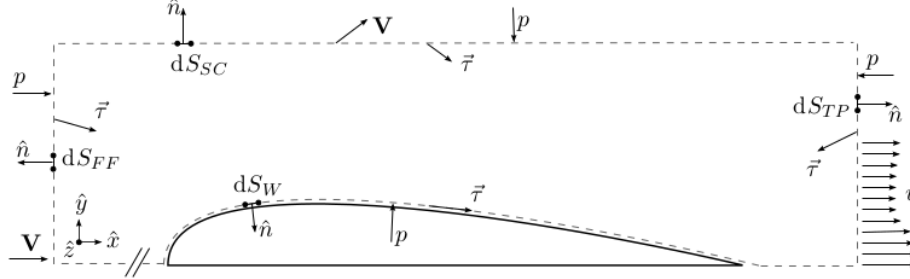


Figure 4.3: Control volume encapsulating a non-propelled body

Examination of Equations (4.19) and (4.20) shows that the net resultant force acting on a non-propelled body may be decomposed into different contributions, with their associated physical interpretations. These are 1) the rate of kinetic energy imparted to the flow, 2) the pressure and viscous stress work rate done on the flow passing through the control volume boundaries, 3) the rate of compression/expansion work done on the fluid inside the volume and 4) the rate of viscous dissipation within the control volume. The 4th and 5th have an alternative description of being the net increase in internal energy across the volume. It is clear that the proportions attributed to each decomposition term are dependent on the definition of the control volume with respect to the body. For example, a control volume with bounds positioned sufficiently far away from the body will have no pressure or viscous stress work rates. Therefore, the control volume definition becomes important when trying to relate back the decomposed terms to conventional force terms (i.e. induced drag, wave drag, profile drag etc.). Such a definition and decomposition has been performed by Drela [5]. This section follows Drela's control volume definition but, unlike Drela, the approach taken originates from the generalised kinetic energy equation in the absolute reference frame (developed in Section 4.1) instead of the relative reference frame definition. Although the outcome is essentially the same, the approach taken here makes the derivation more intuitive and incorporates viscous terms neglected by Drela (on the claims that its contribution is negligible). With reference to Figure 4.3, an appropriate definition of the control volume (considered here in Cartesian coordinates) specifies that 1) the front far-field surface (S_{FF}) is positioned such that $\mathbf{V} = \mathbf{V}_\infty$ and $p = p_\infty$, 2) the Trefftz plane surface (S_{TP}) is normal to the x-axis³, 3) the side cylinder surface (S_{SC}) is parallel to $\mathbf{V}_\infty$, 4) the surfaces are positioned such that all vortex flows exit via S_{TP} and 5) any oblique shock waves exit via S_{SC} . In addition, to aid decomposition of induced drag related terms,

³Drela specifies normal to the free-stream vector, but also has the specification that the aircraft's body axis is chosen such that the free-stream velocity vector is parallel to the x-axis. This work includes a more generalised derivation where the orthogonal coordinate system of the body axis does not necessarily need to be aligned to the free-stream velocity vector

S_{TP} should be positioned as close to the body as possible⁴, otherwise a portion of the vortex energy associated with induced drag will have already been dissipated within the volume. The reader is referred to a paper by Spalart [23], who details the difficulties of positioning the Treftz plane for induced drag decompositions. Applying Equation (4.12) to Figure 4.3, given the control definition above, gives Equation (4.21), which may be re-written in a more convenient form of Equation (4.22) following the groupings made by Drela. However, it is noted here that from the derivation presented in this work (unlike that by Drela) the energy equation is manipulated to give a direct expression of the body force (instead of power in and out flows). This allows for intuitive decomposition from the generalised form.

$$\begin{aligned}
\mathbf{F}_B \cdot \mathbf{V}_\infty &= \iint_{S_W} [(p - p_\infty) \mathbf{V}_\infty - \bar{\boldsymbol{\tau}} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S} \\
&= \iint_{S_{TP}} \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} + (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\boldsymbol{\tau}} \cdot (\mathbf{V} - \mathbf{V}_\infty) \right] \cdot d\mathbf{S} + \\
&\quad \iint_{S_{SC}} \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} + (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\boldsymbol{\tau}} \cdot (\mathbf{V} - \mathbf{V}_\infty) \right] \cdot d\mathbf{S} + \\
&\quad \iiint_V [- (p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\boldsymbol{\tau}} \cdot \nabla) \cdot \mathbf{V}] dV
\end{aligned} \tag{4.21}$$

$$\mathbf{F}_B \cdot \mathbf{V}_\infty = \dot{E}_B = \dot{E}_{k_u} + \dot{E}_{k_{vw}} + \dot{E}_p + \dot{E}_\tau + \dot{E}_W + \dot{E}_{p\nu} + \dot{E}_{\tau\nu} \tag{4.22}$$

Each of the terms in Equation (4.22) may be expanded in terms of the velocity vector and viscous tensor components. A description of each follows closely those defined by Drela, and are presented in Cartesian coordinates.

The stream-wise (or non-vortical) component of kinetic energy imparted to the flow (where subscripts R and A refer to relative and absolute reference frames, respectively):

$$\dot{E}_{k_u} = \iint_{S_{TP}} \frac{\rho}{2} (u_A^2) u_R dS \tag{4.23}$$

The remaining kinetic energy imparted to the flow, normally associated to vortical flows (approximately equivalent to induced drag power, should the treftz plane be positioned such that minimal dissipation of transverse flows has occurred before crossing it):

$$\dot{E}_{k_{vw}} = \iint_{S_{TP}} \frac{\rho}{2} (v_A^2 + w_A^2) u_R dS \tag{4.24}$$

Pressure boundary work done by the flow passing through the treftz plane:

$$\dot{E}_p = \iint_{S_{TP}} (p - p_\infty) u_R dS \tag{4.25}$$

⁴practically this may not be possible, as it may violate the conditions 4 and 5 listed here

Viscous stress boundary work done by the flow passing through the treftz plane (this term was omitted by Drela, on the basis of being negligible, but is included here for completeness):

$$\dot{E}_\tau = \iint_{S_{TP}} -(\tau_{xx}u_A + \tau_{xy}v_A + \tau_{xz}w_A) dS \quad (4.26)$$

Power associated with wave drag as a result of kinetic energy imparted to the flow via the side cylinders of the control volume, as well as pressure and viscous shear stress boundary work rates on these surfaces (once again, viscous stress terms had been neglected by Drela, but included here for completeness):

$$\begin{aligned} \dot{E}_W = & \iint_{S_{SC}} \frac{\rho}{2} (u_A^2 + v_A^2 + w_A^2) v_R dS + \\ & \iint_{S_{SC}} (p - p_\infty) v_R dS + \\ & \iint_{S_{SC}} -(\tau_{yx}u_A + \tau_{yy}v_A + \tau_{yz}w_A) dS \end{aligned} \quad (4.27)$$

Control volume compression work (over and above buoyancy work) done by driving the body through the flow:

$$\dot{E}_{p\nu} = \iiint_V -(p - p_\infty) \nabla \cdot \mathbf{V} d\nu \quad (4.28)$$

Viscous dissipation within the control volume (heat transferred to the flow within the boundary layers, viscous wakes and shock waves):

$$\dot{E}_{\tau\nu} = \iiint_V (\bar{\tau} \cdot \nabla) \cdot \mathbf{V} d\nu \quad (4.29)$$

Unlike the power balance derivations of Drela, this decomposition fully accounts for all energy terms and does not neglect the viscous related boundary work contributions in terms $(\dot{E}_\tau, \dot{E}_W)$. It is worth noting that, in the author's opinion, the last two terms $(\dot{E}_{p\nu}, \dot{E}_{\tau\nu})$ may contain contributions from the presence of shock waves within the control volume. A shock wave will undoubtedly raise the internal energy of the flow through which it passes and therefore, contrary to Drela's definition, the term $\dot{E}_W$ does not fully capture the wave drag.

The purpose of this section was to demonstrate that the absolute reference frame derivations unique to this work, are consistent with the outcomes of the derivations by Drela when applying the same assumptions. Given the argument in the above paragraph it is evident that force contributions contained within the volume integrals $(\dot{E}_{p\nu}, \dot{E}_{\tau\nu})$ are a result of different flow phenomena. In considering a control volume around a body, force resulting energy contributions may be decomposed further to account for contributions by different flow mechanisms. For a non-propelled body, such flow mechanisms are illustrated in Figure 4.4, and may be identified as dissipative phenomena due to boundary layers (and wakes), vortices (for lifting bodies) and shock waves. Section 5.1,

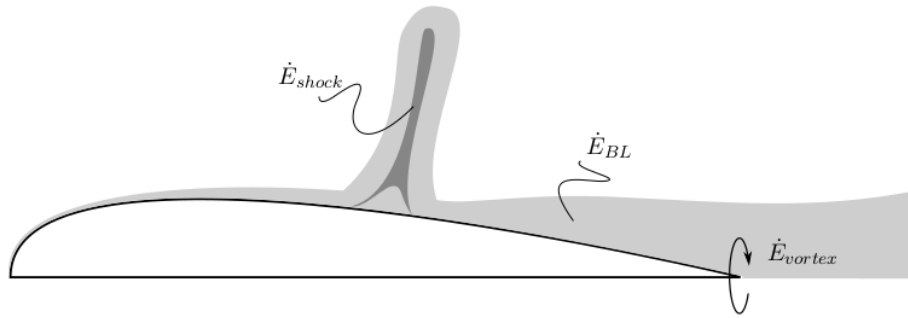


Figure 4.4: Control volume decomposition for non-propelled body

retained over S_F because the resulting vector from $\bar{\tau} \cdot \mathbf{V}$ is unlikely to be perpendicular to $\hat{n}$, and therefore $(\bar{\tau} \cdot \mathbf{V}) \cdot \hat{n} \neq 0$. In Equation (4.30) there is just the $\bar{\tau} \cdot \mathbf{V}$ term left which is associated to S_F , and it is this term that describes the work rate associated with the viscous forces acting on the fan surface. In Equation (4.31), this is accompanied by an additional term $\bar{\tau} \cdot \mathbf{V}_\infty$ which is the work rate associated with the viscous forces acting at the free-stream velocity.

$$\begin{aligned} & \iint_{S_P} \left[\frac{\rho}{2} (\mathbf{V}^2 - \mathbf{V}_\infty^2) \mathbf{V} + (p - p_\infty) \mathbf{V} - \bar{\tau} \cdot \mathbf{V} \right] \cdot d\mathbf{S} + \dots \\ & - \iint_{S_F} \bar{\tau} \cdot \mathbf{V} \cdot d\mathbf{S} + \dots \\ & \iiint_{\nu_2} [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] d\nu = 0 \end{aligned} \quad (4.30)$$

$$\begin{aligned} & \iint_{S_P} \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} + (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty) \right] \cdot d\mathbf{S} + \dots \\ & - \iint_{S_F} [(p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S} - \iint_{S_F} \bar{\tau} \cdot \mathbf{V} \cdot d\mathbf{S} + \dots \\ & \iiint_{\nu_2} [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] d\nu = 0 \end{aligned} \quad (4.31)$$

With reference to Equation (3.11), the following substitution for the force on the propulsor may be made:

$$\mathbf{F}_P \cdot \mathbf{V}_\infty = \iint_{S_F} [(p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S}$$

and now subtracting Equation (4.31) from Equation (4.30) to eliminate the volume integrals and the fan surface viscous power $\bar{\tau} \cdot \mathbf{V}$, the propulsor force is expressed as a combination of surface integrals over S_P :

$$\mathbf{F}_P \cdot \mathbf{V}_\infty = - \iint_{S_P} \left\{ \frac{\rho}{2} [(\mathbf{V}^2 - \mathbf{V}_\infty^2) - (\mathbf{V} - \mathbf{V}_\infty)^2] \mathbf{V} + (p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty \right\} \cdot d\mathbf{S} \quad (4.32)$$

The main purpose of this derivation was to eliminate the volume integrals and $\bar{\tau} \cdot \mathbf{V}$ on surface S_F , because these are unattainable when the propulsor control volume is not included within the CFD domain. An alternative, more direct derivation, may be performed by applying Equation (4.11) directly to the propulsor CV.

4.5 Application to Propelled Bodies

This section applies the energy equation forms developed in this work to a propelled body. The method of derivation is unique to this work, and allows for advantageous



terms of the net-vehicle-force:

$$\begin{aligned}
\mathbf{NVF} \cdot \mathbf{V}_\infty &= \mathbf{F}_B \cdot \mathbf{V}_\infty + \mathbf{F}_P \cdot \mathbf{V}_\infty \dots \\
&= \iint_{S_O} \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} + (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\boldsymbol{\tau}} \cdot (\mathbf{V} - \mathbf{V}_\infty) \right] \cdot d\mathbf{S} \dots \\
&+ \iint_{S_P, \nu_1} \left[\frac{\rho}{2} (\mathbf{V}^2 - \mathbf{V}_\infty^2) \mathbf{V} + (p - p_\infty) \mathbf{V} - \bar{\boldsymbol{\tau}} \cdot \mathbf{V} \right] \cdot d\mathbf{S} \dots \\
&+ \iiint_{\nu_1} [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\boldsymbol{\tau}} \cdot \nabla) \cdot \mathbf{V}] d\nu
\end{aligned} \tag{4.35}$$

Writing Equation (4.35) in a simplified notation:

$$\mathbf{NVF} \cdot \mathbf{V}_\infty = \dot{E}_k + \dot{E}_p + \dot{E}_\tau + \dot{E}_{prop} + \dot{E}_{p\nu} + \dot{E}_{\tau\nu} \tag{4.36}$$

where the terms follow the order of appearance as in Equation (4.35) and are; (1) the kinetic energy deposited in the flow surrounding the control volume, (2) the boundary pressure work done on S_O , (3) the viscous boundary work done on S_O , (4) the power flow rate from the propulsor control volume CV_2 into the control volume encapsulating the aircraft/assembly CV_1 , (5) the compression/expansion volumetric work rate and (6) the rate of volumetric viscous dissipation. As was done in Section 4.3, $\dot{E}_k$, $\dot{E}_p$ and $\dot{E}_\tau$ can be simplified to trefftz plane integrals and decomposed using the control volume definition by Drela [5]. This is not repeated here for the sake of brevity. The only additional power term is that of the propulsor control volume $\dot{E}_{prop}$, which contains within it a kinetic energy, pressure boundary work and viscous boundary work terms. This is the same propulsor power control volume term found by Drela, except this instance includes viscous terms for completeness. Therefore, the derivation method (which is unique to this work) may be found to produce the same decomposed terms presented by Drela, when using his same control volume definition.

4.5.2 Net-Vehicle-Force Imbalance

The expressions developed here are applicable to steady flight, and thus for an entire aircraft $\mathbf{NVF}$ must be zero. The power providing the force in the direction of the flight path (i.e. $\mathbf{NVF} \cdot \mathbf{V}_\infty$), if not zero, results in a corresponding change in aircraft potential energy associated to a change in altitude at a corresponding climb angle. If $\mathbf{NVF} \cdot \mathbf{V}_\infty > 0$ (recalling that positive and negative quantities refer to downstream and upstream direction, respectively), the force imbalance will result in a negative climb angle and drop in altitude. In this scenario, aircraft potential energy is transferred to the flow due to a drop in altitude. If $\mathbf{NVF} \cdot \mathbf{V}_\infty < 0$, then the surplus in NVF results in a positive climb angle and power is consumed in raising the aircraft's potential energy as it climbs. In each scenario, the energy is balanced because of a transfer in potential energy via change in altitude. However, if steady level flight is assumed then $\mathbf{NVF} \cdot \mathbf{V}_\infty = 0$, and therefore there is no change in aircraft potential energy due to altitude.

4.5.3 Net Fuselage Force

Alternatively, if only a portion of the aircraft is under consideration, like a fuselage propulsor assembly, then $\mathbf{NVF}$ is not necessarily zero at steady level flight (although a propulsor may be used just to mitigate the drag of the fuselage). In the remainder of this discussion this force is referred to as the net fuselage force ($\mathbf{NFF}$) to differentiate it from the net vehicle force, but noting that the two are interchangeable. In this case, an ideal isolated force F_{iso} (i.e. one that does not influence the flow-field surrounding the assembly be considered) is required to maintain steady level flight by rectifying the imbalance of a non-zero $\mathbf{NFF}$. In the context of this work, a surplus in $\mathbf{NFF}$ goes towards lowering the thrust required from a set of podded main engines in overcoming the remainder of the drag due to the remaining airframe aerodynamic components not included within the CFD domain. On the other hand, a deficit in $\mathbf{NFF}$ will depend on a portion of the main engine thrust to balance out the overall forces. Decoupling out the propulsor fuselage assembly from the rest of the airframe is an assumption which neglects a lot of aerodynamic interaction, but does allow for more convenient design explorations in simplifying CFD models.

4.5.4 Flow Phenomena Decompositions

As was the case with the non-propelled body, in Section 4.3, it is useful to decompose the flow-field into different physical phenomena. In the propelled scenario, however, there is the additional jet region to be considered as depicted in Figure 4.5. The reason for decomposing the flow-field into these different regions is to aid the understanding of the aerodynamic interactions for tightly integrated airframe-propulsion systems, thereby gaining more insight from design explorations. With reference to Equation (4.35), contributions to the net-vehicle-force by each local flow region may be quantified using the volume integrals with integrands $-(p - p_\infty) \nabla \cdot \mathbf{V}$ and $[(\bar{\tau} \cdot \nabla) \cdot \mathbf{V}]$. The current formulation is limited in that only the viscous dissipation volume integral may be applied to local regions to quantify their specific contribution to rearward force, due to its irreversible nature. On the other hand, the compression/expansion work integrand may be locally non-zero, but reversible, meaning that its contribution may ultimately be mitigated in a different local region in the flow-field. As an example, it is clear that a shock-wave will result in viscous dissipation and contribute to a rearward force, which is quantifiable using the latter integrand. However, it is also clear that the former integrand (in this instance compression work) is non-zero in the local shock region, but that the flow may undergo complete reversible expansion downstream (as suggested by Drela [5]) or partial expansion, meaning that the local quantification is not truly representative of the shocks overall contribution. The quantification of the viscous dissipation contribution by different flow mechanisms, is reliant on spatial partitioning of the flow, which is discussed in more detail in Section 5.1. As a brief description here, the boundary layers and viscous shear wake regions may be identified by a non-dimensional metric comprised of eddy-viscosity terms, and the two volume integrands of Equation (4.35). This particular region selection parameter is unique to this work, and is an improvement

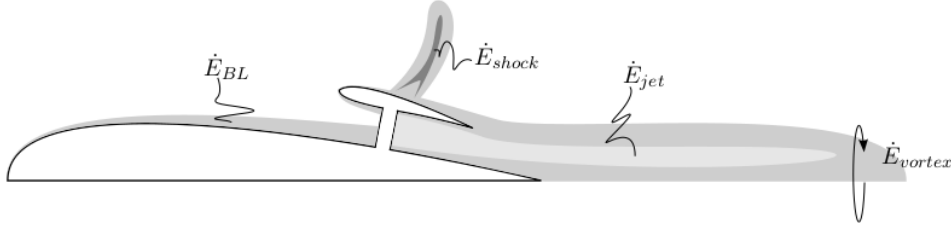


Figure 4.5: Control volume decomposition for propelled body

on the selector proposed by Paparone and Tognaccini [7]. It is deemed to be one of the significant contributions of this thesis. This work does directly use the shock region selector proposed by Paparone and Tognaccini in the same paper, which is based a non-dimensional term based on spatial pressure gradients in the flow. Finally, the jet region is identified according to a rise in stagnation enthalpy from free-stream conditions. This is a rather crude approach, as jet structures are far more complex [38], however it is left as future work. As this work only considers non-lifting configurations, no vortex region selector has been considered. It must also be mentioned here, that there are of course areas where the regions overlap and the contributions cannot be entirely separated out and attributed to the respective regions. In these areas, it must be decided which region takes precedence over the other, and this procedure is discussed in more detail in Section 5.1.

4.6 BLI Performance Evaluation

The same performance metrics as presented in Chapter 3 are used. The energy-based formulation only allows for the determination of NVF , and not any other thrust-force accounting definitions apart from the Reference Net Vehicle Force $F_{N,ref}$. Therefore performance may still be based on TSPC and the modified propulsive efficiency. The advantage of the energy-based formulation is to further breakdown contributions to $F_{N,ref}$ due to different flow mechanisms. In this way, the change in force can be related to contributions due to shocks, jets and boundary layers.

4.7 Summary

This chapter has looked at derivations into energy-based assessments of aircraft performance. The kinetic energy equation has been formulated in the absolute reference frame, which is a significant contribution of this work, in that it addresses the concerns raised by Renard and Deck [4] about the formulation presented by Drela [5]. Subsequently, the internal energy has been incorporated into the kinetic energy equation, yielding an alternative form and means of obtaining certain volume integrals using surface fluxes instead. Intuitive interpretations as to the physical meaning behind mathematical terms,

has been presented through the application of these equations to streamtubes, propulsor control volumes, non-propelled and propelled bodies. Finally, decomposition of the flow-field into different flow mechanisms is discussed along with spatial partitioning to quantify energy dissipation contributions.

4.7.1 Concepts and Formulations

The kinetic energy equation in an absolute frame of reference is given as:

$$\begin{aligned} \oint_S \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \right] \cdot d\mathbf{S} = & \oint_S [-(p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) + \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty)] \cdot d\mathbf{S} + \\ & \iiint_\nu [(p - p_\infty) \nabla \cdot \mathbf{V} - (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] d\nu + \\ & \iiint_\nu [\rho \mathbf{f} \cdot (\mathbf{V} - \mathbf{V}_\infty)] d\nu \end{aligned} \quad (4.12)$$

The internal energy flux substitution for volumetric compression and dissipation integrals is given by:

$$\iiint_\nu [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] d\nu = \oint_S [\rho (e - e_\infty) \mathbf{V} + \mathbf{q}_c + p_\infty \mathbf{V}] \cdot d\mathbf{S} - \iiint_\nu \rho \dot{q} d\nu \quad (4.16)$$

The energy based formulation of the force acting on a non-propelled body is given by the following:

$$\begin{aligned} \mathbf{F}_B \cdot \mathbf{V}_\infty = & \iint_{S_W} [(p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty] \cdot d\mathbf{S} \\ = & \iint_{S_{TP}} \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} + (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty) \right] \cdot d\mathbf{S} + \\ & \iint_{S_{SC}} \left[\frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} + (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) - \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty) \right] \cdot d\mathbf{S} + \\ & \iiint_\nu [-(p - p_\infty) \nabla \cdot \mathbf{V} + (\bar{\tau} \cdot \nabla) \cdot \mathbf{V}] d\nu \end{aligned} \quad (4.21)$$

$$\mathbf{F}_B \cdot \mathbf{V}_\infty = \dot{E}_B = \dot{E}_{k_u} + \dot{E}_{k_{vw}} + \dot{E}_p + \dot{E}_\tau + \dot{E}_W + \dot{E}_{p\nu} + \dot{E}_{\tau\nu} \quad (4.22)$$

The energy-based formulation of propulsor force is given by:

$$\mathbf{F}_P \cdot \mathbf{V}_\infty = - \iint_{S_P} \left\{ \frac{\rho}{2} [(\mathbf{V}^2 - \mathbf{V}_\infty^2) - (\mathbf{V} - \mathbf{V}_\infty)^2] \mathbf{V} + (p - p_\infty) \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty \right\} \cdot d\mathbf{S} \quad (4.32)$$

4.7. Summary

particularly advantageous when incorporating propulsors into numerical flow domains, without having to include it's own flow domain too (i.e. representing it as a set of inlet and outlet boundary conditions). Another advantage is that it provides another form to calculate the NVF , and thereby is a further source of formulation verification.

- A procedure of spatial decomposition by Paparone and Tognaccini [7] into different flow mechanisms has been adopted, modified and improved from formulations on entropy drag, and incorporated into the energy approach of this work. This decomposition allows for volume integrals describing energy dissipation, to be quantified for different flow regions (e.g. shock, jet and BLs). A BL and viscous wake region selector, modified from Paparone and Tognaccini's work, is a significant contribution presented in a later Chapter (see Section 5.1).

4.7.3 Future Work

The following are suggestions as to how the above formulations may be improved, by either including more rigour or better insight into the physical interpretations of the various mathematical terms.

Understanding of $-(p - p_\infty) \nabla \cdot \mathbf{V}$ Contributions

$-(p - p_\infty) \nabla \cdot \mathbf{V}$ is a rather perplexing term, as it becomes difficult to understand net contributions. Unlike the $(\bar{\tau} \cdot \nabla) \cdot \mathbf{V}$ integrand, $-(p - p_\infty) \nabla \cdot \mathbf{V}$ may be positive and negative in different regions of the flow domain, making it difficult to understand what the net effect is, or how different flow mechanisms contribute to the net effect. One of the main questions about this term is whether or not it is reversible or dissipative? Or alternatively, can it be decomposed into reversible and irreversible contributions? Certainly in the scenario where heat is added/extracted to/from a volume, this term has a definite irreversible portion. However, for adiabatic instances is this term reversible? Drela [5] states that through a shock wave, this term undergoes a compression which is expanded reversibly along the streamline. Therefore further work is required to better understand this term, through an improved decomposition.

Proof of $(\bar{\tau} \cdot \nabla) \cdot \mathbf{V} > 0$

$(\bar{\tau} \cdot \nabla) \cdot \mathbf{V} > 0$ is stated by Drela [5], and from experience has been found to hold true (at least within a reasonable numerical tolerance for RANS CFD). However, it may be useful to prove that this is indeed the case. Velocity derivatives may be of either sign in different regions of the flow, and it becomes difficult to see exactly why this statement is true. Proof of this statement may also lead to a better understanding of the physics behind this term and how the dissipation is manifested in different flow regions.

Relations to Entropy Drag Formulations

There is some interesting work on momentum-based far-field drag decompositions in literature [7, 23, 25–28], which utilize entropy drag and enthalpy terms. Future work should aim to bridge the gap between such decompositions and the types of formulations presented in this work and References [5, 6]. This may lead to more definitive decompositions and physical interpretations, as well as means for more extensive verification.

Chapter 5

Practical Implementation in RANS CFD

Chapters 3 and 4 introduced different methods for examining the net-vehicle-force of a tightly integrated airframe-propulsion system, which allow for different decompositions of the net-vehicle-force. The purpose of this work is not to compare advantages and disadvantages of each in order to prescribe a single method. Instead, the aim is to discover how the different decompositions can be used to better describe, identify and analyse different physical phenomena, and how they may compliment one another in vehicle aerodynamic performance analysis. In light of the above, the question arises as to whether these methods are consistent with one another when applied to numerical solutions? and if not, what are the subsequent limitations and impact on performance assessments? Are the inconsistencies irreconcilable, or do they still allow for improved analysis?

Inevitably, the answer to the above questions is highly dependent on the numerical methods used to solve the continuity, momentum and energy equations. Two-equation RANS CFD codes have become the work-horse of design explorations, and subsequently this chapter aims to explore the applicability and limitations associated with implementation of momentum and energy-based methods in RANS solutions. A well known deficiency of two-equation RANS models is their inability to deal with curvature and rotation [39]. This directly impacts the physical representation of the viscous stress and energy dissipation in the flow, and the theory in Chapters 3 and 4 is highly reliant on viscous terms (and hence the viscous stress tensor). As such, the purpose of this chapter is to establish whether it is possible to apply energy-based assessment methods to two-equation RANS CFD solutions, and what the impact is on consistency between results produced by momentum and energy-based methods. Furthermore, an additional objective is to identify modelling considerations that must be taken into account when applying these assessment methods.

The chapter begins by presenting a procedure for spatially decomposing different regions of the flow-field, attributed to different physical mechanism. This is done at the beginning of the chapter, as the spatial decompositions are required by the sections

that follow. Subsequently, flow over a non-propelled fuselage is considered, where viscous stress tensor fields are examined for different near-wall treatments and interpolation schemes. This is a crucial step, which determines the RANS CFD settings used throughout the remainder of the chapter. The rest of the chapter compares near-field momentum-based, far-field momentum-based and energy-based methods of force extraction over non-propelled and propelled bodies. These sections essentially implement and compare the result of the methods presented in Chapters 3 and 4.

The contributions of this chapter are as follows:

- The development of a new BL and viscous region selector, ξ , unique to this work. This is a significant contribution, because this selector accurately extracts the BL without neglecting the laminar regions of BLs closest to the no-slip wall condition. This flaw was observed in the descriptor proposed by Paparone and Tognaccini [7]. The selector, ξ , also has the advantage of being able to omit shock regions in both external and internal jet flows. This is due to the novel formulation, which has made use of the volumetric viscous dissipation and compression work terms found in the energy-based derivations of *NVF*.
- The spatial decomposition precedence procedure, along with the various selectors, is unique to this work (at least to the author's knowledge). The procedure allows for extraction of jet core expansion, core compression and transition regions, which is useful in better understanding the energy formulations of this work. In addition, the application of volumetric viscous dissipation and volumetric compression work integrals to these decomposed regions, is also deemed to be unique to this work.
- The section on viscous stress tensor flow-fields and near-wall treatments, provides unique insight into RANS CFD implications that need to be considered, when seeking consistency between momentum and energy-based methods of performance assessments. This section is mainly aimed at Fluent users, who may now use the outcomes of this chapter to better understand the solver settings and near-wall treatments appropriate for use (when using the methods presented in Chapters 3 and 4). Although this chapter has been based on Fluent results, it does give guidance to user's of other RANS tools.
- The back-to-back comparison of the different momentum- and energy-based methods is a first of its kind (at least to the author's knowledge). This comparison led to a better understanding of how spurious effects influence the force estimations when using different methods and control volume sizes. The control volume studies show how insight into spurious effects may be gained, when using a RANS CFD tool without having access to it's source code.
- The wake energy progression breakdowns presented in this chapter are comprehensive, and unique to this work. The author is unaware of any other studies, which have performed these types of analyses on RANS CFD solutions. These breakdowns were performed for varying control volume sizes, which gave unique insight into the physical interpretations of energy terms.

5.1 Spatial Regions of Flow Mechanisms

This section is placed at the beginning of the chapter, as it contains the method used to identify the BL in numerical solutions, which is a requirement of the subsequent sections. The identification of the BL is coupled to identification of flow mechanism regions, such as shock-waves and jets. Therefore, it seems logical to introduce those identification methods at this point in the chapter. Identification of these regions is also strongly related to the energy-based method and the decomposition concepts introduced in Chapter 4. Therefore, a discussion on the flow mechanism regions is discussed with primary reference to the implications on energy-based assessments.

Chapter 4 presented an energy-based analysis, in which the contributions of energy loss pertaining to various local physical phenomena can be used to decompose sources of energy loss and gain manifesting themselves in the form of a net-vehicle-force. It is therefore desirable to separate out the energy loss associated with each physical phenomena, to gain a better understanding of trade-offs between design parameters in minimising energy losses. However, the energy derivations of Chapter 4 are in the form of surface and volume integrals and, as such, the only means for separating out energy contributions is to do so spatially (i.e. there is no intrinsic way to differentiate between the different phenomena, at least to the author's knowledge). This is far from trivial, and in some instances not possible, due to the inherent coupling between some physical phenomena (BL shock interactions being a prime example). This section discusses possible practical techniques and challenges for spatially partitioning the flow-field of RANS CFD solutions, in order to best approximate energy contributions from various physical phenomena. Following the definitions by Sato [19], the energy contributions to the flow around an aircraft may be categorised in to the following regions:

- **Vortex Region** - energy loss due 3D effects of vortical flow typically associated with lift induced drag
- **Wave Region** - the energy deposited in the flow due to the presence of shock waves
- **Surface Region** - energy dissipated in the BLs along airframe surfaces
- **Wake Region** - the energy associated with the wake produced by the airframe surfaces
- **Jet Region** - the energy introduced to the flow by the propulsor in the form of a jet plume

In light of the above, this section begins with a discussion on BL definition and identification. It is important to be able to locate the extent of the BL in the flow for numerous reasons. The first is that it is required to define BLI propulsor geometries, whose design space is dictated by the amount of the BL thickness ingested. Secondly, it allows for the dissipation losses associated with the BL to be quantified. A new method for identification and spatial partitioning is presented following a review of existing methods.

Following the BL identification subsection, are subsection regarding region identification for shocks and jets (as these are the only two remaining flow mechanism types encountered by the 2D axisymmetric CFD cases in this thesis). Because these regions are coupled, a procedure based on region precedence is introduced. Finally, the section ends with a brief discussion on spurious drag removal.

5.1.1 Boundary Layer and Viscous Regions

The necessity, in the context of this work, for separating out the BLs from the rest of the flow is twofold. Firstly, one of the key questions when considering BLI, is how much of the BL to ingest? The definition of BL thickness, δ , is therefore of significance as it is necessary in providing a point of reference. Secondly, it is valuable to decompose out local areas where the BLs contribute to dissipation and the impact of that contribution to overall vehicle efficiency. As already mentioned, this decomposition aids strategic

which is in turn based on the BL displacement thickness δ^* :

$$\delta^* \equiv \int_0^{l_e} \left(1 - \frac{\rho u}{\rho_e u_e}\right) dl \quad (5.1)$$

δ^* may be interpreted as the thickness corresponding to the deficit in mass-flow-rate between the amount that would have passed over the surface versus the amount that was actually permitted due to the presence of the BL (or alternatively as the distance by which the BL edge streamline is deflected away from the surface) [21]. As such, the BL edge conditions may be obtained from inviscid flow calculations of an *effective body* whose geometry now coincides with the curvature outlined by the displacement δ^* away from the original body. However this is problematic, as the displacement thickness is itself defined by the boundary edge flow conditions. In some cases, where the BL is assumed to be quite thin relative to the body's thickness, a close approximate is obtained by assuming the difference in geometries between the *body* and *effective body* to be negligible. Alternatively Anderson suggests an iterative procedure, which initialises the effective body to be equivalent to the actual body, from which an inviscid flow solution provides a first estimate of the BL edge flow conditions. Subsequently, these may be used (along with the viscous flow solution around the actual body) to estimate the displacement thickness δ^* . The newly calculated δ^* , is then used to redefine the effective body geometry, from which inviscid flow calculations will yield a new estimate of BL edge flow conditions and hence δ^* . This may be repeated until a value for δ^* is converged upon. This is a rather convoluted method to determine the BL thickness δ and is riddled with practical complications such as, how to truncate the effective body geometry at a trailing edge? or how to deal with split and compound BLs, where BLI propulsors are submersed within an airframe BL?

In a similar vein, Drela [22] has developed a series of *Equivalent Inviscid Flow* (EIF) models. The simplest of the models produces acceptable accuracy through the combination of Bernoulli's equation and a series of vortex sheets positioned along the body's surfaces to mimic the vorticity (Drela notes that this may be modified to model compressible cases using a compressible Bernoulli relation and a gridded approach to vorticity representation as opposed to vortex sheets). This simplified model may be used to approximate the BL edge flow conditions. For thicker BLs, the displacement of flow induced by the BL needs to be accounted for, and the EIF model may be improved by mimicking the mass flux normal to the flow surface found in the real viscous flow solution. Drela proposed two models to achieve this, the *Displacement Body Model* and the *Wall Transpiration Model*. The former is analogous to the *effective body* proposed by Anderson [21], where a *displacement body* protrudes into the flow by δ^* (to achieve the mass flow flux normal to the surface) and the EIF is modelled by a series of vortex sheets placed along the displacement body. As with Anderson's approach, this requires some iterative procedure to converge on δ^* . The latter (Wall Transpiration Model), on the other hand, keeps the geometry constant and instead combines a vortex sheet along with a source sheet to mimic the displacement found in the real viscous flow. Although these models have demonstrated accurate predictions of drag polars for 2D airfoils, cal-

culating vortex sheet and source sheet strengths and distributions, is not a trivial task and seems overly complex for the purposes of determining the BL thickness.

The above methods are convoluted and tricky to implement, particularly when complex geometries are involved. Therefore, it is preferable to use an alternative method which extracts the BL directly from the viscous solution, is independent of geometry topology, and insensitive to other flow phenomena which may be present (e.g. shock waves).

Viscous Region Selectors

The alternative to the inviscid flow methods presented above, is to use some metric available in the viscous solution to differentiate between the cells located within the viscous layer from the rest of the flow. Lovely [40] explored a array of different possible detection methods and considered entropy, vorticity and enthalpy as metrics to distinguish between viscous and potential flow. Entropy is not appropriate, as it is generated in other energy dissipative flow phenomena such as shock waves. In addition, the dividing threshold value is somewhat arbitrary and therefore difficult to specify. Vorticity shared some of the same difficulties as entropy and had the additional problem of varying smoothly from surface walls blending into the external flow, which rules out a definitive BL edge marker. Nonetheless as a solution to these problems, and arguing the subjectivity of thickness definitions (such as the 99% convention), Lovely [40] reverts to an integration scheme. The scheme takes advantage of the relationship between vorticity and velocity to estimate the BL edge velocity:

$$u_e = \int_0^\delta \frac{du}{dy} dy \approx \int_0^\delta \omega_z dy$$

where the local Cartesian coordinate system is orientated such that $\hat{y}$ is normal to the surface, and the vorticity is given by:

$$\omega_z = \frac{1}{2} \left(\frac{dv}{dx} - \frac{du}{dy} \right)$$

In theory, the upper limit (i.e. δ) of the above integral does not need to be known as the vorticity becomes zero outside of the BL. Subsequently the displacement thickness may be approximated:

$$\delta^* \approx \int_0^{\omega=0} \left(1 - \frac{u}{u_e} \right) dy$$

Lovely's [40] integration scheme has a few practical difficulties. The first is that vorticity outside of the BL, in a numerical CFD solution, is unlikely to be zero. This then puts into question the accuracy of the edge velocity approximation. The integration scheme is based on surface integrals emanating outwards from surfaces, which makes it difficult to implement throughout the domain over complex geometries. It also will suffer difficulties when attempting to extract BLs from regions where jets and shocks interact with the

BL (particularly in BLI scenarios). Therefore, it is not deemed as a suitable means of BL region extraction in this work.

Paparone and Tognaccini [7] have proposed a more practical solution to BL and viscous region identification. Paparone and Tognaccini also recognised that a dissipation metric, such as entropy, is not suitable as it is unable to distinguish between BL regions and shock regions. Instead, a non-dimensional parameter based on the eddy (turbulent) and laminar dynamic viscosities is more appropriate:

$$F_{BL} = \frac{\mu + \mu_t}{\mu} \quad (5.2)$$

Equation (5.2) has a theoretical value of unity in the flow outside the viscous layers of the flow, meaning that regions of flow where $F_{BL} > 1$ are within the boundary or wake regions. However in practice, outside the viscous regions $F_{BL} \approx 1$, and a cut-off value definition is required. Paparone and Tognaccini suggest the viscous regions be defined as:

$$F_{BL} > K_{BL} \cdot F_{BL,\infty} \quad (5.3)$$

where $F_{BL,\infty}$ is the selector value in the free-stream flow. It is recommended that for an accurate numerical flow solution, values for K_{BL} between 1.1 to 2 be used. This was demonstrated for a 2D case on the NACA 0012 airfoil, and it was shown that by only considering the flow region within the BLs, the numerical spurious drag is neglected and a drag accuracy within 1 drag count is achievable. However, in a more recent study [41] (applying this method to a 3D study of an aircraft in power off conditions), a value of 4.0 was selected based on a sensitivity study of spurious drag removal on $C_D = f(K_{BL})$. It was remarked that "practically, visual inspection is recommended to confirm that the chosen values really do select appropriate wave drag and boundary layer cells". This calls into question the robustness of this selector, particularly for large design space studies where automation of pre- and post-processing is required (as is the case in this work).

Apart from the difficulties in robustly defining a cut-off value for K_{BL} , there is a fundamental flaw in the F_{BL} selector, because $F_{BL} \rightarrow 1$ within a thin portion of the BL closest to the wall. This is expected, as the flow behaves in a laminar way very close to the wall. Therefore, Equation (5.3) excludes a few cell layers close to the wall because of the cut-off value. Additionally, $F_{BL} \rightarrow 1$ around stagnation regions (e.g. airfoil leading edge) where the BL is very thin, and these regions are therefore also excluded through the use of Equation (5.3). This is problematic because these are clearly regions of flow belonging to the BL and, more importantly, are regions with extremely large velocity gradients and subsequently viscous dissipation contributions. This leads to the requirement for a more robust metric and cut-off threshold definition.

Viscous Region Selector with Compression Work Filter

Following the previous discussion on the existing selector metrics and methods found in literature, it is concluded that none of the above are suitable for the requirements of this work. Therefore, a new selector is required which will:

- robustly capture the boundary region whilst filtering out other dissipative flow regions (i.e. shocks and jets)
- is non-dimensional, based on volumetric specific parameters and therefore simple to implement across the domain
- has a robustly and consistently definable cut-off threshold

From examination of Paparone and Tognaccini's F_{BL} metric [7], it was clear that a more appropriate metric should contain terms that capture the combined properties of large eddy viscosity and energy dissipation. However, a combination of the two is insufficient to filter out shock waves, because even though shock waves have small eddy viscosity magnitudes, this low magnitude may be over compensated for by the extremely large intensities of energy dissipation. Therefore, it seems intuitive to filter out any shock waves by including a term inversely proportional to the volumetric compression work done by a shock. This is advantageous, as volumetric compression work within a viscous layer is relatively weak. Paparone and Tognaccini's F_{BL} metric [7], can be improved to have stronger correlations to BLs and weaker correlations to shock waves using a ratio of viscous dissipation and volumetric pressure work (which are the terms comprising internal energy, refer to Chapter 4):

$$\xi = \left(\frac{\mu + \mu_t}{\mu} \right) \left[\frac{(\nabla \cdot \bar{\tau}) \cdot \mathbf{V}}{|p \nabla \cdot \mathbf{V}|} \right] \quad (5.4)$$

Due to the increased strength in correlation to the BL region, and assuming a sufficiently large domain required by external aerodynamic problems, a convenient method for calculating an appropriate cut-off value for ξ was found to be the volume weighted average of ξ over the whole domain¹. The BL region is therefore given by the condition:

$$\xi > \frac{1}{\nu} \iiint_{\nu} \xi \, d\nu = \frac{1}{\nu} \sum_{i=1}^n \xi_i \quad (5.5)$$

where n is the number of cells comprising the entire fluid domain. This is very convenient because, unlike the Paparone and Tognaccini's method, there is no necessity to perform sensitivity studies to determine an appropriate cut-off value². Another feature of the ξ approach, is that it is largely able to discount the shock containing core regions of the jet plume.

Figure 5.1 is a example of the viscous region selector applied to a fuselage concentric propulsor. This particular example shows the robustness of the selector, as the flow contains both shocks and jets. Furthermore, having the propulsor submersed within a BL, and thereby splitting it, adds additional complexity to demonstrate the robustness

¹A volume weighted average was chosen arbitrarily over a mass weighted average as the results were very similar

²It is noted that this has only been tested on 2D and 2D axisymmetric simulations, and the author cannot comment on its validity in 3D flows when vortices are present

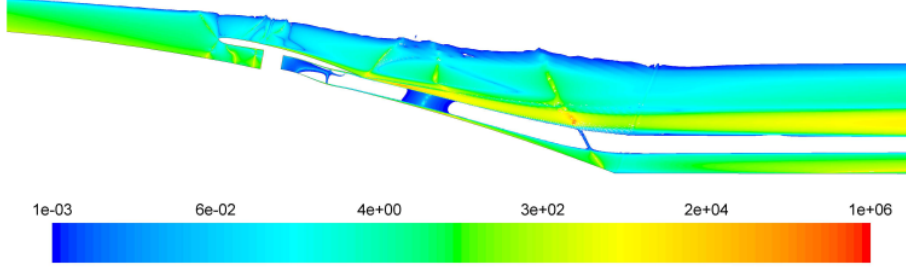


Figure 5.1: An example of the ξ viscous region for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$.

of this selector. Not evident in this figure, is the inclusion of the cells adjacent to the fuselage and nacelle walls, which are omitted by Paparone and Tognaccini's [7] selector. The region obtained by ξ also excludes the portion of the shock external to the BL. The shock and BL interaction regions are closely coupled, and splitting these regions is dependent on the precedence given, which is discussed further in Section 5.1.4. For the most part, the selector successfully omits the jet region (or rather jet core, following the definitions of Franquet [38]) apart from areas where $|p\nabla \cdot \mathbf{V}|$ becomes very small between jet expansion and compression regions. Figure 5.2 illustrates these areas using a metric defined in Section 5.1.2. In these areas, there is some local dissipation due to compressible effects in the jet core and although small, its presence in ξ becomes amplified in adjacent areas where $|p\nabla \cdot \mathbf{V}|$ becomes very small. This is the primary weakness of ξ in robustly isolating the BL regions of the flow. Nonetheless, there will always be close coupling between the jet region and viscous shear (particularly in the jet transition region where mixing is dominant). Dealing with these coupled interactions is discussed in Section 5.1.4 where region precedence is used to overcome these difficulties.

5.1.2 Shock-Wave Regions

A review of different shock region detection methods is presented by Lovely [40]. These are not discussed here, for the sake of brevity, and based on the conclusion that a suitable method has been developed by Paparone and Tognaccini [7]. Paparone and Tognaccini suggest a non-dimensional parameter based on pressure spatial gradients and local speed of sound:

$$\zeta = \frac{\mathbf{V} \cdot \nabla p}{a|\nabla p|} \quad (5.6)$$

Outside of the BLs, expansion regions ($\zeta < 0$) are not associated with energy dissipation. In fact, as is evident from Equation (4.16), expansion ($p\nabla \cdot \mathbf{V} < 0$) mitigates viscous dissipation in the combined form of internal energy. This is evident by the expansion

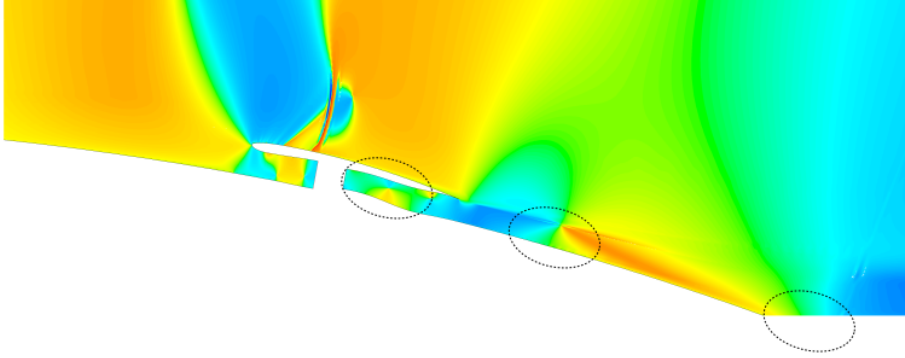


Figure 5.2: Contours of ζ (see Section 5.1.2) showing regions where $|p\nabla \cdot \mathbf{V}|$ becomes very small, for an example fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$.

wave BL interaction region ahead of a shock. Therefore, regions of expansion outside the BLs and jet plumes are filtered out of the domain, and any associated dissipative energy is considered to be numerical spurious contributions due to artificial dissipation and discretization errors.

A particular advantage of the above sensor to filtering out shock regions, is that it is roughly representative of the local Mach number. Therefore, compressive regions ahead of the shock (which contribute to energy dissipation) are located where $\zeta > 1$. The subsequent difficulty in determining a valid cut off value between 0 and 1, is finding the approximate Mach number in the compression region just behind the shock. Paparone and Tognaccini [7] suggest using values calculated from Rankine-Hugoniot relations where applicable, but in other cases the free-stream Mach number provides an adequate estimate. The cut-off value is trickier to determine in regions of BL shock interactions, where shock partitioning takes precedence over BL partitioning. Low cut-off values will incorporate far more cells of the interaction region within the shock partition, and vice versa. Ueno *et. al.* [41] used a procedure by slowly increasing the cut-off value from low values, and monitored drag to determine a point where the viscous drag and wave drag contributions began to plateau. This is possible because most of the wave drag associated dissipation occurs at high values of ζ . However, this is time consuming to perform for large design space explorations, and in the interests of automation it was decided to use a factor of the free-stream Mach number. Therefore, from experience and trial and error, the shock region is defined by:

$$\zeta > 1.1M_\infty \quad (5.7)$$

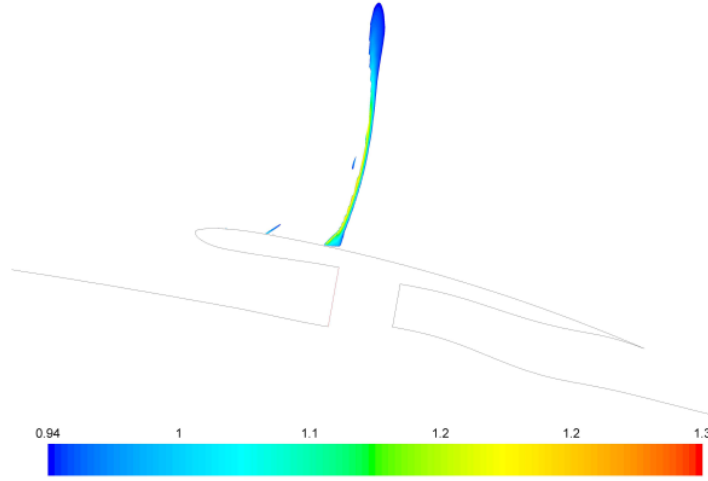


Figure 5.3: An example of the ζ shock region for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.6$.

This is permissible as any parts of the shock found outside of this region, tend to have small contributions to energy dissipation. Figure 5.3 gives an example of how the ζ selector may be used to extract the shock region. This figure shows the shock extracted above the nacelle of a fuselage concentric BLI propulsor. As can be seen, the shock selector extracts the shock region, including portions submersed in the BL. This is of great advantage, as taking precedence over the BL selector ξ allows for the shock's dissipation to be fully quantified. This precedence approach is detailed by Paparone and Tognaccini [7]. It must also be noted that ζ also filters out shocks within the jet core, and can be used to assess shock dissipation contributions of jets if the right precedence procedure is used (this is discussed further in Section 5.1.4).

5.1.3 Jet Region Partitioning

Another region of interest is the jet plume, and its associated losses. Sato [19], for instance, bases a definition of propulsive efficiency on the jet losses (although this definition is not adopted in this work). Jet structures are highly complex, as is evident from the review by Franquet [38], who looks at the expansion of jets without impingement. The jets of BLI propulsors will impinge on surfaces, leading to more complex internal structures. Typically, jets may be separated into a near-field (containing a core and mixing layer), transition and far-field zones. Jet cores contain an array of shock and expansion wave patterns, leading to further substructures. It would be of great interest to be able to identify all of these structures, and understand their associated loss

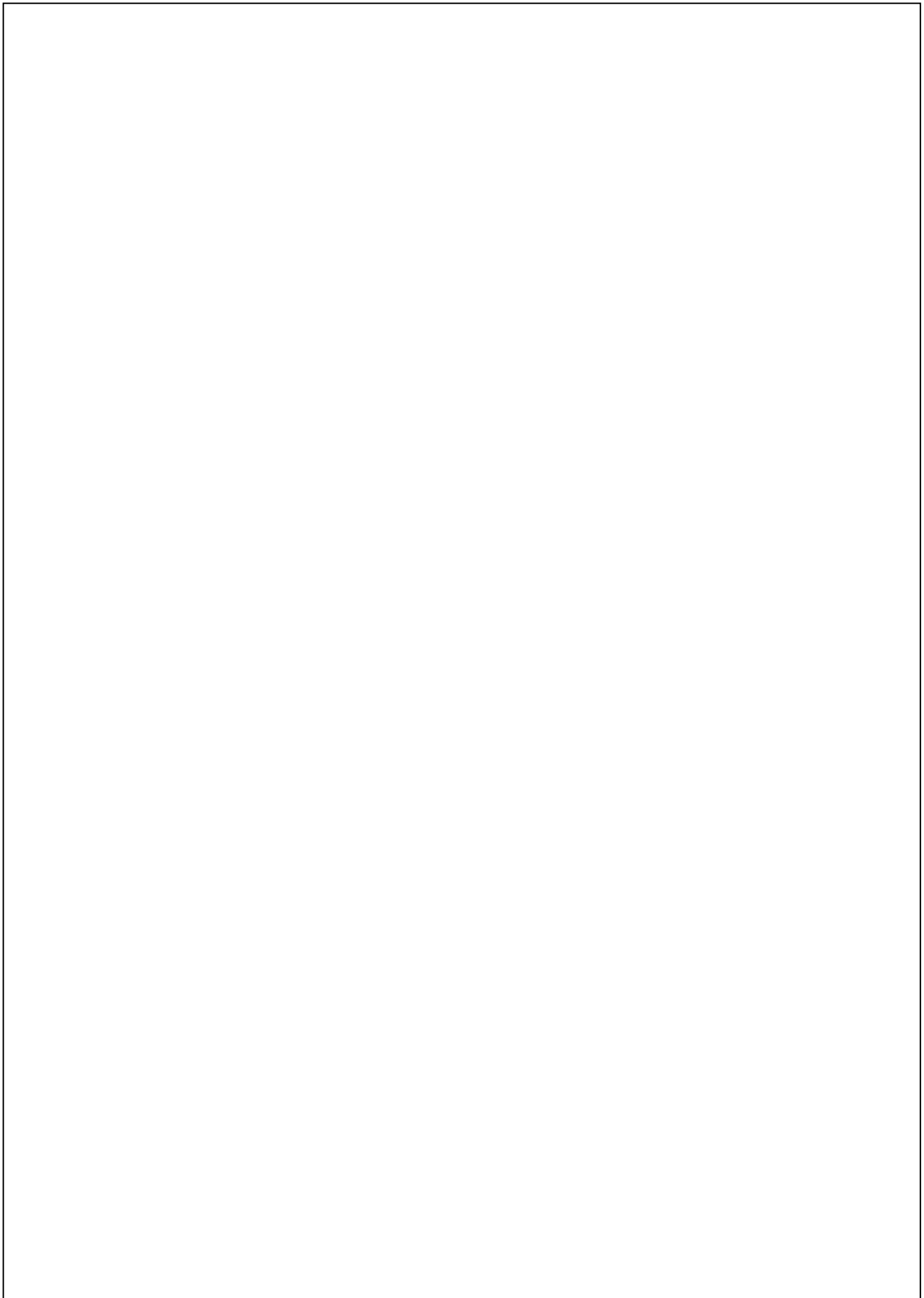




Figure 5.5: ζ contours for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

retains that precedence, but within a procedure that accounts for the jet region too.

The precedence procedure proposed in this thesis, is demonstrated on a fuselage concentric BLI propulsor example where the propulsor is positioned midway down the fuselage, ingests 40% of the BL and has a $FPR \approx 1.9$. The higher FPR is required to demonstrate how the procedure deals with compressible structures in the jet stream. Figure 5.5 illustrates the compression and expansion regions present in this sample case. The steps of the procedure are demonstrated by Figures 5.6 and 5.8 to 5.12.

Step 1: requires that the jet region takes precedence, and the cells allocated to the jet region are determined by Υ , i.e Equation (5.8). The total cells in this region are depicted in Figure 5.6, which shows contours of ζ within the jet region. The purpose of showing ζ contours, is to demonstrate how well Equation (5.8) encapsulates the compression and expansion waves within the jet plume. A schematic of the different shock structures for a free-jet without impingement, is shown in Figure 5.7, and it is clear how some of these structures are present in Figure 5.6. **Step 2:** requires separating out the jet region cells which are under strong compression (shocks). These shocks are present within the jet core region and are isolated using Equation (5.7), which is depicted in Figure 5.8. Quantifying energy dissipation in these cells gives an idea of the losses associated to shocks within the jet plume. **Step 3:** involves separating out the BLs and viscous turbulent regions in the jet flow that remains after removing the shock regions. Figure 5.9 depicts the viscous regions picked out by Equation (5.5). This figure makes the jet transition and core regions apparent, and it is interesting to see how this precedence procedure helps to decompose out the internal structures of the jet. In light of this discussion, the region depicted in Figure 5.9 may be assumed to be representative of the jet transition region (see Figure 5.7 and refer to Franquet [38] for a description

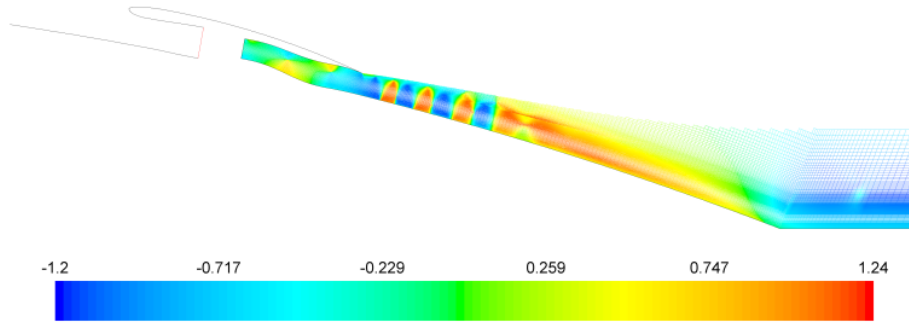


Figure 5.6: ζ jet contours (i.e. within the cells of the domain allocated in the jet region) for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

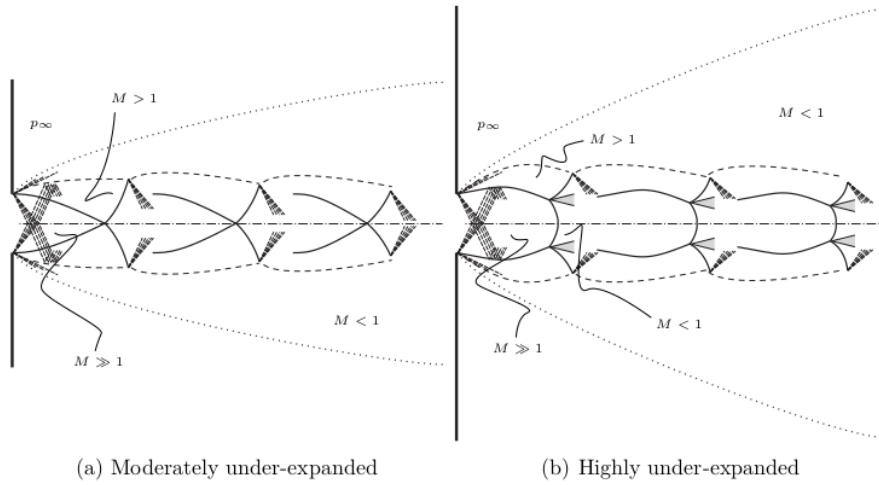


Figure 5.7: Flow structures of an under-expanded jet. Modified from Franquet *et. al.* [38]

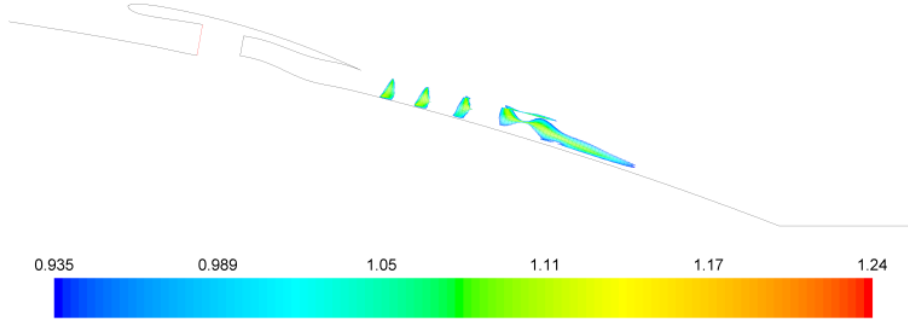


Figure 5.8: ζ contours within the jet's shock regions inside its core. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

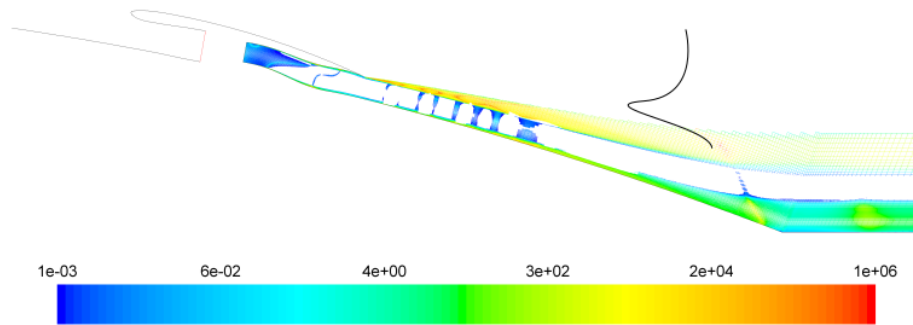
on jet structures).

Step 4: allocates the remaining cells to the non-dissipative (or expansion) region of the jet's core. Within this region there is no mixing, and the flow may be assumed to be expanding isentropically (or close to), resulting in no energy dissipation. Intuitively, the flow is fairly uniform in this region.

Step 5: requires separating out the cells associated to shocks in the flow outside of the jet plume. Precedence is given to the shock within the BL shock interaction region, based on the arguments by Paparone and Tognaccini [7]. Equation (5.7) is applied to the flow outside of the jet region, which extracts the shock over the top of the nacelle as depicted in Figure 5.11.

Step 6: is the final step, which requires isolating the BL and viscous wake regions from the external flow remaining after separating out the jet and external shock. Figure 5.12 depicts this region, where the shock and jet regions can visibly be seen to be absent.

The ability of the precedence procedure to decompose the flow-field into various mechanisms has been successfully demonstrated for a complex scenario of a tightly integrated propulsion airframe system. This is deemed to be unique to this work, along with the ξ region selector introduced in this section. The advantage hereafter, is that the energy dissipation volume integrals (i.e. $\dot{E}_{T\nu}$) may be applied to each region to quantify their individual contributions to the overall NVF . This is very useful in aiding more strategic design space explorations, detailed design and understanding the potential benefits of BLI. It is noted here that this procedure has only been applied to a non-lifting configuration in 2D axisymmetric flow, and that further developments are required to



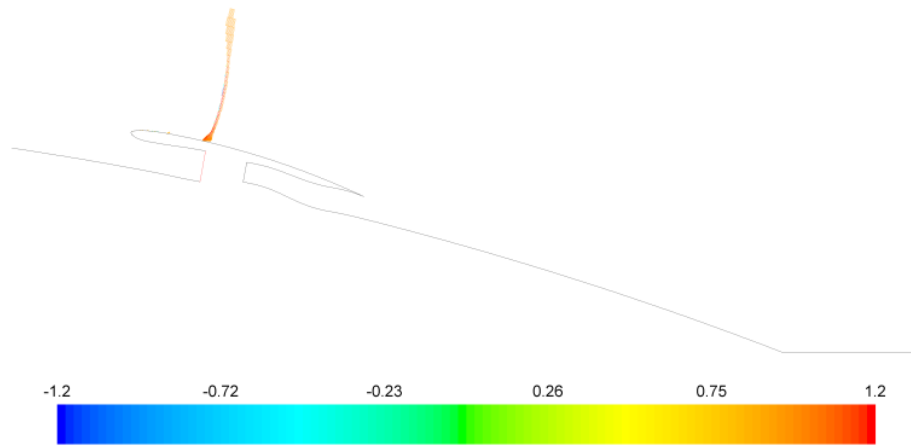


Figure 5.11: ζ contours within the external shock region. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

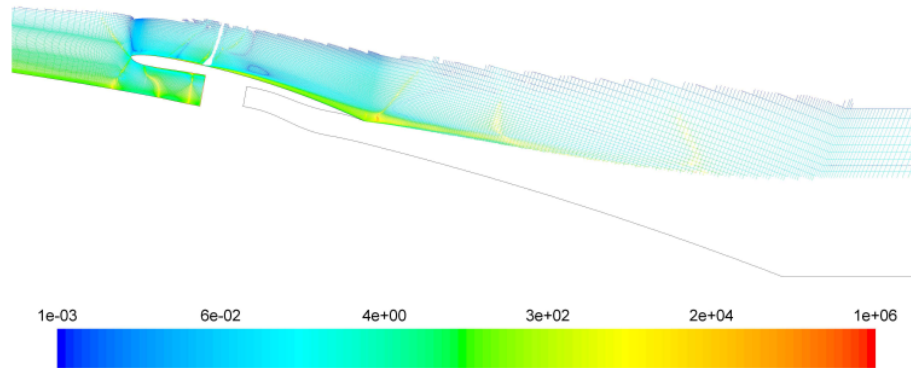


Figure 5.12: ξ contours within the external viscous regions. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

extend it to incorporate 3D effects.

5.1.5 A Note on Spurious Drag Removal

Paparone and Tognaccini [7] claim that spurious drag removal is one of the key advantages to spatial decomposition of the various flow mechanisms in numerical solutions. This is based on the idea that significant amounts of spurious drag generation is contained within the bulk flow regions, i.e. outside of BLs and shocks. Based on this assumption, the logical step is to identify cells outside of the BL and shock regions and determine the energy dissipation in these regions using volume integrals entropy drag. Paparone and Tognaccini [7] then define this as the removable spurious drag, noting that there is still spurious drag in the BL and shock regions which cannot be removed. This is a reasonable approach, however it was found, in this thesis that, the dissipation in the bulk flow regions (using the $\dot{E}_{\tau\nu}$ definition presented in Chapter 4) was negligible and this removal process is unnecessary. This was reaffirmed in a momentum-based far-field analysis of a non-propelled fuselage (detailed in Section 5.3.2) which showed the spurious drag to be concentrated in the BL region. The author notes that this may be code dependent, and other CFD codes may have a stronger presence of spurious drag in the bulk flow. Nonetheless, all codes are likely to contain much higher contributions to spurious drag within the dissipation regions of interest. Unfortunately the author is not aware of any procedures which may be implemented to remove this contribution (within the BLs and viscous wakes) at a post-processing level.

5.2 Viscous Stress Tensor and Near-Field Surface Forces

The theoretical methods presented in Chapters 3 and 4 make extensive use of the viscous stress tensor, which has been mathematically detailed in Appendix A. ANSYS Fluent does not report the viscous stress tensor within the flow domain, but only reports shear stress force vectors on wall surfaces. As an end user, it is difficult to know exactly how the wall shear stress vector is calculated within the solver, and ANSYS Fluent (as will become evident) certainly makes use of smoothing functions and projections to deal with complex flows and curvature to predict, reasonably, the forces at wall surfaces. Nonetheless, as presented in Appendix A, the tensor may be obtained directly from velocity derivatives, viscosity, density and turbulent kinetic energy through Boussinesq's approximation. ANSYS Fluent internally constructs the viscous tensor in this way (at least this is what is reported in its theory guide [42], but there may well be additional corrections applied) which is used in solving the energy equation. Two-equation, eddy-viscosity RANS CFD models inherently struggle with complex curvature and rotating flows, and great care must be taken when deriving results directly from velocity and pressure gradients. Naturally, the resolution of the velocity derivatives is highly dependent on the interpolation scheme and near-wall treatment, and this section investigates the sensitivity of this resolution to different interpolation schemes and $y+$ grids. This is an essential contribution to this work, as it is important to ascertain

(and hopefully achieve) whether RANS CFD is capable of producing continuous viscous tensor flow-fields which, from an energy balance standpoint, produce force estimations consistent with those obtained from momentum-based analyses. It is emphasised here that it is this consistency which is of primary interest in verifying the methods, and not the actual accuracy of the results which would validate their use. ANSYS Fluent offers three-different interpolation schemes; Green-Gauss Cell-Based (GGCB), Green-Gauss Node-Based (GGNB) and Least-Squares Cell-Based (LSCB). ANSYS Fluent claim the GGNB interpolation scheme as being the most accurate, with the GGCB having the least accuracy [42]. Nonetheless, the LSCB scheme is recommended as its accuracy is comparable to the GGNB, but with the advantage of quickening computational time. However, as has already been mentioned, the primary interest of this section is to study consistency in the viscous tensor field and not accuracy of the result, and it is therefore worthwhile exploring each of the gradient schemes.

To establish whether consistency is attainable, the viscous tensor flow-field is examined at surface walls and throughout the BL profile. This study has been performed on a non-propelled 2D axisymmetric fuselage, with geometry corresponding to the baseline aircraft. In this work, the baseline's fuselage geometry was determined such that it matched the drag predictions of an analytical empirical model which forms a part of a drag build-up method to predict the aircraft drag (see Appendix B). This step is a fundamental step for CFD evaluations of a propelled fuselage, in order to ensure consistency. The fuselage drag matching is detailed in Appendix C, and this includes geometry definitions, meshing, GCI studies and the representativeness of the final geometry. In brief, the fuselage geometry is defined as having an elliptical nose, cylindrical mid-body and circular arc tail-cone. This geometry is useful (and preferred to an airfoil) as it includes the types of curvature which will highlight potential issues in directly obtaining the viscous stress tensor from velocity derivatives. This is because of the cylindrical mid-body section, which will not suffer these problems. It is also noted that the simple geometry uses first order curvature at junctions between the three-sections, which is also known to introduce numerical difficulties.

A pressure-based coupled solver has been used (this solver is valid for the flight conditions under consideration [42]), in combination with the $k - \omega$ SST turbulence model. The $k - \omega$ SST turbulence model was selected due to its near-wall treatment method, which (for appropriate y^+) allows for valid integration of flow properties down to the wall surface [42]). Additionally, it is extensively used in external aerodynamic applications because of advantages over its rivals; $k - \epsilon$ and Spalart-Almaras (see Versteeg and Malalasekera [39] for a discussion on this). Nonetheless, the key is the ability to integrate fluid properties down to the wall surface, as this is crucial to being able to apply the energy based methods reliant on decomposing out BL energy dissipation. Although Fluent is a finite volume based method, a structured mesh approach was taken to better align the cells with the flow streamlines, as per the recommendations in the Fluent Theory Guide [42]. A C-grid was used, and BL growth ratios of 1.05 (Fluent recommend a growth ratio < 1.1 for sufficient BL resolution [42]) enforced to ensure sufficient BL resolutions.

The above discussions may be summarised by the following question: Are momentum-based and energy-based methods consistent with one another when applied to numerical two-equation RANS CFD solutions? The answer to this question is dependent on the continuity in the viscous stress tensor field, which is in turn dependent on interpolation schemes, turbulence modelling and near-wall treatment. The $k-\omega$ SST model is a suitable turbulence model allowing for integration through the viscous sub-layer down to the wall. Therefore, in order to answer the above question, the viscous field continuity for the different interpolation schemes available (GGCB, GGNB and LSCB) needs to be examined. This needs to be performed in conjunction with $y+$ sensitivity to account for viscous stress tensor continuity within the BL.

Figure 5.13 depicts the $y+$ values calculated at the surface walls along the fuselage nose, mid-body and tail corresponding to different first cell thickness', which were kept constant along all surfaces. The results are plotted against a non-dimensional length x/c of each fuselage component. The different first cell thickness' used are represented in the plot's legends by area weighted averages of the $y+$ values along each surface. The $k-\omega$ SST model is known to perform well for $y+$ close to (or slightly less than, as very small values start to incur errors) unity [42]. A range of first cell thickness' were selected to examine the sensitivity of the Boussinesq constructed viscous stress tensor to variations in $y+$ values below and above unity. Solutions were obtained for each grid using the three different gradient schemes offered by Fluent; GGCB, GGNB and LSCB. Each scheme produced slightly different $y+$ results, as the $y+$ parameter is calculated according to the viscous stress solution at the wall.

?? illustrates friction coefficients corresponding to Figure 5.13, as reported directly by ANSYS Fluent. These result indicate an insensitivity to $y+$ values for each of the schemes. This insensitivity is more pronounced in the GGCB scheme, as slightly larger deviations are evident in the other schemes, particularly along the tail where the LSCB scheme was used. A very different result is depicted in Figure 5.15, where the friction coefficient (from Boussinesq's approximation) has been calculated from velocity derivatives reported by ANSYS Fluent at the cell nodes. The GGCB scheme, unlike the other two, retains smoothness along each surface and a fair consistency between $y+$ values. The GGNB depicts very strange results over the curved surfaces of the nose and tail, but has maintained some consistency over the cylindrical mid-body. Finally, the LSCB scheme, seems to follow the correct trends, but demonstrates large sensitivities to $y+$ and erratic discontinuous behaviour over the nose and mid-body sections. Figure 5.16 depicts the associated rate-of-strain tensor invariant (please see Appendix A.2), which clearly shows how the schemes (apart from the GGCB) struggle to cope with the curvature of the nose and tail surfaces. Through a comparison of Figures 5.14 to 5.16, it is apparent that ANSYS Fluent contains internal methods for smoothing and correcting shear stress forces, and does not report them as direct outcomes of Boussinesq's approximation using the reported velocity derivatives.

A comparison of the friction coefficients is depicted in Figure 5.17. It is clearly evident that the Boussinesq calculated friction coefficient produces fairly consistent results to the Fluent reported values when using the GGCB scheme. Using this scheme it is seen that

there is a significant sensitivity to $y+$ values, particularly where the $y+ > 1$. However, for $y+ < 1$ the GGCB scheme results in almost no difference between Fluent reported and Boussinesq calculated skin friction coefficients. The only significant error occurs at the leading and trailing edge stagnation points, because in these regions the rate-of-strain approaches zero and any slight difference becomes magnified as a percentage. In addition, the turbulent viscosity at the sharp trailing edge's node is misreported, as can be seen in the top graph of Figure 5.19. Fluent is a finite volume solver and node values are interpolated, which means that this error will occur at the sharp trailing edge. The bottom graph shows how the turbulent viscosity should approach zero. Returning to Figure 5.17, the comparison shows a poor correlation for the GGNB and LSCB schemes, and because of this it is reasoned that the GGCB scheme is best suited for producing consistent viscous stress tensor flow-fields for energy-based assessments. The other conclusion is that $y+$ sensitivity is significant and it is important to ensure $y+ < 1$ over all the surfaces.

A final summary of the integrated near-field forces on the fuselage compared to Fluent reported forces, are detailed in Table 5.1. Stream-wise viscous $\Psi_{\tau,z}$, pressure $\Psi_{p,z}$ and net Ψ_z are given as a % difference from those reported directly by Fluent. Considering the difference in viscous forces, it is evident that the GGNB scheme is unsuitable due to the severe discrepancies located over the curved surfaces of the nose and tail. Then in comparing the GGCB and the LSCB, it may be concluded that similar consistencies are achievable where $y+ < 1$, but that the former outperforms the latter where $y+ > 1$. Based on these results, it is fair to conclude that the GGCB scheme offers the best consistency with the least sensitivity to variations in $y+$. Considering the pressure forces, it can be seen that there is a discrepancy, particularly on the nose portion of the fuselage. A source of error occurs in post-processing where face-centred values are interpolated from node values, along with the normal vector. Subsequently, a small discrepancy arises and is amplified close to the stagnation point, but its contribution to the net force error is insignificant. Additionally, as expected, the pressure coefficient is largely unaffected by the scheme used and the $y+$ value, as is seen in Figure 5.18.

5.3 Drag Prediction and Spurious Contributions

The purpose of this section is to extract the net force on the fuselage using an encapsulating control volume, and to compare the results against near-field values to determine uncertainties. The comparison also looks at momentum and energy-based methods of extraction, and the subsequent uncertainties. It was decided to present the comparison performed on the GGCB case with average $y+ = 0.85$ (refer to Table 5.1). This is because it yielded the smoothest viscous tensor flow-field which resulted in the closest consistency between the near-field forces reported by Fluent and those calculated from velocity gradients using Boussinesq's approximation. In this comparison, different control volumes were considered to examine consistency and the impact of spurious drag on the forces calculated. With reference to Figure 5.20, the first control volume encapsulates the vehicle with the back plane a few cells downstream of the trailing edge to avoid

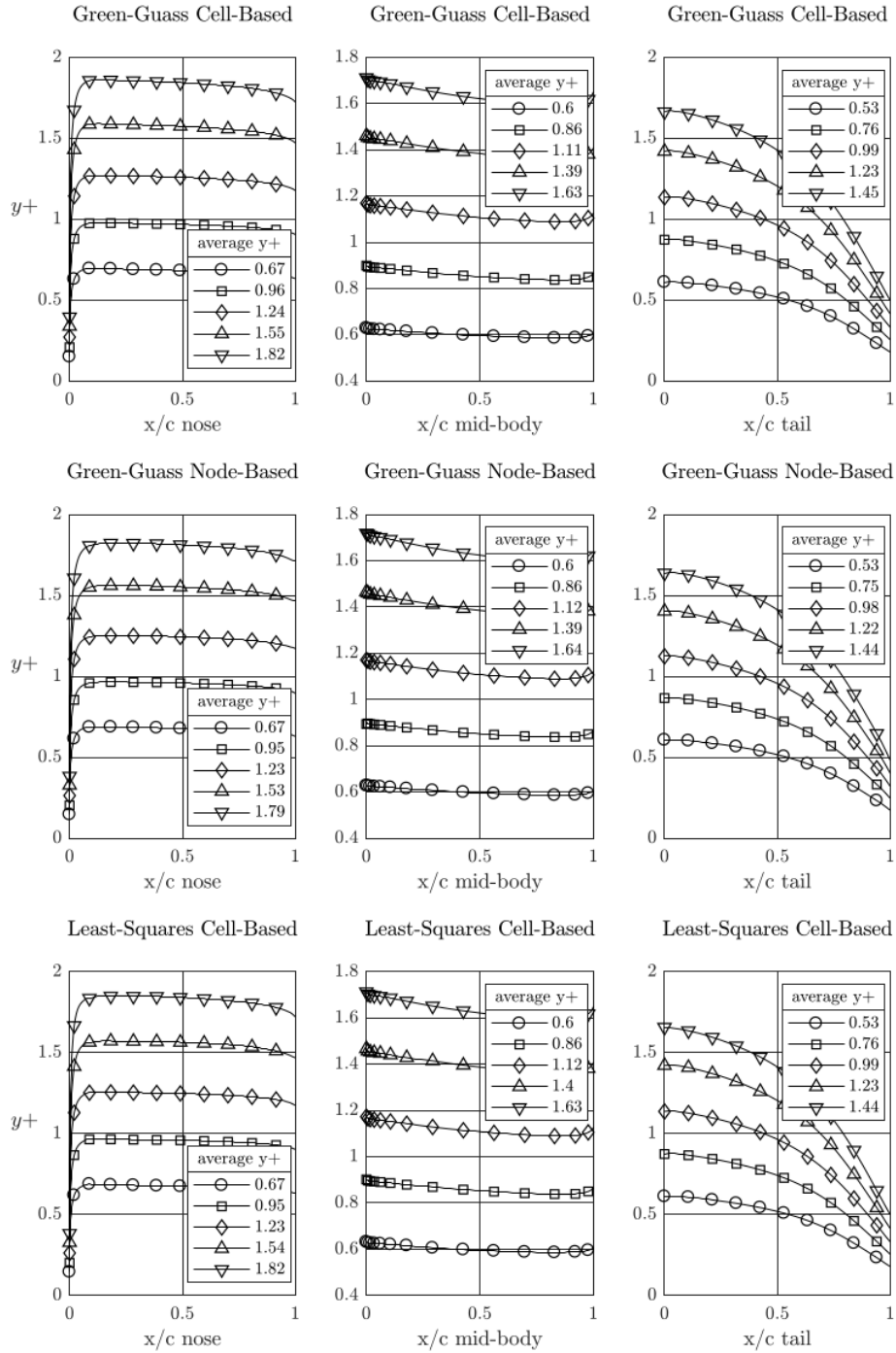


Figure 5.13: y^+ over isolated fuselage for different gradient schemes and first cell heights on the Non-Propelled Fuselage at FL450 M0.85

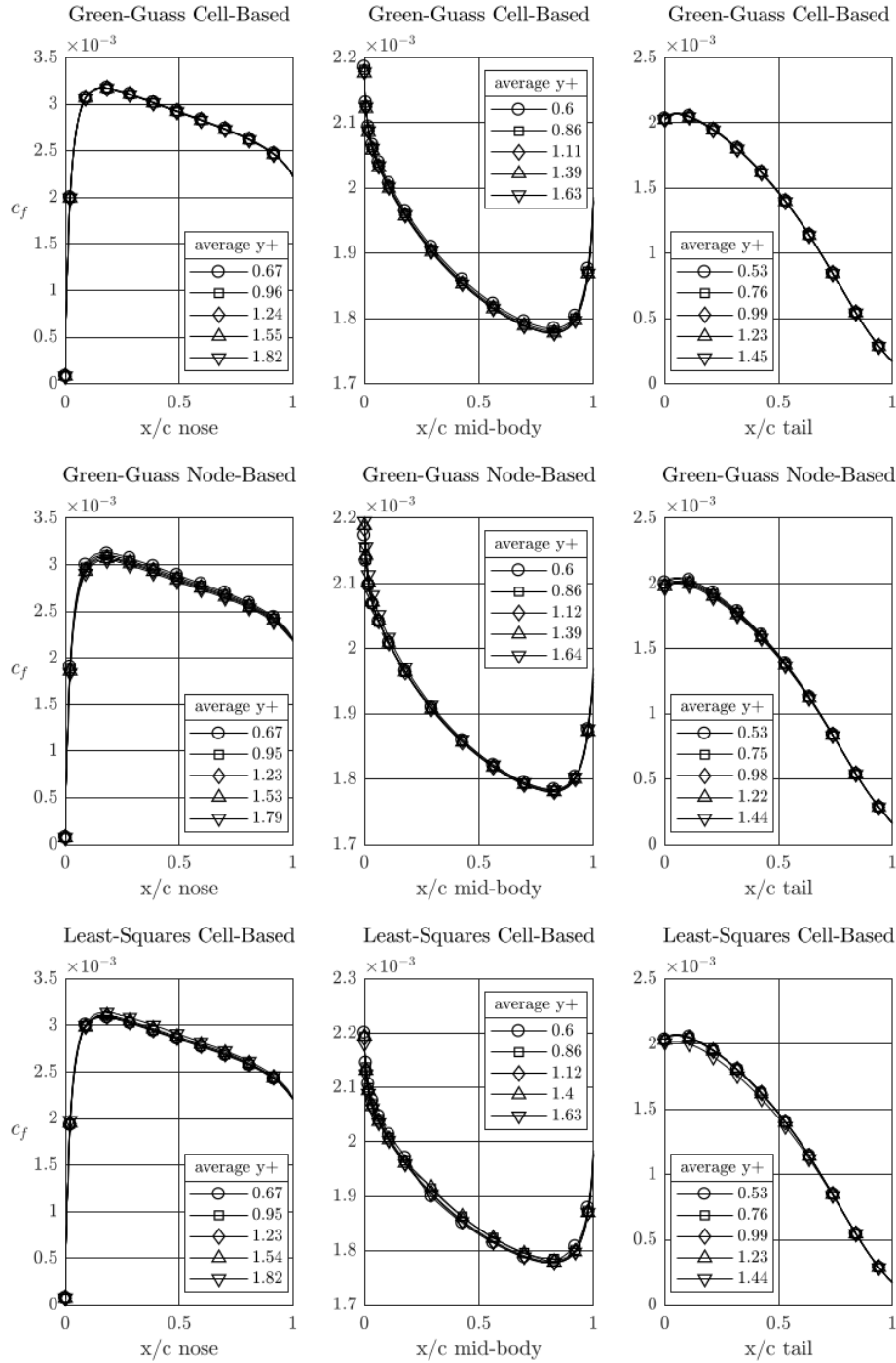


Figure 5.14: Friction coefficient corresponding to Figure 5.13 as reported by ANSYS Fluent on the Non-Propelled Fuselage at FL450 M0.85

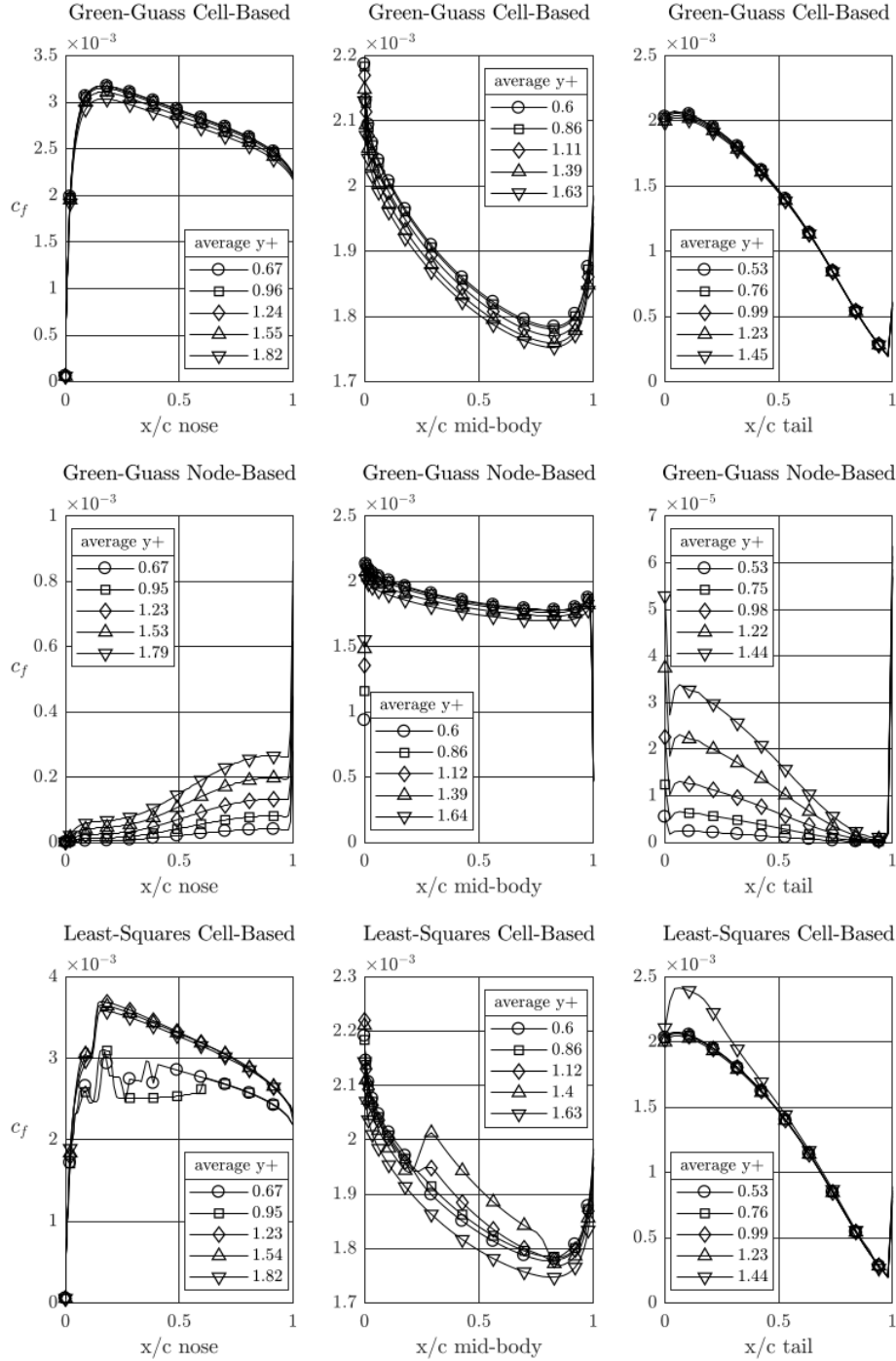


Figure 5.15: Friction coefficient calculated from velocity derivatives using Boussinesq's approximation on the Non-Propelled Fuselage at FL450 M0.85

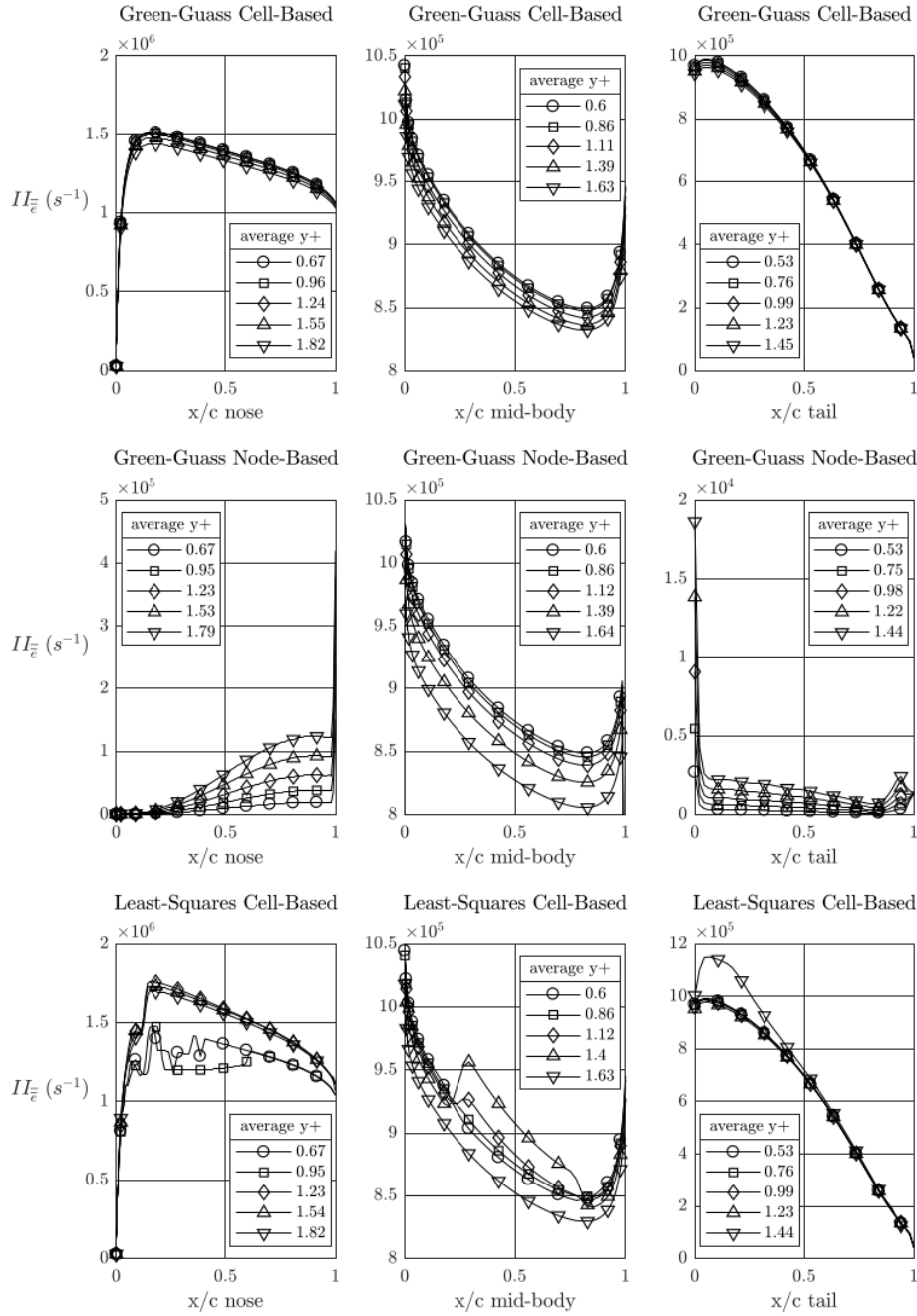


Figure 5.16: Rate-of-strain tensor second invariant corresponding to Figure 5.13 on the Non-Propelled Fuselage at FL450 M0.85

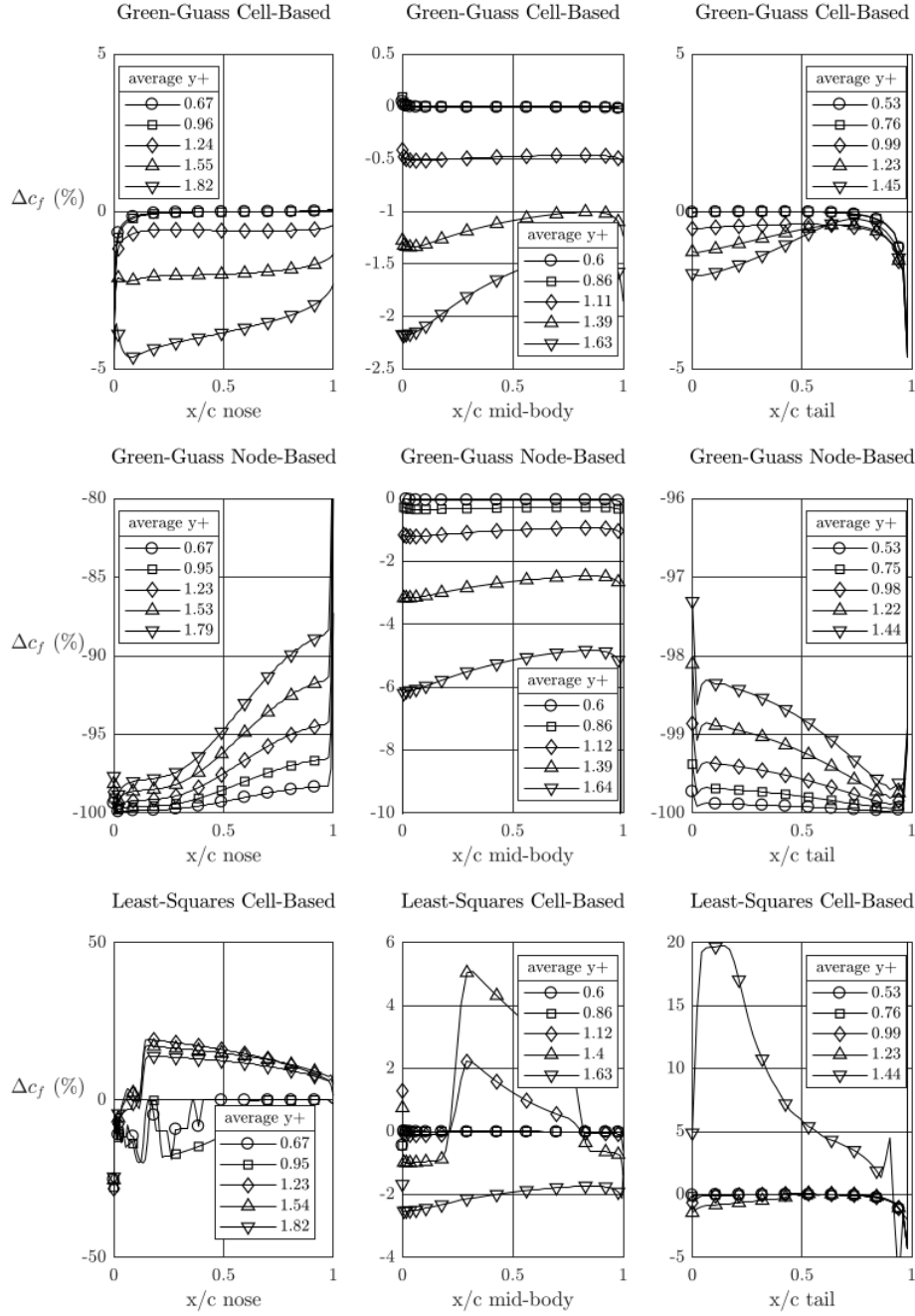


Figure 5.17: Friction coefficient difference between Figures 5.14 and 5.15 on the Non-Propelled Fuselage at FL450 M0.85

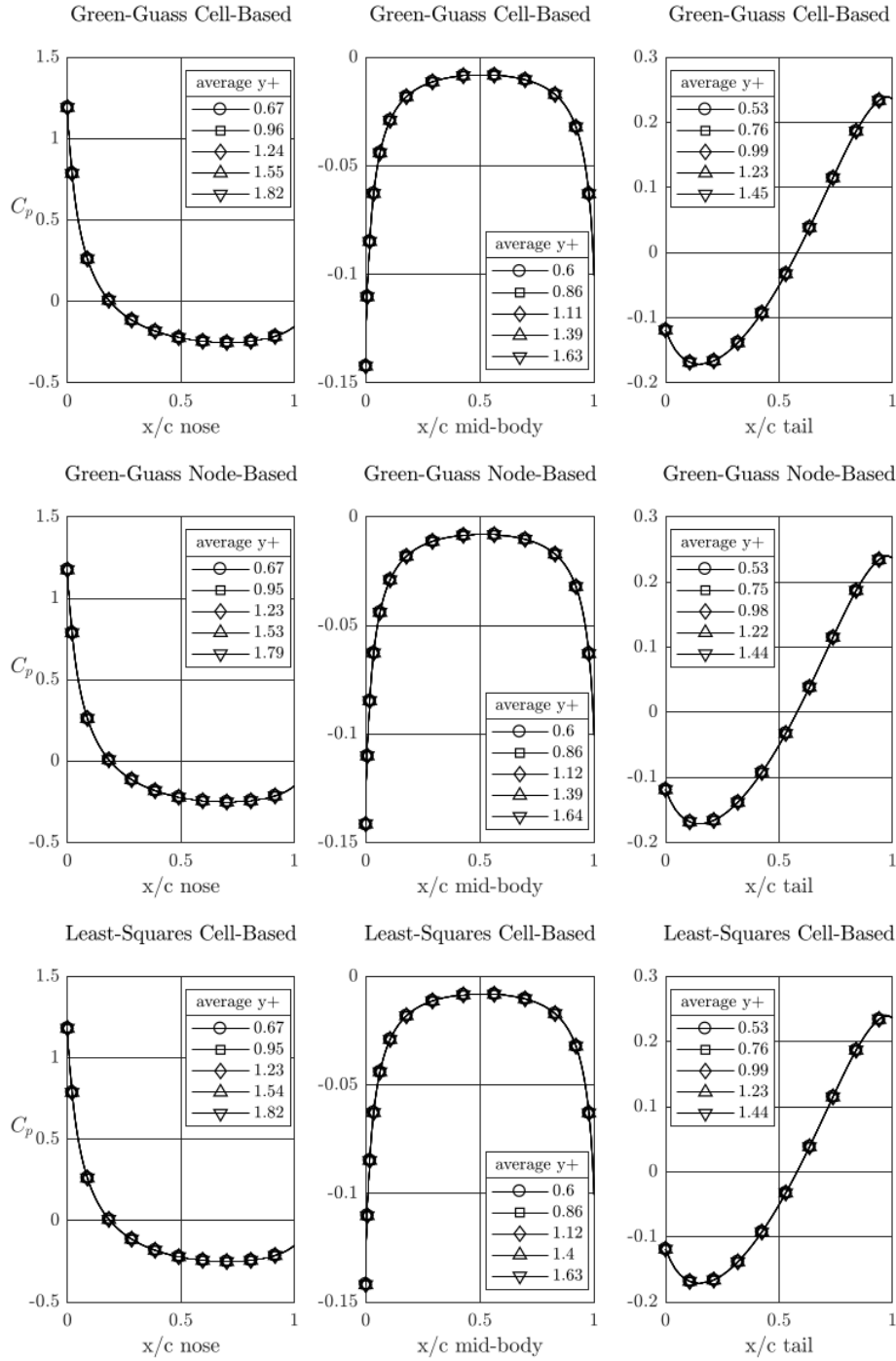


Figure 5.18: Pressure coefficient to Figure 5.13 on the Non-Propelled Fuselage at FL450 M0.85



Table 5.1: Summary of integrated viscous, pressure and total stream-wise forces on the non-propelled fuselage at FL450 M0.85, showing a comparison against Fluent reported values

Scheme	avg. $y+$	$\Delta\Psi_{\tau,x}(\%)$				$\Delta\Psi_{p,z}(\%)$				$\Delta\Psi_z(\%)$			
		Nose	Mid-Body	Tail	Net	Nose	Mid-Body	Tail	Net	Nose	Mid-Body	Tail	Net
GGCB	0.60	-0.87	0.25	-0.56	-0.04	8.85	0.00	-1.59	2.28	0.32	0.25	-0.80	0.10
	0.85	0.04	0.02	-0.08	0.01	8.88	0.00	-1.60	1.50	1.13	0.02	-0.44	0.10
	1.11	-0.42	-0.47	-0.57	-0.48	8.89	0.00	-1.60	2.29	0.73	-0.47	-0.81	-0.31
	1.39	-1.57	-1.13	-1.07	-1.20	8.90	0.00	-1.60	2.29	-0.28	-1.13	-1.20	-0.99
	1.63	-3.13	-1.70	-1.64	-1.93	8.90	0.00	-1.60	2.29	-1.65	-1.70	-1.63	-1.68
GGNB	0.60	-98.96	-0.86	-99.97	-29.80	8.84	0.00	-1.62	2.25	-85.64	-0.86	-76.72	-27.90
	0.85	-97.95	-1.09	-99.95	-29.68	8.94	0.00	-1.63	2.26	-84.79	-1.09	-76.69	-27.80
	1.11	-96.64	-1.80	-99.92	-29.87	9.01	0.00	-1.64	2.27	-83.67	-1.80	-76.64	-27.98
	1.38	-95.01	-3.47	-99.90	-30.69	9.08	0.00	-1.64	2.29	-82.26	-3.47	-76.60	-28.76
	1.62	-93.33	-6.03	-99.87	-32.09	9.15	0.00	-1.65	2.30	-80.79	-6.03	-76.55	-30.08
LSCB	0.60	-2.92	0.01	-0.07	-0.48	9.05	0.00	-1.60	2.29	-1.46	0.01	-0.43	-0.32
	0.85	-6.54	0.02	-0.09	-1.07	9.03	0.00	-1.60	2.29	-4.63	0.02	-0.44	-0.87
	1.11	13.48	0.64	-0.16	2.65	9.00	0.00	-1.60	2.29	12.93	0.64	-0.49	2.63
	1.39	13.05	1.69	-0.68	3.26	8.97	0.00	-1.60	2.29	12.55	1.69	-0.89	3.20
	1.62	10.82	-2.04	11.90	1.88	8.88	0.00	-1.64	2.30	10.59	-2.04	8.70	1.91

badly resolved flow-field at the stagnation point. The BL selector ξ (Equation (5.5)) was used to determine the boundary and viscous wake region, so that control volume size was ensured to fully encapsulate the BL³. From this position, Δs was increased to a location where $p \approx p_\infty$ (and beyond) to examine the influence of far-field extent on the force calculated. Subsequent control volumes were placed further downstream, at intervals Δz , such that they could be summed to represent larger control volumes encapsulating the fuselage. In the analysis, the contributions to force on the fuselage are given in terms of a general force coefficient:

$$C_F \cdot S_{\text{ref}} = \frac{F}{\frac{1}{2}\rho_\infty V_\infty^2} \quad (5.9)$$

or in energy-based assessments:

$$C_{\dot{E}} \cdot S_{\text{ref}} = \frac{\dot{E}}{\frac{1}{2}\rho_\infty V_\infty^3} \quad (5.10)$$

5.3.1 Near-Field Momentum-Based Force Coefficient

The near-field forces have already been presented in Section 5.2, to which the reader is referred. In this work, the Fluent reported near-field force will be used as a reference.

³ ξ is defined in Section 5.1

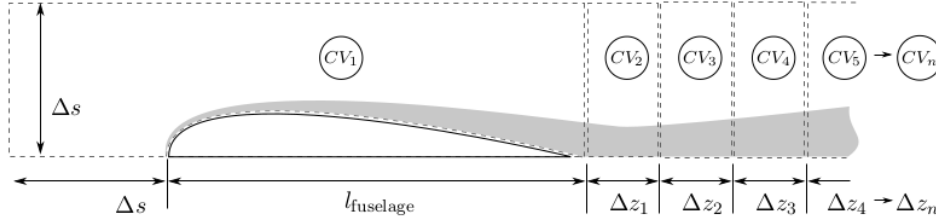


Figure 5.20: Control volumes with varying size and trefftz plane locations.

It is important to state that near-field forces are not favoured in being more correct than far-field (be it momentum or energy-based approaches), but are simply used as a reference for comparison.

5.3.2 Far-Field Momentum-Based Force Coefficient

The force acting on the fuselage body is obtained from the outer control volume surfaces (see Section 3.3 for details). Figure 5.21 illustrates the force coefficient obtained from momentum conservation for various control volumes encapsulating the fuselage. This was performed for Δs values just encapsulating the BL, to two fuselage lengths away from the body (along with a number of different trefftz plane locations, up to seven fuselage lengths away). Figure 5.22 depicts additional points at trefftz plane locations extending further downstream (up to 27 fuselage lengths). Both figures have included near-field force coefficients for reference, showing both Fluent reported values and values calculated using the viscous stress tensor calculated from Boussinesq's approximation.

Focussing on any one of the constant trefftz plane locations, the first noticeable trend is that the far-field method estimates smaller values in comparison to near-field predictions, and that the difference between the two increases with Δs . It is expected that near-field estimations tend to yield larger drag predictions, as has been shown by Desterac and van der Vooren [25]. It is also intuitive that the difference between near and far-field estimations widens as the control volume moves further away from the body. However, what may seem perplexing is that the force coefficient keeps decreasing with Δs , until converging two fuselage lengths away. One may have expected the opposite, as it might be assumed that numerical spurious drag increases with larger control volumes and the inclusion of more cells within the control volume. But a possible explanation for the trend is that the spurious drag (due to artificial dissipation and viscosity) is concentrated within the high shear regions of the flow (i.e. BLs and viscous wakes), and is insignificant in the almost potential flow regions of the bulk flow. This was indeed found to be the case when examining viscous dissipation contours throughout the flow. This argument does not fully explain the decrease in force coefficient, as all Δs locations should be expected to contain the same amount of spurious drag. In explaining the decrease in force as Δs increases, the flow properties at the control volume surfaces become more dependent on the bulk flow solution. This bulk flow solution is dependent

on the displacement thickness of the BLs, as would be the case in predicting drag using *effective bodies* in a potential flow solution (see Anderson [21] for an explanation on drag estimation from *effective body* potential flow solutions). RANS CFD is very capable at solving accurately these bulk flow regions, but poor in accurately resolving the BL. However, uncertainties in displacement thickness produced by the turbulence models is likely to be very small relative to the bulk flow. In other words, the further Δs is placed away from the body, the less sensitive it becomes to uncertainty in displacement thickness. Additionally, the bulk flow is not subject to spurious drag from artificial terms, so the result at larger Δs displacements is that the force estimation is subject to less spurious drag. This also gives explanation as to why near-field solutions suffer from possible drag over-predictions, as suggested in literature [7, 25, 26], because of the influence of spurious effects within the BL.

Shifting focus to any constant Δs location, the resulting trends for increasing the treftz plane downstream location may be discussed. Very close to the fuselage trailing edge, the results are unreliable as the treftz plane is located within a separation region. This is because RANS struggles with regions undergoing separation, and BLs in adverse pressure gradients. Additionally a sharp trailing edge has been used, which also affects the validity of the results in this region. There is an initial jump in the force which settles to a local minimum about one fuselage length downstream. The author is unsure of the reason for this, but suspects that it is possibly related to the shift from a bounded shear flow in the BL to a free-shear region in the wake. Of particular interest is the steady increase in force with increase in Δz due to an increase in spurious drag. This may seem to counter the argument above, where an increase in Δs resulted in an exclusion of spurious effects. However, increasing Δz incorporates more of the viscous wake and this trend makes sense recalling that spurious effects are concentrated in regions of shear. This is clearly evident by the parallel offset in force coefficient between different Δs CV positions. It is noted that one anomaly is present, where the trend-line of $\frac{\Delta s}{l_{fuse}} = 1/15$ begins to deviate significantly for $\frac{\Delta z}{l_{fuse}} > 7$. At this treftz location, Δs is too close to the body to fully encapsulate the viscous wake, thereby introducing errors.

To summarise, the force predicted decreases with the exclusion of spurious drag by pushing the CV out into the bulk flow. However, a large amount of spurious drag is incorporated when shifting the treftz plane downstream, due to encapsulating larger amounts of the shear regions in the wake. Based on the trends observed, it is assumed that the local minimums ($\frac{\Delta z}{l_{fuse}} \approx 1$) represent the force prediction least effected by spurious drag, particularly where Δs is positioned sufficiently far away. From Figure 5.23, it can be seen that this local minimum corresponds to approximately a 4% difference from the Fluent reported near-field values. It is important to note that the near-field has just been used here as a reference, and the comparison does not imply that the near-field is assumed to be the correct value, or more accurate. Nor does this section claim that the local minimum is more correct or accurate. The reasoning simply explains that it is hypothesised that this local minimum is the point least affected by numerical spurious

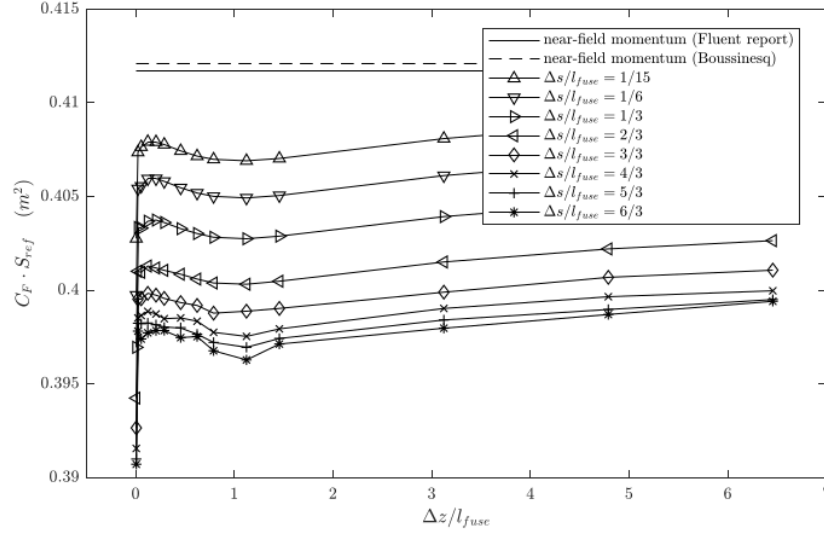


Figure 5.21: Momentum-based far-field extraction of the non-propelled fuselage force coefficient at FL450 M 0.85, for control volumes varying in size. Near-field result is also included for reference.

influences.

It must be noted that the above explanations are the author's hypothesis for the observed trends, and it is very difficult to prove without having access to the CFD source code. Nonetheless, given the evidence at hand, they do appear to be reasonable explanations and the author cannot think of another explanation which fits this evidence as well. The purpose of this section was to illustrate uncertainties present in momentum-based methods. As has already been mentioned, neither near or far-field methods are assumed to be the correct ones. However, examining their difference in force predictions, gives insight into the uncertainties in force prediction. It is important to examine these uncertainties, and this will be revisited for predictions using energy based methods.

5.3.3 Energy-Based Force Coefficient

To recap, the net force acting on a non-propelled body is given as a summation of kinetic energy flux, pressure boundary work, viscous boundary work, volumetric compression

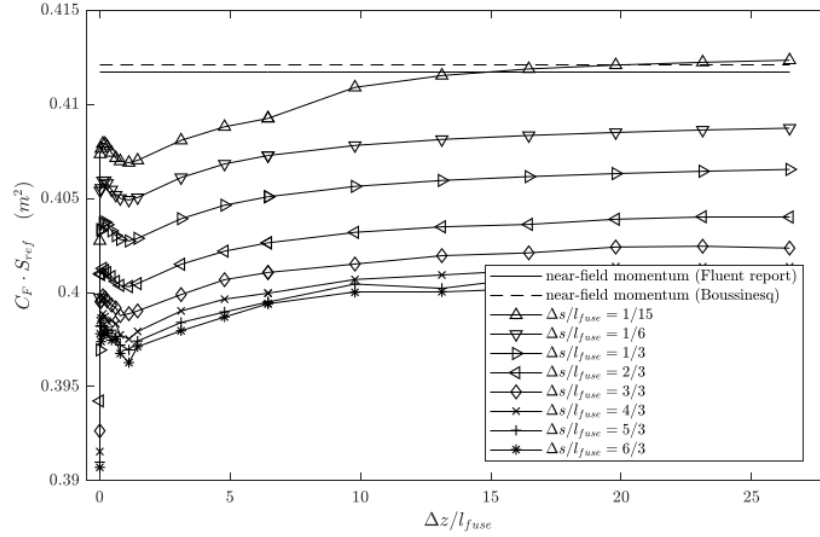


Figure 5.22: Momentum-based far-field extraction of the non-propelled fuselage force coefficient at FL450 M 0.85, for control volumes varying in size (extended to include more treftz plane locations). Near-field result is also included for reference.

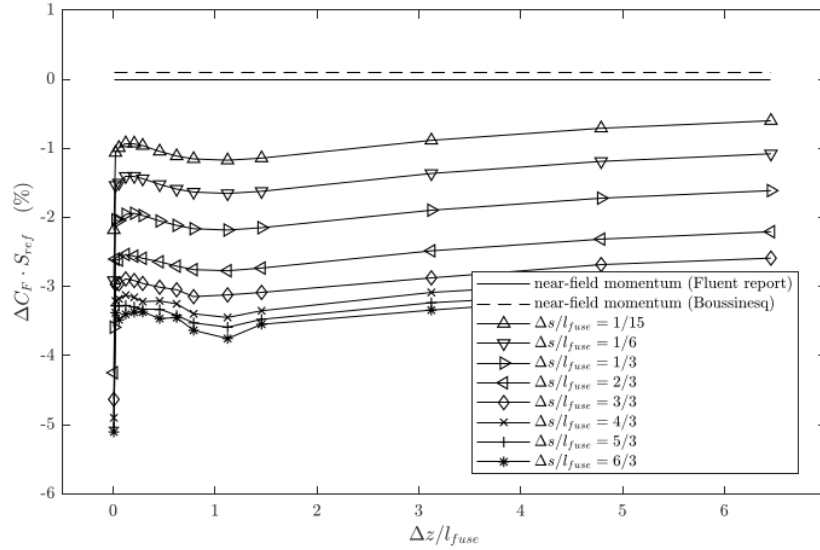


Figure 5.23: Comparison between the near and far-field momentum-based force coefficient extractions of the non-propelled fuselage force coefficient at FL450 M 0.85

work and volumetric viscous dissipation, respectively:

$$\begin{aligned}
 \mathbf{F}_B \cdot \mathbf{V}_\infty &= \dot{E}_k + \dot{E}_p + \dot{E}_\tau + \dot{E}_{p\nu} + \dot{E}_{\tau\nu} \\
 &= \iint_S \frac{\rho}{2} (\mathbf{V} - \mathbf{V}_\infty)^2 \mathbf{V} \cdot d\mathbf{S} + \iint_S (p - p_\infty) (\mathbf{V} - \mathbf{V}_\infty) \cdot d\mathbf{S} \\
 &\quad - \iint_S \bar{\tau} \cdot (\mathbf{V} - \mathbf{V}_\infty) \cdot d\mathbf{S} - \iiint_\nu (p - p_\infty) \nabla \cdot \mathbf{V} d\nu \\
 &\quad + \iiint_\nu (\bar{\tau} \cdot \nabla) \cdot \mathbf{V} d\nu
 \end{aligned} \tag{5.11}$$

This form of the energy-based formulations is preferred to internal energy flux methods, as it is the viscous dissipation and volumetric compression work that is of interest. The overall viscous dissipation may be obtained indirectly from the internal energy fluxes, however the purpose of this section is to examine uncertainties involved in using the volume integrals. The reason for this is that the volume integrals may be used in examining the contributions of different flow mechanisms, as discussed in Section 5.1. The same control volumes, as described by Figure 5.20, are utilized in examining the terms of Equation (5.11). The discussion below follows the order of terms in Equation (5.11), and ends with a summary of the net force and breakdown of contributions.

Kinetic Energy Flux $\dot{E}_k$

Figure 5.24 depicts how the force coefficient (associated with the rate of kinetic energy deposited to the flow) varies as the treftz plane is shifted further downstream. The general trend indicates a peak value at the trailing edge which diminishes asymptotically to zero. This is an intuitive result because the wake gradually returns (or asymptotically returns) to zero in the absolute frame of reference (or free-stream in the relative frame of reference) far downstream of the trailing edge. There is an insignificant difference in values for different Δs locations. This makes sense because outside the BL the velocity is close to zero in the absolute frame of reference, apart from minor differences due to local pressure fields.

Pressure Boundary Work $\dot{E}_p$

The change in pressure boundary work is depicted in Figure 5.25. Unlike the kinetic energy flux, the pressure boundary work has a negative peak which asymptotically diminishes to zero (for the most part). However, there are significant variations for different Δs positions. Too close to the body, and the asymptote crosses over the zero axes. This is because the CV surfaces are within a compressed region of the flow where static pressure hasn't returned to free-stream values. However, this offset is cancelled out by a corresponding offset in the $\dot{E}_{p\nu}$ term, as is depicted in Figure 5.27. At $\frac{\Delta s}{l_{fuse}} = 1$, the static pressure has returned to its surroundings and the asymptote falls on zero. The attenuation of this term is expected as the flow returns asymptotically to the surrounding static pressure. As a negative term, the force coefficient contributes to a thrust direction.

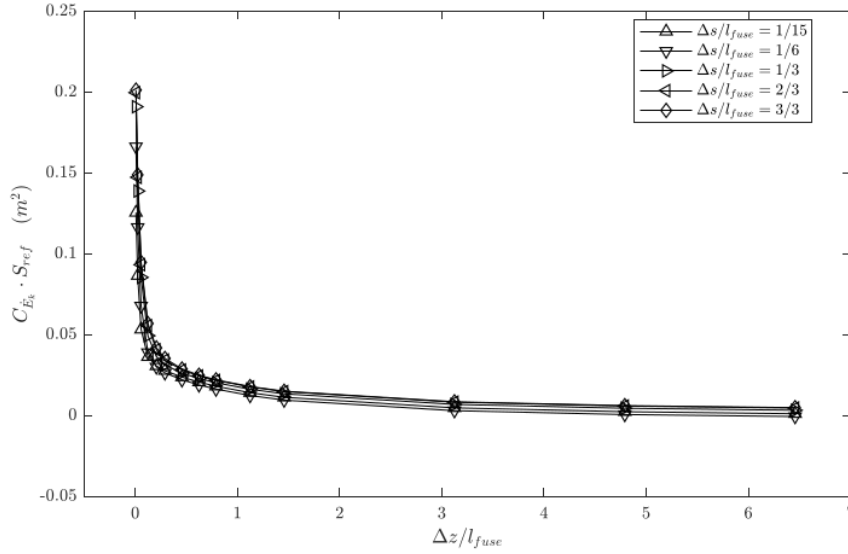


Figure 5.24: Variation in kinetic energy flux $C_{\dot{E}_k}$ corresponding to different treftz plane locations, for the non-propelled fuselage at FL450 M 0.85

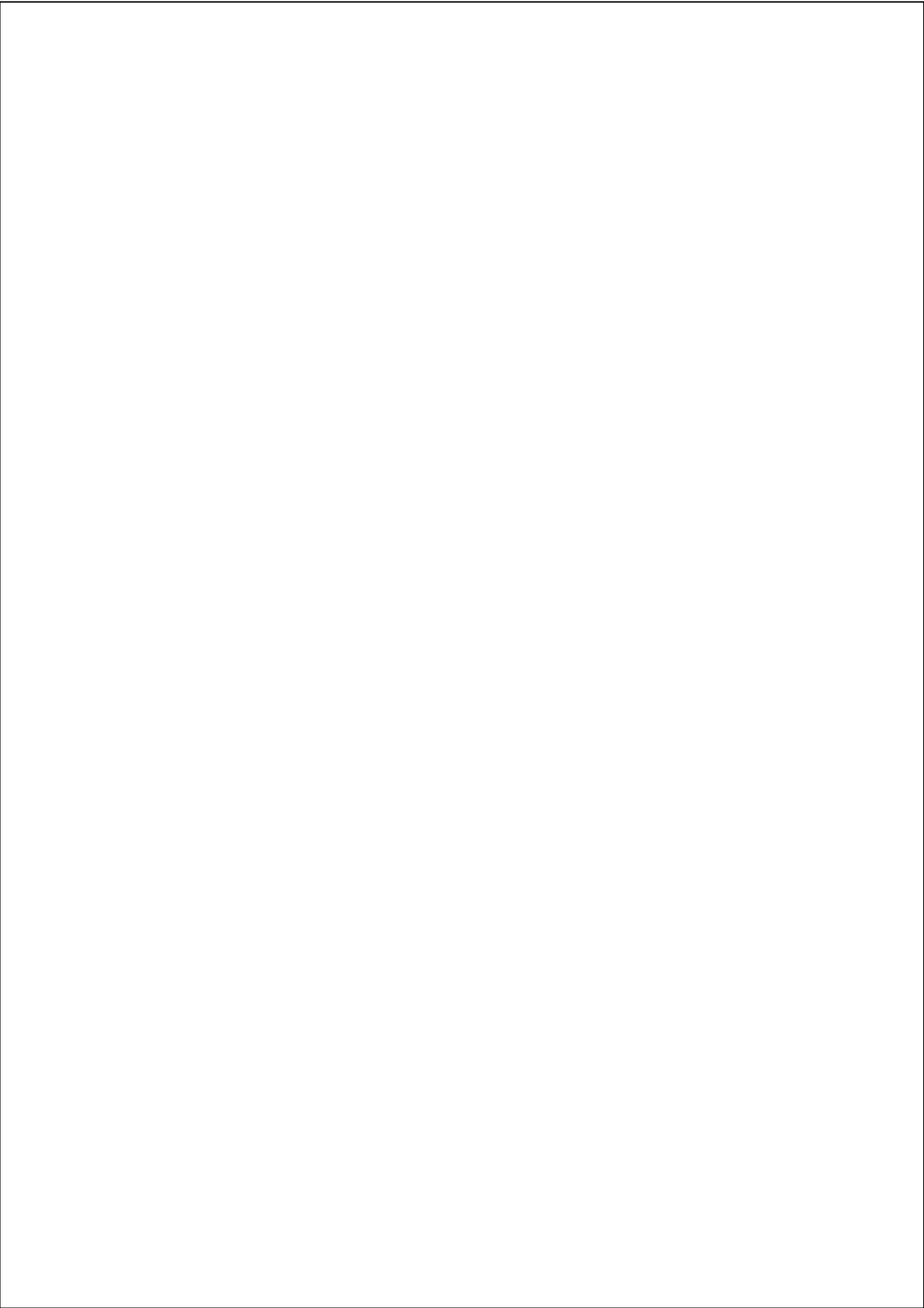
This is because the treftz lies within the compressed portion of the flow and therefore work is done on the control volume by the fluid.

Viscous Boundary Work $\dot{E}_\tau$

Figure 5.26 depicts the viscous boundary work. It is small in comparison to the other energy contributions, as the velocity gradients away from physical walls diminishes quickly. As expected it quickly asymptotes to zero, apart for a case $\frac{\Delta s}{l_{fuse}} = 1/15$, where the side walls of the control volume intersects the viscous wake region at a downstream position $\frac{\Delta z}{l_{fuse}} = 7$. It is a negative value, and is a result of the fluid doing work on the control volume in response to the local flow conditions.

Volumetric Compression Work $\dot{E}_{pv}$

As the fuselage moves through the air, it displaces it and compresses it. In compression, energy is transferred to the flow and there is a peak positive value at the fuselage trailing edge. This compression work term diminishes with an increase in the treftz plane's downstream location. However, it does not asymptote to zero. Firstly, it must be noted that for small Δs locations the asymptote crosses over the axis corresponding to Δs at a position where $p \approx p_\infty$. This offset is countered by the opposing crossover



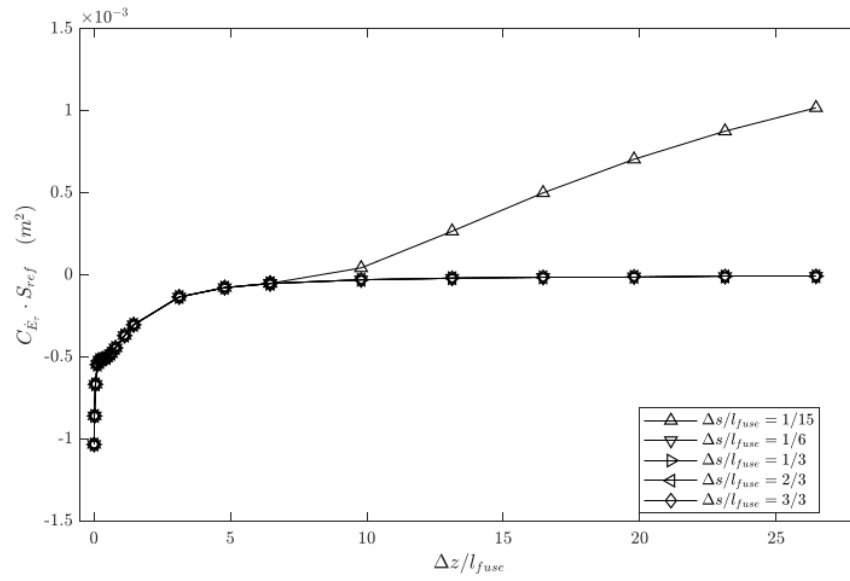


Figure 5.26: Variation in viscous boundary work $C_{\dot{E}_\tau}$ corresponding to different treftz plane locations

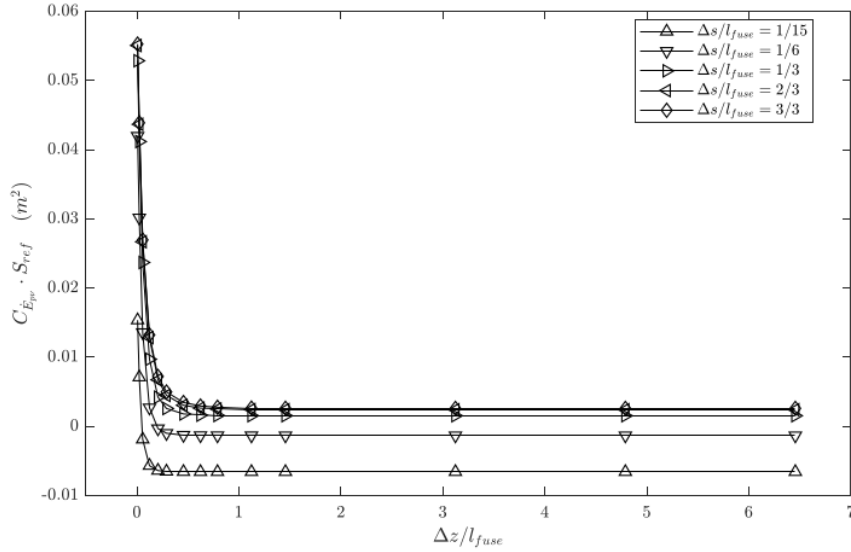


Figure 5.27: Variation in volumetric compression work $C_{\dot{E}_{pw}}$ corresponding to different treftz plane locations

viscous dissipation.

Net Force

Figure 5.29 depicts the net force coefficient using the energy equation. It is depicted alongside the net near-field surface force, and the difference is illustrated in Figure 5.30 using the Fluent reported near-field force as reference. As was the case with the far-field momentum-based analysis, it is seen that unreliable results are obtained when the treftz plane is positioned too closely to the fuselage's trailing edge. However, unlike the momentum-based far-field method, the force coefficient predicted is fairly constant for different downstream locations of the treftz plane, and the trends become almost completely flat after several fuselage lengths downstream. This is a very interesting result, in that it does not exhibit any accumulation in spurious drag, as was the case in the far-field momentum-based results. The exact reason for this is unclear without having exposure to the solver source code. Nonetheless, the evidence suggests that the energy-based result is less sensitive to spurious drag.

Breakdown of Contributions

At this point it is useful to look at the breakdown of the contributions to the net force, as depicted in Figure 5.31. Conceptualising what is happening in the absolute frame

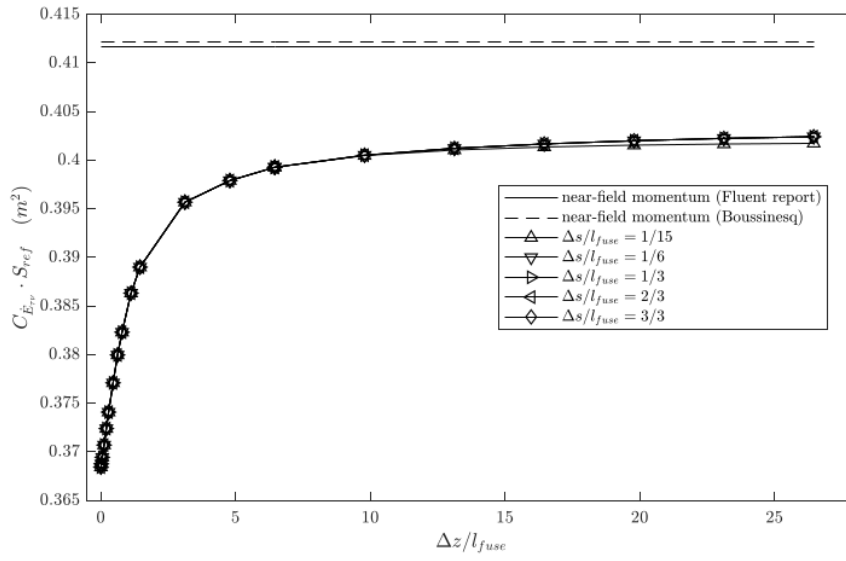


Figure 5.28: Variation in volumetric viscous dissipation $C_{E_{\tau\nu}}$ corresponding to different trefftz plane locations

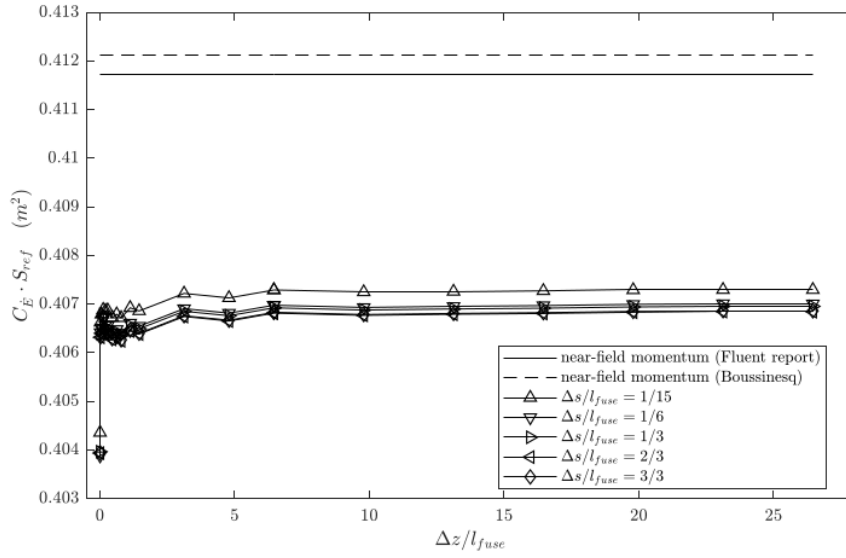


Figure 5.29: Net forces coefficient $C_{\tilde{E}}$ extracted using the volumetric dissipation method

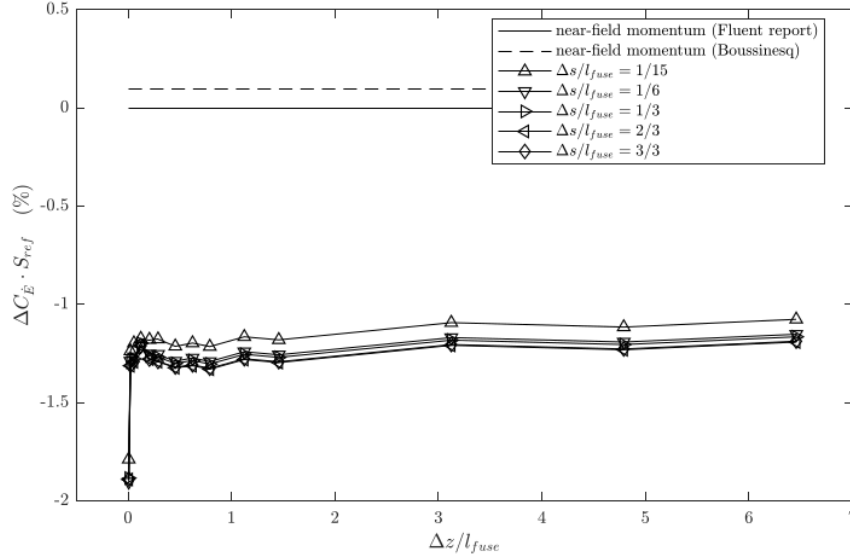


Figure 5.30: Difference in net forces coefficient $C_{\dot{E}}$ extracted using the volumetric dissipation method, using near-field as reference

of reference, helps to make sense of the energy transfers taking place in the flow. In the absolute reference frame, the fuselage is driven through the air, and in the process it compresses the flow locally around it. At the same time, the flow is set in motion and gains kinetic energy. One of the common conceptual mistakes about potential BLI benefit, is that $\dot{E}_k$ at the trailing edge (denoted hereafter as $\dot{E}_{k,TE}$) represents the maximum theoretical energy gains available by through wake filling. This would be true if all of that kinetic energy ended up dissipating irreversibly. However, it is clearly visible from Figure 5.31, that the gain is $\dot{E}_{\tau\nu}$ at a trefftz plane far downstream, is much smaller than the $\dot{E}_{k,TE}$. Instead, a large portion of $\dot{E}_{k,TE}$ is mitigated by the pressure boundary work (and additional small portions from the viscous boundary work) on the control volume. Another important observation of this breakdown, is the contribution by $\dot{E}_{p\nu}$, which is small but not insignificant. This breakdown demonstrates the intuitive approach to making the *NVF* the subject of the energy balance, instead of a *power source* = *power sink* approach.

5.3.4 Methods Comparison and Uncertainties

To examine the uncertainty associated with drag predictions on the isolated fuselage, a comparison must be made between the near-field, far-field momentum and energy-based methods. Figure 5.32 depicts this comparison, using Fluent's near-field report as reference. Again it is emphasised here that using this reference does not mean that it is

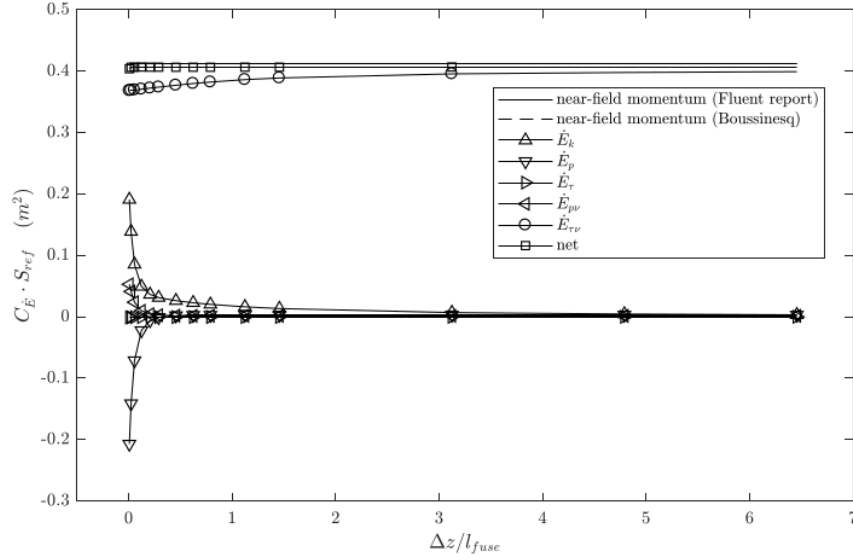


Figure 5.31: Breakdown of different C_E contributions corresponding to different trefftz plane locations, for $\frac{\Delta s}{l_{fuse}} = 1$

considered to be the most correct or accurate. None of the methods, at this point, are considered to be more accurate than the other, as there is insufficient evidence to make such a conclusion. Establishing this is left for future work, where an extensive number of validation cases need to be examined. Nonetheless, the comparison does help with determining an uncertainty, and from Figure 5.32 it can be seen that the uncertainty is around 4%.

5.4 NVF Calculations

The purpose of this chapter is to demonstrate how the *NVF* of a propelled body may be obtained using different post-processing methods, and to understand the uncertainties involved. So far this chapter has looked at how to isolate different flow mechanisms to quantify their contributions to the *NVF* force, as well as investigating the viscous stress tensor, consistency between different methods predicting forces on a non-propelled body and the presence of spurious drag. What remains is to look at the inclusion of a propulsor, and the associated practical considerations required. As such, this section begins by examining the forces on a propulsor represented as a set of outlet and inlet boundary conditions in the domain. Subsequently, the procedure for calculating the thrust-force accounting systems of Section 3.6 is explored and demonstrated. Finally, the *NVF* is obtained for a propelled case from far-field momentum based and energy-

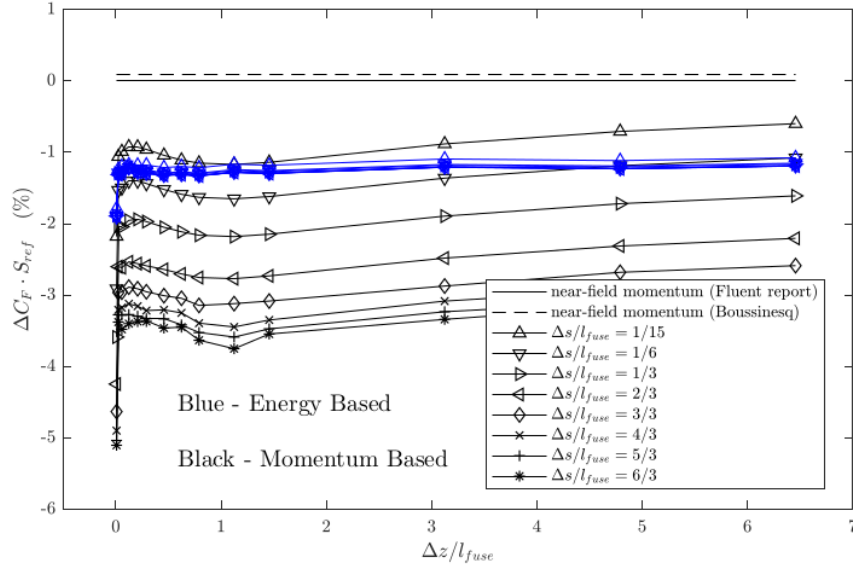


Figure 5.32: Comparison of the difference in net forces coefficient C_E extracted using near-field, far-field momentum and energy-based methods

based CV methods to evaluate the consistency between methods.

5.4.1 Near-Field Boundary Conditions and Propulsor Forces

In this thesis, the propulsor is modelled as a combination of a static pressure outlet (fan inlet) and mass-flow inlet (fan outlet) boundary conditions. The reason for this is that the fan design is unknown, and therefore it is not feasible to include an actual fan within the flow domain. As mentioned in the literature review, some works make use of actuator disks to model the fan. This was deemed incorrect for a number of reasons. The first is that the area is constant across the actuator disk, and therefore some physical laws must be violated in order to achieve the pressure rise required across the fan (in compressible flow, Fluent discontinuously decreases entropy across the actuator disk). Additionally, this discontinuity violates Gauss' theorem, and its use would not allow for the relationships between surface and volume integrals, which have been used extensively in Chapters 3 and 4. Instead, the combination of outlet and inlet boundary conditions allow for better representation of the propulsor. Controlling the area change between the boundary conditions, allows for desired pressure ratios to be achieved (or at least approached) for a specified polytropic efficiency. Estimating the propulsor geometry to meet these requirements is covered in Chapter 6. Given this fan representation, the fan forces and shaft power need to be calculated. As can be seen from Equations (3.11),

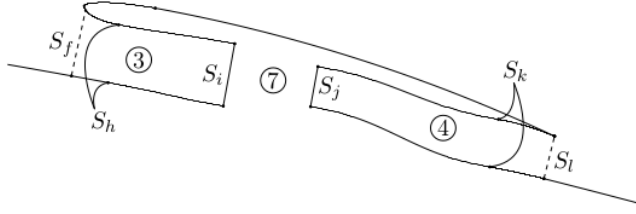


Figure 5.33: Control volume divisions for a BLI propulsor represented as a set of outlet (S_i) and inlet (S_j) boundary conditions. This particular geometry is for a propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

(3.35) and (4.32), the forces and shaft power are obtained from surface integrals along the propulsor control volumes. Integrating along boundary conditions may be risky, and the aim of this subsection is to understand the implications associated with these integrations. Possible sources of error may occur where the boundary condition meets the duct walls, due to interactions with the BL formation. Figure 5.33 depicts the propulsor within the CFD domain. The net force acting on the fan may be obtained from Equation (3.10):

$$\Psi_{S_F} = \Omega_{S_P} = \Omega_{S_{i7}} + \Omega_{S_{j7}} \quad (3.10)$$

Values for $\Omega_{S_{i7}}$ and $\Omega_{S_{j7}}$ may be obtained through direct integration along the boundary conditions, or alternatively from conservation of momentum using Equation (3.9), where:

$$\Omega_{S_{i7}} = -\Omega_{S_{i3}} = \Omega_{S_{f3}} - \Psi_{S_{h3}} \quad (5.12)$$

and:

$$\Omega_{S_{j7}} = -\Omega_{S_{j4}} = \Omega_{S_{l4}} - \Psi_{S_{k4}} \quad (5.13)$$

In a similar vein, the shaft power (before accounting for mechanical efficiency) may be obtained from direct integrations on the boundary condition surfaces using Equation (3.35), or alternatively from a similar integration of the total energy change between S_f and S_l . The error of the direct integration, in relations to conservation, within the intake and exhaust ducts is summarised in Table 5.2. In this instance, the error is small and typically within 0.5%. This corresponds to an error of around 20N and 4kW. Although these numbers are relatively small, it is recommended to use the values determined from conservation in the duct, as these are a result of better resolved flow. This is because other cases in the design exploration were found to have larger errors, up to 1.5% in both power and force.

5.4.2 Thrust-Force Accounting Approach

Having now covered the integrations to obtain the force on the propulsor, the focus turns to the post-processing calculation procedure required to obtain the various thrust-force

Table 5.2: Example of the propulsor boundary conditions integration error for a BLI propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

	$\Omega_{S_{i,x}} (N)$	$\Omega_{S_{i,x}} (N)$	$F_{p,x} (N)$	$\dot{W}_{CV7} (W)$
Direct integration	-2105	-4787	-6891	1.895E+06
Indirect conservation	-2101	-4770	-6871	1.891E+06
Difference	-4	-17	-21	3.879E+03
Error (%)	0.18	0.35	0.30	0.21

accounting systems presented in Section 3.6. This subsection details the method implemented in Matlab (which interacts with Fluent) to obtain the wall, flux and streamtube forces (i.e. Equation (3.9)) for each of the control volumes of Figure 3.4.

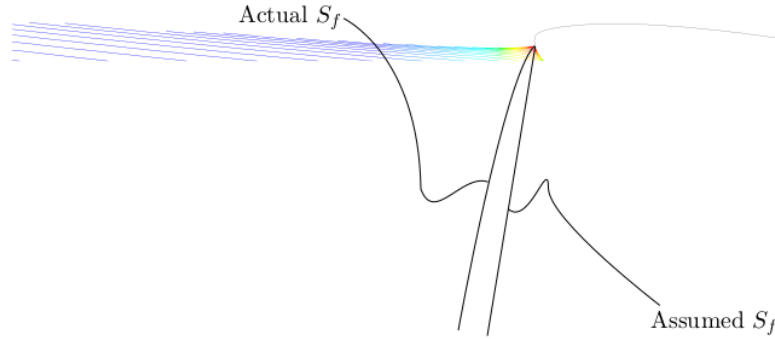
Surface S_a

This surface is briefly mentioned first, as it is defined as being sufficiently ahead of the assembly such that $p = p_\infty$, $\mathbf{V} = \mathbf{V}_\infty$ and $\bar{\tau} \cdot \hat{n} = 0$. Additionally, it is defined as being normal to the x axis, and knowing the mass-flow-rate through the propulsor determines its radius. Nonetheless, $\Omega_{x,a1}$ is zero given its upstream definition.

Surfaces S_f , S_h and S_p

The highlight area surface S_f is defined in this work as stemming from the stagnation point on the nacelle lip, which divides up the internal and external flows, extending to a point on the fuselage where it is perpendicular to the fuselage surface. This assumes that S_f is a conical frustum, which is an approximate to a more correct definition. In reality, the highlight surface is better defined as following a contour connecting the highest static pressures in the local region, which is a surface perpendicular to the streamlines. Figure 5.34 depicts an example of the difference between the real and approximate definitions for S_f , which shows a small disparity between the two. As the difference is relatively small, the approximate S_f definition is used in this work for the sake of practicality, as calculating surface geometries according to stream functions is unnecessarily complex for this application. It is also reasoned that the implications on streamtube force will be insignificant.

Based on the approximate definition, it was necessary to find a way to automatically locate the stagnation point on the lip. To do so, the node on the lip surface corresponding to the highest static pressure is located, along with adjacent nodes. Lines running perpendicular to the fuselage from each of these nodes are constructed to determine the



C_p

Figure 5.34: Example of streamlines through the real and assumed highlight surfaces S_f , shown within contours of pressure coefficient. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

mass-flow-rate across each of these conical frustums as an initial estimate of surface S_f . Knowing the actual mass-flow-rate through the surface S_i , allows for interpolation of the stagnation point on the lip to ensure continuity. Having defined S_f , and hence the intersecting points on the lip and fuselage, allows for the Matlab tool to split the nacelle and fuselage surfaces to create surfaces S_h (inner and outer separately) and S_p (knowing the nacelle after-body's trailing edge point).

Surfaces S_c , S_g , S_d , S_b and S_e

The next step is to locate surface S_c . In this work, S_c was defined as the plane where pre-diffusion first begins ahead of the intake. For a completely new design, this is very difficult to determine. However, this thesis approaches the design from a retrofit perspective, and therefore the flow may be compared to that over the non-propelled fuselage case. Pre-diffusion results in a rise in static pressure of the flow, and therefore the Matlab tool is able to identify the location of S_c by examining a change in the pressure coefficient (in comparison to the non-propelled case) along the fuselage surface upstream of the intake. A 1% increase in C_p was found to be a robust threshold for approximating the start of pre-diffusion along the fuselage surface. Thereafter, a conical frustum is created perpendicular to the fuselage at this point, and linear interpolation used to find its radius corresponding to the mass-flow-rate at S_i . Similar to S_f , this is an approximate to the surface and a more correct definition would require locating the

contour perpendicular to the streamlines in this region. However, it is assumed that the additional accuracy is outweighed by the complexity involved in creating a tool to automatically determine the actual surface geometry from stream functions.

Having defined S_c allows for the remaining fuselage surface to be split into surfaces S_d and S_g . Ideally surfaces S_b and S_e , being streamlines, could be determined from their stream functions given the locations of S_a , S_c and S_f . However, this is not a trivial task to implement and deemed impractical for the purposes of this work. It is only the streamtube unbounded force Φ which is required to calculate the thrust-force accounting definitions, and this can now be determined using conservation (i.e. Equation (3.9) and with reference to Figure 3.4) to yield:

$$\Phi_{S_{b1}} = (\Omega_{S_{a1}} + \Omega_{S_{c1}}) - \Psi_{S_{d1}} = -\Phi_{S_{b6}} \quad (5.14)$$

$$\Phi_{S_{e2}} = (\Omega_{S_{c2}} + \Omega_{S_{f2}}) - \Psi_{S_{g2}} = -\Phi_{S_{e6}} \quad (5.15)$$

Surfaces S_l , S_k , S_m , S_n and S_o

The nozzle exit surface S_l is defined by the conical frustum extending perpendicularly from the tail-cone surface to the sharp trailing edge of the nacelle. This is again an approximation, as the correct surface should follow the contour found to be perpendicular to the jet's streamlines in this region. However as depicted in the example of Figure 5.35, the approximation is very close to the actual jet exit surface. Therefore the approximate frustum is used for practicality, and is based on the assumption that differences from the real surface are negligible. Having found S_l , allows for the tail-cone surface to be split into surfaces S_k (inner and outer) and S_m , all that remains is to determine S_n and S_o . To do so, the surface S_n is defined as the position downstream where the static pressure has returned to free-stream conditions. The Matlab tool does this by examining the pressure coefficient along the domain axis and finding the point where the static pressure is within 1% of the free-stream value. The disk of S_n , perpendicular to the x axis, is found through linear interpolation to match the mass-flow-rate of S_j . Surface S_o is not geometrically defined, as this would require calculating its stream function. Instead, only its force is required, and this is determined using conservation from Equation (3.9) with reference to Figure 3.4:

$$\Phi_{S_{o5}} = (\Omega_{S_{l5}} + \Omega_{S_{n5}}) - \Psi_{S_{m5}} = -\Phi_{S_{o6}} \quad (5.16)$$

Determining NVF and Thrusts and Forces

Having defined all the flux and wall surfaces, allows for Ω and Ψ force terms to be obtained from direct integration. Subsequently, the thrusts and forces of the different definitions are determinant, and may be tabulated. Table 5.3 is an example of the thrusts and forces calculated in the stream-wise direction, corresponding to the formulations of Table 3.1.

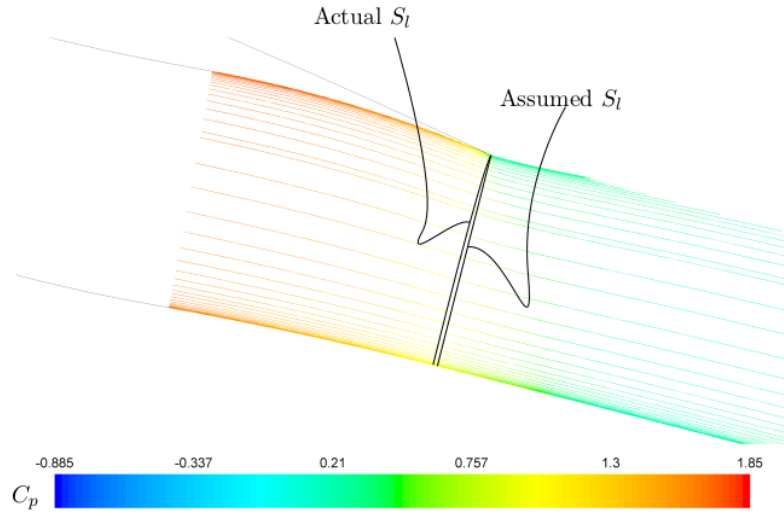


Figure 5.35: Example of streamlines through the real and assumed nozzle exit surfaces S_l , shown within contours of pressure coefficient. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

Table 5.3: Thrust-force accounting system table example. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

Definition	Thrust (x dir.)	Force (x dir.)	$NVF(x$ dir.)
Propulsor net thrust	-6870.87	4689.30	-2181.57
Intrinsic Net Thrust	-5401.33	3219.76	-2181.57
Standard Net Thrust	-3610.98	1429.41	-2181.57
Overall Net Thrust	-1789.15	-392.42	-2181.57
Pre-Diff. Net Thrust	-5515.29	3333.72	-2181.57
Pre-Diff. Net Overall Thrust	-3693.46	1511.89	-2181.57
Ref. Net Vehicle Thrust	-5239.78	3058.21	-2181.57

5.4.3 Far-Field Momentum-Based NVF Coefficient

The NVF derived for the various thrust-force accounting methods in Section 5.4.2, is based on near-field momentum-based force post-processing on the walls of the airframe. An alternative is to use momentum conservation to obtain the NVF from control volumes encapsulating the aircraft, as per Equations (3.14) and (3.15). The same approach as Section 5.3.2 is followed, whereby the impact of different control volume dimensions on NVF outcome is assessed. This is demonstrated on the same fuselage concentric propulsor case, where the propulsor is placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$. This allows for comparison against the near-field based method presented in the previous subsection on thrust-force accounting definitions.

Figure 5.36 shows the NVF coefficient $C_{F_{NVF}}$ obtained for various control volumes, with reference to Figure 5.20 (except now with the inclusion of a propulsor). Based on the coordinate system definition, the negative $C_{F_{NVF}}$ means that a Reference Net Vehicle Thrust is produced. For any given downstream treftz location $\Delta z/l_{fuse}$, it is observed that the NVF decreases as the control volume is located further away from the fuselage (i.e. increasing $\Delta s/l_{fuse}$). Around $\Delta s/l_{fuse} = 6/3$, the NVF obtained converges on a set of values for the different treftz locations. This is consistent with the trends observed for the non-propelled fuselage in Section 5.3.2, where the decrease in rearward force was explained as a diminishing influence of spurious effects on surface integrals in the bulk (potential) flow regions. This reinforces the hypothesis that spurious effects influence near-field forces on physical walls. Considering a given $\Delta s/l_{fuse}$ location, it can be seen that NVF values increase with downstream treftz location, as more of the viscous wake (and subsequently spurious effects) is incorporated within the volume. This also reinforces the idea that spurious effects are concentrated within the viscous wake regions of the flow, following the arguments already discussed in Section 5.3.2. These ideas on the influence of spurious effects are reinforced because the trends follow the same direction for both non-propelled and propelled bodies.

The uncertainties in $C_{F_{NVF}}$ associated to Δs spacing are summarised in Figure 5.37, where the forces are reported as a percentage difference from the converged location values (i.e. at $\Delta s/l_{fuse} = 6/3$) and indicate a 1.5-2% disparity. However, it is noted that this only represents the uncertainty for the control volumes considered, and not the entire spectrum. In quantifying the uncertainty across the spectrum, smaller control volumes around the fuselage-propulsor assembly should be considered. In fact, the near-field values are approached as $\Delta s \rightarrow 0$ and $\Delta z \rightarrow 0$, and the uncertainty is given by the difference between the near-field forces and the far-field forces at a location where the force is insensitive to change in Δs and Δz . This is discussed later in Section 5.4.5.

5.4.4 Energy-Based NVF Coefficient

Having discussed NVF outcomes from far-field momentum-based methods, the focus is now shifted to the energy-based method. The same approach is taken, in order to understand the impact of obtaining the NVF from different control volume dimensions, and is demonstrated on the same example case. In addition, this subsection considers a

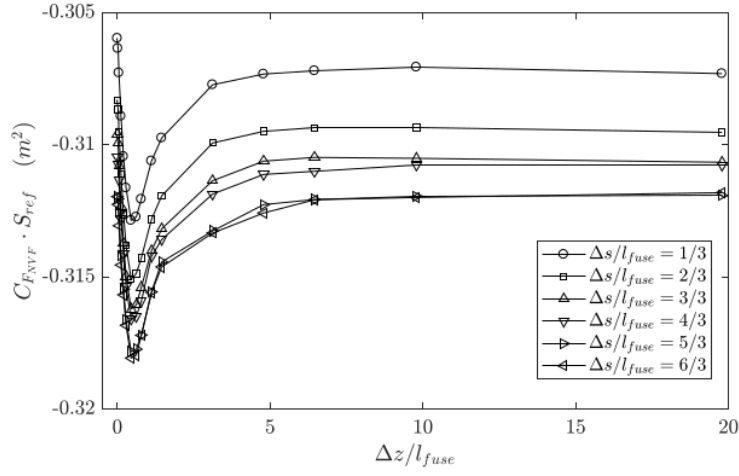


Figure 5.36: Momentum-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

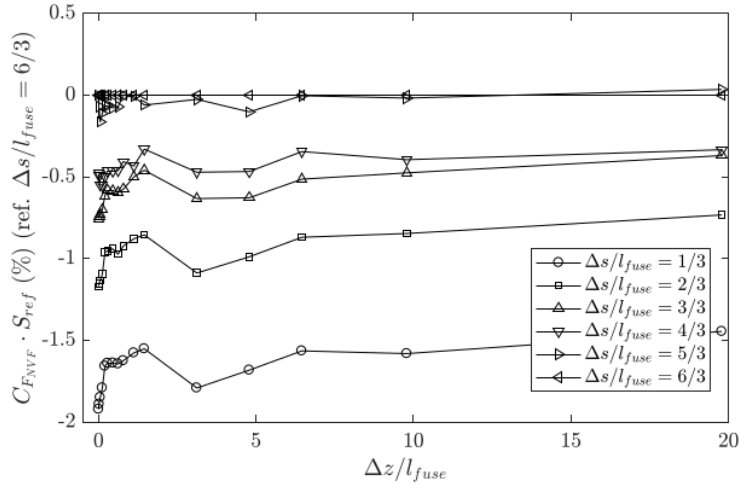


Figure 5.37: Control volume uncertainties of the momentum-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

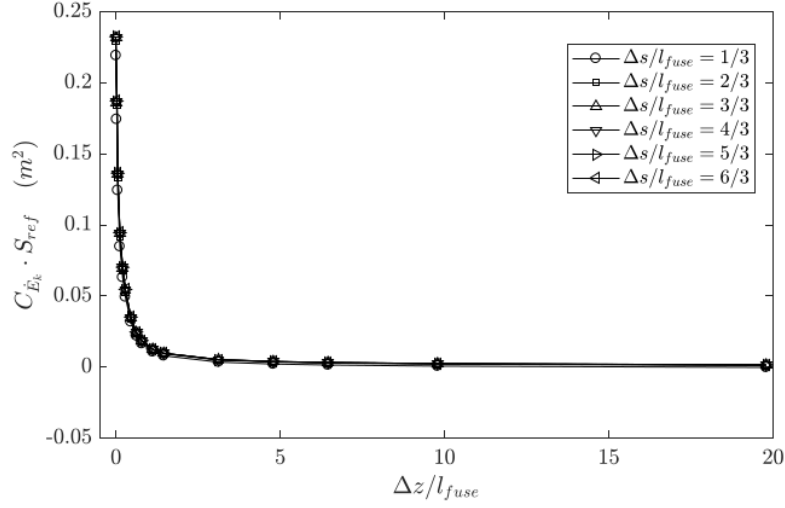


Figure 5.38: Kinetic energy flux force coefficient $C_{\dot{E}_k}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

breakdown of the various energy-based contributions.

Kinetic Energy Flux $\dot{E}_k$

Figure 5.38 depicts the kinetic energy flux contribution to the overall force. A similar trend to the non-propelled case is observed, where $C_{\dot{E}_k}$ shows a positive peak diminishing to zero as the treftz plane is located further downstream. It is a positive value, as it represents the amount of kinetic energy imparted to the flow, of which a portion is dissipated to heat at treftz locations further downstream. The remaining amount is mitigated by the pressure boundary work contributions, as will be discussed further in a later section on the breakdown of the overall contributions to NVF .

Pressure Boundary Work $\dot{E}_p$

Figure 5.39 depicts the pressure boundary work acting on the various control volumes. Like the non-propelled fuselage case, the contribution is strongly negative close to the fuselage trailing edge, and diminished to zero at treftz locations further downstream where $p \rightarrow p_\infty$. The negative contribution corresponds to the thrust direction, and is a result of the locally compressed flow. There is very little difference for changes in Δs , as the closest position is already in a location where $p \rightarrow p_\infty$. As has already been mentioned, this $\dot{E}_p$ contribution mitigates a large portion of the $\dot{E}_k$ contribution.

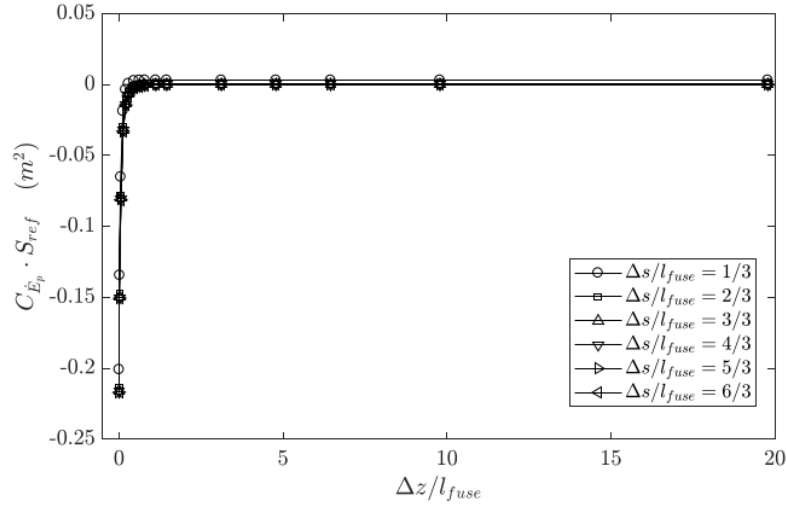


Figure 5.39: Pressure boundary work force coefficient $C_{\dot{E}_p}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

Viscous Boundary Work $\dot{E}_\tau$

Figure 5.40 illustrates the viscous boundary work contribution to the NVF . Unlike the non-propelled fuselage case (see Figure 5.26) $\dot{E}_\tau$ is positive, therefore contributing to rearward force. This boundary work is far less intuitive than pressure boundary work, and it is difficult to explain the reason for negative (in the non-propelled case) and positive (in the propelled case) contributions. One explanation is that the viscous stress tensor on at the jet boundary gives rise to a shear stress vector acting in the other direction because of the high plume velocity relative to the spilt BL with which it mixes.

Returning briefly to the trend corresponding to changes in Δz , for the non-propelled case. The contribution began at a peak negative value at the fuselage trailing edge, and diminished asymptotically as the treftz plane location shifted further downstream. Whereas in the propelled case, the contribution begins with a small positive contribution which grows to a local maximum before diminishing asymptotically to zero. The difference observed in the propelled trend is attributed to jet plume and how it begins with a laminar core and transitions into a turbulent mixing layer further downstream. The peak represents the point at the end of the jet core's extent, and only the turbulent mixing transition region remains. The energy contribution of the turbulent transition region then diminishes as the shear layers mix out.

Finally, it is mentioned that there is little difference for changes in Δs , because viscous contributions are negligible outside of the BLs. Additionally the contributions

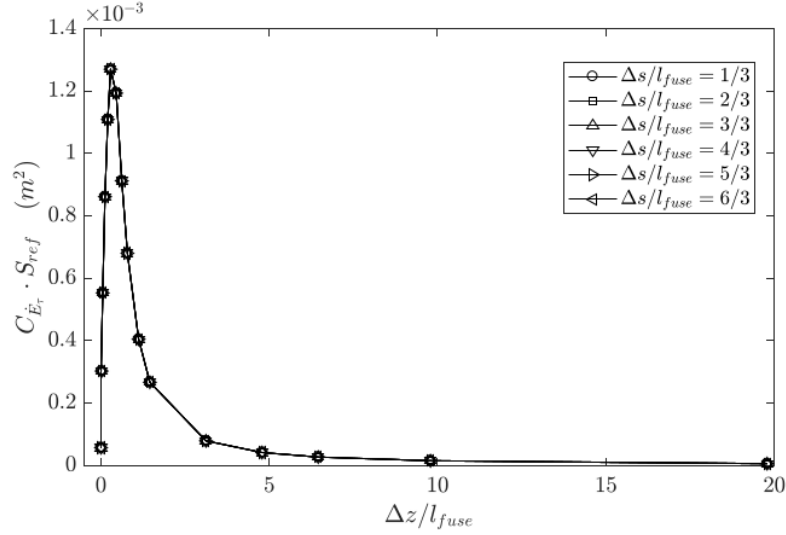


Figure 5.40: Viscous boundary work force coefficient $C_{\dot{E}_\tau}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

of this term are generally very small and may be considered insignificant relative to the NVF .

Propulsor Force Related Work $\dot{E}_{prop}$

Figure 5.41 depicts the force on the propulsor, which is obtained from Equation (4.32). Because it obtained from integrations over the fan inlet and outlet boundaries, it is a constant and unaffected by changes in the encapsulating control volume dimensions (i.e Δs and Δz), but is depicted here for completeness. It is clearly a negative value, thereby contributing a forward force to the NVF . Its contribution to NVF is large, as expected, and is discussed further in a breakdown summary later in this section.

Volumetric Compression Work $\dot{E}_{pv}$

The volumetric compression work is depicted in Figure 5.42, and unlike the non-propelled fuselage case (see Figure 5.27) is solely negative for all Δz locations. Like the non-propelled case, however, it does not diminish to zero, but rather asymptotes on non-zero constant. The contribution in the non-propelled case (at a suitable Δs) is positive, contributing to the rearward force on the body. For the propelled fuselage, it is a negative constant, which means that it contributes to the thrust direction in the NVF .

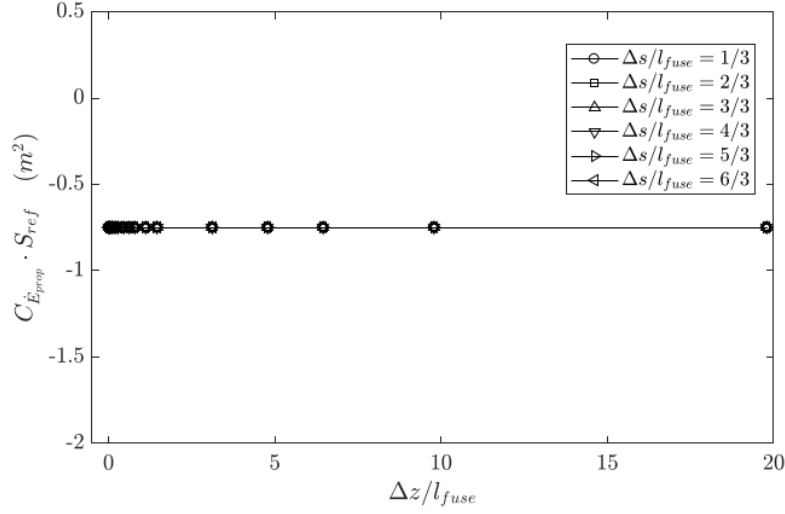


Figure 5.41: Propulsor $C_{\dot{E}_{prop}}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

This evidence opposes the argument by Drela [5] who claims this term is insignificant unless heat is added at high pressure. In fact, its contribution is rather significant to the thrust direction component of NVF , and disregarding would be incorrect. This demonstrates the value of formulating the energy equation with NVF as the subject of the equation, as opposed to the *power source = power sink* approach by Drela, and further emphasises the contribution to knowledge of this work. In this propelled fuselage case, the constant thrust direction contribution is insensitive to changes in Δs , because $p \rightarrow p_\infty$ at the first control volume location.

Volumetric Dissipation $\dot{E}_{\tau\nu}$

The energy dissipation for the different control volumes is depicted in Figure 5.43. The first observation is that the dissipation is insensitive to changes in Δs . This is because the dissipation is concentrated in high shear and/or turbulent regions. The second is that, as expected, it increases with Δz from a positive offset at the trailing edge to an asymptote value. The positive contribution to the NVF correlates to a rearward force, which is expected as this directly represents energy lost to the flow. The asymptotic increase is a result of including more of the wake within the control volume, by increasing Δz , but by a diminishing amount as the wake returns to ambient conditions.

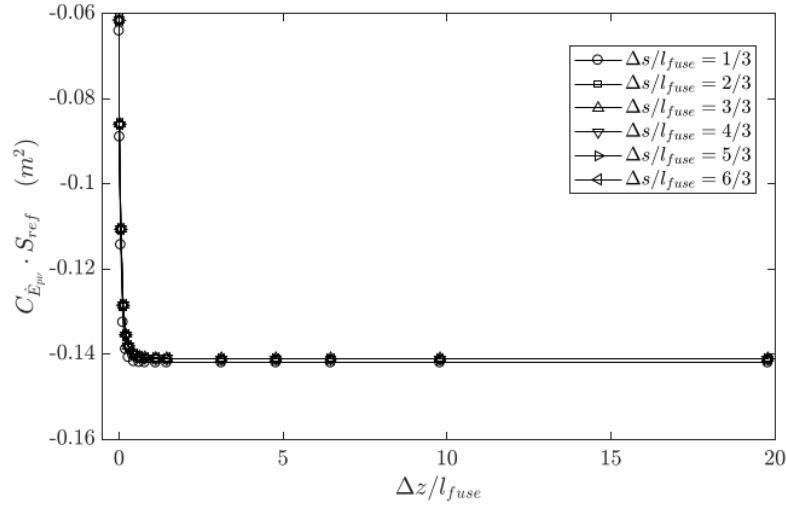


Figure 5.42: Volumetric compression work $C_{\dot{E}_{pv}}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

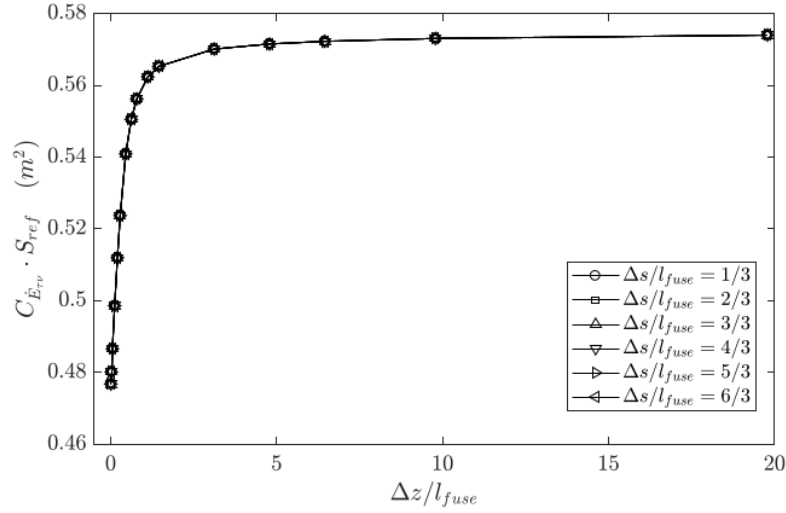


Figure 5.43: Volumetric dissipation $C_{\dot{E}_{\tau\nu}}$ contribution, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

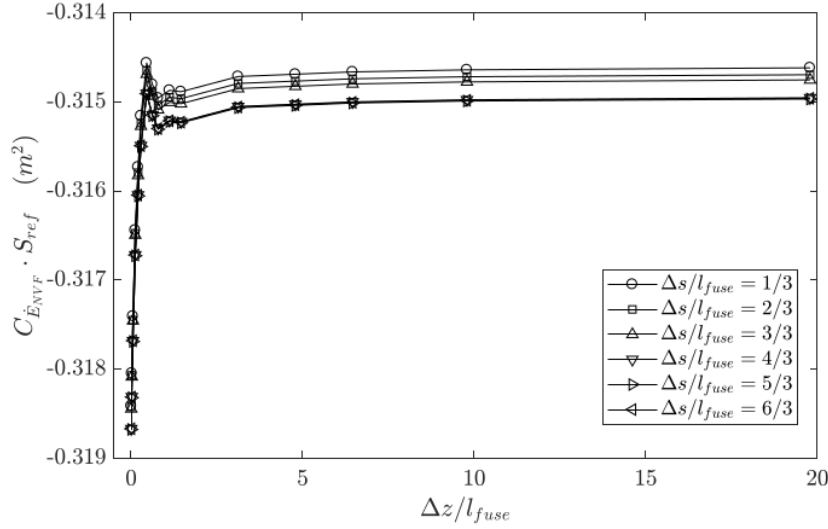


Figure 5.44: Energy-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

NVF Coefficient $C_{\dot{E}_{NVF}}$

The sum of all the contributions results in a *NVF* coefficient, which is depicted in Figure 5.44. Due to the control volumes selected, the *NVF* is relatively insensitive to changes in Δs , and the results converge when $\Delta s / l_{fuse} = 4/3$. Like the non-propelled fuselage, there is a sharp increase in *NVF* from the trailing edge, which is an unreliable region with separations and transitions from BLs to unbounded shear layers. Thereafter, there is a subtle asymptotic increase by placing the treftz plane further downstream, suggesting that there is some gain in dissipation due to spurious effects. This increase was not as clear when the non-propelled fuselage was being considered. Nonetheless, the asymptote is reached very quickly, and the *NVF* appears as an almost constant value, with little deviations.

Figure 5.45 illustrates the uncertainties associated with changing Δs . This is done in reference to the largest, converged distance $\Delta s / l_{fuse} = 6/3$. It is evident that a very close agreement is achieved, with an uncertainty less than 0.2%. The shapes of the uncertainty trends in Figure 5.45 at $\Delta s / l_{fuse} = 1/3, 2/3$ and $3/3$, suggest that they are due to differences between the pressure boundary work $C_{\dot{E}_p}$ and volumetric compression work $C_{\dot{E}_{pv}}$, which were the only parameters exhibiting some sensitivity to Δs . This makes sense because in clipping out the control volume region, the cells intersected by the control volume surface are either included or excluded depending on how much of their cell volume lies within the control volume region. This introduces increasing

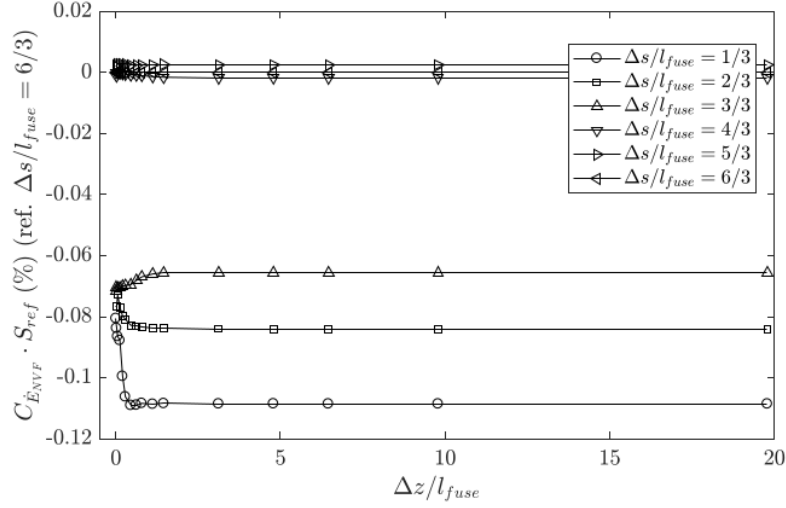


Figure 5.45: Control volume uncertainties of the energy-based far-field force coefficient, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

errors at smaller Δs values, but not for larger control volumes where the contribution of individual cells is insignificant (because the flow approaches returning to ambient conditions).

Breakdown of Contributions

A breakdown of the NVF and its components, is depicted in Figure 5.46, where the results corresponding to $\Delta s / l_{fuse} = 6/3$ are shown. First, considering the terms that diminish to zero. The viscous boundary work has an insignificant contribution, which was expected as the shear forces are dramatically reduced away from wall. The viscous boundary work, on the other hand, has a large initial contribution as does the kinetic energy flux. This breakdown makes clear the observation that only a portion of the kinetic energy flux $\dot{E}_k$ ends up dissipating (i.e. transferring to $\dot{E}_{\tau\nu}$), and that a significant portion is mitigated by the pressure boundary work $\dot{E}_p$. This was observed for the non-propelled fuselage too, and is an important example of the advantage in having NVF as the subject of the energy balance.

Considering now the terms that do not diminish to zero. The propulsor's contribution is constant with changes in Δz and, for the most part, so is the NVF coefficient. The viscous dissipation's increases from its initial offset (due to dissipation in the airframe BLs) with Δz as energy is transferred from either $\dot{E}_k$, $\dot{E}_p$, $\dot{E}_{p\nu}$ or a combination thereof.

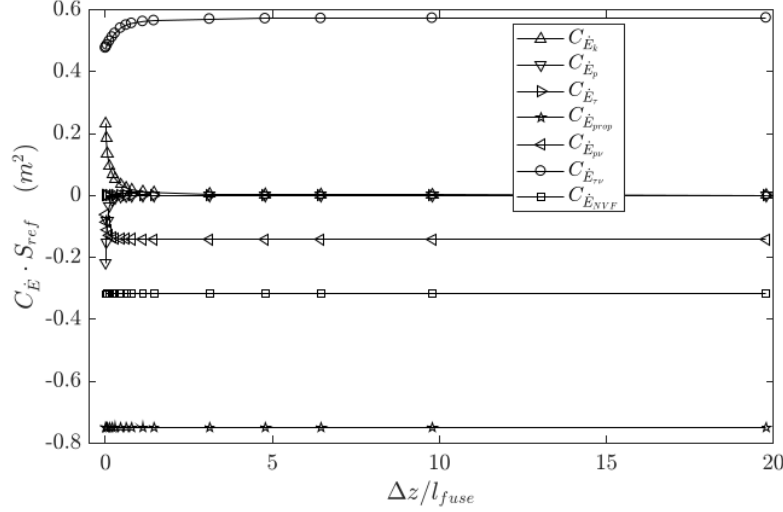


Figure 5.46: Energy-based far-field force coefficient breakdown, for different trefftz plane locations and $\Delta z/l_{fuse} = 6/3$. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

If portions of either of the pressure terms dissipates, it will do so indirectly via a transfer to kinetic energy. This is because the dissipation term is a function of the viscous stress tensor, which is in turn dependent on velocity gradients. Nonetheless, after several fuselage lengths downstream, the only non zero terms are the positive viscous dissipation, the volumetric compression work and the propulsor power inflow. A very important observation is the significance of $\dot{E}_{pv}$'s contribution, as it accounts for almost a half of the *NVF*. This once again illustrates how this term cannot be assumed to be insignificant, as suggested by Drela [5]. What else is very interesting, is that it can be seen that an almost natural division of thrust-force has formed, where $\dot{E}_{prop}$ and $\dot{E}_{pv}$ are thrust contributors and the rearward force is solely due to $\dot{E}_{rv}$. This requires further work to investigate if $\dot{E}_{pv} < 0$ for all propelled cases. If this is shown to be true, then thrust and force may be unambiguously defined.

5.4.5 Methods Comparison and Uncertainties

In this section three methods have been used to post-process the *NVF* from a RANS solution of a propelled fuselage. The first is classified as a near-field method, where forces on aircraft surfaces are obtained by direct pressure and skin friction integrations. The second is a far-field momentum-based approach, where conservation is used to determine the *NVF*. Finally, the energy-based method, determines the *NVF* based on volume integrals of viscous dissipation and compression work, as well as energy transfer through flux

surfaces. Uncertainties for the latter two were considered, by examining variations in the NVF calculated from different sized control volumes. It is now desirable to quantify uncertainties of NVF calculation through a comparison of all three methods. To do so, this work uses the near-field NVF as a reference, but it must be noted that this is out of convenience and it is not considered to be the most accurate or correct of the methods.

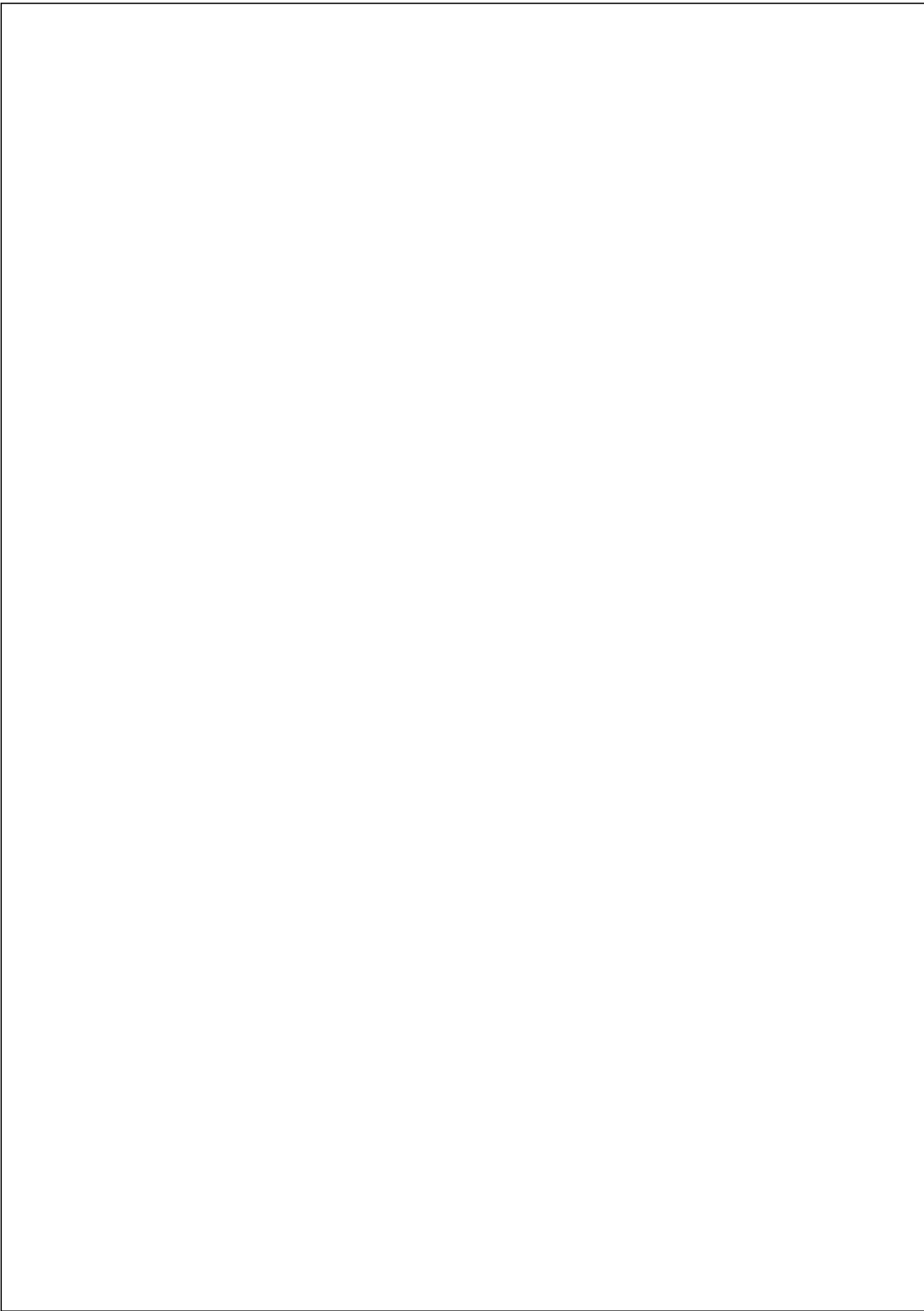
The uncertainties associated with the three methods are summarised by the two graphs of Figure 5.47, where the upper and lower compare the far-field momentum-based and energy-based methods against the near-field values, respectively. In considering first the far-field momentum-based method comparison, it must be recalled that as the control volume is shrunk around the body, the NVF will approach the near-field value. However, the NVF range must be considered to a control volume size where it becomes insensitive to changes in Δs . Hence the associated uncertainty of the far-field momentum-based method is 9%. Considering now the energy-based method in comparison to the near-field, the difference is less than 9%, which means that the far-field momentum based method dictates the overall combined uncertainty. However, considering now only the far-field momentum-based and energy-based results, it can be seen that for a suitable control volume (i.e. where the result is insensitive to further increases in Δs and Δz) that the difference between the two is less than 1%. Therefore, there is a very good agreement between these far-field momentum-based and energy-based methods.

In the non-propelled fuselage case, there was a small difference between all three methods. The possible reason for the disparity with the near-field solution for the propelled case, may be due to the complex flows over the surfaces. The inclusion of the propulsor has introduced shocks within the jet plume and over the top of the nacelle. RANS struggle with BL shock interactions, as well as the separation induced at the trailing edge of the nacelle after-body. In the near-field method, the friction force calculated is dependent only on the velocity gradient within the first cell height, and this may cause the disparity due to errors in the BL within these troublesome regions. Given this discussion, it must once again be emphasised, that none of the methods are being prescribed as more accurate than the other. Instead, the intention is to show how uncertainties must be taken into consideration. Future work should look at extensive validation studies, to try and ascertain the causes of disparities. Nonetheless, in the context of this work, all three methods are used to quantify the uncertainty associated with BLI performance, with the near-field providing the most conservative predictions.

5.5 Propulsor Power

So far this chapter has been concerned mainly with extracting forces from different methods. However, of ultimate interest is the amount of power used to generate those forces. The formulation of the total power into the control volume was introduced in Section 3.7 and is given by Equation (3.35). This equation obtains the power from surface integrals on propulsor control volume surface S_P (see Figure 3.2). Following conservation, the same power may be extracted from the same surface integrals, but





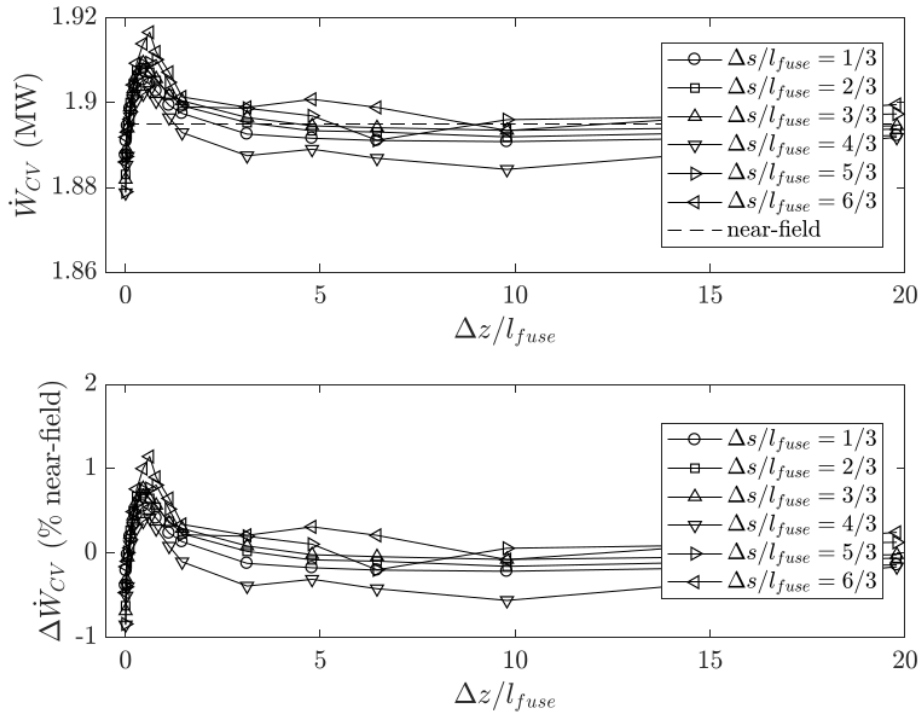


Figure 5.48: Comparison of near-field versus far-field predictions of propulsor power, for different control volumes around a propelled fuselage. Demonstrated on a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

was that not all of the surplus in kinetic energy, at the fuselage trailing edge, was dissipated to heat. This highlights the misconception that all of the kinetic energy surplus is realized as drag, and it can be recovered (ideally) through BLI.

The same three methods were used to predict the *NVF* of a propelled fuselage. The near-field momentum-based method was used to present the *NVF* in the form of different thrust-force accounting systems. A practical procedure for achieving this, has been documented. Subsequently, the *NVF* was also obtained from far-field momentum-based and energy-based methods, which were compared against one another and the near-field method to establish sources of uncertainty. It was found that the far-field momentum-based and energy-based results were in close agreement (within 1%), but both differed significantly from the near-field predictions (a difference of upto 9%). The large discrepancy from near-field values was attributed to the increased flow-field complexity over the airframe surfaces, due to interactions between BLs, shocks and jets. Thereafter, the section moves on to consider the wake's energy progression, where it was discovered that a large thrust force may be attributed to the volume compression work integral, $\dot{E}_{p\nu}$. This is a very important finding, and advances the work of Drela [5] who originally speculated this terms contribution to be negligibly close to zero.

The chapter is concluded with a final look at the differences in total energy inputs between near- and far-field derived values. It is observed that the discrepancies are insignificantly small, and that energy has been successfully conserved throughout the domain.

5.6.1 Concepts and Formulations

This chapter introduces the concept of CFD domain spatial partitioning, by establishing methods to identify flow-field regions defined by different physical mechanisms. A precedence procedure has also been developed, in order to separate out coupled flow regions in the most physically meaningful way. Overall jet regions (containing jet substructures) are first separated out using the stagnation enthalpy jet region sector:

$$\Upsilon = \left| \frac{\Delta H - H_\infty}{H_\infty} \right| > 1.1 \quad (5.8)$$

The highly compressed regions of the jet's core are then separated out using the shock region selector:

$$\zeta = \frac{\mathbf{V} \cdot \nabla p}{a|\nabla p|} > 1.1M_\infty \quad (5.6)$$

The BLs and viscous wake is extracted from the remaining jet flow, using the viscous region selector:

$$\xi = \left(\frac{\mu + \mu_t}{\mu} \right) \left[\frac{(\nabla \cdot \bar{\tau}) \cdot \mathbf{V}}{|p\nabla \cdot \mathbf{V}|} \right] \quad (5.4)$$

$$\xi > \frac{1}{\nu} \iiint_\nu \xi \, d\nu = \frac{1}{\nu} \sum_{i=1}^n \xi_i \quad (5.5)$$

This leaves behind the isentropic expansion regions of the jet core. The external flow (i.e. outside of the overall jet region) is then partitioned (in the following order) into an external shock region (using Equation (5.6)) and BLs and viscous wakes (using Equation (5.5)), which leaves behind the bulk potential flow of the rest of the domain.

This chapter has also introduced a procedure for representing the *NVF* using various thrust-force accounting definitions. The description of this procedure is too lengthy to summarise here, and the reader is referred to Section 5.4.2.

5.6.2 Contributions

The contributions of this chapter are as follows:

- The development of a new BL and viscous region selector, ξ , unique to this work. This is a significant contribution, because this selector accurately extracts the BL without neglecting the laminar regions of BLs closest to the no-slip wall condition. This flaw was observed in the descriptor proposed by Paparone and Tognaccini [7]. The selector, ξ , also has the advantage of being able to omit shock regions in both external and internal jet flows. This is due to the novel formulation, which has made use of the volumetric viscous dissipation and compression work terms found in the energy-based derivations of *NVF*.
- The spatial decomposition precedence procedure, along with the various selectors, is unique to this work (at least to the author's knowledge). The procedure allows for extraction of jet core expansion, core compression and transition regions, which is useful in better understanding the energy formulations of this work. In addition, the application of volumetric viscous dissipation and volumetric compression work integrals to these decomposed regions, is also deemed to be unique to this work.
- The section on viscous stress tensor flow-fields and near-wall treatments, provides unique insight into RANS CFD implications that need to be considered, when seeking consistency between momentum and energy-based methods of performance assessments. This section is mainly aimed at Fluent users, who may now use the outcomes of this chapter to better understand the solver settings and near-wall treatments appropriate for use (when using the methods presented in Chapters 3 and 4). Although this chapter has been based on Fluent results, it does give guidance to user's of other RANS tools.
- The back-to-back comparison of the different momentum- and energy-based methods is a first of its kind (at least to the author's knowledge). This comparison led to a better understanding of how spurious effects influence the force estimations when using different methods and control volume sizes. The control volume studies show how insight into spurious effects may be gained, when using a RANS CFD tool without having access to it's source code.
- The wake energy progression breakdowns presented in this chapter are comprehensive, and unique to this work. The author is unaware of any other studies,

which have performed these types of analyses on RANS CFD solutions. These breakdowns were performed for varying control volume sizes, which gave unique insight into the physical interpretations of energy terms.

5.6.3 Future Work

The suggested future work, following the outcomes of this chapter, are listed below:

Flow-Field Decompositions

The flow-field spatial decompositions have only been tried on 2D and 2D axisymmetric cases. Future work needs to comprehensively examine whether the procedure presented in this work, is equally valid and practical within 3D flows. In addition, future work should include a method for separating out lift induced effects, such as wing tip vortices. This is a crucial step, which is required to analyse lifting bodies.

Validation Cases

This chapter repetitively mentioned, during the back-to-back comparison of NVF predictions, that no particular method was prescribed as being correct or more accurate than the other. Instead, the focus of this chapter was to examine whether consistency between the methods could be established, and to quantify associated uncertainties. Future work should focus on applying these methods to an extensive list of validation cases, using different codes. This will not only validate the methods, but give improved insight as to which method may yield more correct results.

Understanding the Origins Spurious Effects

The topic of spurious effects was tackled using a black box approach, where the author did not have access to the source code of Fluent. A gap in the knowledge is how numerical spurious effects influence near- versus far-field (an energy) force predictions. It would be extremely useful for future studies to look into the theory and algorithms used in RANS CFD solvers, to explain better the reasons for the trends observed in this work.

Chapter 6

Fuselage Concentric BLI Propulsion Performance

Different methods for analysing BLI propulsion were presented in Chapters 3 and 4. These included (1) near-field momentum-based, (2) far-field momentum-based and (3) energy-based assessment approaches. The implementation of these methods to RANS CFD solutions, has been covered in detail in Chapter 5. This chapter now applies these methods in a design exploration, which looks at rear-fuselage BLI propulsion of business jets. In particular, the potential benefits of a fuselage concentric BLI propulsor, are assessed through parametric sweeps.

The chapter begins by presenting the outcomes of lower fidelity preliminary parametric assessments, which were used as a guide for the higher fidelity CFD parametric studies. Thereafter, the CFD tools and overall methodology are introduced, which gives a description of geometry definitions, determination of parametric variables and the procedure implemented to obtain solutions allowing for consistent performance comparisons. The propulsor is examined from the CFD parametric solutions, and its performance analysed in the context of actual FPRs, fan efficiencies, duct pressure losses, distortion characteristics and shaft power. Subsequent sections present the post-processed results for each of the vehicle assessment methods mentioned above. The near-field momentum-based assessment looks closely at surface forces, through pressure and friction coefficients, and examines the overall performance from the perspective of various thrust-force accounting definitions. The far-field momentum-based approach looks at the Reference Net Vehicle Thrust, and uses it to obtain an alternative estimate of the system-level performance. This is repeated for the energy-based method, but incorporates additional analyses which look at energy breakdowns and their contributions to *NVF*. The contributions of this chapter are as follows:

- Design exploration of BLI applied to a business jet which, to the author's knowledge, has not yet been published in public literature. This includes trends which show how the benefit of a retrofit design is optimised through managing thrust-split. This is based on a unique turbofan performance map, developed specifically to analyse thrust-split retrofit designs. In addition, the author is unaware of any

other publicly available studies which have considered the effect of different BLI propulsor tail-cone positioning.

- A tool and procedure for designing a retrofit fuselage concentric BLI propulsor, based on CFD solutions of the baseline fuselage. This tool serves to approximate geometries to meet basic design requirements within reasonable accuracy. This tool also automates geometry, mesh and solution generation. It interacts iteratively with the solution to ensure specified polytropic efficiencies for consistent comparisons. This includes a method for modelling the propulsor in CFD, which is preferred to actuator disk models used in other studies (please refer to Chapter 2).
- An analysis of the NVF using different BLI appropriate thrust-force accounting definition is a first of its kind (at least to the author's knowledge). It is demonstrated that better understanding is gained by using multiple thrust-force accounting definitions, as opposed to common practice which tends to adopt one definition for dividing up airframe surfaces between engine and airframe manufacturers.
- Clear evidence that the volumetric compression work, $\dot{E}_{pv}$, has a significant thrust contribution, which cannot be assumed to be small or negligible. This advances the work by Drela [5], who speculated its contribution to only be significant where heat is added at pressure within the control volume. This is now known not to be the case, and making NVF the subject of the energy equation gives insight as to why not.
- Application of a unique flow-mechanism partitioning procedure (see Section 5.1.4), has allowed for detailed extraction of jet structures, to which the volumetric dissipation, $\dot{E}_{\tau v}$, and volumetric compression work, $\dot{E}_{pv}$, have been applied. Analyses of these terms has confirmed force contributions of isentropic expansion, dissipative compression and viscous transition regions. To the author's knowledge, this form of energy breakdown is unique to this work and has provided very useful insight into contributions to NVF .
- Identification of a new method for separating out, more definitively, the shock contribution from BL-shock interaction regions. It is proposed that this may be done by iteratively determining the threshold of ζ (refer to Section 5.1.2) that corresponds to a minimum net amount of compression work in the viscous BLs. This is a unique improvement to the regional partitioning methods presented by Paparone and Tognaccini [7]. Demonstration of this proposed method has been left for future work.
- The final comparison of BLI benefit has been performed using various methods, based on different physical laws. This multi-physics based approach to BLI performance is unique to this work, and is opposed to prescribing a single method for future use. Instead, improved physics based and numerical understanding of CFD solutions is gained through comparisons between methods. In addition, these comparisons allow for uncertainties in predicted benefits to be quantified.

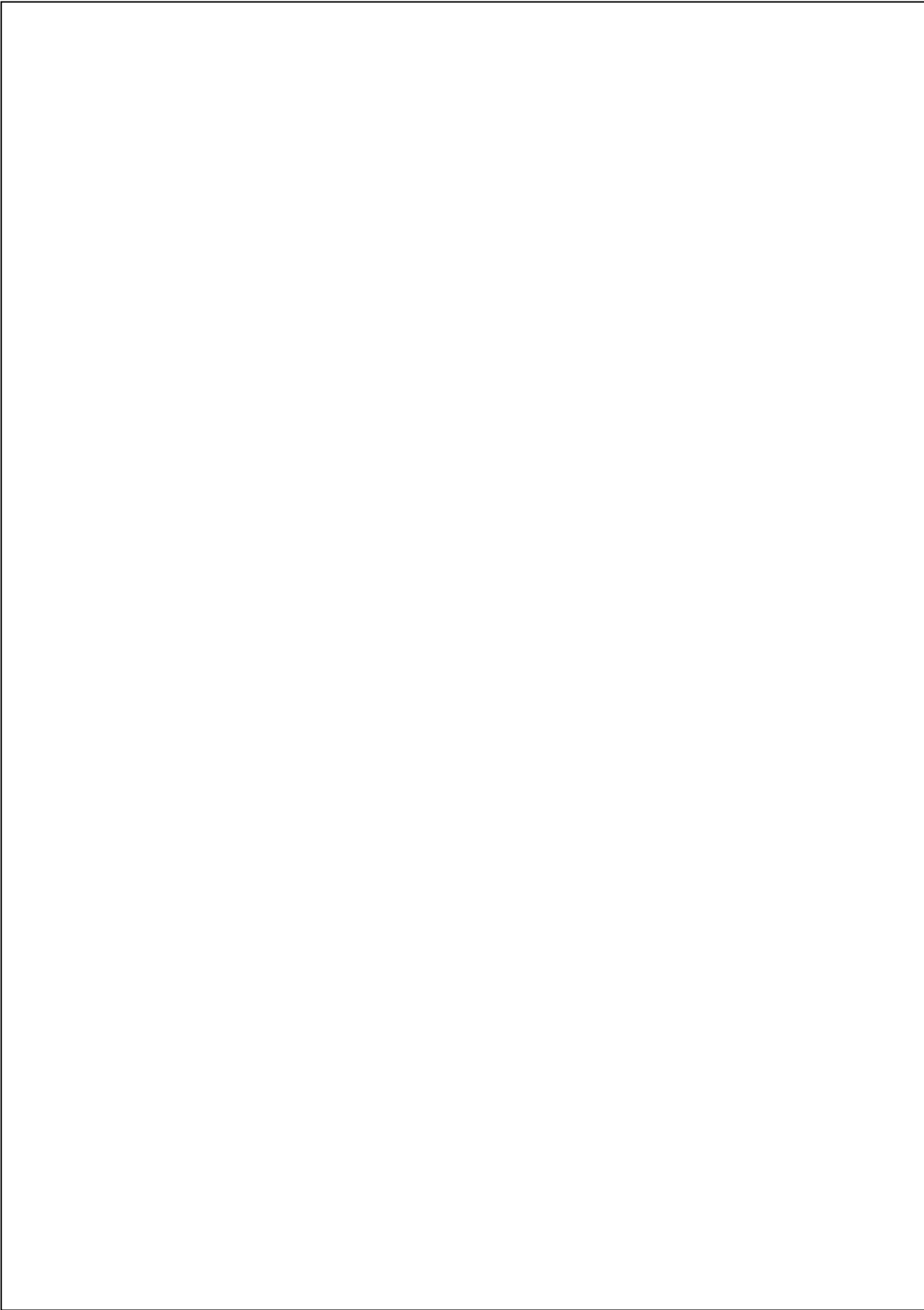
6.1 Outcomes of Preliminary Parametric Studies

The overall aim of this research, is to explore the design space of rear-fuselage BLI configurations, with particular application to a business jet. The configuration is to be powered by two main podded turbofan engines (i.e. free-stream ingesting), which mechanically power BLI fans whilst producing a proportion of the thrust themselves. A schematic of what the configuration might look like, is sketched in Figure 6.1. The constraint on the design exploration is to take a retrofit approach, whereby the same engines as the baseline aircraft are to be used to power the configuration. In other words, the same main engines are to be operated at an off-design condition to meet remaining thrust requirements specified by a BLI thrust-split, as well as to power the BLI fans. Following the retrofit design constraint, the fuselage of the BLI configuration is to maintain its geometry as close to the baseline aircraft's as possible. In doing so, the thrust-split is defined as the ratio of the Reference Net Vehicle Thrust over the entire net thrust requirement of the aircraft. The Reference Net Vehicle Thrust, $F_{N,ref}$, has been defined in Section 3.6. The total thrust requirement is determined by a drag-build up method of the baseline aircraft's components, which is presented in Appendix B. The following questions now arise:

- What is the optimum thrust-split?
- How much of the BL should be ingested?
- What are the sensitivities to BLI induced intake pressure losses?
- What are the implications of flow distortion on BLI fan performance?
- How does BLI impact the external aerodynamics?
- How does BLI affect the mission profile?

To begin answering some of these questions a preliminary parametric study, based on lower fidelity tools, is performed and is documented in Appendix F. The subsequent purpose of these studies, is to pave the way for the higher fidelity CFD assessments that follow in this chapter.

Briefly, the preliminary parametric study makes a number of assumptions. In keeping consistent with the Reference Net Vehicle Thrust accounting definition (see Section 3.6) it has been assumed that the total drag of the aircraft does not differ from the baseline. This discounts the aerodynamic coupling between the BLI propulsor and airframe, as well as the profile drag added by the BLI nacelle. Furthermore, any changes in induced drag, due to additional propulsion system weight, are ignored. Finally, assuming $MFR = 1$ makes it possible to equate the Reference Net Vehicle Thrust to the Intrinsic Net Thrust (i.e. $F_{N,ref} = F_{N,int}$). This is useful, because only the flow conditions at the highlight surface S_f are required to calculate $F_{N,ref}$ for a given FPR. In addition, because $MFR = 1$, there is no pre-diffusion and the ingested BL's profile is not affected by the presence of the propulsor. The BL profile along the fuselage is calculated from the 7th power







Lip Design

Figure 6.4 shows the intake and cowl geometry definitions in more detail. The highlight's conical frustum hypotenuse, h_i , is defined as being perpendicular to the tangent of the tail-cone circular arc at the specified position along the tail-cone, which is at an angle β_1 to the axis of symmetry. A straight line defines the distance along the fuselage between the highlight and throat, denoted by its conical frustum hypotenuse h_{th} . This is equivalent to the length of the nacelle's lip, which has been defined as an ellipse, of fineness ratio $EFR = \left(\frac{b}{a} : 1\right)$. A study by Luidens *et. al.* [43] on inlet designs, considered ellipse ratios of (1.5:1), (2.5:1) and (3:1), where the (3:1) ellipse was found to give the best performance at higher flight Mach numbers. Although Luidens *et. al.* study was applied to axisymmetric podded nacelles, Berrier *et. al.* [44] also found that (3:1) ratio outperformed a (2:1) ellipse for $M > 0.83$ in a design study on BLI flush mounted inlets. It must be noted that the impact on performance, in terms of pressure recovery, was deemed to be small in relation to other parameters. Nonetheless, the design cruise speed of the baseline aircraft in this thesis is M 0.85, and thus it was decided to use a (3:1) ellipse for all the geometry variations considered.

The lip thickness, a , is defined by a contraction ratio $CR = \frac{A_i}{A_{th}}$. Extensive use of a contraction ratio of 1.25 has been found in studies on podded axisymmetric intake and cowl designs [43, 45–47]. Berrier *et. al.* [44] considered slightly larger contraction ratios between 1.26 - 1.3, for designs of flush mounted BLI intakes. Despite subsequent research

podded installations, it is argued that the NACA-1 series provides a reasonable design allowing for consistent comparisons. Cowl design based on NACA-1 series derivatives have been studied by Langley [45] and more recently by Re [46]. Both investigated the effect of highlight-to-crest diameter ratios on performance for axisymmetric cowls of podded installations at subsonic ranges M 0.6 to M 0.92. Subsequently, the cowl geometry in this work has been based on profiles (namely cowl 6 from Langley [45]), which are deemed to give improved performance at the high subsonic Mach numbers flown by the baseline business jet.

Intake Duct

The intake duct is defined as carrying the flow from the throat to the fan inlet, as depicted in Figure 6.4. The purpose of the intake duct is to further diffuse the flow to meet design Mach inlet requirements at the fan face, should this not have been achieved yet within the pre-diffusion region ahead of the intake. Additionally, the duct must be able to route the flow in such a way that the tail-cone geometry is conformed to in a continuous manner at the highlight and exhaust planes. Therefore, a method for constructing ducting curves, allowing for a variety of area distributions and centre-line offsets, was required. This method also needed to be robust and simple to implement, given the number of geometries to be investigated in this design exploration. There are many different types of design approaches for BLI ducts in literature, but each specific to their own particular design applications. References [44, 48, 49] design the ducts for application in a BWB, and it is not apparent as to how this design came about. Anabtawi *et. al.* [52] have used a method dependent on Stratford-like pressure gradients [53], but did not describe how this was done. Instead, the work of Bell and Mehta [54] provides a more practical, and simple to implement approach, to designing ducts. Bell and Mehta performed experimental research on contracting small low-speed wind-tunnels, using polynomials to describe different shapes. A 5th order polynomial was found to reduce the onset of separation, and is given here in local intake xy coordinates:

$$y(x) = y_{th} - (y_{th} - y_{fan,i}) \left[6(x')^5 - 15(x')^4 + 10(x')^3 \right] \quad (6.1)$$

where x' is the non-dimensional location $\frac{x}{L_{intake}}$, and y_{th} , $y_{fan,i}$ are the duct locations (following the coordinate system in Figure 6.4) at the throat and fan inlet for a given centre-line offset Δh .

After-Body

Figure 6.5 depicts the geometry definitions of the after-body and exhaust duct. The majority of work on BLI nacelle design has focussed on the intake region, and little is available on suitable after-body designs. For instance, References [44, 48–50] appear to use circular arcs with the radius determined according to an after-body length and boat-tail angle. This is a simplistic approach, but does have the advantage of being easy to implement, robust and easily transferable to different geometries. Furthermore, it is



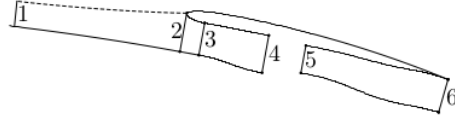


Figure 6.6: Fuselage concentric BLI propulsor station numbers.

Table 6.1: Summary of propulsor station calculation inputs

Input	Description
BLR	Portion of the BL to be ingested (variable)
Pos. h_i	The highlight position along the tail-cone (variable)
FC	Flight Conditions (FL450 M 0.85)
M_4	Fan average inlet Mach number, $M_4 \leq 0.65$ [56]
CR	Lip contraction ratio $\frac{A_i}{A_{th}}$ (1.25 assumed)
DS	Diffusion split ratio $\frac{M_1 - M_3}{M_1 - M_4}$ ($DS = 1$ assumed)
PR_{13}	Pressure recovery up to the throat area (99% assumed)
PR_{34}	Intake pressure recovery (98% assumed)
PR_{45}	Exhaust and nozzle pressure recovery (99% assumed)
FPR	Fan pressure ratio (variable)
η_{poly}	Fan polytropic efficiency (89% assumed)
η_{noz}	Nozzle isentropic efficiency (95% assumed)

quired to generate a geometry that represents these design space combinations. The pre-processing tool first approximates the areas of the different aerodynamic stations of the propulsor (see Figure 6.6) which are the inputs to defining the geometry as per Section 6.2.1. Secondly, the thermodynamic properties at fan inlet and outlet are required to initialise the inlet boundary conditions of the CFD domain. The process of calculating the thermodynamics at the propulsor aerodynamic stations, is summarised by Figure 6.7. The objective is to first determine flow profiles of the BL portion to be ingested. Thereafter, a set of inputs, summarised in Table 6.1, allow for calculation of the average flow properties at each station. A station vector of static and stagnation flow properties are solved iteratively at each station, assuming an ideal gas and based on the flow properties by McBride *et. al.* [57]. This station vector includes (amongst others) static and stagnation pressures, temperatures and enthalpies, as well as densities, areas and velocities.

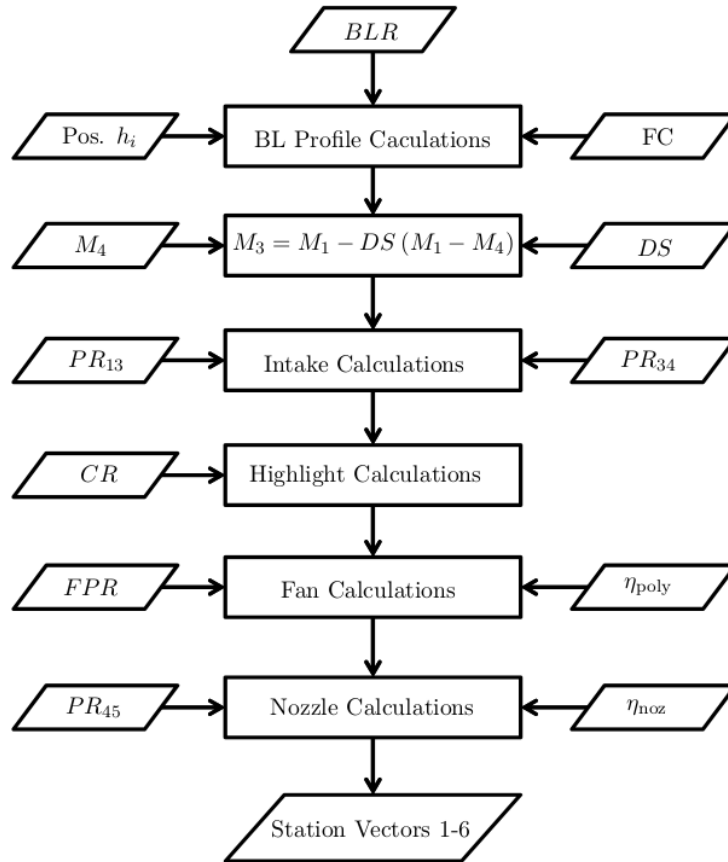


Figure 6.7: Flowchart of pre-processing tool station calculations.

BL Profile Calculations

Station 1 marks the beginning of the pre-diffusion stream-tube, and its station defines how much of the BL is ingested. However, it is not possible to know Station 1's position prior to solving the flow around the geometry, which is not yet known. As such, an assumption must be made in order to approximate the properties for this station. The approximation assumes that Stations 1 and 2 coincide, which amounts to a $MFR = 1$. Now the position of Station 1 is known, as it is specified at the start of the highlight area, and this is an input to the design exploration. Based on this assumption, the boundary layer height δ at this position must first be determined from the baseline fuselage solution, which has been documented in Appendix C. The tool written in Matlab interacts with Fluent's post-processing applications to determine δ using the viscous layer descriptor ξ presented in Section 5.1.1. An interior surface is created in Fluent and constructed perpendicular to fuselage surface at the highlight point. The flow properties extracted from this interior surface, are interpolated linearly to locate δ 's position corresponding to the threshold of ξ defined by Equation (5.5), and this returns BL profiles of all flow properties made available by Fluent. Further linear interpolation is applied to obtain profiles corresponding to the requested BLR, and this sets the mass-flow-rate through the propulsor. Subsequently mass-flow-averaging of the profiles allows for the remaining station vectors of the propulsor to be determined.

Highlight and Intake Calculations

The throat and fan inlet stations (i.e. 3 and 4) are determined according to a design constraint on the maximum allowable average Mach number at the fan face. This is obtained from continuity based on estimates of pressure recoveries in the pre-diffusion and intake streamtubes (i.e. PR_{13} and PR_{34}), and is governed by a diffusion split ratio DS . The diffusion split allows for examination of different MFR s for the same BLR and M_4 . However this work assumes a $DS = 1$ to reduce the number of cases within the design space¹. The station vectors, and importantly areas, are iteratively calculated from known stagnation pressures, stagnation temperatures and Mach numbers. The highlight area may now be determined according to the specified CR . This allows for an iterative procedure to determine its station vectors from stagnation pressure, temperature and area.

Fan and Nozzle Calculations

The fan outlet stagnation properties (at station 5) are calculated based on a specified FPR and η_{poly} , and are standard calculations found in most texts on gas turbine performance [56,58]. Thereafter, the axial velocity across the fan is assumed constant in order to calculate the area at the fan exit from the stagnation properties. This is deemed a reasonable first iteration design approach [56]. Finally, the station vector at the nozzle

¹The impact of variations of DS is left as future work. This parameter may have a significant influence on results due to the effect of MFR on external aerodynamics around the nacelle.

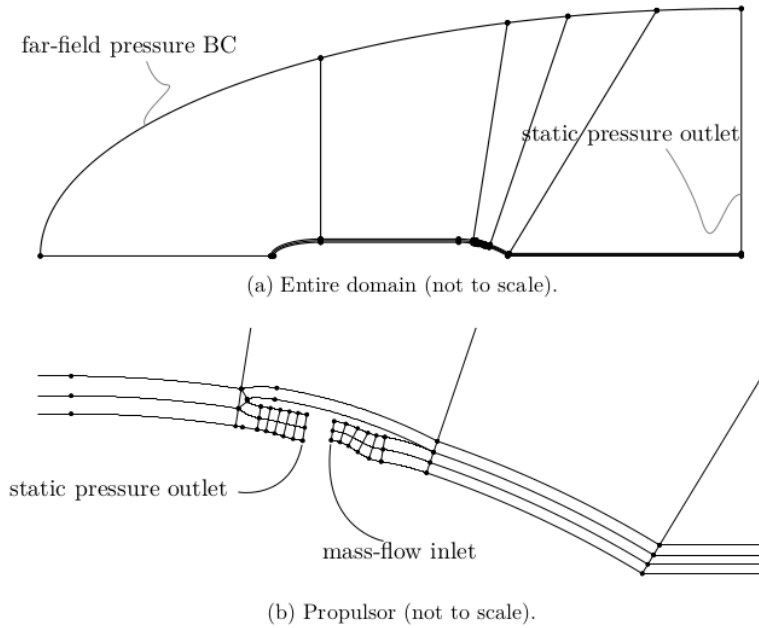


Figure 6.8: Domain blocking and boundary conditions (not to scale).

exit (station 6) is calculated using an estimate isentropic efficiency and exhaust pressure recovery. The calculation is based on the procedure found in Reference [58].

6.2.3 Meshing Tool

A Matlab controlled ANSYS ICEM script has been written for automatic generation of the mesh. Firstly, Matlab generates the blocking structure outside of ICEM, an example of which is depicted in Figure 6.8. The domain surrounding the assembly has an elliptical inlet, far-field pressure boundary condition, which extends to a downstream trefftz plane with static pressure outlet boundary condition. The domain in general is blocked with a C-grid. Figure 6.8b illustrates the blocking around the propulsor. A C-grid is placed around the nacelle, and the ducts are split to reduce cell skewness. An example of a meshed propulsor is depicted in Figure 6.9.

6.2.4 In-Solution and Post-Processing Tools

The Matlab script interacts with Fluent to load the mesh and initialise the case with boundary conditions calculated by the pre-processing tool. The solver settings have already been discussed in Section 5.2. To help initialise the final solution, the fan's static pressure outlet boundary condition is first set without the target mass-flow requirement,

Figure 6.9: Propulsor mesh example for a fuselage concentric propulsor placed midway along the tail-cone, ingesting 40% of the BL and operating at a $FPR \approx 1.9$.

thereby not conserving mass-flow across the fan. Discretization accuracy is gradually increased from first order upwind to second order upwind to ensure convergence where possible. Convergence is monitored based on profiles of total pressure either side of the fan as well as net surface forces over the airframe and nacelle. Having converged, the outlet static pressure target mass flow is now set. Fluent then allows the static pressure to vary to match the specified mass-flow-rate, restoring mass flow conservation across the propulsor. After re-convergence, Fluent returns inlet and outlet profiles to Matlab, which then calculates the FPR and polytropic efficiency across the fan. Based on a Regular Falsi method, a new fan exit total temperature is calculated to ensure the specified polytropic efficiency is adhered to. The inlet boundary condition total temperature is repetitively updated until the solution converges on the correct polytropic efficiency. It must be noted that the mass-flow-rate through the propulsor is kept constant so as to represent the initial BLR specified. However, the intended FPR will not be achieved because of errors in the approximations made by the pre-processing tool in calculating propulsor areas. Nonetheless, it was found that the FPR achieved did not vary too drastically, and results could be interpreted from sufficiently broad sweeps. Finally, a post-processing tool was developed in Matlab, and interacts with Fluent to allow for automated calculations of formulations found in Chapters 3 and 4.

6.2.5 System Level Performance at Cruise

In order to obtain the system level performance, the Reference Net Vehicle Thrust $F_{N,ref}$ is used to determine the Thrust-Split of the aircraft's total required thrust. In this case

$F_{N,ref}$ represents the change to the fuselage's net force (NFF), and any surplus/deficit is compensated by the main engines. The main engines must also produce additional thrust to counter the drag of the other airframe components (estimated from the drag build up method in Appendix B). Hence, the thrust split is given as the ratio:

$$TS = \frac{F_{N,ref}}{F_{N,vehicle}}$$

Furthermore, the power required to drive the BLI propulsor is calculated from the CFD solution and after accounting for a transmission efficiency estimate, the power off-take from the main engines is also known. The turbofan model described in Appendix D is then used to determine the corresponding fuel flow at the off-design point which meets the thrust and power off-take requirements demanded of the engine. The new fuel flow rate is then used to calculate a new SFC for the entire propulsion system (i.e. combined main engine and BLI propulsors):

$$SFC = \frac{\dot{m}_f}{F_{N,vehicle}}$$

which may be better presented as a percentage of the baseline engines SFC in the absence of a BLI propulsor. This process is simplified by using the decoupled turbofan performance map presented in Figure 6.33, which gives correlations of changes in SFC for different combinations of thrust-split and BLI fuselage propulsive efficiencies (following the modified definition of propulsive efficiency, Equation (3.38)).

6.2.6 Design-Space Exploration Parametric Cases

Two different tail-cone positions are considered in this work, as depicted in Figure 6.10. Position A is placed at the top of the tail-cone, and its highlight located where the fuselage and tail-cone meet. Position B has the propulsor's highlight located midway down the tail-cone. This is as far back as the geometry constraints would allow when considering ingesting 100% of the BL. The purpose for considering different positions, is to further understand the implications (and possible synergies) of the raised static pressure on the tail-cone both ahead (due to pre-diffusion) and aft (because of the jet) of the fan.

The next main governing design space parameter is the amount of BL ingested, i.e. BLR. It was not deemed necessary to ingest more than the BL thickness δ , as beyond this the flow approaches free-stream conditions and benefits diminish rapidly. The BLR range investigated was between 20-100% at 20% increments. For each tail-cone position and BLR value, the FPR was varied between 1.4 - 1.9 at 0.1 increments. It is noted that this was the FPR input to the pre-processing code, and the resultant FPR differs due to the actual flow solution. Nonetheless, the purpose for varying the FPR is to vary the NVF achieved for different BLRs. The polytropic efficiency is fixed to allow for consistent fan power comparisons.

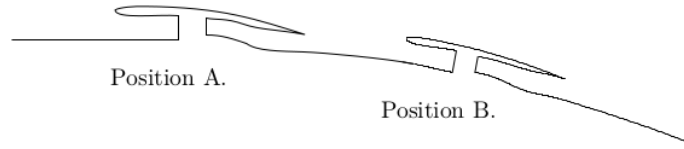


Figure 6.10: Tail-cone propulsor positions examined

6.3 Propulsor Performance

This section examines the performance of the propulsor obtained from the CFD solutions, as a result of the geometry and boundary conditions produced by the pre-processing tool. First, the actual FPR is compared to that predicted by the pre-processing tool, along with corresponding isentropic efficiencies. Secondly, the fan's average inlet Mach number (along with its corresponding mass-flow-ratio) is examined to see how well the pre-processing tool was able to meet the design requirement of $M_4 = 0.65$. Subsequently, trends in pressure losses within the pre-diffusion, intake and duct streamtubes are discussed (also with reference to the estimated values used in the pre-processing tool). This is followed by a discussion on radial distortion seen at the fan face. Finally, the trends in propulsor power are presented. The above are all described with reference to both tail-cone positions considered, see Figure 6.10, as well as the various BLR and FPR combinations.

6.3.1 FPR and Isentropic Efficiency

The actual fan pressure ratios achieved by the geometry to match the specified polytropic efficiency of 89%, are depicted in Figure 6.11, along with the corresponding isentropic efficiencies. Considering first tail-cone Position A, it is observed that the actual FPR is lower than the pre-processing tool inputs, but that the values are fairly constant for different BLRs (which in-turn results in fairly constant isentropic efficiency values). This is due to errors in approximated nozzle exit area by the pre-processing tool, which assumes that the static pressure external to the propulsor is equal to the free-stream conditions. This is clearly not the case, as the combined presence of the nacelle on the curved tail-cone results in a different local static pressure. This affects Position B more prominently, because the local static pressure gradients further down the tail-cone are steeper and varying BLR changes the nozzle's position along the tail-cone by substantial amounts. For example, the nozzle exit for BLR = 20% is fairly close to the highlight's midway position along the tail-cone, whereas a BLR = 100% positions the nozzle exit close to the tail-cone's trailing edge. The local static pressures surrounding these regions is very different, and explains the trends seen in Figure 6.11c due to the area approximation error. To demonstrate this effect, Figure 6.12 illustrates the pressure contours surrounding the propulsor in two different ingestion scenarios. This figure clearly shows that the nozzle exit plane is submersed in different local static pressure regions, which are also different from the free-stream condition. Future work may wish

to iterate between the flow solution and pre-processing tool to ensure that the correct area is calculated to deliver the exact FPR requested at the desired polytropic efficiency.

6.3.2 Pre-Diffusion and Fan Inlet Mach Number

The pre-processing tool approximates an intake geometry to try and ensure the fan inlet design Mach number is not exceeded. It was preferable to have as large a mass-flow-ratio (MFR) as possible so as to avoid excessive shock strengths over the top of the nacelle, which result in a large separation bubble at the nacelle after-body's trailing edge. In doing so, it was necessary for the pre-processing tool to construct the geometry such that the maximum permissible Mach number at the fan face is reached. The mass-weighted-average design Mach number specified was 0.65, as this is the value recommended for current fan technology [56]. The pre-processing tool approximates the ingestion profile at Station 1, which results in slight deviations away from this design requirement. Figure 6.13 gives an indication as to how closely the correct fan area was approximated. Also shown are the corresponding MFRs based on the areas of the pre-diffusion and highlight surfaces S_c and S_f , respectively. A correlation between the MFR and discrepancies is observed, whereby the closer MFR is to 1 the smaller the discrepancy. This makes sense, as a $\text{MFR} = 1$ assumption was made when calculating the fan areas in the pre-processing tool (see Section 6.2.2). The design Mach number is not exceeded for Position A, but is for Position B. However the amount by which it is exceeded is sufficiently small. Future work may wish to create an iterative loop between the pre-processing tool and the solution in order to match exactly the intended Mach number at the fan face.

6.3.3 Propulsor Pressure Losses

The pre-processing tool is reliant on estimations of the total pressure losses throughout the system, and this affects the geometry constructed. A constant total pressure loss has to be assumed, as these are unknown prior to having the CFD solution. One of the purposes of this work is to map out the variations in total pressure losses throughout the propulsor system. Figure 6.14 illustrates the trends of total pressure recovery in the pre-diffusion, intake and exhaust streamtubes for both tail-cone positions. The first trend observed is that pre-diffusion pressure recovery decreases with an increase in BLR. One reason is the correlation with MFR, whereby better pressure recovery is realised for MFRs closer to 1. This is because less diffusion occurs and the BL is subject to a lower adverse pressure gradient (BLs are sensitive to adverse pressure gradients, more prone to separation and incur larger pressure losses [59]). The trends within the intake and exhaust duct follow the opposite direction, whereby pressure recovery improves with increased BLR. This is because the relative amount of mass-flow-rate contained within the BL (compared to the mass-flow-rate of the entire streamtube) decreases with increasing BLR, and therefore a smaller proportion of streamtube's mass-flow-rate is subject to losses. This is the reason for the similar trends observed in both intake and exhaust ducts. Comparing Position A and B, it is observed that pressure loss magnitudes

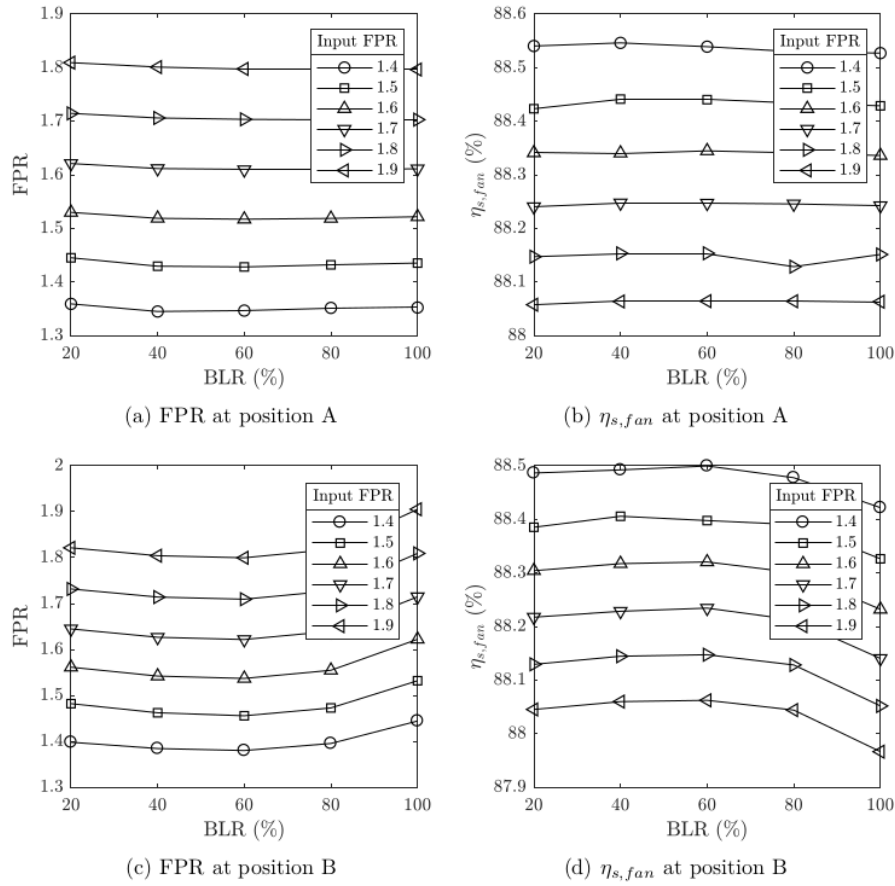


Figure 6.11: Fan pressure ratio and isentropic efficiency achieved to match polytropic efficiency of 89% for a geometry estimated to produce a specified input FPR



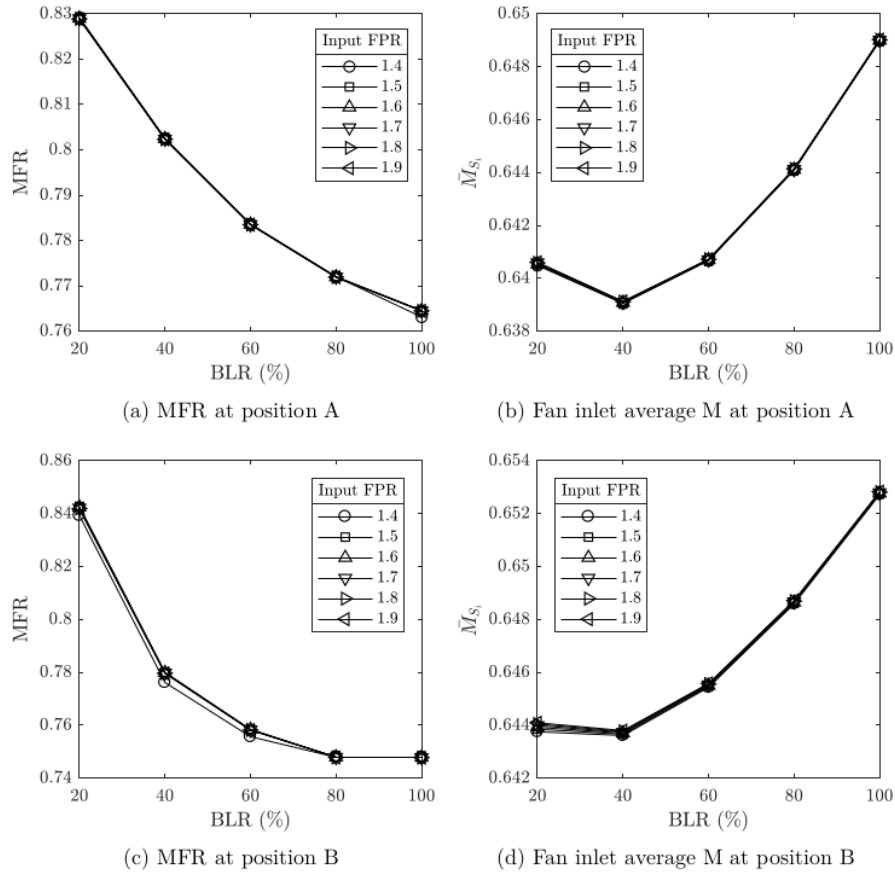


Figure 6.13: The mass-weighted-average fan inlet (surface S_i) Mach number achieved by the geometry (design target set to $\bar{M}_{S_i} = 0.65$), with corresponding MFR

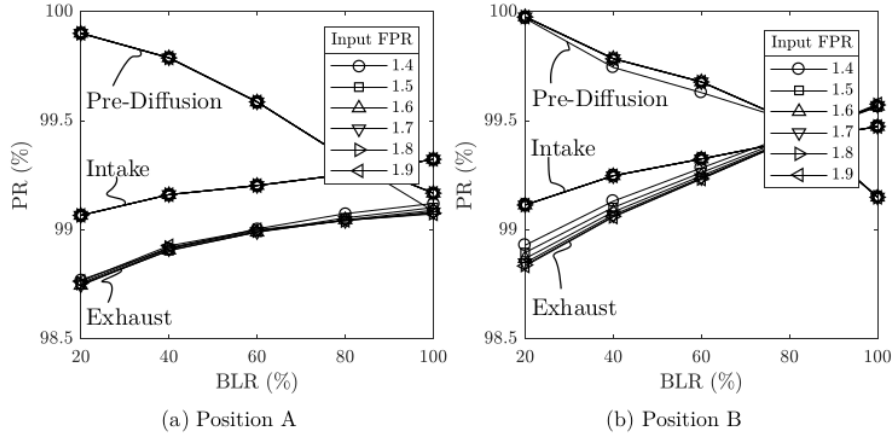
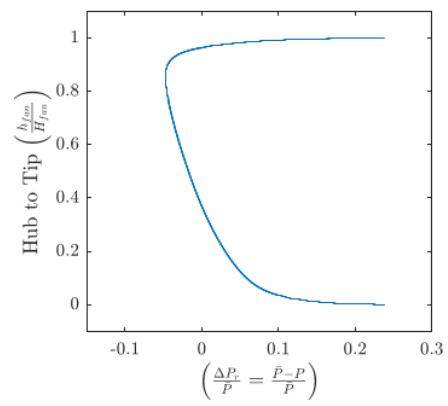


Figure 6.14: Pressure recovery within the pre-diffusion, intake and exhaust internal stream-tubes

are very similar. As already mentioned in the previous two subsections, future work may wish to incorporate an iterative loop between pre-processing and in-solution tools, and these pressure recovery values become important in predicting geometries that will result in the correct FPR for a given polytropic efficiency, and M_4 .

6.3.4 Distortion Descriptors and Intensity

In assessing the fan performance, it is also important to consider the impact of flow distortion. The concentric BLI propulsors of this study will only suffer radial distortion, due to the 2D axisymmetric geometry. The level of distortion at the fan face is measured using Radial Distortion Intensities descriptors, profiles of which are depicted in Figure 6.15 for each inlet condition. Classical distortion patterns have been studied experimentally and using parallel compressor models by Cousins and Davis [61,62], who took into consideration both hub and tip radial distortions and their impact on surge margin. Cousins and Davis' parallel compressor model indicated a 1.9% and 6% reduction in the surge margin for tip and hub distortion, respectively [61]. However, as the paper mentions, the parallel compressor model becomes less valid for hub distortions, partly due to the averaging method used in mapping fan speed lines (as the tip of a fan stalls first). Nonetheless, as shown in Figure 6.15, the distortion entering the concentric BLI propulsor is hub dominant and faces the concerns of reduced surge margin. However, radial distortion can be designed for through appropriate blade twist, and it is therefore assumed that negative effects could be mitigated through good design. Examination of Figure 6.15 indicates that larger BLR ratios result in larger distortion intensities, due to the greater difference in minimum and maximum total pressures. However, it must be noted that the variation from hub-to-tip is far more linear than the intensities at



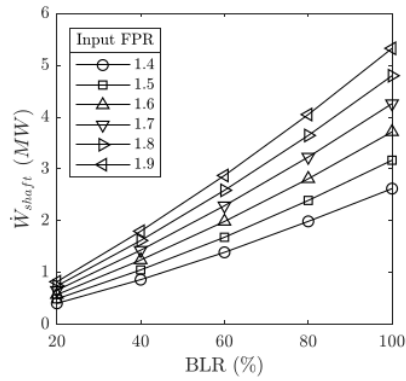
smaller ingestion thicknesses. This non-linearity may lead to more difficult fan design. Unfortunately at this stage of the work it is not possible to predict the efficiency and structural implications due to the distortion. In addition most research has focussed on circumferential (and circumferential-radial coupled) dominant distortion in BLI applications [63, 64]. In summary, the impact of flow distortion is coupled to fan design, which falls outside the scope of this thesis.

6.3.5 Shaft Power

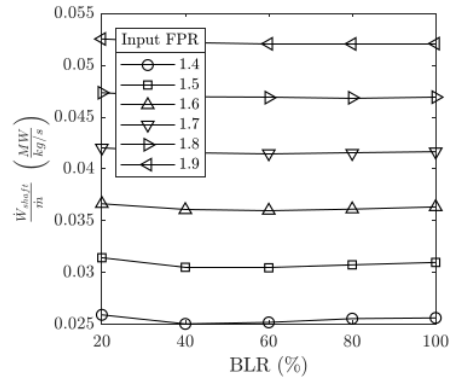
Figures 6.16a and 6.16c show the fan shaft power (excluding any mechanical efficiencies) for the different tail-cone positions, ingestion thickness' and input FPRs. As expected, the power increases with increase in BLR because of the increased mass-flow-rate. Additionally, the power naturally increases with fan pressure ratio as larger amounts of work are imparted to each unit of mass-flow-rate. In fact, dividing the power by the mass-flow-rate yields an almost constant mass-specific-power for a given pressure ratio, as is depicted in Figures 6.16b and 6.16d. This was the purpose of opting to use a constant polytropic fan efficiency to allow for consistent comparisons. At Position B, the trends in Figure 6.16d are not flat, but correspond to the approximation errors in FPR as was discussed previously in Section 6.3.1. This may further motivate future work to incorporate an iterative interaction between the pre-processing tool and the solution, to arrive at exactly the specified pressure ratios, resulting in exact mass specific powers for a given polytropic efficiency. This would provide improved consistency for comparisons of the same FPR at different BLRs.

6.3.6 Summary

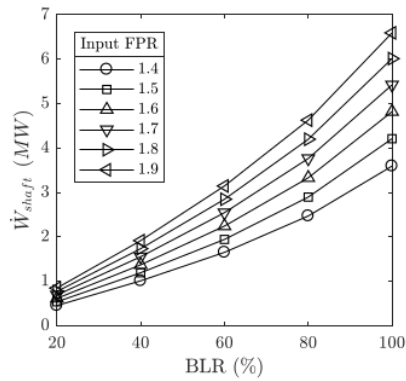
This section has presented the performance of the fan post-solution, and the results have been compared to the design requirements given to the pre-processing tool. The approximation methods of the pre-processing tool were able to generate geometries which resulted in FPRs fairly close to the input values. However, there were greater variations for propulsors at Position B over Position A, which resulted in variable mass-specific fan powers. The average inlet fan Mach number was kept within a reasonable tolerance of the maximum allowable value, and the pressure losses in the system were found to be relatively small and close to the estimated values. In future work, it may be worthwhile iterating between the pre-processing tool and the solution, to exactly match the required FPR and fan inlet Mach number. This would improve the consistency when comparing different BLRs for a given FPR. This was not done in this work due to the increased complexity and computational time involved. Furthermore, the approximation errors are not deemed large enough to significantly affect the results of this initial design exploration.



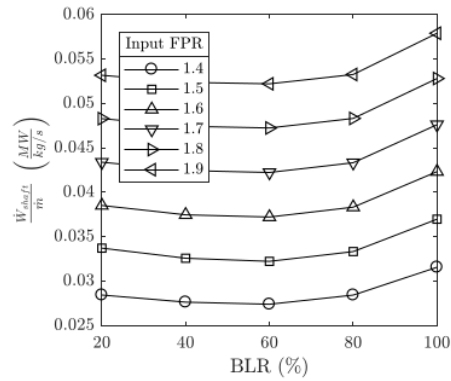
(a) Pos. A shaft power



(b) Pos. A mass-specific power



(c) Pos. B shaft power



(d) Pos. B mass-specific power

Figure 6.16: Fan shaft power in explicit and mass specific forms, to illustrate the purpose of a constant polytropic efficiency.

6.4 Near-Field Momentum-Based Assessment

This section presents the results of the design exploration using the near-field momentum-based method introduced in Chapter 3. The section begins with the examination of surface forces, in terms of pressure and friction coefficients, over the fuselage and nacelle walls. Subsequently, the thrust-force accounting systems of Table 3.1 are compared for the different tail-cone, BLR and FPR combinations of this design exploration. Thereafter, the Reference Net Vehicle Thrust, $F_{N,ref}$, is used to determine system-level performance, whereby the modified propulsive efficiency and thrust-split are used to determine changes in SFC from a decoupled turbofan performance map.

6.4.1 Surface forces

The purpose of this section is to examine the pressure and friction coefficient distribution over the fuselage and nacelle surfaces. This section first considers the pressure coefficients over the fuselage and nacelle, before examining the friction coefficient over the fuselage surfaces

Fuselage Pressure Coefficients Upstream of Fan Inlet

Figures 6.17 and 6.18 depict the pressure coefficients along the fuselage, upstream of the fan face, for Positions A and B respectively. Figures 6.17a and 6.18a, show the pressure coefficient from the fuselage's leading edge to the fan face (the baseline non-propelled fuselage is depicted up to the tail-cone trailing edge). It can be seen from these figures, that the pressure distribution matches that of the baseline for most of the fuselage length. Figures 6.17b and 6.18b zoom in on the regions where the pressure coefficient begins to differ significantly from the non-propelled baseline, and the variation with BLR becomes clearly visible². The dashed lines indicate the positions where the flux surfaces S_c , S_f and S_i (refer to Figure 3.4) intersect the fuselage. The locations of peak pressure are also indicated, which corresponds to a more correct definition of the highlight plane intersection (see the discussion in Section 5.4.2). This shows that the error in the approximated S_f definition is fairly small.

Examining the pre-diffusion region (between S_c and S_f), it is clear to see that the pressure coefficient is raised above the baseline and increases with BLR. This is due to the decrease in MFR required to reduce the flow velocity to the design Mach number at the fan face. Increasing C_p has no direct pressure force consequence on the fuselage at Position A, because the fuselage has a purely radial facing area. At Position B, on the other hand, the rearward facing area means that increasing the C_p results in a change in force in the thrust (upstream) direction. At first this seems advantageous, but it must be remembered that local pressure also affects the nacelle, and therefore any thrust contributions may be mitigated. After the pre-diffusion streamtube, the pressure coefficient drops as the flow is accelerated around the nacelle lip to the throat. In this

²These figures actually include plots of all FPR variations, but the FPR has an insignificant influence on the upstream pressure. Therefore all FPR lines, for a given BLR, collapse onto one another

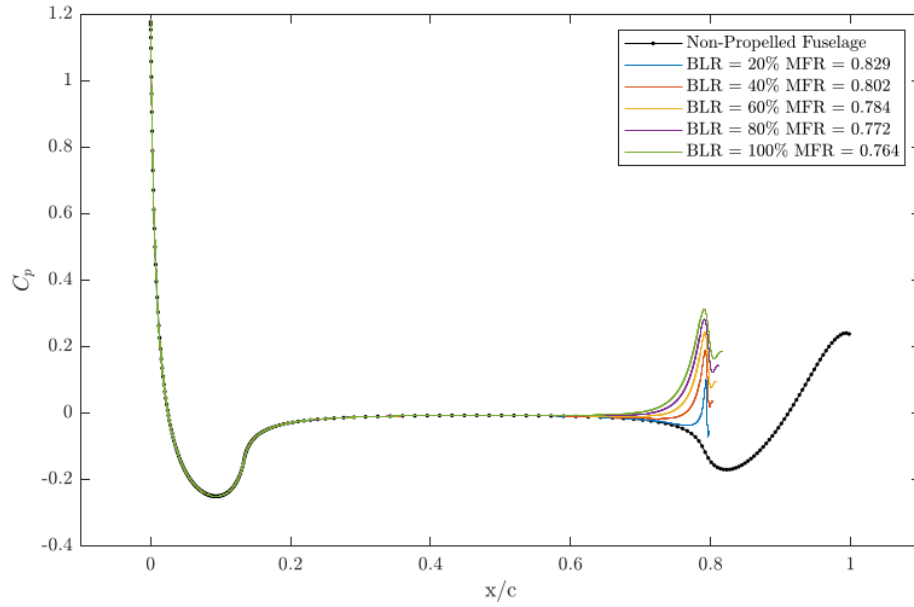
design exploration, the intake ducts were chosen to have constant area. As a result, the acceleration carries on beyond the throat, which leads to a further drop in C_p . A slight pressure wave develops and C_p oscillates in the duct up until the fan face.

Fuselage Pressure Coefficients Downstream of Fan Exit

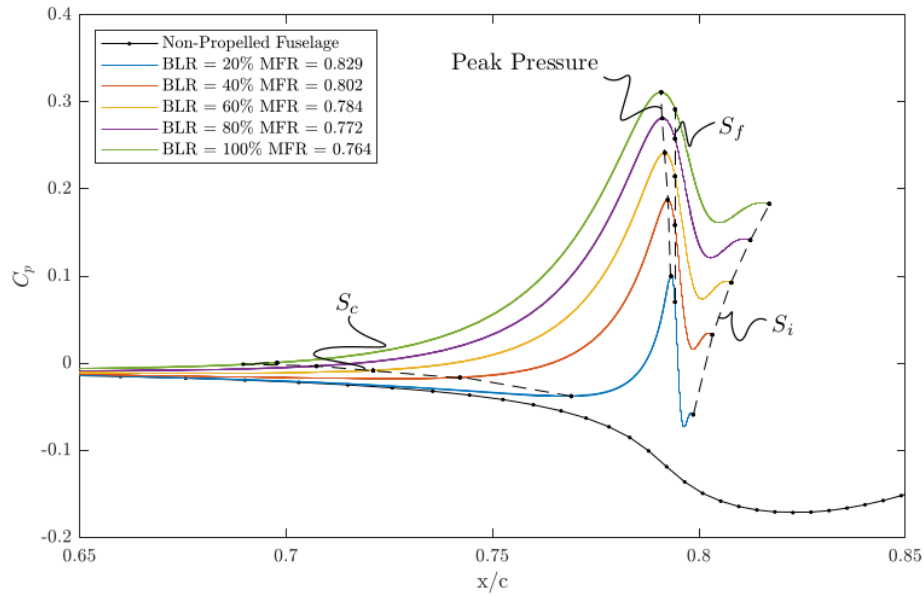
Figures 6.19 and 6.20 illustrate the pressure and friction coefficients along the fuselage surfaces S_k and S_m downstream of the fan exit surface S_j , passing through the nozzle exit plane S_l to the tail-cone's trailing edge. This is depicted for tail-cone propulsor positions A and B, respectively. Considering first Position A, the general shape of the pressure coefficient illustrates how the jet undergoes expansion to a static pressure coinciding with that of the non-propelled fuselage. Having a greater amount of stagnation pressure at this point, the flow then undergoes diffusion due to the conical axisymmetric shape of the tail-cone. This leads to a final static pressure at the fuselage trailing edge which is greater than that of the non-propelled case. Because of the increased stagnation pressure, the boundary layer within the jet has a greater amount of energy and does not separate as easily. This results in a suppressed separation bubble at the trailing edge in comparison to the non-propelled fuselage, and subsequently greater pressure recovery. The result is an overall increase in the pressure coefficient over the tail-cone which, as a rearward facing geometry, leads to a larger pressure force in the upstream direction. The increase in pressure coefficient over the tail-cone, along with the trailing edge separation suppression, may be an advantageous synergy achieved through integrating the propulsor on to the fuselage. However, it is noted that these seemingly positive effects may be mitigated when considering how the local pressure affects the nacelle surfaces. The same observations may be made for Position B propulsors, except that the C_p of the expanding jet does not always reach a local minimum coincident with the baseline. Comparing the individual cases more closely, there is an obvious oscillatory pattern which develops with an increase in FPR. At low FPRs the flow is fully expanded to local static pressure within a close vicinity of the nozzle exit plane. As expected, expansion and compression waves begin to grow in number and strength with an increase in FPR. The higher the FPR, the more under-expanded the nozzle becomes. Harrington [65], examined experimentally the static pressure along the plugs of convergent nozzles over a range of Mach numbers, where a similar pattern in static pressure may be observed. Franquet *et. al.* [38] provide a review of the structures within free under-expanded jets. Although this flow is an impingement scenario (due to the presence of the tail-cone surface) the oscillatory nature of the static pressure can be understood through analogies to the structures within free jets. The general structure of a moderately and highly under-expanded free jet is summarised in Figure 5.7, where it can be seen that the jet undergoes repetitive isentropic expansion and dissipative compression.

Nacelle Pressure Coefficients

Figures 6.21a and 6.21b depict the pressure coefficients over the internal and external nacelle surfaces for propulsor Positions A and B, respectively. Considering first the



(a) Fuselage nose to fan face



(b) Pre-diffusion region to the fan face

Figure 6.17: Pressure coefficients along the fuselage upstream of the fan, for Position A (with comparison against the non-propelled fuselage baseline).



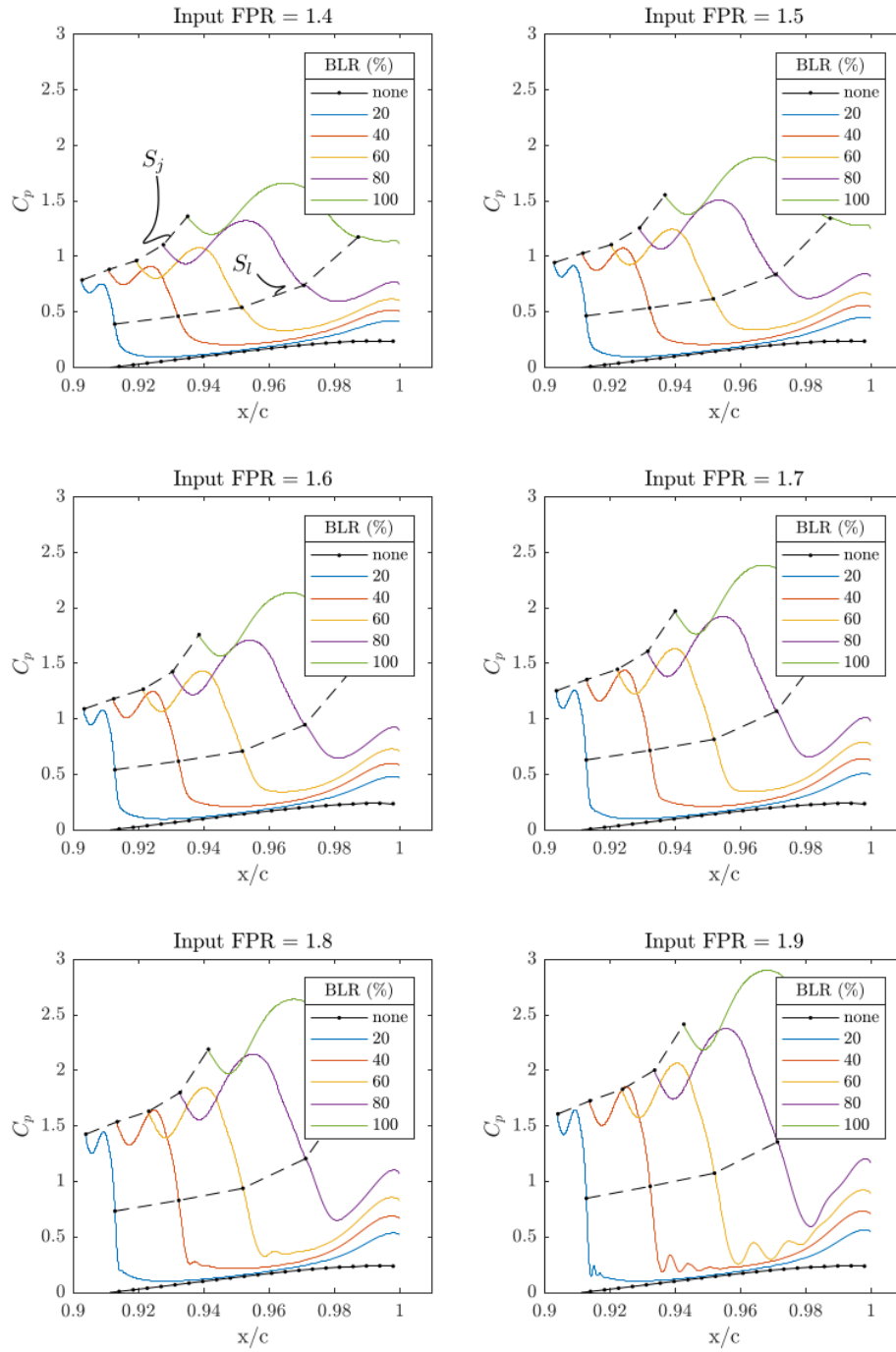


Figure 6.20: Pressure coefficients along the fuselage downstream of the fan, for Position B (with comparison against the non-propelled fuselage baseline).

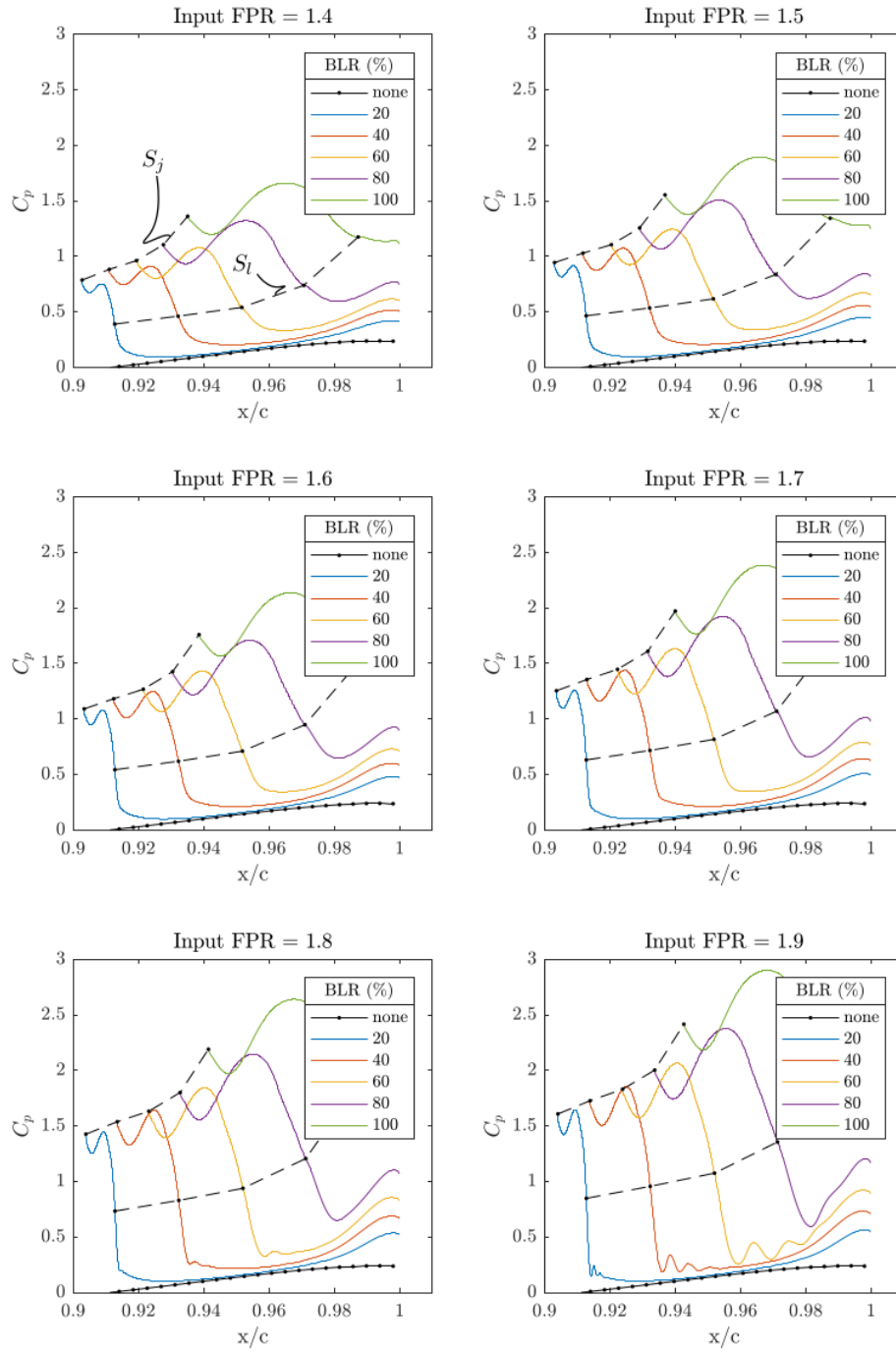


Figure 6.20: Pressure coefficients along the fuselage downstream of the fan, for Position B (with comparison against the non-propelled fuselage baseline).

nacelle intake duct surface S_h , a large C_p is present at the stagnation point, drops rapidly as the flow accelerates to the intake throat and then moderately increases as the flow diffuses to the fan face. The FPR does not influence this internal flow upstream of the fan, and hence the curves for a given BLR lie directly on top of one another. The portion of the curve from the stagnation point (maximum C_p) to the throat (minimum C_p) corresponds to the lip surface and, having a forward facing area, results in a rearward force for $C_p > 0$. The inclination of the surface S_h is different for propulsor Positions A and B, where the former has a slight rearward facing area and the latter a large forward facing area. This means that a very slight thrust force is obtained for Position A propulsors where $C_p > 0$, and visa versa. The opposite is true for the Position B propulsors, and a $C_p > 0$ represents a rearward acting force.

Now considering the flow from the lip stagnation point over the external nacelle surface, S_p . It can be observed that the pressure coefficient drops rapidly as it accelerates over the fore-body, creating a suction surface. The flow eventually reaches sonic velocity resulting in a shock over the after-body and a sharp rise in pressure coefficient. However, the rise pressure coefficient is not substantial enough, and the after-body is also predominantly a suction surface. For Position A propulsors, the cowl has a forward facing area and the portion undergoing suction contributes a forward force. For Position B propulsors, on the other hand, the suction over the cowl contributes to a rearward force given the inclination of the nacelle. In conforming to the fuselage tail-cone curvature, the after-body trailing edge is below that of the intake lip stagnation point. Therefore the after-body has a larger backward facing area than the fore-body's forward facing area, which results in large net rearward force on this surface. To allow for different FPRs at a given BLR, the geometry of S_p is adapted to the corresponding nozzle exit area. In so doing, higher FPRs suffer larger shock wave effects and a lower final C_p closer to the trailing edge due to losses associated with dissipation. Finally, considering the nacelle exhaust duct surface S_k , it is seen that C_p is very large due to the increase in pressure aft of the fan. This internal surface has a forward facing area and is thus subject to a substantial rearward force. It can be observed that there is a highly dominant net rearward force effect on the nacelle, which more than compensates for the forward acting forces achieved over the fuselage surfaces due to locally induced static pressures.

Combined Airframe Pressure Forces

The reason for looking so closely at the pressure coefficients, was to establish whether there are potential synergistic benefits in raising the static pressure over the tail-cone. This was one of the motivations for investigating propulsors at different tail-cone positions. However, Figures 6.22a and 6.22c illustrate clearly how the rearward pressure forces acting on the nacelle far outweigh the forward forces on the fuselage. This can be seen by the net resultant force of the combined fuselage and nacelle, which is shown more clearly in Figures 6.22b and 6.22d. It must be noted that the severe shock over the nacelle certainly increases suction and thus the rearward forces. Therefore an improved design may yield better performance, but it is unclear as to whether it is possible to achieve a net forward pressure force (or even a reduction in the rearward force, in com-

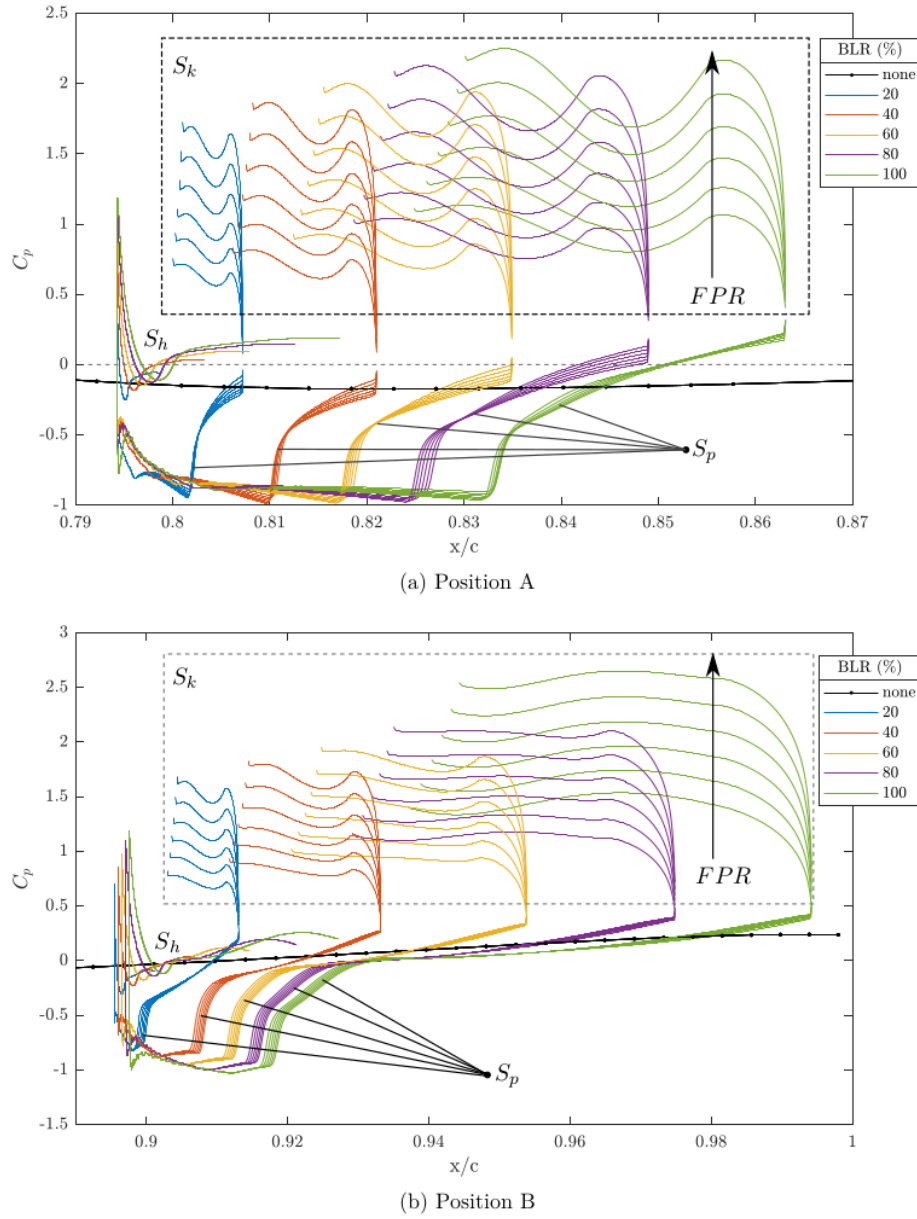


Figure 6.21: Pressure coefficients over nacelle surfaces for different FPRs, showing; (1) external nacelle surfaces S_p , (2) nacelle intake duct surfaces S_h and (3) nacelle exhaust duct surfaces S_k . The non-propelled fuselage case is shown for reference.



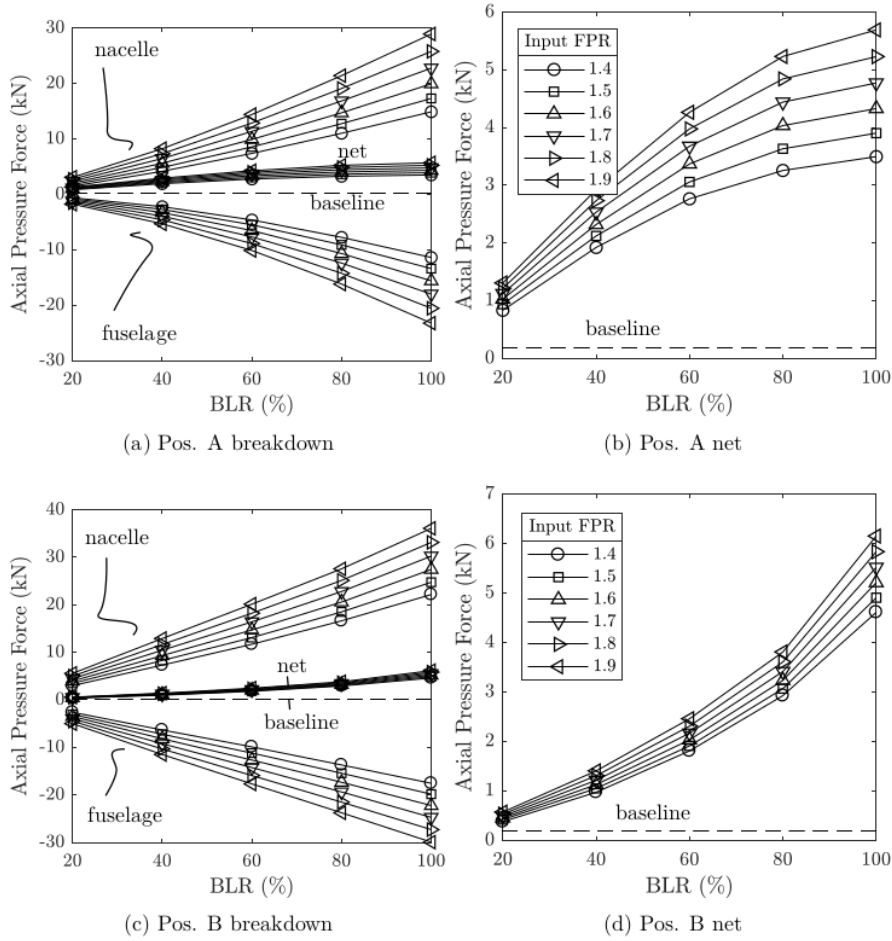
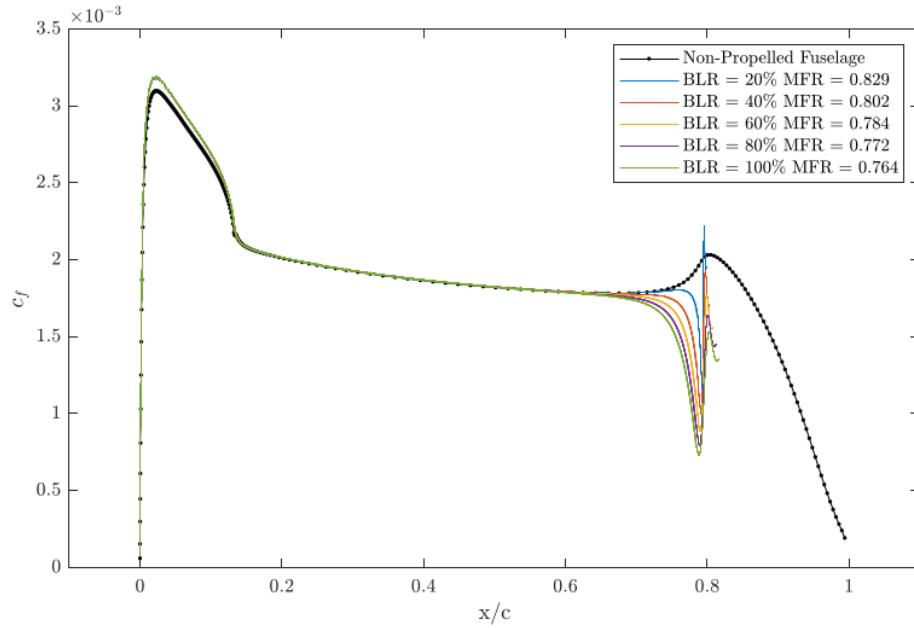
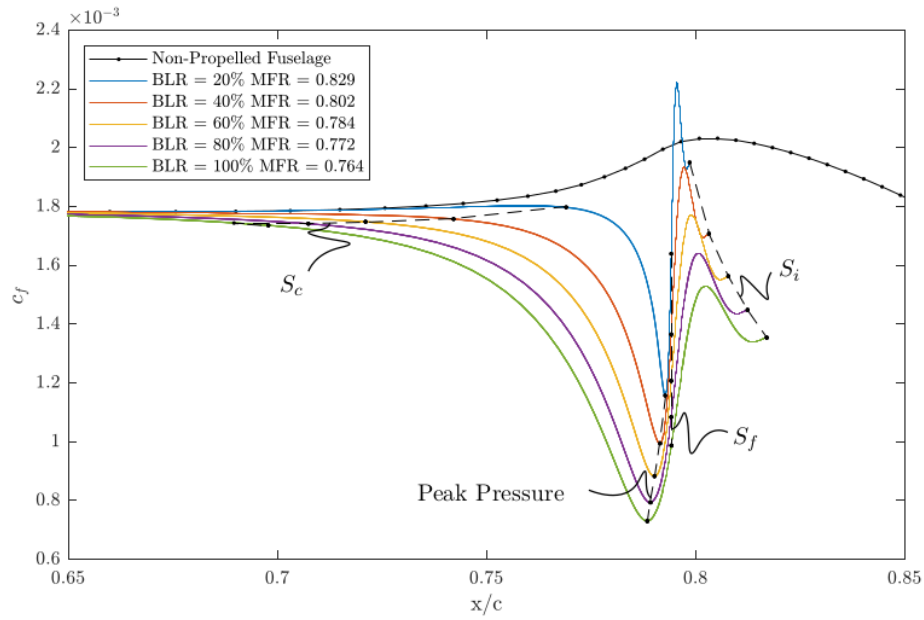


Figure 6.22: Axial pressure force contributions for the nacelle (S_p , $S_{h,nac}$, $S_{k,nac}$), fuselage ($S_{d,g,m}$, $S_{h,fus}$, $S_{k,fus}$) and combined airframe surfaces (with comparison against the baseline).

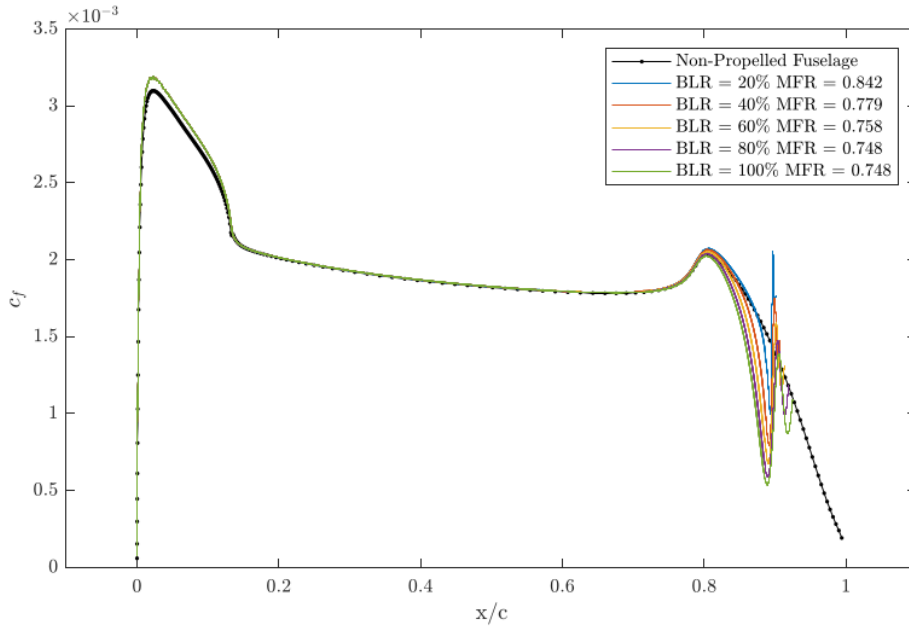


(a) Fuselage nose to fan face

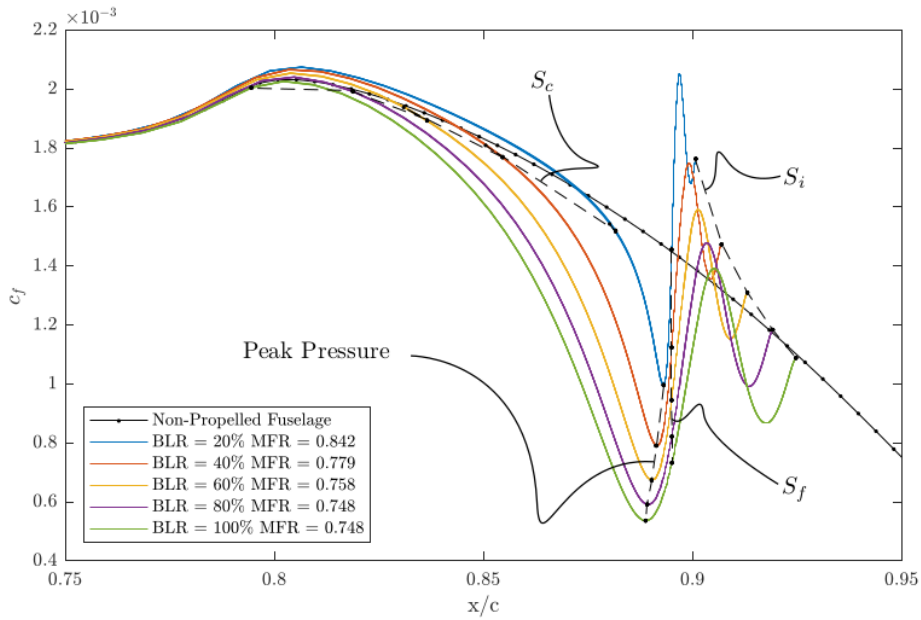


(b) Pre-diffusion region to the fan face

Figure 6.23: Position A friction coefficients upstream of fan inlet along fuselage surface. Comparison against non-propelled baseline fuselage.



(a) Fuselage nose to fan face



(b) Pre-diffusion region to the fan face

Figure 6.24: Position B friction coefficients upstream of fan inlet along fuselage surface. Comparison against non-propelled baseline fuselage.

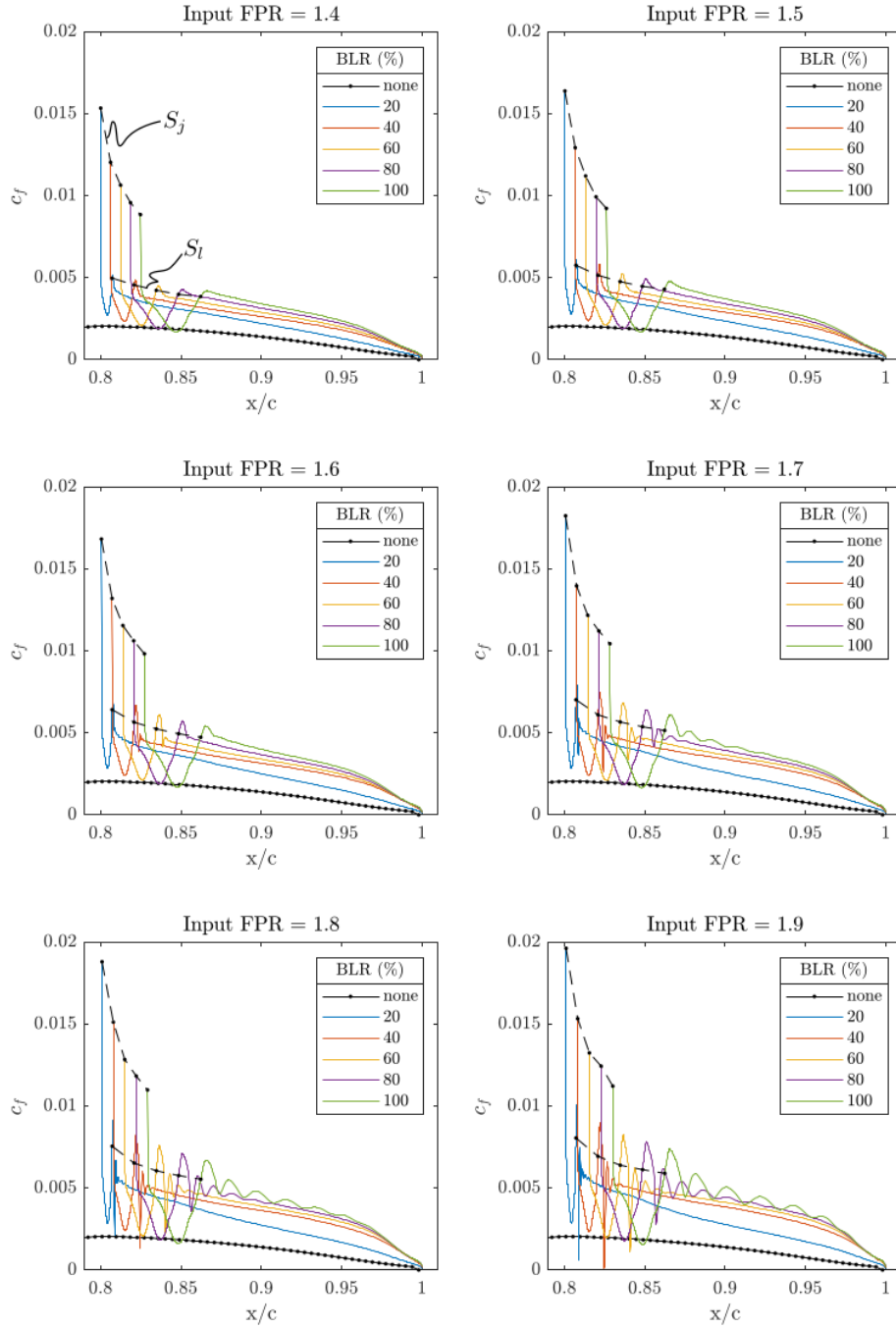
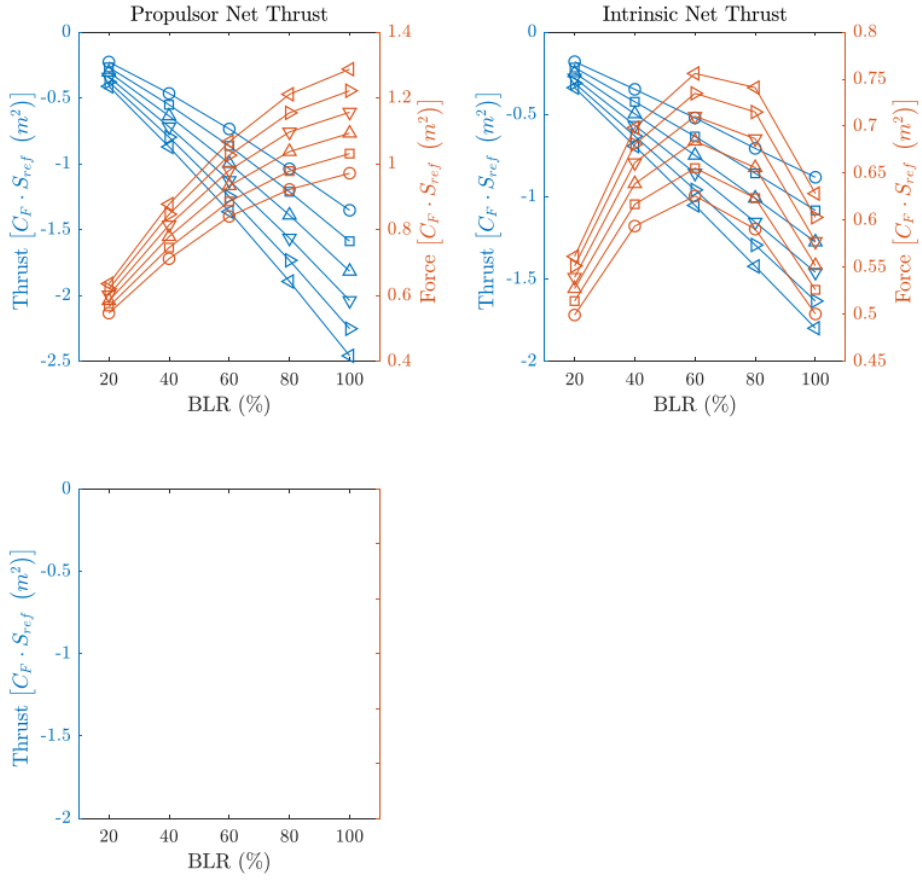


Figure 6.25: Position A friction coefficients downstream of fan exit along fuselage surface







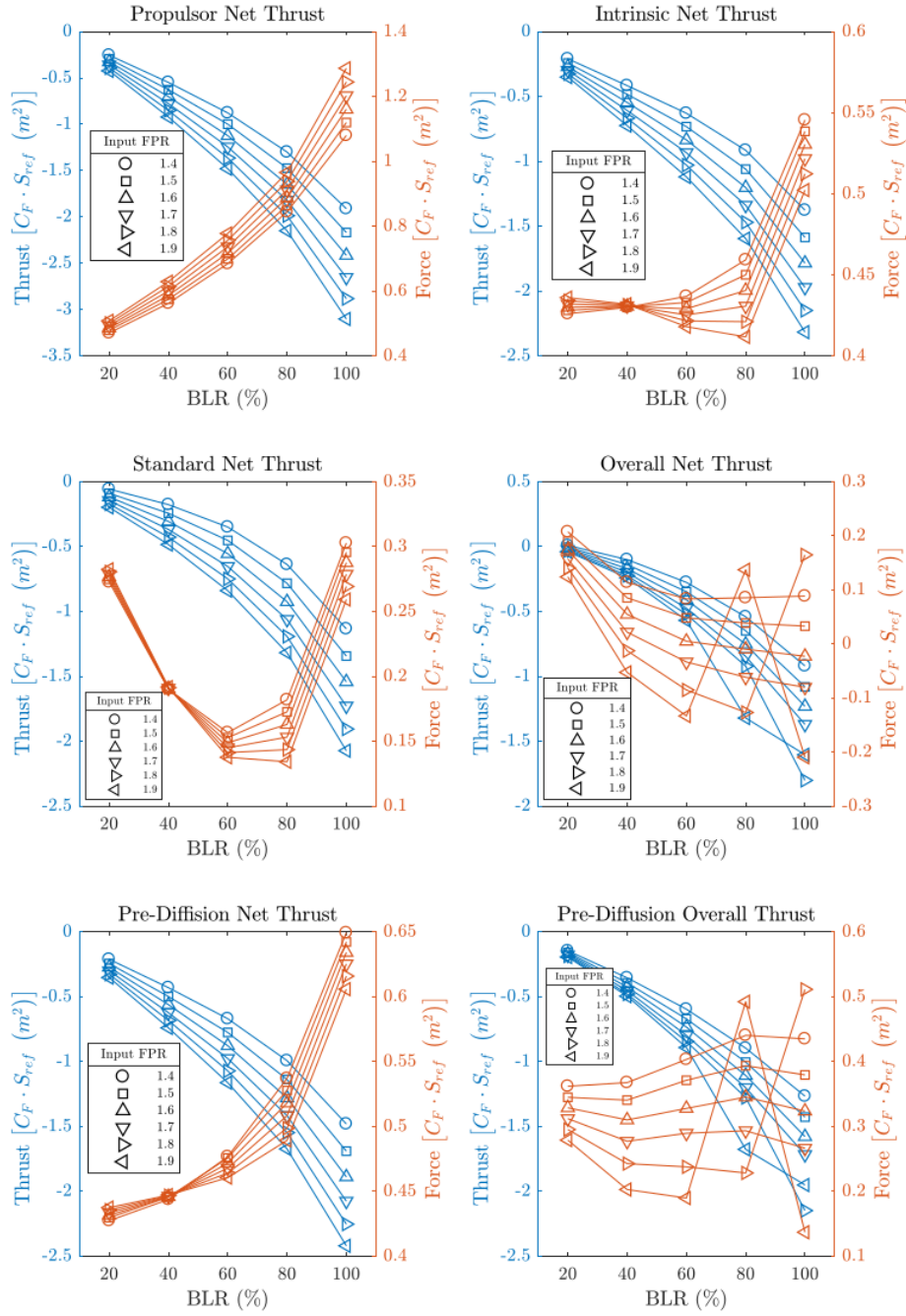


Figure 6.28: Design exploration thrust-force accounting breakdown at Position B

minimum force contribution which shifts to higher BLRs at larger FPRs. Thus it may be deduced that propulsors at Position A are more likely to achieve a benefit at either very small or very large BLRs, whereas Position B has an optimum BLR for a given FPR.

Pre-Diffusion Net Thrust

The next logical definition to consider is the Pre-Diffusion Net Thrust, which passes the fuselage forces within the pre-diffusion streamtube to the thrust contribution. This results in a seemingly counter-intuitive increase in rearward force, which grows with larger BLRs. The reason for this is the necessary inclusion of the forces on the pre-diffusion streamtube surface S_e . This streamtube surface has a forward facing area, which increases as MFR is reduced at higher BLRs. As MFR is reduced further, the frontal area of the streamtube is enlarged and a greater rearward pressure force incurred. This is true for both propulsor positions, A and B, where the parabolic shapes are stretched with increase in BLR. At very small BLRs, the MFR is close to one, and thus the effects are negligible. This demonstrates the use of this thrust-force definition in understanding the implications of mass-flow-ratio on rearward forces.

Standard Net Thrust

The Standard Net Thrust definition follows on from the Pre-Diffusion Net Thrust definition, by passing the fuselage surface forces upstream of the pre-diffusion streamtube to the definition's thrust component. However, the definition's force component must now include the forces acting on the upstream streamtube surface S_b . This addition is negligible at Position A because it has a mostly radial facing area, and the pressure forces cancel out. Nonetheless, there is a change in trends from the Pre-Diffusion Net Thrust definition corresponding to variations in BLR. This is because the pre-diffusion begins further upstream at higher BLRs, and therefore less fuselage friction force is passed to the thrust component. This equalises out the symmetry of the force parabolic trend. A similar effect is observed at Position B, and also accounts for the less dominant forward oriented pressure forces on the streamtube surface S_b , which now has a rearward facing area.

Pre-Diffusion Overall Thrust

This definition alters the Pre-Diffusion Net Thrust definition by passing forces on the tail-cone surface S_m to the definition's thrust component. Having done so, the force contribution must now account for the forces acting on the jet streamtube surface S_o . The amount of wetted area that is passed to the thrust component reduces with an increase in BLR, due to the size of the propulsor, and one might expect to see larger reductions in force at smaller BLRs. However, the opposite occurs due to the forces on the post-exit streamtube S_o , which has a large rearward facing area in regions of high static pressure. An example of the rearward facing area of this surface may be

seen in Figure 5.4, which shows the jet streamtube for a propulsor at Position B. The rearward facing area of S_o increases with BLR, and the static pressure it is subject to increases with FPR. This explains the decrease in the definition's force contribution with increases in BLR and FPR. Hence this definition is useful in explaining the effect of post-exit streamtube forces, and shows how the jet plume has a rearward facing area, unlike podded axisymmetric engines, which pushes the aircraft forward under increased local static pressure.

Overall Net Thrust

Finally the Overall Net Thrust definition is considered. It is similar to the Standard Net Thrust, except that the surface forces on S_m are passed to the definitions thrust component. The reason behind the change in trends from the Standard Net Thrust definition follows the previous discussion regarding the Pre-Diffusion Overall Thrust. A comparison against the Pre-Diffusion Overall Thrust definition indicates an overall reduction in the definition's force component, which is expected due to the transfer of upstream fuselage forces to the thrust component. This highlights the forward contribution of forces on surface S_o due to its rearward facing area.

Summary

This subsection demonstrates the use of the various thrust-force accounting systems, not in prescribing a particular one to be applied in isolation, but in making use of all of them to filter out different near-field force implications through comparisons between each definition. This demonstrates how the thrust-force accounting method developed in this work contributes to knowledge in showing that BLI aerodynamics may be better understood by looking at all appropriate thrust-force accounting definitions, and not simply as a means of selecting one definition to split up which surfaces belong to the engine manufacturer and which belong to the airframer. Future work may wish to look more comprehensively at comparisons between the individual force contributions of each surface (including unbounded streamtube surfaces). This is considered to be useful at a more detailed design level, and is not included here for brevity.

6.4.3 System Level Performance

Having examined the near-field forces and thrust-force accounting systems, the overall performance of the system can be analysed. Firstly, the performance is examined at a BLI propulsor level, where trends of thrust-split, TSPC and propulsive efficiency are presented. Secondly, the entire propulsion system is considered by incorporating the main turbofan engine's off-design cycle performance, which is dependent on the propulsive efficiency and thrust-split of the BLI propulsors.

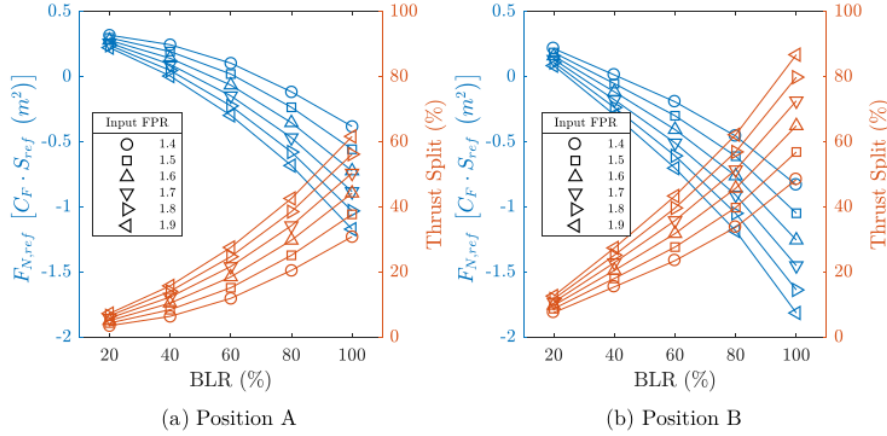


Figure 6.29: Reference Net Vehicle Thrust and Thrust Split for both Positions A and B.

DeCoupled BLI Propulsor Performance

The overall forces on the fuselage propulsor assembly may be summarised with reference to the baseline fuselage. The Reference Net Vehicle Thrust $F_{N,ref}$ is most convenient for this purpose and allows for the aircraft's thrust-split to be determined. Correlations between thrust-split and $F_{N,ref}$ are depicted in Figures 6.29a and 6.29b for positions A and B, respectively. It is clear that the thrust produced at Position B is far greater than that at Position A, and this corresponds to a larger TS. Naturally, the TS increases with BLR and FPR because of the increased mass-flow-rate and jet total pressure. Propulsors at Position B are able to reach a TS around 90%, whereas only 60% is achievable at Position A. This is partly because of the larger BL at Position B which contains more mass flow. Figure 6.30 indicates the shaft power associated to each propulsor position, as well as each BLR and FPR combination. The shaft power is depicted alongside TS, to indicate the correlation between the two, where a larger TS corresponds to a larger power. It now becomes clear that Position B is able to generate more thrust from the same amount of power. This is better summarised in Figure 6.31, which depicts the Thrust Specific Power Consumption (TSPC). Subsequently, TSPC may be translated to a propulsive efficiency based on the unconventional definition introduced in Section 3.7.2, and this is depicted in Figure 6.32. Considering first Position A, it is observed that a relatively low propulsive efficiency range of 35-60% is achieved. The propulsive efficiency curves have a generally parabolic shape, where a local minimum occurs at a BLR of 40%. Increasing the BLR increases the mass-flow-rate through the propulsor, which results in an improved propulsive efficiency. This increase steepens between 60-100% BLR, indicating that the presence of the nacelle in the BL is detrimental and the only propulsive efficiency improvements come from ingesting more mass flow.

More relevant, and expected, trends are observed for Position B, where higher overall

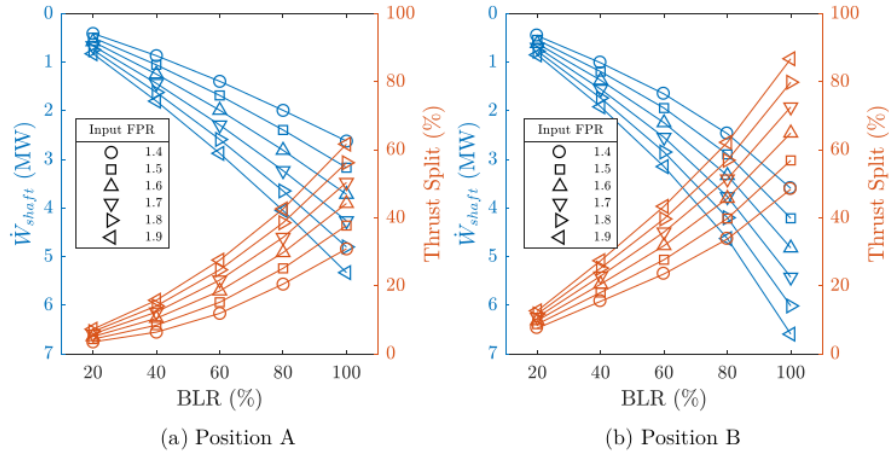


Figure 6.30: Fan shaft power and Thrust Split for both Positions A and B

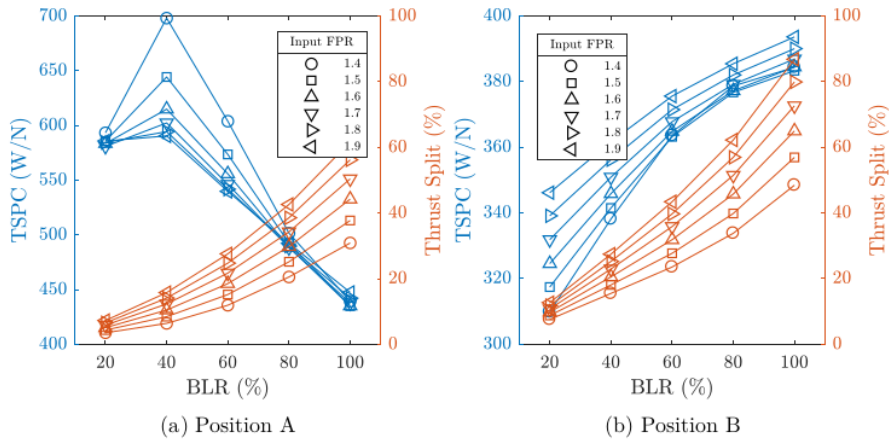


Figure 6.31: TSPC and Thrust Split for both Positions A and B

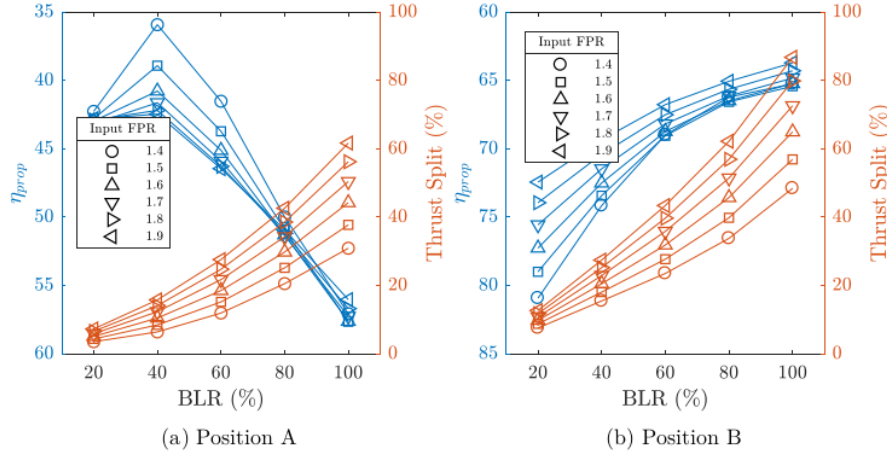


Figure 6.32: Propulsive efficiency and Thrust Split for both Positions A and B

propulsive efficiencies (in the range of 65-80%) are achieved. The propulsive efficiency is highest at the lowest BLR and FPR combination. In general, the propulsive efficiency decreases with an increase in BLR and FPR. This was expected, as the benefits of BL ingestion take advantage of the reduced BL momentum, which correlates to a decrease in BLR due to the BL profile. In addition, lowering the FPR improves the propulsive efficiency by reducing specific thrust, which is the same effect used to improve conventional propulsive efficiency in podded installations. Subsequently, there is a combined benefit of improving propulsive efficiency through lowering FPRs and ingesting smaller (but lower momentum) portions of the BL. This explains why the propulsive efficiency is so spread out for different FPRs at BLR = 20%. This combined effect diminishes as BLR is increased, and the average momentum of the BL starts to approach that of the free-stream condition. This explains why the propulsive efficiency bunches up at BLR = 100%. Extrapolating to higher BLR values, one can imagine that the trend will start to flatten out for a given FPR. This is a result of the BL profile having a diminishing effect, and the propulsor operating closely to the same conditions as a podded installation.

SFC Benefits

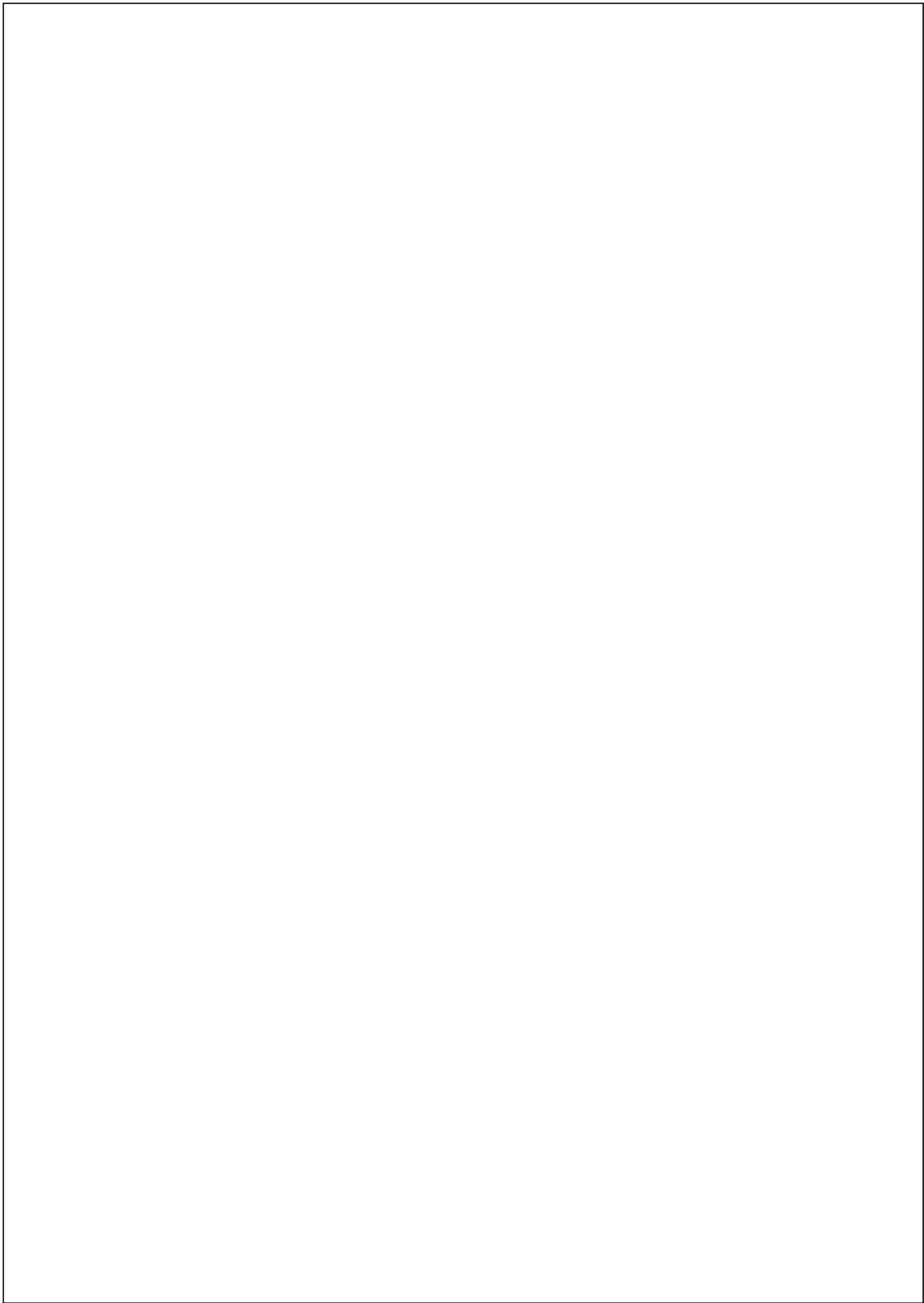
In this work a retrofit approach is adopted, and it was decided to examine the potential benefits achievable in a thrust-split configuration, where the baseline engine powers BLI propulsors and provides a portion of the thrust. Based on this approach the performance benefits, at an off-design cycle level, can be explored parametrically by varying the BLI propulsive efficiency (as per the unconventional definition of Equation (3.38)) and thrust-split. Figure 6.33 is a decoupled performance map corresponding to these parametric combinations (generated using the engine model documented in Appendix D), and il-

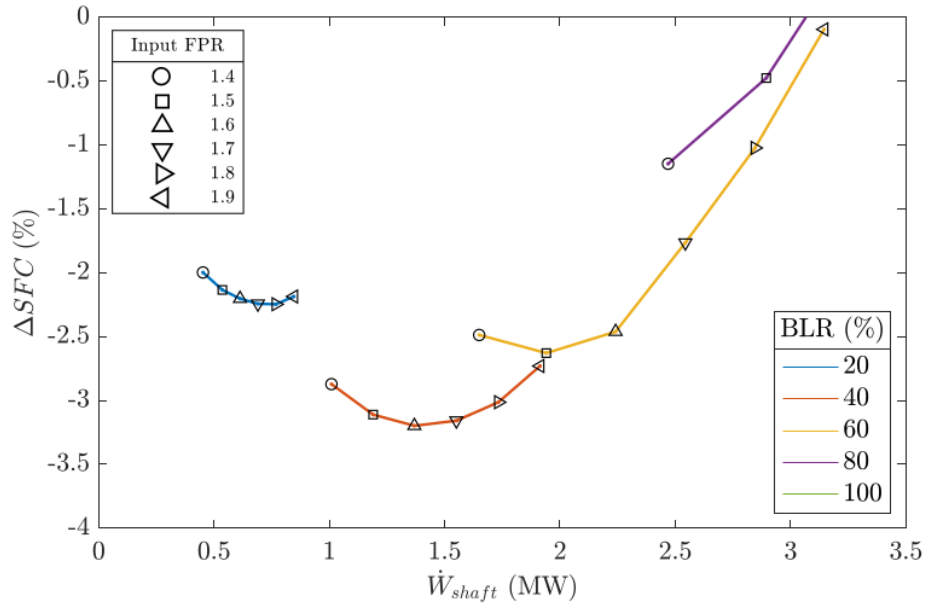
illustrates SFC improvements as a function of shaft power, thrust-split and propulsive efficiency. The format is shown as a carpet plot of propulsive efficiency and thrust split.

The thrust-split range is capped at 70%, because the off-design cycle does not converge and becomes unphysical when trying to produce such small amounts of thrust for increasing power off-take demands. It can also be seen that no benefits are achievable for propulsive efficiencies below 60%. This map essentially shows the reductions in SFC achievable when passing some of the thrust to a propulsor which has a better propulsive efficiency³ than its own. However, even for relatively good propulsive efficiencies, there is a detriment to the cycle performance in taking additional power from the LP spool. As a result, there is an optimum trade-off where the maximum amount of thrust passed to a propulsor of greater efficiency, is limited by the detrimental effects on the cycle. These optimums are marked on Figure 6.33, and it is observed that the greater the BLI propulsive efficiency, the larger the thrust-split at the optimum. For this particular engine, there is a theoretical maximum SFC benefit of around 17% corresponding to a propulsive efficiency of 100% at a thrust-split around 60%. Unfortunately, a 100% propulsive efficiency is not possible, and the values achieved in this design exploration have maximums around 80-85%. The map in Figure 6.33 is now used to quantify the benefits achieved by the BLI propulsors studied in this design exploration. From Figure 6.32 it is clearly visible that the propulsive efficiency of propulsors at Position A are too low to realise any benefit. Therefore, only the SFC benefits of the propulsors at Position B are quantified here.

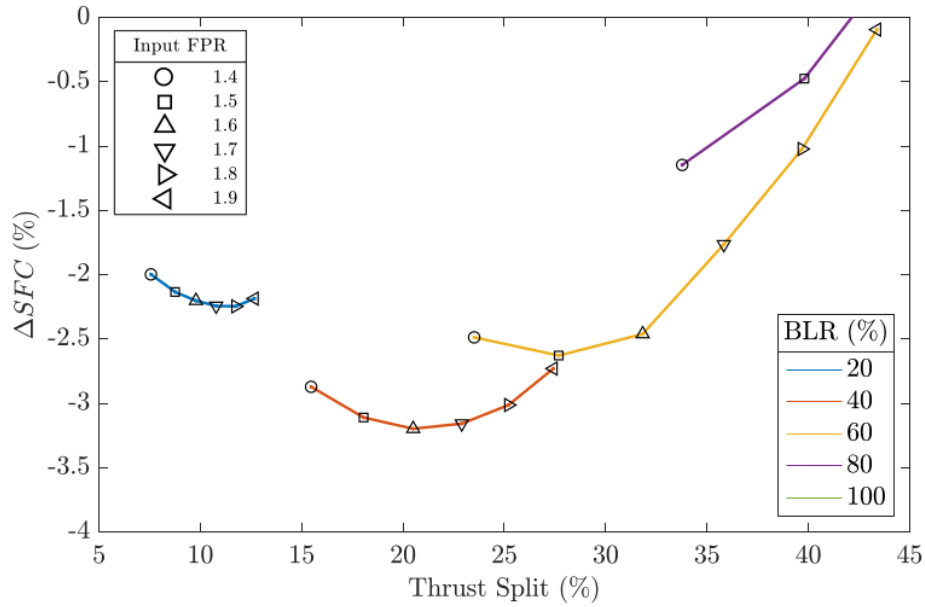
Figures 6.34a and 6.34b illustrate the SFC benefit versus the shaft power and thrust split respectively. The results show the achievable benefits for the different BLRs and FPRs considered. For each BLR, there is a FPR that corresponds to an optimum benefit, and this FPR decreases with an increase in shaft power or thrust split. This is because the propulsive efficiencies are greater at lower BLRs and smaller thrust-splits, as is evident from Figure 6.32b. These low thrust-split regions fall to the far left of the optimum line in Figure 6.33. Therefore, it is beneficial to increase the thrust for a given propulsive efficiency (ideally to the optimum point) to take advantage of the improvements in cycle performance. However, a larger thrust is only achieved by increasing FPR which has a detrimental effect on the propulsive efficiency, as seen in Figure 6.32b. Thus there is a trade-off at some position where further increasing the FPR reduces the propulsive efficiency to a point where SFC begins to increase again. An alternative is to increase the BLR to ingest more flow and produce more thrust. However, increasing the BLR also results in a reduction in propulsive efficiency. This introduces another trade-off between approaching the cycle optimum, but moving to regions of reduced propulsive efficiency. In summary, the optimum propulsive efficiency location is different to the cycle optimum. BLR and FPR allow for the trade-off between the two locations to be optimised. From Figure 6.34, this can be seen to coincide with a BLR = 40% operating at a FPR = 1.6, resulting in a maximum SFC reduction of just over 3% for a thrust-split around 20%.

³This statement is based on the definition in this work and not the conventional definition of propulsive efficiency





(a) SFC benefit versus power off-take



(b) SFC benefit versus thrust split

Figure 6.34: Near-field momentum-based SFC benefits for propulsors at Position B.

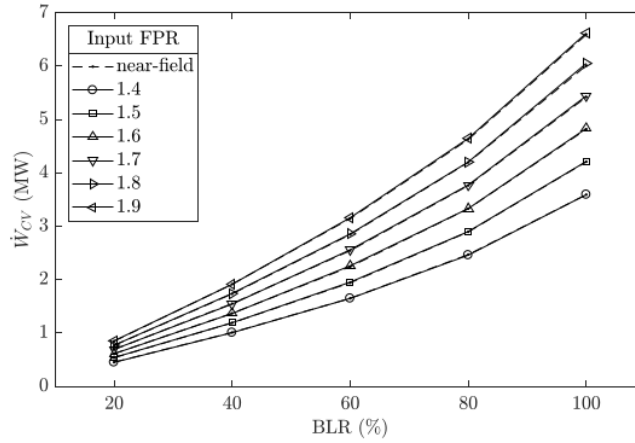


Figure 6.35: Far-field versus near-field propulsor power, for propulsors located at Position B along the tail-cone.

reference to Figure 5.20. This section begins with a brief comparison between near and far-field calculations of propulsor power. Thereafter, the decoupled fuselage concentric BLI propulsor performance is assessed, before incorporating the main engine's cycle performance to predict benefits in SFC. The results are presented alongside the near-field values for reference.

6.5.1 Propulsor Power

This subsection briefly looks at the consistency in propulsor power obtained from the near and far-field integrations of Equation (5.17), before examining the forces and system level performance. The propulsor power for the different BLR and FPR combinations is summarised in Figure 6.35. It is observed that there is a very close correlation, which is typically within 1%. This gives further confidence that the energy through the control volume is properly conserved.

6.5.2 Decoupled BLI Propulsor Performance

Figure 6.36 summaries the performance of the BLI propulsion at Position B. The resultant NVF and thrust-split are depicted in Figure 6.36a, where dashed and solid lines represent the near and far-field values, respectively. This figure indicates a slightly larger NVF , and corresponding thrust-split, in comparison to the near-field. This is expected following the discussions in Section 5.4, where it was found that near-field methods tend to predict larger surface forces over walls than is obtained from far-field methods. The offset in NVF is carried through to the Reference Net Vehicle Thrust $F_{N,ref}$, as observed in Figure 6.36b. Although these differences in force are relatively small, they



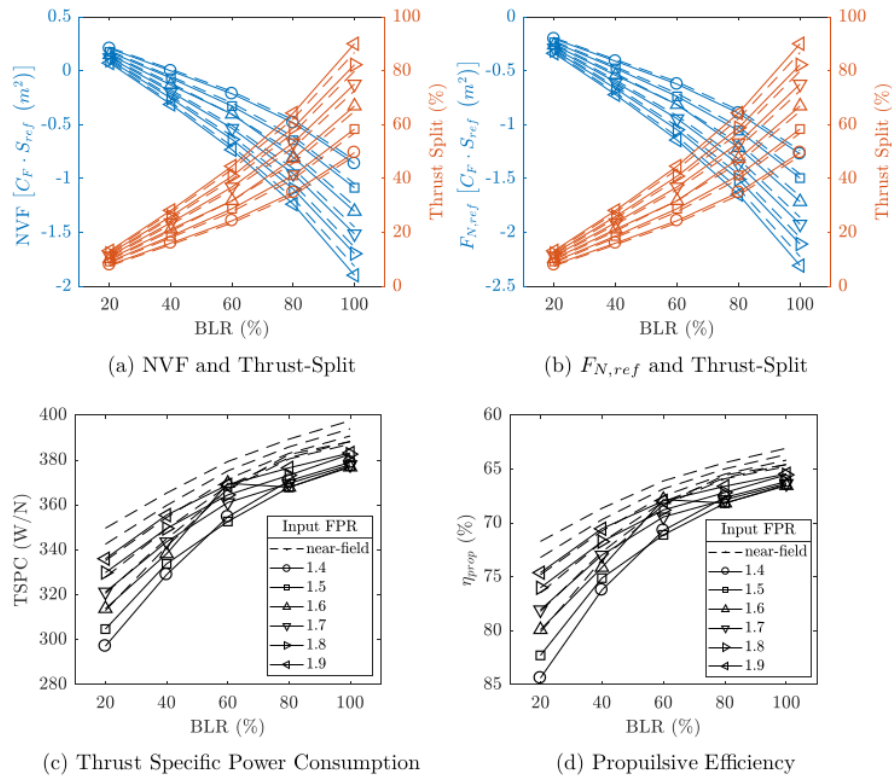
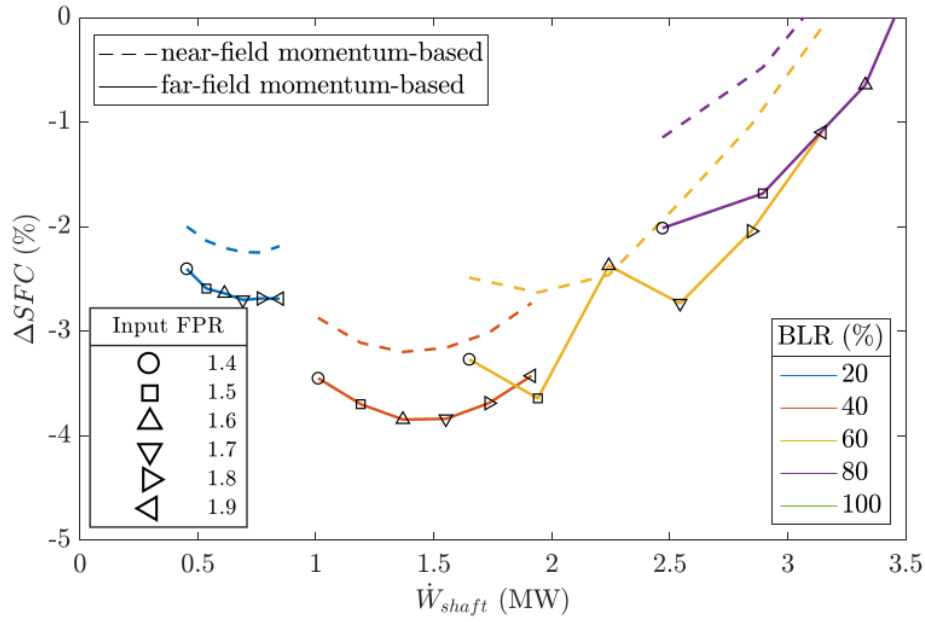
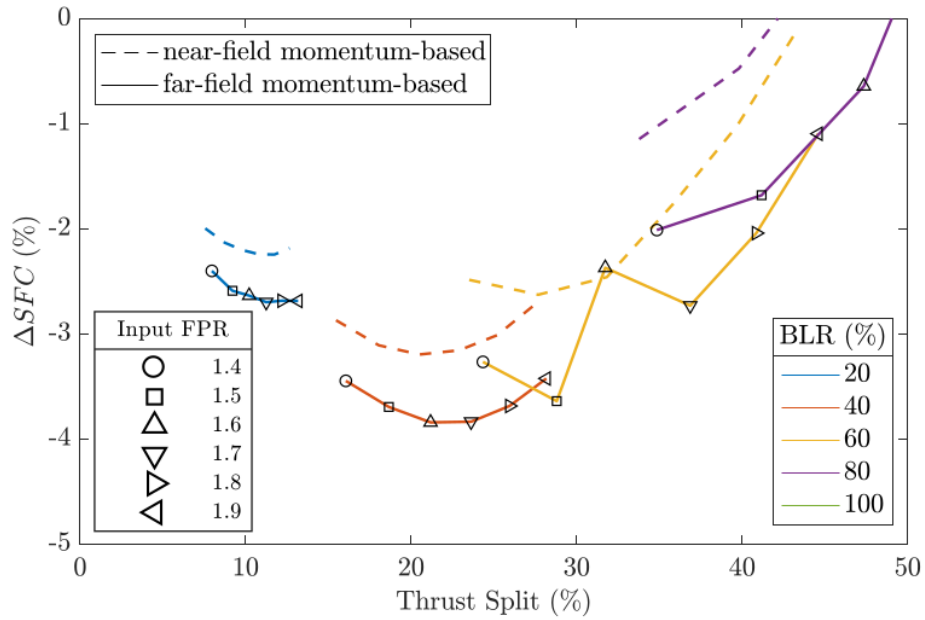


Figure 6.36: Decoupled BLI propelled fuselage performance, using far-field momentum-based methods (compared against dashed lines of the near-field values), for propulsors at Position B.



(a) SFC benefit versus power off-take



(b) SFC benefit versus thrust split

Figure 6.37: Far-field momentum-based SFC benefits for propulsors at Position B, with reference to near-field momentum-based values.

control volume has been used to perform the evaluations, and is defined as having the following geometric extents; $\Delta s/l_{fuse} = 2$ and $\Delta z/l_{fuse} = 10$. The selection of the control volume size is based on the outcomes of Section 5.4. This section begins by presenting the decoupled propelled fuselage performance, where energy-based values are shown with reference to momentum-based results (both near and far-field). Subsequently, turbofan off-design performance is incorporated to obtain estimated benefits in SFC, and further comparisons are made with momentum-based outcomes. Further to this, the energy method is used to breakdown and quantify NVF contributions, in order to decipher and better understand sources of benefits and losses. Firstly, the breakdown is performed for varying trefftz plane locations downstream of the fuselage, in order to track the progression of energy terms in the wake. Secondly, the entire CFD domain is decomposed into different flow field regions, so as to quantify contributions to NVF by different physical mechanisms. These decompositions look at the jet region (along with core expansion/-compression and viscous transitions regions) in detail, along with the shock, BL and shear regions external to the jet.

6.6.1 Decoupled BLI Propulsor Performance

Figure 6.38 summarises the energy-based propelled fuselage performance, and shows results of NVF , thrust-split, TSPC and BLR alongside near-field momentum-based results. Figure 6.39 shows the same results, except this time alongside the outcomes from the far-field momentum-based assessment. The energy-based NVF and thrust-split are greater than the near-field values. There is a discrepancy of around 5% in propulsive efficiency across all BLRs and FPRs. This suggests that discrepancies between near and far-field momentum-based results are less than those between near-field and energy-based results. This is expected, as the far-field momentum based method approaches the exact near-field formulation for a control volume of infinitesimal size. However, it can be seen from Figure 6.39 that the far-field momentum and energy-based methods produce results that are very close to one another. There are very small differences between NVF , thrust-split and Reference Net Thrust, as can be observed from Figures 6.39a and 6.39b. The corresponding difference in TSPC and propulsive efficiency is also observed to be very small, where propulsive efficiency values differ by 1-2%. The reason as to why the far-field momentum-based results are closer to the energy-based results, rather than the near-field results, is attributed to the filtering out of spurious effects (which has been discussed in Chapter 5).

6.6.2 SFC Benefits

Benefits in SFC, corresponding to the energy-based outcomes of thrust-split and propulsive efficiency, are depicted in Figure 6.40. The results are shown alongside those of the near and far-field momentum-based values, from which very similar trends may be observed. In general, the far-field momentum-based and energy-based results are observed to be more consistent with one another than with the near-field values. At BLR = 20-40%, the differences in SFC between the energy and far-field momentum-based

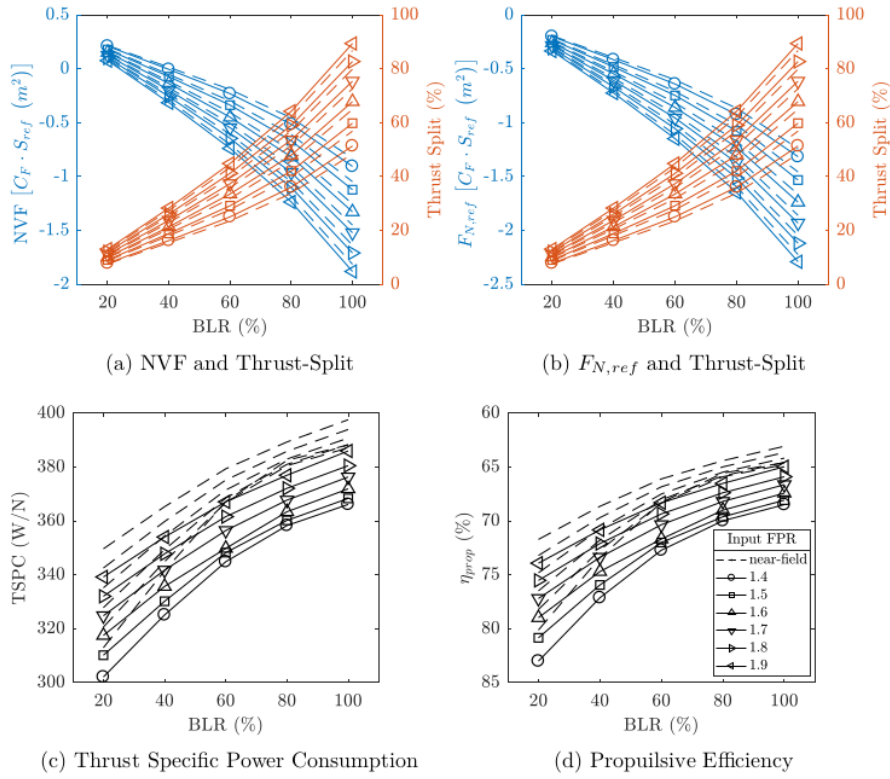


Figure 6.38: Decoupled BLI propelled fuselage performance, using far-field energy-based methods (compared against dashed lines of the near-field momentum-based values), for propulsors at Position B.

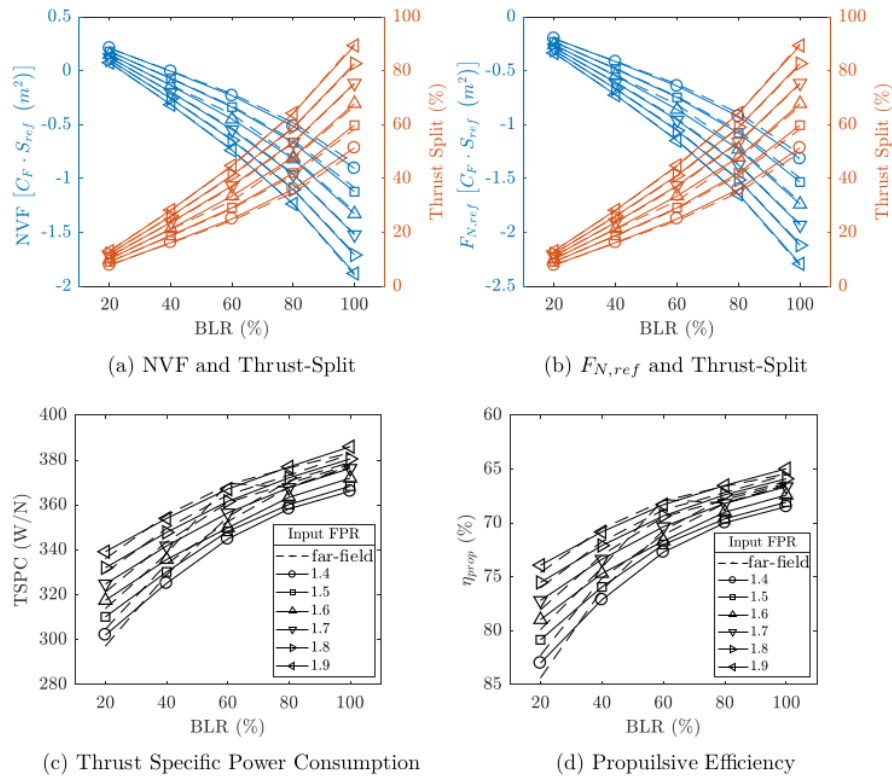
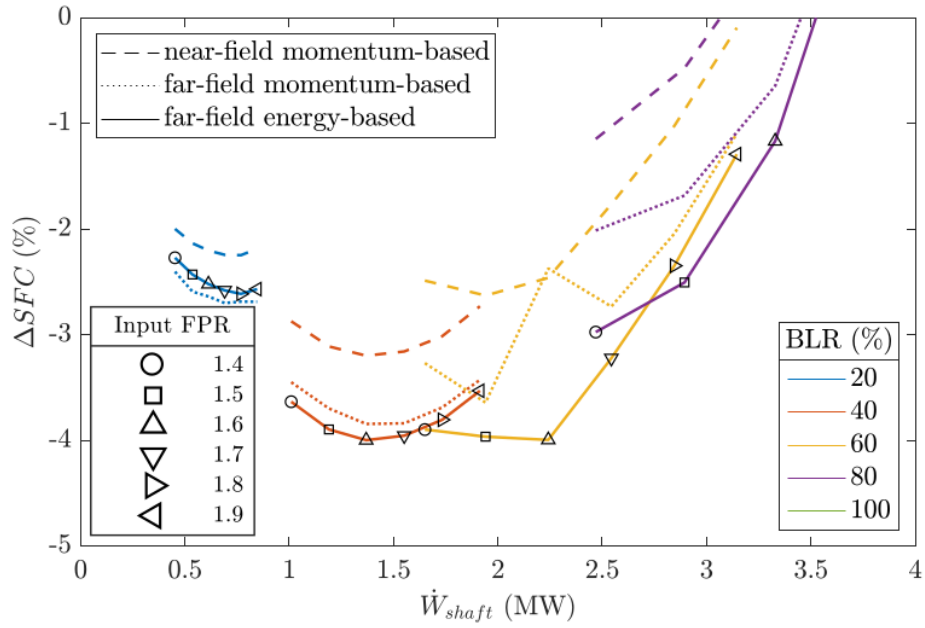
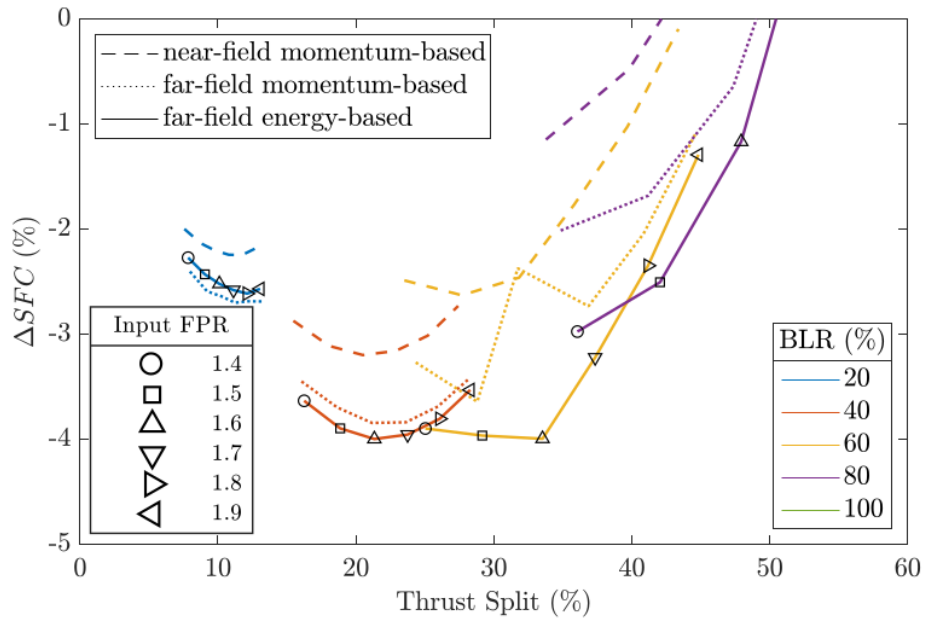


Figure 6.39: Decoupled BLI propelled fuselage performance, using far-field energy-based methods (compared against dashed lines of the far-field momentum-based values), for propulsors at Position B.





(a) SFC benefit versus power off-take



(b) SFC benefit versus thrust split

Figure 6.40: Far-field energy-based SFC benefits for propulsors at Position B, with reference to near and far-field momentum-based values.

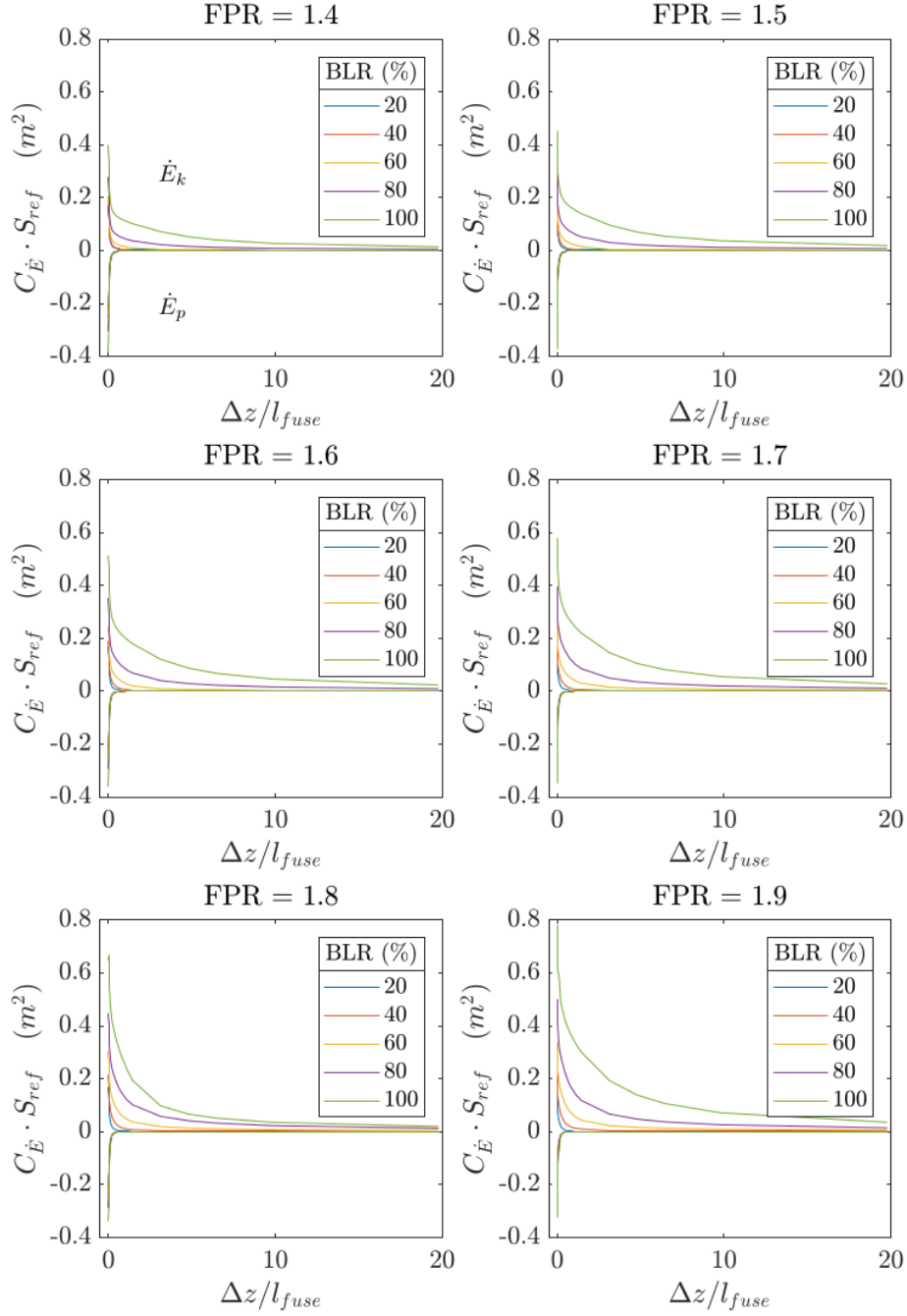


Figure 6.41: Breakdown of kinetic energy flux $\dot{E}_k$ and pressure boundary work $\dot{E}_p$ at different treftz locations, for propulsors at Position B.

Asymptotically Increasing Energy Terms

Figure 6.42 depict the contributions of viscous dissipation, $\dot{E}_{\tau\nu}$, and volumetric compression work, $\dot{E}_{p\nu}$, which asymptotically increase in magnitude with the wake progression. The former is shown as trends above zero, where the latter's trends are depicted below zero. The volumetric dissipation intuitively increases with BLR and FPR, as is expected from observing previous trends of kinetic energy flux in Figure 6.41. The rate at which this occurs is dependent on the viscous shear layers and tends to be slower at greater BLR and FPRs where the initial kinetic energy is larger. It is useful at this point to highlight that the initial amount of kinetic energy flux in Figure 6.41 does not equate to the final change in volumetric viscous dissipation. This emphasises the point made in Section 5.3, that the kinetic energy surplus at the aircraft trailing edge is not necessarily a recoverable energy upon which a BLI propulsor may act.

Also intuitively, it is observed that the volumetric compression work has magnitudes which increase with BLR and FPR. The rate at which the asymptote is reached is much quicker than that of the viscous dissipation, as it is predominantly dependent on expansion processes of the flow. This was also observed in the trends governing pressure boundary work. An important observation of these trends is that $\dot{E}_{p\nu}$ is most definitely far from zero, and is a significant contribution to the thrust direction (sometimes equal to more than half of the dissipation). This is counter to the hypothesis of Drela [5], and once again demonstrates the advantage of formulating the energy equation with *NVF* as its subject.

Summary

It is worth briefly emphasising here the benefit of analysing the wake progression energy breakdowns. The first benefit, is that they make clearer the pathways between energy transfers. For example, it is observed that not all of the kinetic energy surplus is dissipated to heat. This is a common misconception in the understanding of BLI benefit, where it is assumed that the drag correlating the kinetic energy surplus at the aircraft trailing edge is the theoretical maximum recoverable through re-energisation by a BLI propulsor. A further benefit, is in the analysis of the volumetric compression work, which has previously been assumed to be close to zero and negligible [5]. In fact this term is observed to have a significant thrust contribution, and ignoring its contribution will give incorrect *NVF* estimations. Without having examined this term at different downstream trefftz locations, it would have been difficult to determine if its contribution would diminish to zero remain finite. This demonstrates the advantage that energy breakdowns have in supplementing the understanding of *NVF* contributions.

6.6.4 Flow Field Decompositions

The previous subsection discussed the advantages of performing an energy breakdown analysis with varying trefftz plane locations, as a means for supplementing the understanding of physical contributions to the *NVF*. Another advantage, is being able to

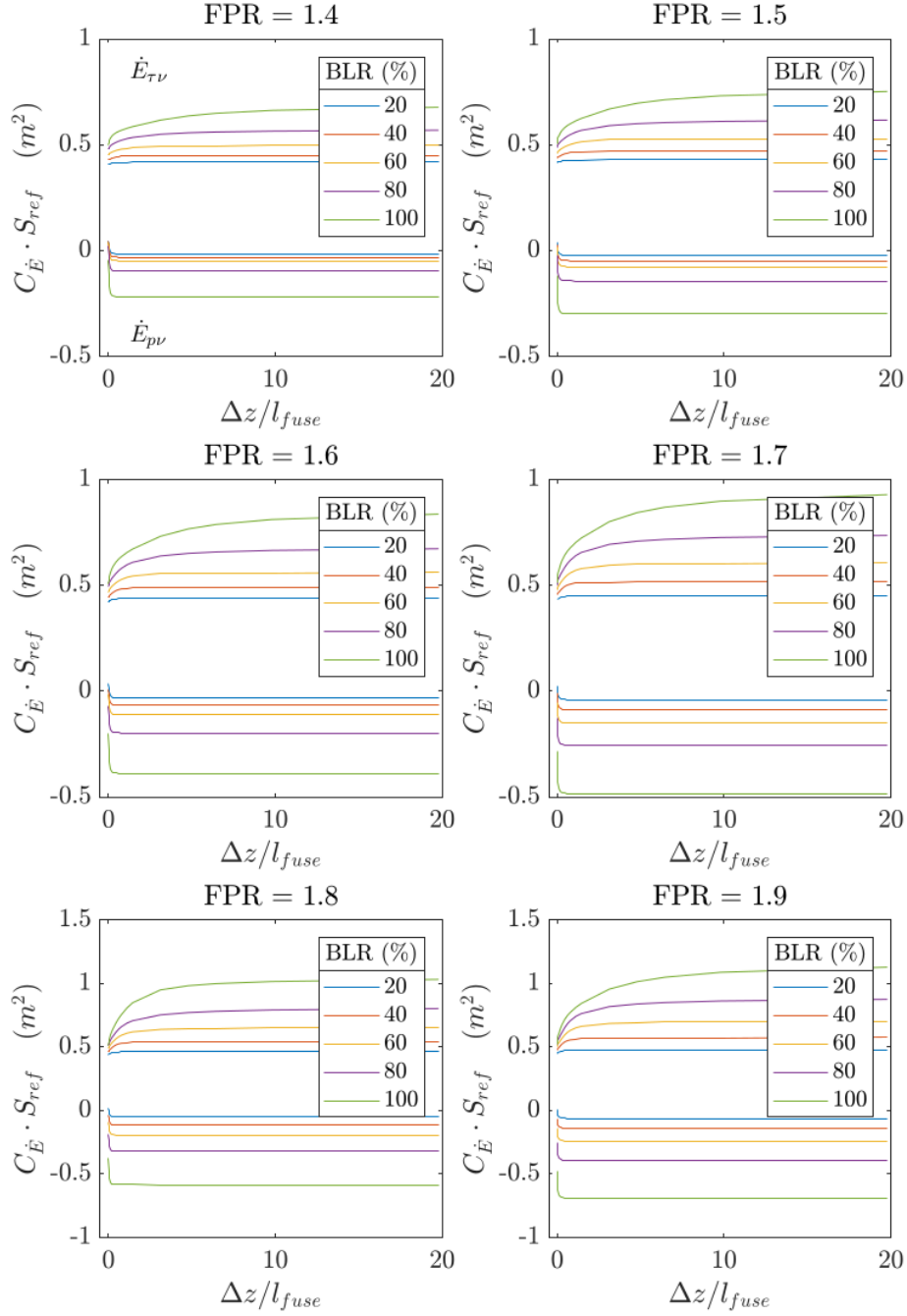


Figure 6.42: Breakdown of volumetric dissipation $\dot{E}_{\tau\nu}$ and volumetric compression work $\dot{E}_{p\nu}$ at different trefftz locations, for propulsors at Position B.

further decompose these contributions and quantify proportions attributed to different flow mechanisms, such as jets, shocks and BLs. This is made possible by the volume integrals of viscous dissipation, $\dot{E}_{\tau\nu}$, and volumetric compression work, $\dot{E}_{p\nu}$, which can be quantified in localised regions of the flow. The procedure of separating out jet (including core constituents and transition layers), shock and BL (also downstream viscous wakes) regions has been described in detail in Section 5.1.4. This procedure has been applied to all the Position B cases of the design exploration, in order to quantify and further understand the contributions of different flow regions to the *NVF*. This subsection first looks at viscous dissipation and then volumetric compression work.

Volumetric Dissipation $\dot{E}_{\tau\nu}$

In analysing contributions to the overall viscous dissipation, the flow is separated into two main regions; (1) the jet region and (2) the flow outside of the jet. The jet is itself subdivided into a core and transition region (the structure of the jet has already been introduced in Section 5.1). The core region may be subdivided further into expansion and compression regions. The viscous dissipation of the jet, and its constituents, are depicted in Figure 6.43, where Figures 6.43a to 6.43d show the contributions of the expansion core, compression core, transition and combined regions, respectively. The expansion within the jet core is isentropic, and as a result the viscous dissipation contribution is approximately zero, as is seen from the scatter in Figure 6.43a. On the other hand, the compression regions within the jet core are anisentropic [66], and have finite non-zero contributions increasing with BLR and FPR. The contributions in the compression core region are still small, and only non-zero for $FPR \geq 1.6$. The contribution increases with FPR because of the stronger compression and shock waves, which are also larger in number and affect more mass flow with increased BLR. The viscous dissipation in the core is completely outweighed by the amount of dissipation in the transition region, as depicted in Figure 6.43c. This is the mixing region between the jet and the external flow, and, as such, is comprised of layers high in shear. The jet transition region contribution grows with FPR, due to the larger velocity differentials between the jet core and external flow regions. Furthermore, increasing the BLR results in shear regions containing larger amounts of mass flow. Finally, it can be deduced from Figures 6.43c and 6.43d that the transition region dominates the overall contribution of the net dissipation of the entire jet region. Referring back to the absolute values for the entire propelled fuselage, Figure 6.42, it can be seen that the jet transition region accounts for more than half the total viscous dissipation.

Shifting focus to the flow external to the jet, the components of viscous dissipation may be attributed to the shock over the nacelle, the airframe BLs, viscous wakes, and the remaining bulk (potential) flow outside of these regions. Figure 6.44a illustrates the very small viscous dissipation contributions of the bulk flow, which is theoretically an isentropic region. The values are not quite zero, as there will always be some shear and shock portions included in this flow, due to the threshold definitions used in the partitioning procedure (see Section 5.1). Nonetheless, the contribution is an order of magnitude smaller than actual shock regions, and a few orders of magnitude smaller

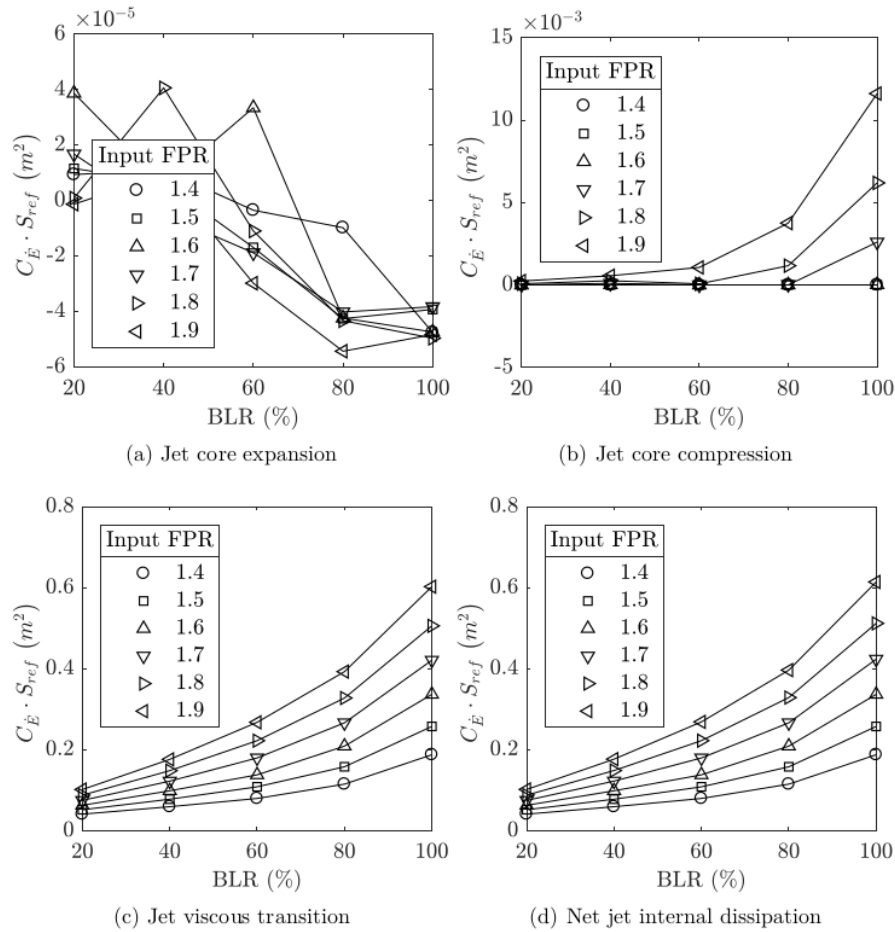


Figure 6.43: Volumetric viscous dissipation within the jet flow regions, for propulsors at Position B

than the BL and viscous wake regions.

Figure 6.44b illustrates the dissipation contribution of the shock over the nacelle. The contribution is sensitive to BLR, where larger ingestion results in bigger nacelle geometries and stronger shocks. However, there is no clear trend explaining correlations to FPR, as two crossover points (around BLR = 40 and 80%) are observed. Although FPR does effect the nozzle exit area, and therefore after-body curvature, it is evident that the BLR (and therefore MFR) are dominant parameters in determining the shock strength and its associated losses. A much larger contribution to the overall dissipation, comes from the BLs over the airframes surfaces in combination with the shear layers outside of the jet transition region. This is depicted in Figure 6.44c, where there is an almost linear correlation with BLR and FPR. Increasing the BLR results in larger nacelle geometries and larger surface area scrubbed by the flow. It also results in stronger shocks, and thereby more severe separation bubbles at the after-body's trailing edge. Separation represents high shear flow, and therefore large dissipation contributions. Higher FPRs result in larger velocity differentials between the jet core and outer flow, which affects the shear outside the jet transition region. Referring back to Figure 6.42, it is seen that the flow external to the jet accounts for nearly half of the total dissipation of the entire propelled fuselage assembly.

In summary it is observed that the majority of the dissipation contributions come from high shear layers, and that shocks contribute very little directly to the overall dissipation magnitudes. However, the key word here is directly. The shock BL interactions over the nacelle are responsible for the severe separation bubbles at the after-body's trailing edge, which are high shear regions. Therefore, the shocks do have significantly large indirect contributions to the net viscous dissipation. It is almost impossible to decouple out what proportion of the shear layers may be attributed to the shock, and this is left for future work. The author is not sure how one may achieve such a differentiation, but it may be performed qualitatively by manually partitioning out the portions of the BL affected by the shock's presence. An alternative possible approach is proposed in the following discussion on volumetric compression work.

Volumetric Compression Work $\dot{E}_{pv}$

Having established that the net contribution of the volumetric compression work is non-zero and in the thrust direction, it is worthwhile to consider each of the jet and external flow regions in an attempt to track the thrust component sources. This is indeed an important contribution of this work, as this form of *NVF* contribution has been previously misunderstood and therefore never quantified or attributed to different flow mechanisms. The volumetric compression work integral, $\dot{E}_{pv}$, has been applied to the different jet regions, the results of which are depicted in Figure 6.45. The quantity attributed to the jet core expansion region is shown in Figure 6.45a, and it becomes immediately evident that this term represents the thrust contribution by the jet⁴. This

⁴This is the thrust contribution of the jet plume, which is a significant proportion of the total thrust on the propelled fuselage. The rest of the thrust is realised at the surfaces of the fan blades inside the

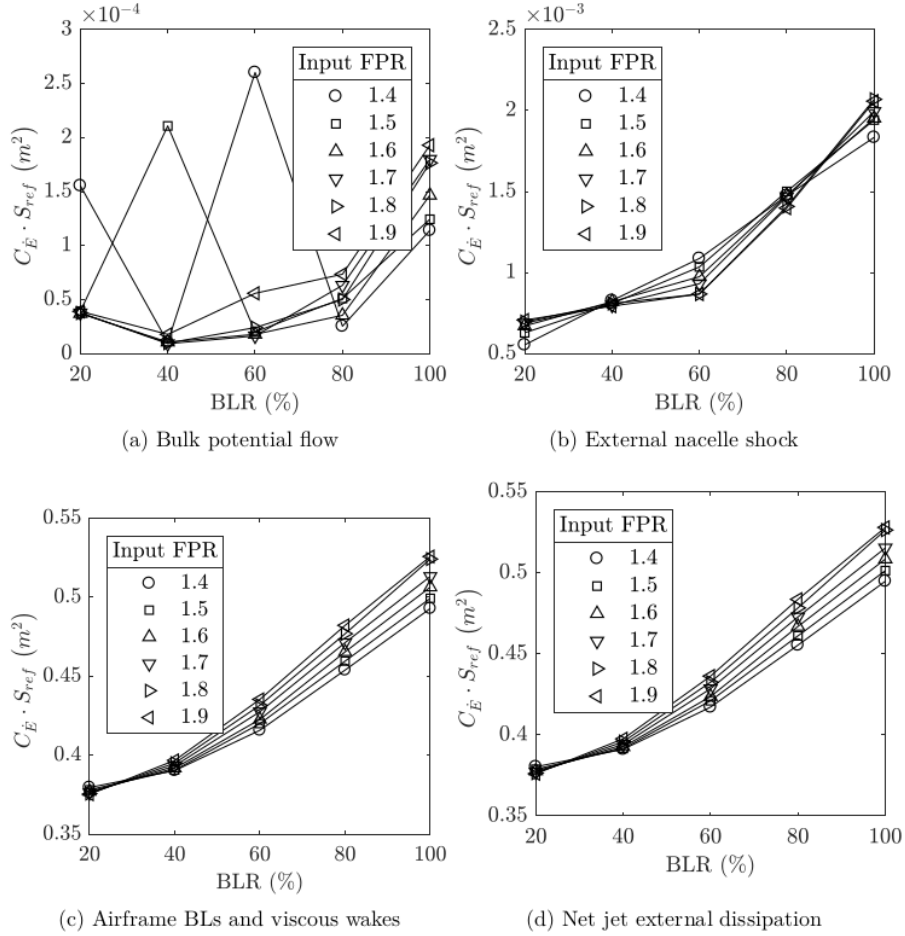


Figure 6.44: Volumetric viscous dissipation in flow regions outside of the jet flow, for propulsors at Position B

is actually intuitive, because in these regions the flow has a static pressure well above ambient and is also expanding. As such, it can be visualised how this expansion at high pressure is propelling the fuselage-propulsor assembly forward. In this context, it may be more appropriate to rename the term volumetric expansion work. Following this explanation, it is clear why the contributions to forward force increase with BLR and FPR. Figures 6.45b and 6.45c show negligible contributions from the jet core compression and jet transition regions, respectively. The jet core compression regions actually act in the opposite direction, but are very small even at high FPRs. This almost negligible force is quite understandable, as it may be visualised how the contracting flow, despite its high pressure, is unable to exert a force on the fuselage assembly through means of a pressure field. The jet transition region will overlap slightly with some of the jet core expansion regions, and therefore has a very small contribution to a forward force. However, the net contribution, Figure 6.45, is comprised almost entirely of the jet core expansion region.

The graphs of Figure 6.46 depict contributions within the flow field external to the jet region. In considering the shock over the nacelle, Figure 6.46b, it is useful to also examine the contribution in the bulk flow region, Figure 6.46a, at the same time. These two plots clearly show how the compression work done ahead of the shock, is expanded almost reversibly behind the shock, leaving behind little-to-no net result. There is some compression work contained within the BL region, as shown in Figure 6.46c, but this is likely to be in the shock BL interaction zone (within the lambda structure) where the partitioning threshold has been unable to exactly decouple the two regions. In that regard, future work may consider implementing an automated iterative method, which adjusts the shock region threshold ζ , such that compression work in the BLs is minimised. This opens up the opportunity to separate the BL shock interaction region more definitively and robustly than has been done before, and thereby further improve methods of region separating proposed in works such as Paparone and Tognaccini [7]. The outcomes leading to this suggestion for future work of separating BL shock regions in numerical CFD solutions, is another contribution of this thesis. Overall, there is a significant net contribution of the flow external to the jet region, which acts in the direction of a rearward force. This net contribution is attributed to the shock wave because of the localised compression, and is also clear because of the lack of sensitivity to changes in FPR. This is explained further by the clear correlation to BLR, where increasing BLR results in stronger shocks (because of a reduction in MFR) and thus larger compression work contributions. Nonetheless, the rearward force contribution of the external shock is small in comparison to the forward force contribution of the jet core expansion region. Additionally, the contributions from the external shock and jet core expansion regions can be seen to be almost fully decoupled, which further emphasises the value in the spatial decompositions of this work.

One of the major contributions of this work is indicated by this demonstration, which shows very clearly how well the partitioning metrics and procedures of Section 5.1 are able to decompose the jet flow and its internal structures. Furthermore, and perhaps

propulsor control volume

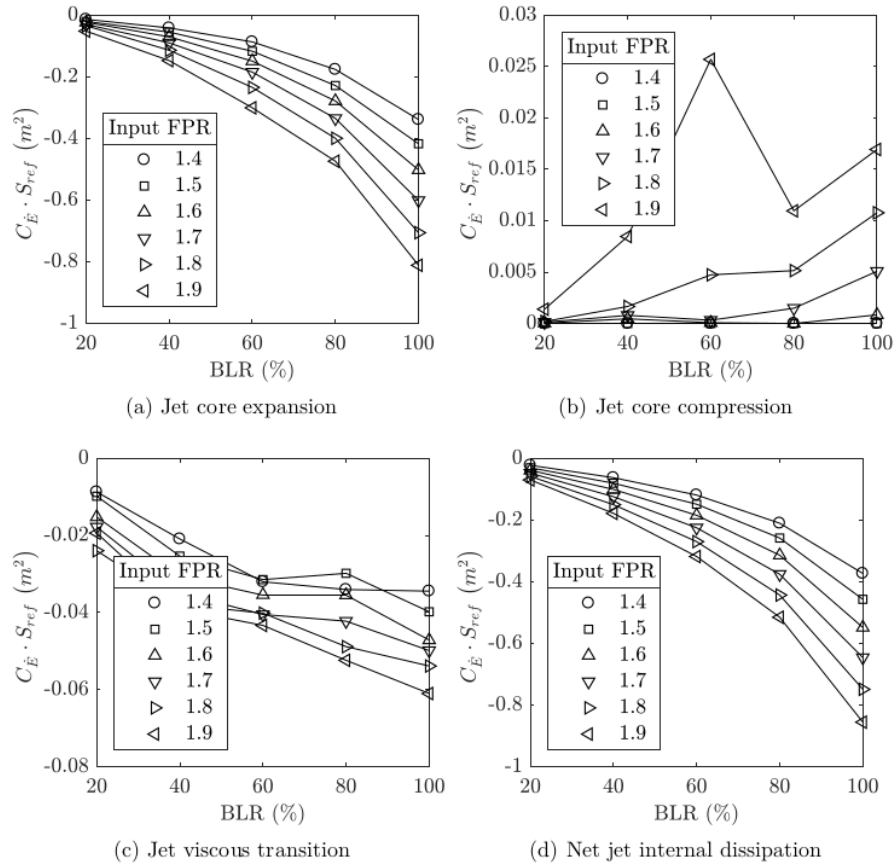
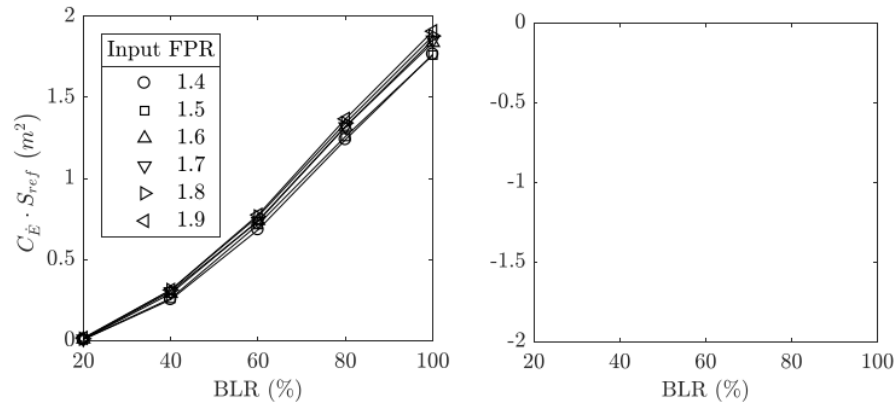


Figure 6.45: Volumetric compression work within the jet flow regions, for propulsors at Position B



(a) Bulk potential flow

more importantly, is the identification of the jet expansion regions responsible for significant thrust generation. This is made evident by the much smaller contribution of the flow external to the jet region, which also does confirm how the flow passing through the shock is compressed reversibly, and re-expanded elsewhere in the bulk flow.

6.7 Comparisons

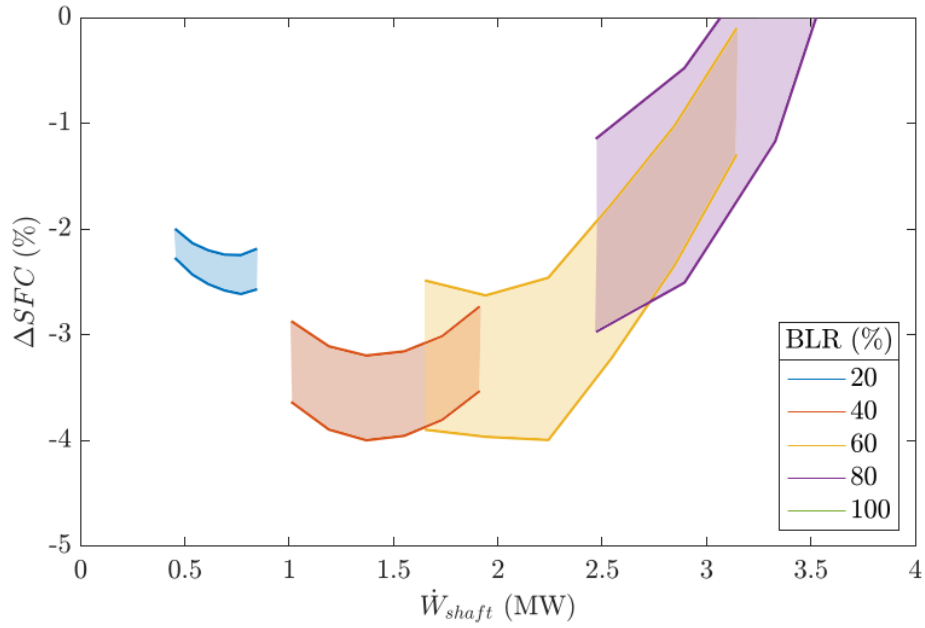
The final objective of this chapter was to quantify the uncertainties in SFC benefit associated with the different methods of performance evaluation. Figures 6.47a and 6.47b illustrate the SFC benefit ranges as a function of the BLI shaft power and thrust-split, respectively. The more conservative results have been found from near-field momentum based methods, whereas the energy-based methods have yielded more optimistic values. The more optimistic energy-based results are due to a higher predicted NVF , and hence the slight shift to larger thrust-splits. Nonetheless, it may be concluded that an optimum SFC benefit between 3-4% is achievable for the retrofit design approach taken in this design exploration.

The results of Figure 6.47 may also be compared to the lower fidelity preliminary parametric studies. In a summary of the preliminary parametric results, Figure 6.2, a propulsor with 1-2% intake pressure losses, was found to have an optimum SFC benefit of around 5%. This corresponded to a thrust-split of 30% and BLR around 60%. The SFC benefits from CFD, for similar intake pressure losses, are 1-2% lower at a reduced thrust-split of 20% and BLR of 40%. Based on this comparison, and the assumptions made, it may be concluded that the preliminary parametric approach gives reasonable estimates of BLI performance. That is to say that the predictions are not unrealistic, and although more optimistic, could possibly be achieved (or even surpassed) through improved nacelle design and denser design explorations.

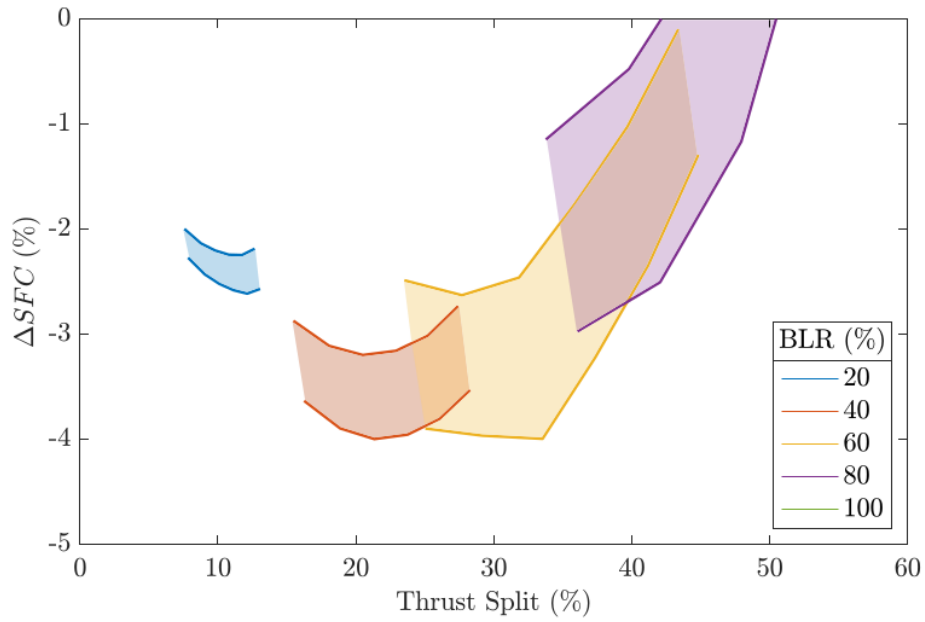
6.8 Summary

This chapter has detailed a design exploration study carried out on a business jet, where a retrofit approach was taken. The retrofit approach assumes that the baseline aircraft's engines are operated at an off-design condition (and are still podded to ingest free-stream flow) to produce a portion of the thrust, whilst powering a BLI fan. A lower fidelity tool was used to perform preliminary parametric studies to help guide the direction of the CFD studies. These preliminary studies indicated that SFC benefits in the range of 5-7.5% could be obtained if less than the BL thickness was ingested. They also showed that optimum benefits are obtained at small thrust splits, between 20-40%, for a fan ingesting the BL around the full circumference of the fuselage.

In the CFD design exploration, it was decided to examine the performance at two propulsor positions along the tail-cone. The first, Position A, was located at the top of the tail-cone where it meets the fuselage's cylindrical mid-body. The second, Position B, was located midway along the tail-cone's overall length. A pre-processing tool was developed to approximate geometries that would achieve a desired combination of BLR, FPR



(a) SFC benefit versus power off-take



(b) SFC benefit versus thrust split

Figure 6.47: Uncertainty in SFC benefits for propulsors at Position B, based on near-field momentum-based, far-field momentum-based and far-field energy-based methods.

and fan inlet Mach number at any tail-cone location. This was coupled to an in-solution tool, with the purpose of iteratively updating CFD boundary conditions to ensure that the fan polytropic efficiency was kept constant for all parametric combinations, which was necessary for consistency. These tools were used to generate CFD solutions for BLR ranges between 20-100% (at increments of 20%) and FPR ranges between 1.4-1.9 (at increments of 0.1), for each position A and B. This yielded a total of 60 cases.

The propulsor performance was examined to see how accurately the pre-processing tool approximated geometries to meet the desired BLR, FPR and fan inlet Mach number requirements. The FPR was found to differ within acceptable amounts from the desired values, and any deviations were not deemed to impact the outcome of the final results (this is because a range of FPRs was required to obtain optimums for a given BLR, and the range is important not the discrete points). Sufficient pre-diffusion was achieved, and the fan inlet Mach number was found to be within an acceptable tolerance of the design limit. In addition, estimates of pressure recoveries were found to correspond closely to the values obtained from the CFD solutions. Finally, the shaft power driving the BLI propulsor was extracted from the flow, which was later used in assessing the propulsive efficiency of the propelled fuselage.

Subsequent sections of the chapter, analysed the propelled fuselage performance using the different methods presented in Chapters 3 and 4. Near-field momentum-based methods were used to analyse the pressure and friction coefficients along the airframe surfaces. This allowed for further force analysis by examining the different thrust-force accounting definitions formulated in Chapter 3. A comparison between the trends of the different definitions, was used to gain insight into the physics behind *NVF* contributions. Finally, the system-level benefits in SFC were determined by incorporating the cycle performance of the main turbofan engines. The near-field study indicated that only propulsors at Position B were able to achieve a benefit in SFC, where an optimum of just over 3% was achieved. This corresponded to a thrust-split around 20%, where 40% of the BL was ingested. A subsequent far-field momentum-based analysis was performed, and yielded more optimistic propulsive efficiency benefits (corresponding to a larger SFC saving of around 3.8%).

The last section of the chapter applied the energy-based assessment method to the solutions at Position B. Slightly more optimistic propulsive efficiency benefits were found in comparison to the far-field momentum-based results. More importantly, this method was used to perform energy-based breakdowns of the *NVF*. This was first done for varying trefftz plane locations to track the wake's progression. Subsequently, full flow-field decompositions were performed, where the jet was separated into its different structures. It was found that a significant portion of the thrust may be attributed to volumetric expansion work within the jet core. This was followed by further decompositions of the flow external to the jet, which gave insight into the losses incurred by the airframe shocks and BLs.

A final comparison of SFC benefits from all three methods indicated an optimum of between 3-4% from the baseline aircraft's SFC. This was found to occur at a thrust-split of around 20% when ingesting 40% of the BL. Although significant, the 1% uncertainty

in SFC benefit was deemed to be acceptable.

6.8.1 Contributions

The contributions of this chapter are as follows:

- Design exploration of BLI applied to a business jet which, to the author's knowledge, has not yet been published in public literature. This includes trends which show how the benefit of a retrofit design is optimised through managing thrust-split. This is based on a unique turbofan performance map, developed specifically to analyse thrust-split retrofit designs. In addition, the author is unaware of any other publicly available studies which have considered the effect of different BLI propulsor tail-cone positioning.
- A tool and procedure for designing a retrofit fuselage concentric BLI propulsor, based on CFD solutions of the baseline fuselage. This tool serves to approximate geometries to meet basic design requirements within reasonable accuracy. This tool also automates geometry, mesh and solution generation. It interacts iteratively with the solution to ensure specified polytropic efficiencies for consistent comparisons. This includes a method for modelling the propulsor in CFD, which is preferred to actuator disk models used in other studies (please refer to Chapter 2).
- An analysis of the NVF using different BLI appropriate thrust-force accounting definition is a first of its kind (at least to the author's knowledge). It is demonstrated that better understanding is gained by using multiple thrust-force accounting definitions, as opposed to common practice which tends to adopt one definition for dividing up airframe surfaces between engine and airframe manufacturers.
- Clear evidence that the volumetric compression work, $\dot{E}_{pv}$, has a significant thrust contribution, which cannot be assumed small or negligible. This advances the work by Drela [5], who speculated its contribution to only be significant where heat is added at pressure within the control volume. This is now known not to be the case, and making NVF the subject of the energy equation gives insight as to why not.
- Application of a unique flow-mechanism partitioning procedure (see Section 5.1.4), has allowed for detailed extraction of jet structures, to which the volumetric dissipation, $\dot{E}_{\tau\nu}$, and volumetric compression work, $\dot{E}_{pv}$, have been applied. Analyses of these terms has confirmed force contributions of isentropic expansion, dissipative compression and viscous transition regions. To the author's knowledge, this form of energy breakdown is unique to this work and has provided very useful insight into contributions to NVF .
- Identification of a new method for separating out, more definitively, the shock contribution from BL-shock interaction regions. It is proposed that this may be done

by iteratively determining the threshold of ζ (refer to Section 5.1.2) that corresponds to a minimum net amount of compression work in the viscous BLs. This is a unique improvement to the type of regional partitioning methods presented by Paparone and Tognaccini [7]. Demonstration of this proposed method has been left for future work.

- The final comparison of BLI benefit has been performed using various methods, based on different physical laws. This multi-physics based approach to BLI performance is unique to this work, and is opposed to prescribing a single method for future use. Instead, improved physics based and numerical understanding of CFD solutions is gained through comparisons between methods. In addition, these comparisons allow for uncertainties in predicted benefits to be quantified.

6.8.2 Future Work

This chapter has possibly opened up more questions than it has answered, and recommendations for future work seem endless. Some of the more pressing recommendations are listed below.

CFD Tools and Methodology

Coupling between the pre-processing tool and in-solution tool. A feedback loop between the pre-processing tool and in-solution tool is required to eliminate geometry approximation errors. A procedure should be implemented to iteratively update the propulsor geometry, to ensure that exact BLR, FPR and fan inlet Mach numbers are achieved. Enhancing the tool's capabilities in this way, would allow for improved consistency between performance comparisons.

Denser Design Explorations

The design exploration presented in this chapter, was limited in the number of tail-cone positions considered. It was found that the tail-cone position had a strong influence as to whether SFC benefits were achievable at all. It is highly recommended that future work explores tail-cone positions more comprehensively. In addition, the propulsive efficiency and SFC benefit was found to be very sensitive to changes in BLR. This design exploration looked at ingestion intervals of 20%, where an optimum was found at 40% BLR. However, it is unclear whether this 40% is the true optimum. Observing the trends in SFC benefit (see Figure 6.47), suggests that an actual optimum may lie somewhere between 40-50% .

The outcomes of the design exploration have highlighted regions of benefit. Knowing these regions, it would be worthwhile to extend the exploration to consider parametric

Turbofan Resizing and Tail-Cone Reshaping

It is evident that there is a compromise in the achievable benefits from a retrofit design perspective. The optimum propulsive efficiency point (of the propelled fuselage) does not coincide with that of the turbofan engine. As such, there is a trade-off between the efficient operation of the engine and improvements in propulsive efficiency. There will always be some trade-off, however, resizing the turbofan engine allow for the trade-off to be minimised by bringing the two optimums closer together.

The retrofit approach taken in this work, constrained the propulsor geometry to conform to the baseline fuselage curvature as closely as possible. This constraint naturally results in a compromise in performance. As an example, an unfavourable after-body curvature was required of propulsors at tail-cone Position A, which lead to strong dissipative shocks over the nacelle and large separation bubbles at its trailing edge. Slight modifications to the tail-cone would allow for less sever after-body curvature, thereby eliminating these inefficiencies.

Nacelle Geometry and Design Optimisation

A relatively simplified approach to defining the nacelle geometries, was taken. The nacelle design could be drastically improved using more flexible and detailed methods. Class Shape Transformation functions, as used by Goulos *et al.* [55], would be suitable for design optimisation studies. In this work, a 3-4% SFC benefit was achieved for a knowingly suboptimal nacelle design. Therefore, it is speculated that drastic improvements on this benefit could be achieved if the nacelle design were to be optimised.

Chapter 7

Summary, Conclusions and Recommendations

7.1 Summary and Conclusions

7.1.1 Research Rationale

Advancements in propulsion system performance, have been reliant on improvements in propulsive efficiency. Traditionally, this has been achieved by increasing the bypass ratio of podded (i.e. free-stream ingesting) turbofan engines, which effectively reduces the specific thrust. This results in a subsequent enlargement of the fan and nacelle diameters, as the engine ingests greater amounts of mass flow. Larger nacelle diameters start to incur penalties, due to the implications on external aerodynamics. This is particularly true for aircraft flying at high subsonic cruise Mach numbers, as in the case of a business jet. This is because, compressible drag rise is dependent on the aircraft's area distribution. This is one of the main reasons why business jets are normally propelled by high specific thrust turbofan engines. Based on the above, it was reasoned that the business jet may benefit substantially from a BLI propulsion system, whereby the specific thrust may be reduced without incurring such heavy penalties in external drag rise. Therefore, the aim of the research was to perform a design exploration study on BLI applied to a business jet, with particular emphasis on external aerodynamic implications.

7.1.2 Assessment Methods

The initial focus of the research, was to develop methods which would allow for the aerodynamic coupling between propulsor and airframe to be thoroughly analysed and understood. To achieve this, a multi-physics, control-volume based approach was taken. This approach led to the development of near-field momentum-based, far-field momentum-based and energy-based net-vehicle-force formulations. The near- and far-field momentum based formulations, allowed for a set of different thrust-force accounting systems to be defined. Comparisons between the multiple thrust-force accounting definitions,

allowed for improved understanding of NVF contributions by different airframe and nacelle surfaces. Energy-based methods, on the other hand, led to the decomposition of the flow-field into various physical mechanisms. These included flow phenomena internal and external to the jet plume. Decompositions within the jet plume isolated regions of isentropic expansion and dissipative compression (shocks) within the jet's core, as well as a transition layer comprised of turbulent mixing. Decompositions external to the jet plume, isolated regions of external shocks, BLs and viscous wakes, as well as the bulk (potential) flow region found in the rest of the domain. Region selectors, unique to this work, were formulated to identify these different flow phenomena within the CFD domain. A precedence procedure was developed to separate out regions with coupled flow phenomena (e.g. BL shock interactions). In addition, the energy based formulation allowed for wake progression breakdowns to be performed, whereby cumulative energy content is analysed at different downstream treftz plane locations.

7.1.3 Practical Implementation in RANS CFD

Before these methods could be used within a design exploration, it was first necessary to investigate the practical implications associated with applying them to RANS CFD solutions. One of the key aspects involved examining viscous stress tensor fields, for different near-wall treatments and interpolation schemes. This was done by comparing the surface forces reported by the solver, versus those obtained directly from the viscous stress tensor. It was found that only the $k - \omega$ SST turbulence model, in combination with a Green-Gauss Cell-Based gradient scheme, was able to produce a continuous viscous stress tensor field, which yielded surface forces consistent with those reported by the solver. Having resolved this, the different assessment methods were applied to the solution of a non-propelled body, and the results compared. The far-field momentum-based and energy-based methods were applied to control volumes having varying extents. This showed that the far-field momentum method was particularly sensitive to numerically introduced spurious effects. Moving the control volume surfaces outward from the body (for a constant treftz plane location), revealed that spurious forces could be somewhat filtered out. This was attributed to the fact that spurious effects are prominent within the highly dissipative flow regions, and that the presence in the bulk potential flow is insignificant. This was confirmed when the treftz plane location was moved further downstream, and rearward force was observed to steadily increase as more of the viscous wake was contained within the control volume. Following this explanation, it was reasoned that the near-field forces are influenced most by the spurious effects, which was confirmed by observations whereby the far-field forces increased steadily toward near-field values as the control volume was brought ever closer to the body. The forces obtained from the energy-based formulation, on the other hand, were observed to be insensitive to changes in control volume extent, and predicted the drag to be much smaller than the values obtained by the momentum-based methods. In comparing the net forces, good agreement was found between the forces predicted by all three methods, and found to be within 4% of one another. Similar trends were found when applying the same control volume treatment to a propelled-fuselage case. A very close agreement was observed

between far-field momentum-based and energy-based results (within 1%), for a control volume extent that minimised spurious effects. However, much larger discrepancies (up to 9%) were observed between these methods and the near-field results. This discrepancy was attributed to the increase in flow-field complexity, which now contained BL, shock and jet interaction regions. This suggests that the far-field and energy-based methods yield more consistent results.

7.1.4 Design Exploration Study

A design exploration study was performed by retrofitting a business jet with a fuselage concentric propulsor, powered by the baseline engines, which remained podded in the free-stream flow. A preliminary parametric study was first performed to gauge conditions under which BLI benefit may be likely. A ram drag approach to analysing BLI benefit, was based on a flat plate analogy to model the boundary layer over the fuselage, which was assumed to be cylindrical. Thrust-split, BLR, fan efficiency and intake pressure recoveries, were varied parametrically to assess potential benefits. It was observed that an optimum SFC benefit between 5-7.5% is achievable at thrust-splits between 30-35%, when ingesting 65-90% of the BL thickness. These results guided the subsequent parametric studies performed using CFD. Two tail-cone positions were examined, with the first at the top of the tail-cone, and the second positioned midway along the tail-cone. Benefits were only realised for the latter position, where a 3-4% improvement in SFC was realised for a thrust-split around 20%, by ingesting 40% of the BL thickness. The 1% uncertainty in SFC benefit is due to the difference between the results obtained from the three different assessment methods. Energy breakdowns and decompositions were performed on all of the cases. One of the significant outcomes of this research was revealing that a significant proportion of the thrust force may be attributed to the isentropic expansion region within the jet plume's core. This is just one example of the many interesting observations made possible by the energy decompositions performed in Chapter 6, which are too lengthy for inclusion here.

7.2 Knowledge Contributions

There are numerous contributions to knowledge contained within this thesis. These are listed in the introduction and summary sections of each chapter within the main body of this document. A few of the prominent contributions have been listed in the introductory chapter to this thesis, and are repeated below for emphasis:

1. A fully customisable, momentum-based, control volume approach, has been developed to produce a unique set of thrust-force accounting definitions, which are applicable to tightly-integrated airframe-propulsion systems. This work includes a step-by-step procedure for extracting these definitions from RANS CFD solutions. In addition, it is demonstrated that a comparison between the definitions allows for improved insight into the mechanisms behind BLI propulsion.

2. A method of BLI performance quantification, based on a modified propulsive efficiency for retrofit design approaches. This avoids the flaws associated with the conventional definition, when applying it to BLI assessments (the conventional propulsive efficiency may theoretically exceed 100%, but this is not possible with the modified definition used in this work).
3. Formulation of the kinetic and total energy equations in the absolute reference frame. The derivation is unique to this work, and having the energy equation in the absolute reference frame provides an intuitive perspective into the physics behind the mathematical terms. This work directly addresses concerns of Renard and Deck [4] about Drela's [5] formulation in the aircraft's body axes system. In so doing it accounts for pressure and friction forces on solid surfaces, as well as the kinetic energy imparted to the fluid (see Section 4.1.1 for more details). This leads to an energy-based control volume formulation with NVF as the subject, which sheds light on the contributions of various energy terms. Other energy formulations [5, 6] take a *energy source = energy sink* format, which makes it difficult to understand the contributions of some terms.
4. A procedure of spatial decomposition by Paparone and Tognaccini [7] into different flow mechanisms has been adopted, modified and improved from formulations on entropy drag, and incorporated into the energy approach of this work. This decomposition allows for volume integrals describing energy dissipation and volumetric compression work, to be quantified for different flow regions (e.g. shock, jet and BLs). A BL and viscous wake region selector, modified from Paparone and Tognaccini's work, is a particularly significant contribution (see Section 5.1). In addition, the precedence procedure developed, allows for further decomposition of the jet into core expansion, core compression and viscous transition regions, and is unique to this work.
5. This thesis provides clear evidence that the volumetric compression work term of energy-based formulations, has a significant thrust contribution, which cannot be assumed small or negligible. This advances the work by Drela [5], who speculated its contribution to only be significant where heat is added at pressure within the control volume. This is now known not to be the case, and making NVF the subject of the energy equation gives insight as to why not. The majority of the volumetric compression work term's contribution is found to be located within the isentropic expansion regions of the jet core. This discovery was made possible by the flow decomposition procedure developed in this thesis. More importantly, it shows that there is a natural thrust-force definition that avoids ambiguity. This definition attributes all rearward forces to volumetric viscous dissipation in the flow, whereas thrust is attributed to the surface forces on fan blades, along with the volumetric compression (or rather expansion) work in the jet plume.
6. Design exploration of BLI applied to a business jet which, to the author's knowledge, has not yet been published in public literature. This includes trends which

show how the benefit of a retrofit design is optimised through managing thrust-split. This is based on a unique turbofan performance map, developed specifically to analyse thrust-split retrofit designs. In addition, the author is unaware of any other publicly available studies which have considered the effect of different BLI propulsor tail-cone positioning. The final comparison of BLI benefit has been performed using various methods, based on different physical laws. This multi-physics based approach to BLI performance is unique to this work, and is opposed to prescribing a single method for future use. Instead, improved physics based and numerical understanding of CFD solutions is gained through comparisons between methods. In addition, these comparisons allow for uncertainties in predicted benefits to be quantified.

7. To the author's knowledge, no other study has yet performed the full energy breakdowns and decompositions, from RANS CFD solutions, like those presented in this thesis. This work includes comprehensive analyses of energy progression and transfer within the wake. It also performs a full decomposition of physical mechanisms in the jet internal and external flow-fields.

7.3 Recommendations for Future Work

Recommendations for future work is listed in a summary section within each chapter, and are repeated here for convenience.

7.3.1 Momentum-Based Assessment Methods

From Thrust-Force Accounting to Thrust-Drag Accounting

At this stage of the work, the concept of "drag" was neglected. Definitions of drag are based on Prandtl's extension of D'Alembert's paradox [36, 37]. This definition requires the removal of the potential flow pressure forces from the real viscous force, when calculating drag contributions over a partial body. The author was unable to find a way to do this, that is physically and mathematically rigorous, or indeed practical. This is because (atleast how it seems to the author), the elimination of terms is reliant on the assumption that streamtube geometries are equivalent in potential and viscous flow. It also relies on the additional assumption that unbounded streamtube surfaces do not experience any viscous force due to shear. It would be interesting to see if there is a possible way to formulate drag, according to the afore mentioned definition, without having to make these assumptions. This would lead to better definitions of thrust-drag accounting, which allow for improved evaluation of drag contributions by different airframe and propulsion system surfaces.

Performance Metrics

One of the primary difficulties in BLI performance evaluation, is finding an alternative definition for propulsive efficiency from the conventional definition. The conventional

propulsive efficiency is dependent on a thrust-drag accounting system which is no longer applicable to BLI configurations. This work presented a modified propulsive efficiency reliant on the reference net thrust, which works well in the context of retrofitting an existing airframe design. However, more important for future analysis is an efficiency metric which can be used when starting a new vehicle design which does not have a baseline for reference. The author was unable to create such a definition, but ideally one is required which describes a vehicles overall propulsive efficiency. This is difficult because flight power of the *NVF* is zero in steady level flight. In summary, a vehicle efficiency metric is required.

7.3.2 Energy-Based Assessment Methods

Understanding of $-(p - p_\infty) \nabla \cdot \mathbf{V}$ Contributions

$-(p - p_\infty) \nabla \cdot \mathbf{V}$ is a rather perplexing term, as it becomes difficult to understand net contributions. Unlike the $(\bar{\tau} \cdot \nabla) \cdot \mathbf{V}$ integrand, $-(p - p_\infty) \nabla \cdot \mathbf{V}$ may be positive and negative in different regions of the flow domain, making it difficult to understand what the net effect is, or how different flow mechanisms contribute to the net effect. One of the main questions about this term is whether or not it is reversible or dissipative? Or alternatively, can it be decomposed into reversible and irreversible contributions? Certainly in the scenario where heat is added/extracted to/from a volume, this term has a definite irreversible portion. However, for adiabatic instances is this term reversible? Drela [5] states that through a shock wave, this term undergoes a compression which is expanded reversibly along the streamline. Therefore further work is required to better understand this term, through an improved decomposition.

Proof of $(\bar{\tau} \cdot \nabla) \cdot \mathbf{V} > 0$

$(\bar{\tau} \cdot \nabla) \cdot \mathbf{V} > 0$ is stated by Drela [5], and from experience has been found to hold true (at least within a reasonable numerical tolerance for RANS CFD). However, it may be useful to prove that this is indeed the case. Velocity derivatives may be of either sign in different regions of the flow, and it becomes difficult to see exactly why this statement is true. Proof of this statement may also lead to a better understanding of the physics behind this term and how the dissipation is manifested in different flow regions.

Relations to Entropy Drag Formulations

There is some interesting work on momentum-based far-field drag decompositions in literature [7,23,25–28], which utilize entropy drag and enthalpy terms. Future work should aim to bridge the gap between such decompositions and the types of formulations presented in this work and References [5,6]. This may lead to more definitive decompositions and physical interpretations, as well as means for more extensive verification.

7.3.3 Practical Implementation in RANS CFD

Flow-Field Decompositions

The flow-field spatial decompositions have only been tried on 2D and 2D axisymmetric cases. Future work needs to comprehensively examine whether the procedure presented in this work, is equally valid and practical within 3D flows. In addition, future work should include a method for separating out lift induced effects, such as wing tip vortices. This is a crucial step, which is required to analyse lifting bodies.

Validation Cases

This chapter repetitively mentioned, during the back-to-back comparison of NVF predictions, that no particular method was prescribed as being correct or more accurate than the other. Instead, the focus of this chapter was to examine whether consistency between the methods could be established, and to quantify associated uncertainties. Future work should focus on applying these methods to an extensive list of validation cases, using different codes. This will not only validate the methods, but give improved insight as to which method may yield more correct results.

Understanding the Origins Spurious Effects

The topic of spurious effects was tackled using a black box approach, where the author did not have access to the source code of Fluent. A gap in the knowledge is how numerical spurious effects influence near- versus far-field (an energy) force predictions. It would be extremely useful for future studies to look into the theory and algorithms used in RANS CFD solvers, to explain better the reasons for the trends observed in this work.

7.3.4 Fuselage Concentric BLI Propulsion Performance

CFD Tools and Methodology

Coupling between the pre-processing tool and in-solution tool. A feedback loop between the pre-processing tool and in-solution tool is required to eliminate geometry approximation errors. A procedure should be implemented to iteratively update the propulsor geometry, to ensure that exact BLR, FPR and fan inlet Mach numbers are achieved. Enhancing the tool's capabilities in this way, would allow for improved consistency between performance comparisons.

Denser Design Explorations

The design exploration presented in this chapter, was limited in the number of tail-cone positions considered. It was found that the tail-cone position had a strong influence as to whether SFC benefits were achievable at all. It is highly recommended that future work explores tail-cone positions more comprehensively. In addition, the propulsive efficiency and SFC benefit was found to be very sensitive to changes in BLR. This

design exploration looked at ingestion intervals of 20%, where an optimum was found at 40% BLR. However, it is unclear whether this 40% is the true optimum. Observing the trends in SFC benefit (see Figure 6.47), suggests that an actual optimum may lie somewhere between 40-50% .

The outcomes of the design exploration have highlighted regions of benefit. Knowing these regions, it would be worthwhile to extend the exploration to consider parametric evaluations of nacelle geometry. For instance, this work fixed the lip contraction ratio, cowl shape and diffusion split (among other parameters). These were preselected values, and their impact on performance is unknown.

Turbofan Resizing and Tail-Cone Reshaping

It is evident that there is a compromise in the achievable benefits from a retrofit design perspective. The optimum propulsive efficiency point (of the propelled fuselage) does not coincide with that of the turbofan engine. As such, there is a trade-off between the efficient operation of the engine and improvements in propulsive efficiency. There will always be some trade-off, however, resizing the turbofan engine allow for the trade-off to be minimised by bringing the two optimums closer together.

The retrofit approach taken in this work, constrained the propulsor geometry to conform to the baseline fuselage curvature as closely as possible. This constraint naturally results in a compromise in performance. As an example, an unfavourable after-body curvature was required of propulsors at tail-cone Position A, which lead to strong dissipative shocks over the nacelle and large separation bubbles at its trailing edge. Slight modifications to the tail-cone would allow for less severe after-body curvature, thereby eliminating these inefficiencies.

Nacelle Geometry and Design Optimisation

A relatively simplified approach to defining the nacelle geometries, was taken. The nacelle design could be drastically improved using more flexible and detailed methods. Class Shape Transformation functions, as used by Goulos *et al.* [55], would be suitable for design optimisation studies. In this work, a 3-4% SFC benefit was achieved for a knowingly suboptimal nacelle design. Therefore, it is speculated that drastic improvements on this benefit could be achieved if the nacelle design were to be optimised.

Appendix A

Viscous Terms

This appendix serves to show the expanded forms of the viscous shear stress tensor, as well as associated terms found in the momentum and energy equations. The expanded forms are given here in Cartesian and Cylindrical, as required for post-processing any 2D and 2D antisymmetric CFD simulations in this work.

A.1 Tensor Notation, Operations and Identities

This section describes the tensor notation and identities used throughout this work. They are documented here, as they are particularly pertinent to deriving viscous related terms. First (vectors) and second order tensors are defined as having the following form

$$\mathbf{u} = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} \quad \overline{\overline{A}} = \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{21} & A_{22} & A_{23} \\ A_{31} & A_{32} & A_{33} \end{bmatrix}$$

Subsequent sections present tensor operations and identities applicable in this work. These are given in both Cartesian and Cylindrical coordinates.

A.1.1 Gradient of a Vector

The gradient of a vector returns a second order tensor, in Cartesian coordinates:

$$\nabla \mathbf{u}(x, y, z) = \begin{bmatrix} \frac{\partial u_x}{\partial x} & \frac{\partial u_y}{\partial x} & \frac{\partial u_z}{\partial x} \\ \frac{\partial u_x}{\partial y} & \frac{\partial u_y}{\partial y} & \frac{\partial u_z}{\partial y} \\ \frac{\partial u_x}{\partial z} & \frac{\partial u_y}{\partial z} & \frac{\partial u_z}{\partial z} \end{bmatrix}$$

or alternatively in Cylindrical coordinates:

$$\nabla \mathbf{u}(r, \theta, z) = \begin{bmatrix} \frac{\partial u_r}{\partial r} & \frac{\partial u_\theta}{\partial r} & \frac{\partial u_z}{\partial r} \\ \frac{1}{r} \frac{\partial u_r}{\partial \theta} - \frac{1}{r} u_\theta & \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} - \frac{1}{r} u_r & \frac{1}{r} \frac{\partial u_z}{\partial \theta} \\ \frac{\partial u_r}{\partial z} & \frac{\partial u_\theta}{\partial z} & \frac{\partial u_z}{\partial z} \end{bmatrix}$$

A.1.2 Divergence of a Vector

The divergence of vector returns a scalar, in Cartesian coordinates:

$$\nabla \cdot \mathbf{u}(x, y, z) = \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z}$$

or alternatively Cylindrical coordinates

$$\nabla \cdot \mathbf{u}(r, \theta, z) = \frac{1}{r} \frac{\partial}{\partial r} (r u_r) + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z}$$

A.1.3 Tensor Products and Identities

Here it is assumed that $\bar{\bar{A}}$ is a symmetrical second order tensor, which is true of stress tensors in both Cartesian and Cylindrical coordinates. The dot product between a symmetrical second order tensor and a first order tensor

$$\bar{\bar{A}} \cdot \mathbf{u} = \begin{bmatrix} A_{11}u_1 & A_{12}u_2 & A_{13}u_3 \\ A_{21}u_1 & A_{22}u_2 & A_{23}u_3 \\ A_{31}u_1 & A_{32}u_2 & A_{33}u_3 \end{bmatrix}$$

The scalar (or double dot, or inner) product of two second order tensors, and may be defined differently depending on the convention chosen. Assuming $\bar{\bar{A}}$ is symmetric, but not necessarily tensor $\bar{\bar{B}}$, the following identity can be shown:

$$\bar{\bar{A}} : \bar{\bar{B}} = A_{ij}B_{ji} = A_{ij}B_{ij}$$

This is represented in expanded form:

$$\bar{\bar{A}} : \bar{\bar{B}} = A_{11}B_{11} + A_{22}B_{22} + A_{33}B_{33} + A_{12}(B_{12} + B_{21}) + A_{23}(B_{23} + B_{32}) + A_{13}(B_{13} + B_{31})$$

The divergence of a tensor vector product is found in the energy formulations, and gives viscous dissipation term. Assuming $\bar{\bar{A}}$ is a symmetrical second order tensor. The identity in its general form may be expressed as:

$$\nabla \cdot (\bar{\bar{A}} \cdot \mathbf{u}) = (\nabla \cdot \bar{\bar{A}}) \cdot \mathbf{u} + (\nabla \cdot \mathbf{u}) \cdot \bar{\bar{A}}$$

Due to symmetry, the above identity may be expressed as follows:

$$\nabla \cdot (\bar{\bar{A}} \cdot \mathbf{u}) = \mathbf{u} \cdot (\nabla \cdot \bar{\bar{A}}) + \bar{\bar{A}} : \nabla \mathbf{u}$$

which gives the result:

$$(\nabla \cdot \bar{\bar{A}}) \cdot \mathbf{u} = \bar{\bar{A}} : \nabla \mathbf{u}$$

A.2 Rate-of-Strain

The rate-of-strain describes the deformation of a fluid element in motion, and subsequently the forces between adjacent elements [67]. Considering any orthogonal coordinate system, the rate-of-strain is captured by a second order tensor. The tensor is comprised of terms accounting for isotropic expansion and strain dissociated with changes in volume. In its general form, the rate-of-strain tensor is given by Equation (A.1), which produces a symmetric 3×3 matrix.

$$\bar{\bar{e}} = \frac{1}{2} [\nabla \mathbf{V} + (\nabla \mathbf{V})^T] \quad (\text{A.1})$$

In Cartesian coordinates, the rate-of-strain tensor is expanded to give Equation (A.2).

$$\bar{\bar{e}}(x, y, z) = \begin{bmatrix} e_{xx} & e_{xy} & e_{xz} \\ e_{yx} & e_{yy} & e_{yz} \\ e_{zx} & e_{zy} & e_{zz} \end{bmatrix} = \frac{1}{2} \begin{bmatrix} 2 \frac{\partial V_x}{\partial x} & \frac{\partial V_x}{\partial y} + \frac{\partial V_y}{\partial x} & \frac{\partial V_x}{\partial z} + \frac{\partial V_z}{\partial x} \\ \cdot & 2 \frac{\partial V_y}{\partial y} & \frac{\partial V_y}{\partial z} + \frac{\partial V_z}{\partial y} \\ \cdot & \cdot & 2 \frac{\partial V_z}{\partial z} \end{bmatrix} \quad (\text{A.2})$$

This may be rewritten more simply in Einstein notation, Equation (A.3).

$$e_{ij} = \frac{1}{2} \left(\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} \right) \quad (\text{A.3})$$

The cylindrical coordinate system used in this work has the azimuthal angle θ rotating about the same z axis in Cartesian coordinates. The rate-of-strain tensor in cylindrical coordinates is given by Equation (A.4).

$$\bar{\bar{e}}(r, \theta, z) = \begin{bmatrix} e_{rr} & e_{r\theta} & e_{rz} \\ e_{\theta r} & e_{\theta\theta} & e_{\theta z} \\ e_{zr} & e_{z\theta} & e_{zz} \end{bmatrix} = \frac{1}{2} \begin{bmatrix} 2 \frac{\partial V_r}{\partial r} & r \frac{\partial}{\partial r} \left(\frac{V_\theta}{r} \right) + \frac{1}{r} \frac{\partial V_r}{\partial \theta} & \frac{\partial V_r}{\partial z} + \frac{\partial V_z}{\partial r} \\ \cdot & 2 \left(\frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{V_r}{r} \right) & \frac{1}{r} \frac{\partial V_z}{\partial \theta} + \frac{\partial V_\theta}{\partial z} \\ \cdot & \cdot & 2 \frac{\partial V_z}{\partial z} \end{bmatrix} \quad (\text{A.4})$$

In examining the rate-of-strain throughout a flow-field, it is difficult to consider all nine terms of the second order tensor. Instead, it is useful to look at the scalar trace and invariants of the tensor. Using Einstein notation, the second and third invariants of the rate-of-strain tensor are [68]:

$$II_{\bar{e}} = -\frac{1}{2}\bar{e}_{ij}\bar{e}_{ij} \quad (\text{A.5})$$

$$III_{\bar{e}} = -\frac{1}{3}\bar{e}_{ij}\bar{e}_{jk}\bar{e}_{ki} \quad (\text{A.6})$$

Equation (A.5) is relevant in this work as it is proportional to local kinetic energy viscous dissipation rates [68]. The invariant of Equation (A.6) correlates to strain skewness, which also contributes to viscous dissipation. This work makes use of Equation (A.5) in examining RANS CFD resolved flows to determine consistency in the use of Boussinesq's approximation within the energy-based methods.

A.3 Deviatoric Stress

The overall stress on a fluid element is comprised of two contributions, one of which is isotropic (uniform in all orientations, or spherically symmetrical) and another non-isotropic. The latter is the deviatoric stress tensor, which is zero in a stationary fluid and is thus only relevant in moving fluids. This deviatoric stress is also the non-isotropic portion of the rate-of-strain, and the relation between the two is given by Equation (A.7).

$$\bar{d} = 2\mu \left[\bar{e} - \frac{1}{3}(\nabla \cdot \mathbf{V})\bar{I} \right] \quad (\text{A.7})$$

A deviatoric stress tensor with zero trace, as per Stokes' Hypothesis [69], is obtained by the inclusion of the divergence term $(\nabla \cdot \mathbf{V})$ in Equation (A.7). Drela [22], refers to this deviatoric stress tensor as the viscous stress tensor. The deviatoric stress tensor in Cartesian coordinates is obtained from Equations (A.2) and (A.7) to give Equation (A.8), with the divergence term given in Equation (A.9).

$$\bar{d}(x, y, z) = \begin{bmatrix} d_{xx} & d_{xy} & d_{xz} \\ d_{yx} & d_{yy} & d_{yz} \\ d_{zx} & d_{zy} & d_{zz} \end{bmatrix} = \mu \begin{bmatrix} 2\frac{\partial V_x}{\partial x} - \frac{2}{3}\lambda & \frac{\partial V_x}{\partial y} + \frac{\partial V_y}{\partial x} & \frac{\partial V_x}{\partial z} + \frac{\partial V_z}{\partial x} \\ \cdot & 2\frac{\partial V_y}{\partial y} - \frac{2}{3}\lambda & \frac{\partial V_y}{\partial z} + \frac{\partial V_z}{\partial y} \\ \cdot & \cdot & 2\frac{\partial V_z}{\partial z} - \frac{2}{3}\lambda \end{bmatrix} \quad (\text{A.8})$$

$$\lambda(x, y, z) = (\nabla \cdot \mathbf{V}) = \frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} + \frac{\partial V_z}{\partial z} \quad (\text{A.9})$$

The deviatoric stress tensor in Cartesian coordinates is obtained from Equations (A.2) and (A.7) to give Equation (A.8), with the divergence term given in Equation (A.9).

$$\bar{\bar{d}}(r, \theta, z) = \begin{bmatrix} d_{rr} & d_{r\theta} & d_{rz} \\ d_{\theta r} & d_{\theta\theta} & d_{\theta z} \\ d_{zr} & d_{z\theta} & d_{zz} \end{bmatrix} = \mu \begin{bmatrix} 2 \frac{\partial V_r}{\partial r} - \frac{2}{3} \lambda & r \frac{\partial}{\partial r} \left(\frac{V_\theta}{r} \right) + \frac{1}{r} \frac{\partial V_r}{\partial \theta} & \frac{\partial V_r}{\partial z} + \frac{\partial V_z}{\partial r} \\ \cdot & 2 \left(\frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{V_r}{r} \right) - \frac{2}{3} \lambda & \frac{1}{r} \frac{\partial V_z}{\partial \theta} + \frac{\partial V_\theta}{\partial z} \\ \cdot & \cdot & 2 \frac{\partial V_z}{\partial z} - \frac{2}{3} \lambda \end{bmatrix} \quad (\text{A.10})$$

$$\lambda(r, \theta, z) = (\nabla \cdot \mathbf{V}) = \frac{1}{r} \frac{\partial}{\partial r} (r V_r) + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} \quad (\text{A.11})$$

A.4 Boussinesq Eddy Viscosity Assumption

The Boussinesq eddy viscosity assumption, along with two equation turbulence models, solves the Reynolds averaging closure problem by assuming proportionality between the Reynolds stresses and the mean strain rate tensor via the eddy (or turbulent) viscosity [70]. A familiar form, in Einstein notation and applicable to Cartesian coordinates, is given by Equation (A.12).

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (\text{A.12})$$

Evidently, the Reynolds stresses can be rewritten in a similar generalised form, Equation (A.13) as to the deviatoric stress tensor, Equation (A.7).

$$\bar{\bar{d}}_R = 2\mu_t \left[\bar{\bar{e}} - \frac{1}{3} (\nabla \cdot \mathbf{V}) \bar{\bar{I}} \right] - \frac{2}{3} \rho k \bar{\bar{I}} \quad (\text{A.13})$$

Subsequently, the viscous shear tensor (taking into account Reynolds stresses) can be written as the sum of Equations (A.7) and (A.13). Thus the generalised form of the viscous stress tensor used in this work is given by Equation (A.14).

$$\bar{\bar{\tau}} = 2(\mu + \mu_t) \left[\bar{\bar{e}} - \frac{1}{3} (\nabla \cdot \mathbf{V}) \bar{\bar{I}} \right] - \frac{2}{3} \rho k \bar{\bar{I}} \quad (\text{A.14})$$

With reference to Equation (A.9), the viscous stress tensor in Cartesian coordinates is given by Equation (A.15).

$$\begin{aligned} \bar{\bar{\tau}}(x, y, z) &= \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix} \\ \tau_{xx} &= (\mu + \mu_t) \left[2 \frac{\partial V_x}{\partial x} - \frac{2}{3} \lambda \right] - \frac{2}{3} \rho k, \quad \tau_{xy} = \tau_{yx} = (\mu + \mu_t) \left(\frac{\partial V_x}{\partial y} + \frac{\partial V_y}{\partial x} \right) \\ \tau_{yy} &= (\mu + \mu_t) \left[2 \frac{\partial V_y}{\partial y} - \frac{2}{3} \lambda \right] - \frac{2}{3} \rho k, \quad \tau_{yz} = \tau_{zy} = (\mu + \mu_t) \left(\frac{\partial V_y}{\partial z} + \frac{\partial V_z}{\partial y} \right) \\ \tau_{zz} &= (\mu + \mu_t) \left[2 \frac{\partial V_z}{\partial z} - \frac{2}{3} \lambda \right] - \frac{2}{3} \rho k, \quad \tau_{zx} = \tau_{xz} = (\mu + \mu_t) \left(\frac{\partial V_x}{\partial z} + \frac{\partial V_z}{\partial x} \right) \end{aligned} \quad (\text{A.15})$$

With reference to Equation (A.11), the viscous stress tensor in Cartesian coordinates is given by Equation (A.16).

$$\begin{aligned} \bar{\bar{\tau}}(r, \theta, z) &= \begin{bmatrix} \tau_{rr} & \tau_{r\theta} & \tau_{rz} \\ \tau_{\theta r} & \tau_{\theta\theta} & \tau_{\theta z} \\ \tau_{zr} & \tau_{z\theta} & \tau_{zz} \end{bmatrix} \\ \tau_{rr} &= (\mu + \mu_t) \left[2 \frac{\partial V_r}{\partial r} - \frac{2}{3} \lambda \right] - \frac{2}{3} \rho k, \quad \tau_{r\theta} = \tau_{\theta r} = (\mu + \mu_t) \left(r \frac{\partial}{\partial r} \left(\frac{V_\theta}{r} \right) + \frac{1}{r} \frac{\partial V_r}{\partial \theta} \right) \\ \tau_{\theta\theta} &= (\mu + \mu_t) \left[2 \left(\frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{V_r}{r} \right) - \frac{2}{3} \lambda \right] - \frac{2}{3} \rho k, \quad \tau_{\theta z} = \tau_{z\theta} = (\mu + \mu_t) \left(\frac{1}{r} \frac{\partial V_z}{\partial \theta} + \frac{\partial V_\theta}{\partial z} \right) \\ \tau_{zz} &= (\mu + \mu_t) \left[2 \frac{\partial V_z}{\partial z} - \frac{2}{3} \lambda \right] - \frac{2}{3} \rho k, \quad \tau_{zr} = \tau_{rz} = (\mu + \mu_t) \left(\frac{\partial V_r}{\partial z} + \frac{\partial V_z}{\partial r} \right) \end{aligned} \quad (\text{A.16})$$

A.5 Viscous Forces, Work and Dissipation

A.5.1 Viscous Stress Forces

The viscous force per unit area, along any point along a surface (be it a closed wall surface or open flux surface), is obtained from the dot product between the viscous stress tensor and the normal vector in the appropriate coordinate system (i.e. $\bar{\bar{\tau}} \cdot \hat{n}$). In Cartesian coordinates:

$$\bar{\bar{\tau}} \cdot \hat{n}(x, y, z) = \begin{bmatrix} \tau_{xx} n_x + \tau_{xy} n_y + \tau_{xz} n_z \\ \tau_{yx} n_x + \tau_{yy} n_y + \tau_{yz} n_z \\ \tau_{zx} n_x + \tau_{zy} n_y + \tau_{zz} n_z \end{bmatrix} \quad (\text{A.17})$$

In cylindrical coordinates:

$$\bar{\tau} \cdot \hat{n}(r, \theta, z) = \begin{bmatrix} \tau_{rr} n_r + \tau_{r\theta} n_\theta + \tau_{rz} n_z \\ \tau_{\theta r} n_r + \tau_{\theta\theta} n_\theta + \tau_{\theta z} n_z \\ \tau_{zr} n_r + \tau_{z\theta} n_\theta + \tau_{zz} n_z \end{bmatrix} \quad (\text{A.18})$$

A.5.2 Viscous Boundary Work

The work done on the boundary of a control volume due to viscous stresses is given by the dot product of the viscous stress tensor, local velocity vector and the normal vector to the control volume surface, i.e. $(\bar{\tau} \cdot \mathbf{V}) \cdot \hat{n}$, given that the viscous stress tensor is symmetrical. In Cartesian coordinates:

$$\begin{aligned} (\bar{\tau} \cdot \mathbf{V}) \cdot \hat{n}(x, y, z) &= (\tau_{xx} V_x + \tau_{xy} V_y + \tau_{xz} V_z) n_x + \cdots \\ &= (\tau_{yx} V_x + \tau_{yy} V_y + \tau_{yz} V_z) n_y + \cdots \\ &= (\tau_{zx} V_x + \tau_{zy} V_y + \tau_{zz} V_z) n_z \end{aligned} \quad (\text{A.19})$$

In cylindrical coordinates:

$$\begin{aligned} (\bar{\tau} \cdot \mathbf{V}) \cdot \hat{n}(r, \theta, z) &= (\tau_{rr} V_r + \tau_{r\theta} V_\theta + \tau_{rz} V_z) n_r + \cdots \\ &= (\tau_{\theta r} V_r + \tau_{\theta\theta} V_\theta + \tau_{\theta z} V_z) n_\theta + \cdots \\ &= (\tau_{zr} V_r + \tau_{z\theta} V_\theta + \tau_{zz} V_z) n_z \end{aligned} \quad (\text{A.20})$$

A.5.3 Viscous Dissipation

The term representing viscous dissipation within the energy formulations may be rewritten in expanded form in Cartesian and Cylindrical coordinates using the following relation (due to the symmetry of the viscous stress tensor):

$$(\nabla \cdot \bar{\tau}) \cdot \mathbf{V} = \bar{\tau} : \nabla \mathbf{V} = \bar{\tau} : (\nabla \mathbf{V})^T \quad (\text{A.21})$$

In Cartesian coordinates

$$\begin{aligned} (\nabla \cdot \bar{\tau}) \cdot \mathbf{V} &= \bar{\tau}_{xx} \frac{\partial V_x}{\partial x} + \bar{\tau}_{yy} \frac{\partial V_y}{\partial y} + \bar{\tau}_{zz} \frac{\partial V_z}{\partial z} + \bar{\tau}_{xy} \left(\frac{\partial V_x}{\partial y} + \frac{\partial V_y}{\partial x} \right) + \\ &\quad \bar{\tau}_{xz} \left(\frac{\partial V_x}{\partial z} + \frac{\partial V_z}{\partial x} \right) + \bar{\tau}_{yz} \left(\frac{\partial V_y}{\partial z} + \frac{\partial V_z}{\partial y} \right) \end{aligned} \quad (\text{A.22})$$

In cylindrical coordinates:

$$\begin{aligned} (\nabla \cdot \bar{\tau}) \cdot \mathbf{V} &= \bar{\tau}_{rr} \frac{\partial V_r}{\partial r} + \bar{\tau}_{\theta\theta} \left(\frac{1}{r} \frac{\partial V_\theta}{\partial \theta} - \frac{1}{r} V_r \right) + \bar{\tau}_{zz} \frac{\partial V_z}{\partial z} + \\ &\quad \bar{\tau}_{r\theta} \left(\frac{\partial V_\theta}{\partial r} + \frac{1}{r} \frac{\partial V_r}{\partial \theta} - \frac{1}{r} V_\theta \right) + \bar{\tau}_{rz} \left(\frac{\partial V_z}{\partial r} + \frac{\partial V_r}{\partial z} \right) + \\ &\quad \bar{\tau}_{\theta z} \left(\frac{1}{r} \frac{\partial V_z}{\partial \theta} + \frac{\partial V_\theta}{\partial z} \right) \end{aligned} \quad (\text{A.23})$$

Appendix B

Drag Estimation

The aim of the work presented in this appendix was to find suitable empirical drag build-up correlations which allow for reverse engineering the baseline aircraft component's geometries. Rolls-Royce supplied the author with a set of drag polars to describe a state-of-the-art business jet. This is an unknown fictional aircraft, and therefore nothing of the geometry was known. In order to reverse engineer the geometries, different empirical methods were used within a global optimiser, which was allowed to vary the different geometries of various components. This appendix describes that process and the outcomes achieved.

At a conceptual design level (or when detailed airframe geometry is unknown) reasonable drag estimates from basic aircraft geometries are required for aircraft performance calculations. In the absence of aircraft geometry details, high-fidelity drag estimation methods are impractical and instead empirical formulations are required. The contribution of drag can be subdivided in many different ways, but for this work that subdivision was chosen to be consistent with the baseline aircraft data supplied. Thus it is assumed that drag is comprised of the following terms:

- Lift independent (skin friction) C_{D_0}
- Lift dependent (induced) C_{D_i}
- Wave drag C_{D_W}
- Trim drag C_{D_T}
- Combined term for excrescence and interference

There are numerous methods for semi-empirical based estimations of each of the above terms. This appendix makes reference to Sforza [71], who has incorporated a comprehensive summary and comparison of the available methods.

B.1 Lift Independent Drag

The lift independent drag (or commonly referred to as skin-friction drag) may be calculated from empirical formulations in what is referred to as drag build-up methods, which is a sum of the drag of individual components of the airframe. For each component i , C_{D_0} is determined as a function of a skin friction factor C_f , form factor F , interference factor Q , the surface's wetted area S_{wet} and an appropriate reference area S_{ref} :

$$C_{D_0,i} = \left(\frac{C_f F Q S_{wet}}{S_{ref}} \right)_i \quad (\text{B.1})$$

The skin-friction coefficient for each component is calculated from empirical formulations, which are functions of the component's Reynolds Re and Mach M numbers. Different empirical relations, or area-weighted combinations thereof, are used to account for laminar or turbulent flow over the component's surfaces. The geometrical form factor F and interference factor Q are obtained for each component using empirical relations unique to each particular component. The total profile drag is then simply obtained as the sum of the profile drag for each component, i.e.:

$$C_{D_0} = \sum C_{D_0,i} \quad (\text{B.2})$$

There are a number of different empirical relations to describe the skin-friction coefficient from flat plate theory, which are also dependent on the Reynold's number range being considered. Additionally, there are also methods for the inclusion of compressibility effects. These methods are reviewed in Reference [71]. In fitting the aircraft data supplied, the different methods in [71] were considered, and the best match was found to come from the Schlichting-Prandtl formula for turbulent flows within the corresponding Reynolds number range:

$$C_{F,turb} = \frac{0.455}{(\log_{10} Re_x)^{2.58}} \quad 10^6 < Re_x < 10^9 \quad (\text{B.3})$$

At lower Mach numbers, outside of the above Reynolds range, the code assumes a laminar BL with the friction coefficient given by the well-known form:

$$C_{F,lam} = \frac{1.328}{\sqrt{Re_x}} \quad (\text{B.4})$$

Introducing compressibility effects worsened the ability of the optimiser to find a geometry to match the aerodynamic curves provided. Therefore such terms were omitted and it was assumed that the effects of compressibility would solely come from the method used to account for wave drag. A list of different form factors may be found in Reference [71], and were compared to examine the fit of the data. However, the fit appeared to be insensitive to the form factors used, and the geometries were merely altered to give a reasonable fit. That is to say, the different form factors changed offsets but not shapes of the drag polar trends. The different form factors all appeared to give reasonable

geometries. Therefore, because the actual geometry of the aircraft was unknown, it was not possible to determine which of the form factors were best suited. This work thus assumed the use of those provided by Jenkinson *et al.* [72], in order to remain consistent with Cranfield's in-house code HERMES [73], should it be required to later verify the performance code created for this work (see Appendix E).

The form and interference factors for each of the aircraft components are listed below, as presented by Jenkinson *et al.* [72]:

B.1.1 Fuselage

The fuselage form factor is dependent on its length, midsection width and height. In this work, 2D antisymmetric CFD is performed on the fuselage, and therefore it is assumed that the fuselage has a circular cross-section.

$$F = 1 + \frac{2.2}{\lambda^{1.5}} - \frac{0.9}{\lambda^3}$$

$$\lambda = \frac{L_{\text{fuselage}}}{[4A/\pi]^{0.5}}$$

$$Q = 1.0$$

B.1.2 Wing and Tail Surfaces

The wing's form factor is based on its sweepback angle at 50% of the chord, as well as the thickness-to-chord ratio (t/c). The thickness-to-chord ratio is not specified, so an average value is used.

$$F = (F^* - 1) \cos^2 \Lambda_{0.5c} + 1$$

$$F^* = 1 + 3.3 (t/c) - 0.008 (t/c)^2 + 27.0 (t/c)^3$$

$$Q = 1.0$$

the form factor for tail surfaces uses the following substitutions:

$$F^* = 1 + 3.52 (t/c)$$

$$Q = 1.2$$

B.1.3 Nacelle

The engine nacelle drag for a fuselage installation is:

$$FQ = 1.5$$

drag sensitivities to Mach and lift variations. The methods available in literature have also been summarised in Reference [75], with a detailed comparison and will therefore not be described further here. Nonetheless, the different approaches are used, from area ruling to the more common (and practical) analytical empirical formulations relating drag rise to a function of the Mach number in relation to either the critical Mach or drag divergence Mach numbers. The critical Mach number is that at which formation of shock waves begin to appear, but the overall drag on the aircraft is not yet increased by a significant amount. The divergence number is the Mach above the critical value where the drag resulting from the formation of shocks becomes dominant and the aircraft's overall drag begins to rise steeply. There are two definitions of the drag divergent number; 1) Boeing's definition is where the drag rise has reached 20 counts and 2) the Douglas definition is based on the rate of drag rise with Mach (i.e. $dC_D/dM = 0.1$). Each of the open literature methods relies on either one of these definitions. The different methods, as listed by Reference [71], which could be applied analytically (this was required for computational reasons when using the optimiser) were implemented, but none were able to capture the shapes of the data provided. A different method was then attempted, which is based on an extension of Korn's equation (Equation B.8) and Lock's 4th power law (Equation B.9) [76]. The method assumes that wave drag is purely a result of that incurred on the wing and does not include any contributions from other components. The extension of Korn's equation is used to determine the drag divergence number (Douglas definition) based on wing sweep, thickness-to-chord ratio and a technology factor κ_A (0.87 and 0.95 for conventional and supercritical airfoils respectively).

$$M_{DD} \cos \Lambda_{1/2} + \frac{C_L}{10 \cos^2 \Lambda_{1/2}} + \frac{t/c}{\cos \Lambda_{1/2}} = \kappa_A \quad (\text{B.8})$$

$$C_{D_W} = 20 (M - M_{cr})^4 \quad M > M_{cr} \quad (\text{B.9})$$

However, this improved method was still unable to capture the wave drag of the data supplied. A new method, which was vaguely referred to as the Rolls-Royce compressibility method was found in the Piano user manual [77]. It provided a default curve for estimating wave drag as a function $M - M_{DD}$, where the definition of drag rise is that of Boeings. It was eluded to that sensitivities to lift and Mach number could be applied, but did not mention how this was done. Upon inspection of the default curve, it was found that it could be fitted using a polynomial. For physical representativeness the curve was fitted with a 4th degree polynomial to relate to Lock's 4th power law mentioned above. This unfortunately still did not match the data, but did seem to capture the trends more accurately. As a result, it was assumed that the sensitivities to lift and Mach, referred to in the Piano manual, would result in an effective modification of the polynomial's coefficients. Thus to match the data, the optimiser was allowed to vary the polynomial coefficients to account for the sensitivities. This was coupled with the Korn equation extension above (Equation B.8) and a conversion from the Douglas to Boeing definition. The optimiser was also permitted to vary Korn's technology factor κ_A . This resulted in a curve which modelled the accuracy of the data provided and related the compressibility to the geometry of the aircraft via Equation B.8.

B.4 Drag Polar Matching

Rolls-Royce provided performance data of a fictitious aircraft, representative of a state-of-the-art business jet. Aerodynamic performance, aircraft weights and engine performance data were supplied. To use this baseline aircraft for the subsequent BLI studies it was necessary to reverse engineer aircraft geometries and engine parameters to match the data provided. Open literature information available on the Gulfstream G650, powered by the Rolls-Royce BR725, was used as guidance for the data matching. This section briefly describes how the aerodynamic data was matched.

The aircraft data supplied gives the lift-to-drag ratio as a function of lift coefficient and Mach number at various altitudes. The drag is dependent on the aircraft geometry, and thus for a given C_L the drag coefficient C_D is known. In addition, the drag breakdown of the aircraft was supplied at a given design cruise condition. In order to determine the aircraft geometry corresponding to the data, a drag build-up method was coupled to a global optimisation procedure so that a least-square's fit of the data could be obtained. The drag breakdown at design cruise was used to help constrain the problem, and information available on the G550 (owing to the detailed sample model provided by Piano [78]) and G650 aircraft were used to give guidance as to the possible ranges of geometric parameters. From the single drag break-down point given, the terms of drag accounted for in the data were identified to include:

- Lift independent C_{D_0}
- Lift dependent (induced) C_{D_I}
- Wave drag C_{D_W}
- Trim drag C_{D_T}
- Combined term for excrescence and interference

The single drag break-down point, FL450 M0.85, also listed values for the following airframe components:

- Wing
- Winglet
- Fuselage
- Stabilizer
- Fin
- Nacelles
- Pylons

There are numerous semi-empirical drag build-up methods available as well as numerous wave drag estimations techniques as already mentioned. Therefore, the matching procedure began with first identifying appropriate semi-empirical methods for fitting the data at speeds below the drag divergence Mach. This then allows for easier identification of an appropriate wave drag estimation method to match the data at higher speeds. 19 geometric parameters were identified as a general requirement of the drag build-up methods, and are as follows:

- Trapezoidal area A (wing, stabilizer and fin)
- Aspect ratio AR (wing, stabilizer and fin)
- Thickness-to-chord ratio t/c (wing, stabilizer and fin)
- Sweep angle Λ (wing, stabilizer and fin)
- Taper ratio λ (wing, stabilizer and fin)
- Diameter (fuselage and engine nacelles)
- Length (fuselage and engine nacelles)

Following an examination of the possible semi-empirical methods, an accurate fit on the data was achieved using a slight modification to the method of Jenkinson *et. al* [72]. The modification required the removal of the compressibility term added to the Prandtl-Schlichting skin friction coefficient, altering it from Equation B.10 to Equation B.11.

$$C_f = \frac{0.455}{(\log Re_c)^{2.58} (1 + 0.144M^2)^{0.65}} \quad (B.10)$$

$$C_f = \frac{0.455}{(\log Re_c)^{2.58}} \quad (B.11)$$

An example of results obtained from the optimiser in low speed are summarised in Figure B.1. However, the optimiser would have gone through multiple possible geometric combinations, and thus there would be multiple geometric solutions that produce drag close to that of the low speed data provided. Needless to say, the solution settled on won't necessarily result in a geometry that estimates accurately the wave drag contributions at higher speeds.

The next step of the matching procedure was to use the optimiser to try and match the inclusion of wave drag. Only the high speed portion of the data was initially fitted to identify an appropriate method for estimating wave drag. Having identified a method that accurately captures the aerodynamics at low speed and high speed regions, the entire data set was then matched to determine the aircraft geometry. The final results of the overall fit are illustrated in Figure B.3, with errors (at the drag break-down point given, FL450 M0.85) presented in Table B.1. As is evident, the method accurately captures the drag divergence point for each C_L and M combination and also the respective

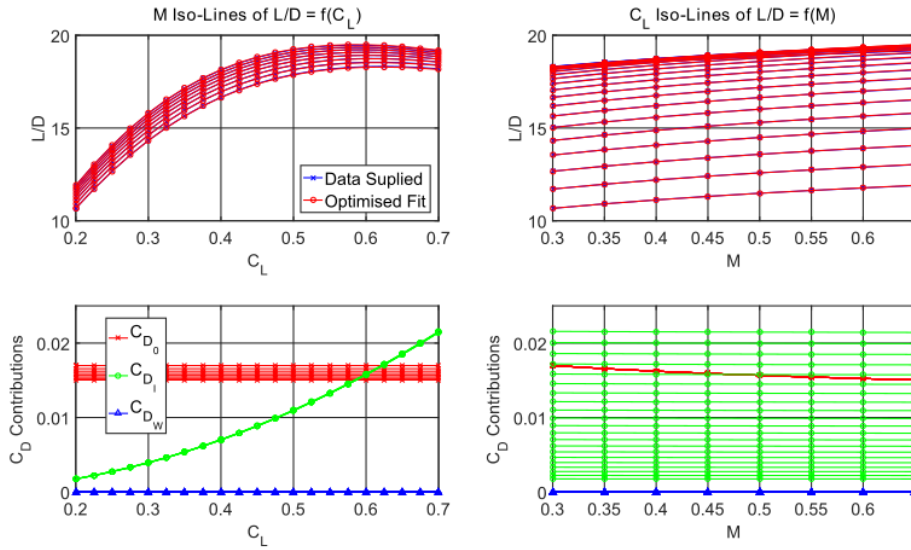


Figure B.1: Baseline Aerodynamics Matching at Low Speeds. (portion of results shown corresponds to FL450)

sensitivities as the flight condition moves further into the compressible region. The resultant geometry is therefore concluded to be equivalent to that of the data supplied.

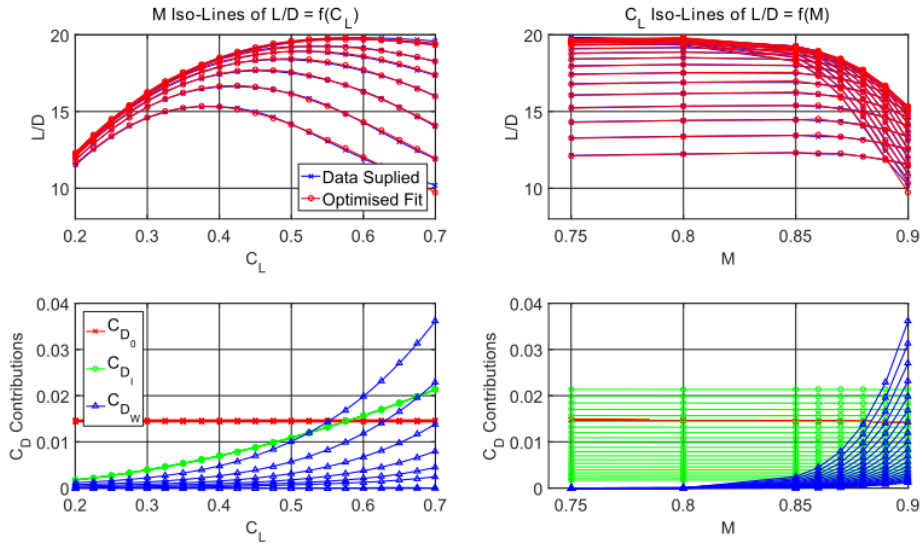


Figure B.2: Baseline Aerodynamics Matching at High Speeds. (portion of results shown corresponds to FL450)

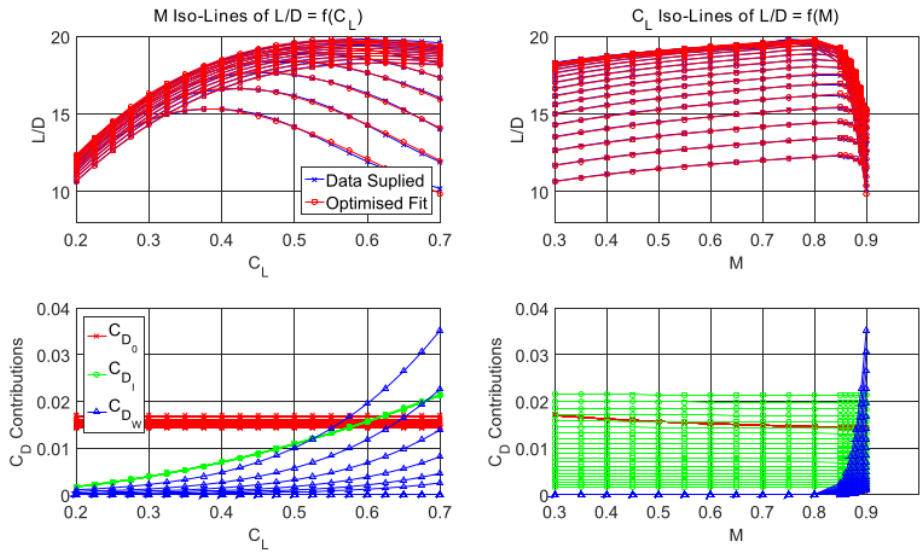


Figure B.3: Baseline Aerodynamics Matching at All Speeds. (portion of results shown corresponds to FL450)

Table B.1: Error's in drag contributions of the matched geometry at the given drag break-down point of FL450 M0.85

CD0 . Sref (m^2)	Data	Calculated	Error (%)
Fuselage	0.404	0.424	4.93
Nacelles	0.164	0.1716	4.68
Wing + Winglets	0.7812	0.7549	-3.37
Stabilizer	0.1439	0.1439	0
Fin	0.0962	0.1006	4.67
Interference	0.1573	0.153	-2.75
Total	1.7466	1.748	0.08
C_{D_0}	0.0145	0.0145	0.09
C_{D_I}	0.0065	0.0065	-0.58
C_{D_W}	2.50E-04	2.50E-04	0
C_D	0.0212	0.0212	-0.12
C_L	0.3852	0.3852	0
L/D	18.16	18.18	0.12
M_{DD}	0.887	0.887	0

Appendix C

Isolated Fuselage Aerodynamic Matching

In Appendix B, analytical empirical formulations for calculating aircraft aerodynamics (as part of a component drag build-up method) were used to reverse engineer the baseline aircraft geometry from aerodynamic data supplied. The drag obtained from the empirical model is used to set the overall thrust requirements of the aircraft, which in turn also sets the baseline engine performance at cruise. A retrofit type approach is taken to include BLI propulsion on the fuselage of the aircraft, and therefore any changes in fuselage assembly force and power affect the thrust and power off-take requirements for the main turbofan engines, and hence the benefit in SFC achieved. Following this explanation, it is necessary to ensure that the CFD studies of BLI fuselage performance are based on a fuselage geometry consistent with that calculated in the drag build-up method. To do so, it is necessary to find a geometry producing a drag matching the empirical formulation found in Appendix B. This appendix addresses how the fuselage geometry was found to best match the empirical analytical drag estimation.

C.1 Model Setup

It was assumed that the fuselage may be represented by an axisymmetric body divided up into three parts: 1) an elliptical nose, 2) cylindrical mid-body and a 3) tail-cone, whose side-walls comprises a revolved circular arc. This geometry is depicted in Figure C.1, where an ellipse fineness ratio determines the nose length and the tail-cone length is given as a ratio of the overall length:

$$\Pi_{\text{nose}} = \frac{L_{\text{nose}}}{R_{\text{fuselage}}} \quad (\text{C.1})$$

$$\Pi_{\text{tail-cone}} = \frac{L_{\text{tail-cone}}}{L_{\text{fuselage}}} \quad (\text{C.2})$$

The computational flow domain around the fuselage is depicted in Figure C.2. The

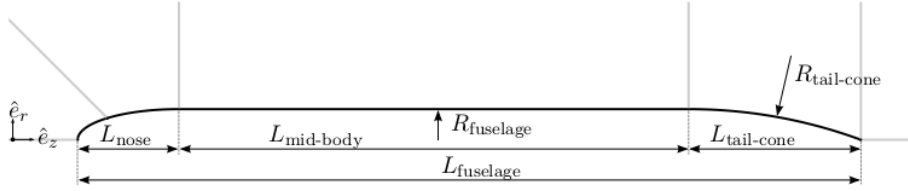


Figure C.1: Isolated fuselage geometry: representative axisymmetric body of revolution

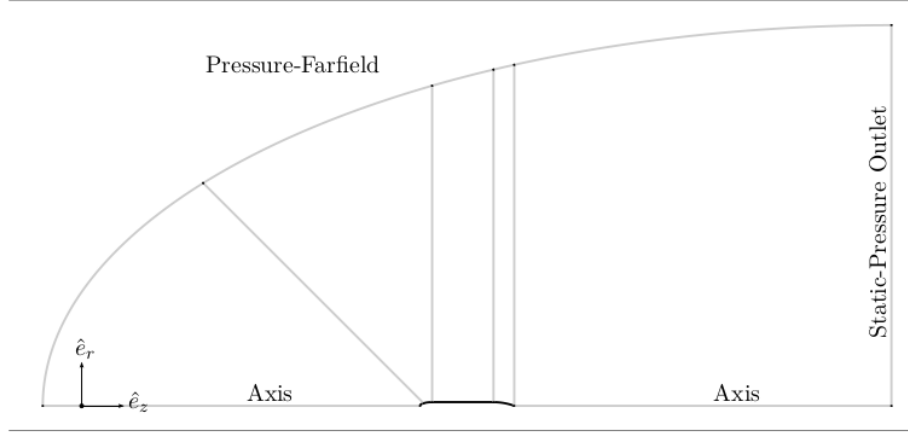


Figure C.2: Isolated fuselage computational domain with boundary conditions

domain is axisymmetric, and its shape chosen for blocking purposes where an L-grid was placed around the fuselage. Fluent's Pressure-Farfield boundary condition is used to model the free-stream conditions of the model. A Static-Pressure Outlet is utilized at the trefftz plane. As part of a grid convergence study, the distances at which these boundaries should be placed are also examined (i.e. such that they are not too close and thus affect the outcome of the solution). For the reasons explained in Section 5.2, $k - \omega$ SST was the selected turbulence model, along with GGCB gradient scheme implemented in Fluent's coupled pressure-based solver.

C.2 Grid Independence Study

A grid convergence study was performed to determine appropriate domain sizes and mesh densities. The grid convergence studies presented in this work follow the procedures recommended by the Journal of Fluids Engineering [79], and with guidance from Roache [80]. A structured meshing approach was taken, using an L-grid to mesh the fuselage's geometry in ICEM [81]. An example of the mesh is depicted in Figure C.3. Tables C.1

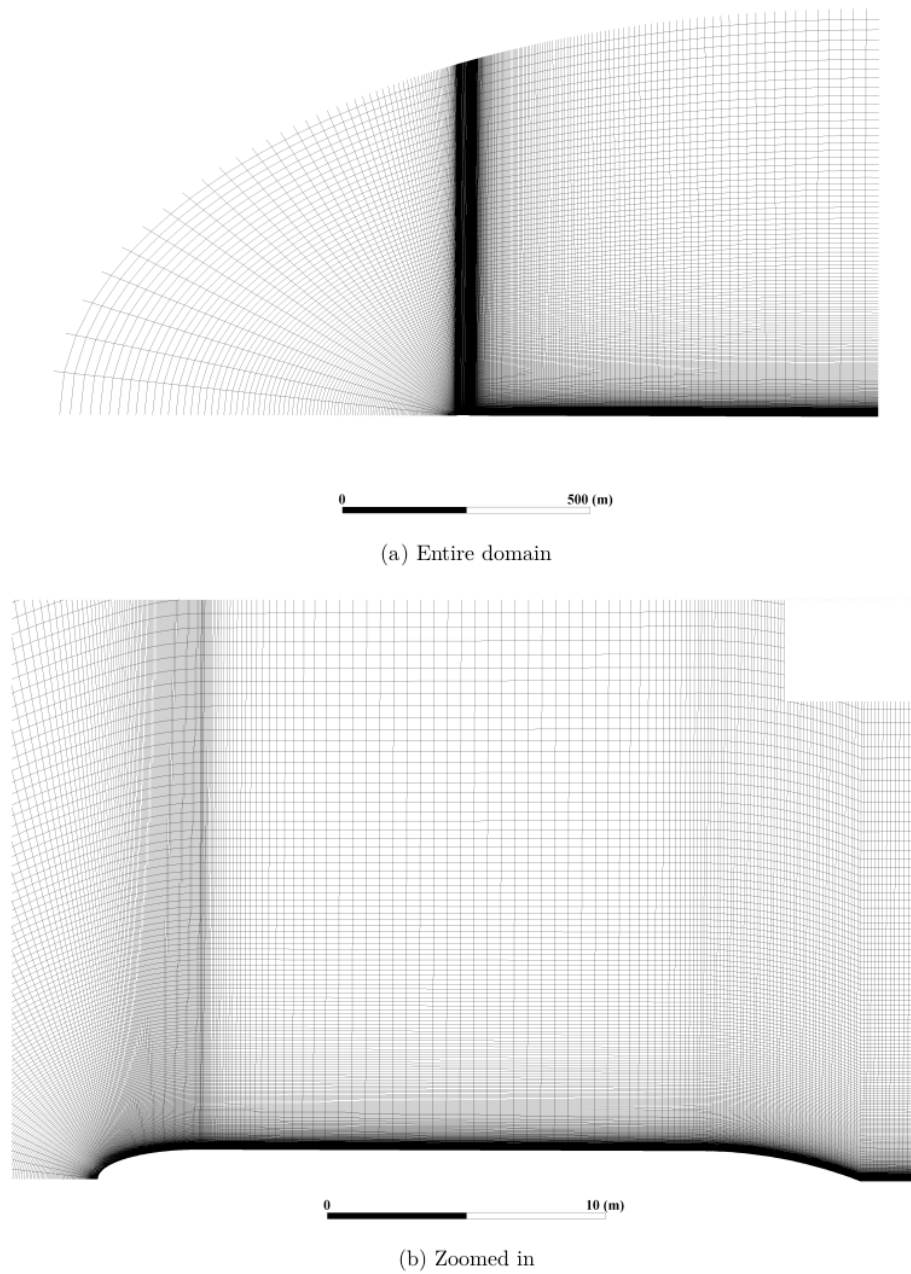


Figure C.3: Isolated fuselage structured mesh example from L-grid

to C.3 detail the number of cells, combined 2D volume and mesh densities for the cases considered. The domain size was specified as a distance away from the fuselage, measured in number of fuselage lengths. A fine, medium and coarse mesh was considered for each domain size.

Table C.1: Isolated fuselage GCI study: number of cells N

Domain Size	Fine	Medium	Coarse
$20 \times L_{\text{fuselage}}$	278355	123551	54725
$30 \times L_{\text{fuselage}}$	269784	119880	53132
$40 \times L_{\text{fuselage}}$	287347	127868	56750

Table C.2: Isolated fuselage GCI study: domain 2D volume $\sum_{i=1}^N \nu_{i,2D}$

Domain Size	Fine	Medium	Coarse
$20 \times L_{\text{fuselage}}$	2373.2	2380.5	2392.3
$30 \times L_{\text{fuselage}}$	2362.3	2360.9	2317.1
$40 \times L_{\text{fuselage}}$	2362.1	2361.2	2327.4

Table C.3: Isolated fuselage GCI study: mesh density $\left(\sum_{i=1}^N \nu_{i,2D}\right)^{1/2}$ with refinement ratio $(\cdot)$

Domain Size	Fine	Medium	Coarse
$20 \times L_{\text{fuselage}}$	0.0923 (1.50)	0.1388 (1.50)	0.2091
$30 \times L_{\text{fuselage}}$	0.0936 (1.50)	0.1403 (1.49)	0.2088
$40 \times L_{\text{fuselage}}$	0.0907 (1.50)	0.1359 (1.49)	0.2025

In each case the first cell wall height was kept constant to ensure an average $y^+ < 1$. The wake at the fuselage's trailing edge is used to perform the GCI study. The wake

profile properties for each grid, including Richardson extrapolations, are depicted in Figure C.4. Uncertainties in these properties, calculated using the GCI, are presented in Figure C.5. These figures clearly show that the grids produce the same flow-field within very small errors of one another. Subsequent cases are based on the medium mesh with a domain size equal to 30 times the fuselage length.

C.3 Drag Matching Procedure and Results

In matching the CFD calculated drag to the analytical model, the overall length and maximum diameter of the fuselage was fixed. The noses' elliptical fineness ratio was altered to match the trends of Mach dependent drag (due to compressibility effects) predicted by the empirical analytical formulation. The nose elliptical fineness ratio governs the flow acceleration and final speed as it approaches the constant cylindrical section of the mid-body, thus determining drag rise with Mach number. The length of the tail-cone, on the other hand, dictates the diffusion over the back of the fuselage and hence drag due to flow separation and wake. Having matched Mach dependent drag rise trends by manipulating the nose elliptical fineness ratio, the resulting drag offset could be achieved by manipulating the tail-cone length. It is noted that variations in both parameters obviously lead to variations in the total surface area of the fuselage and hence how much skin friction takes place. Additionally, impacts on drag contributions are not solely independent and so the matching procedure did require some trial-and-error along with fine tuning.

Each geometry combination was examined over a range of cruise altitudes (FL390, FL410, FL450, FL490) and Mach ranges (0.7 0.8 0.85 0.9)¹ The standard deviation between the CFD results and empirical model at each altitude (i.e. for Mach numbers 0.7, 0.8, 0.85, 0.9) was calculated to measure how well the Mach dependent drag rise had been captured. The nose fineness ratio was altered to minimise the Mach dependent standard deviation. Simultaneously, the altitude dependent standard deviation (drag offset) was minimised by manipulating the proportion of the aircraft length comprised of the tail cone.

The results of the final geometry are depicted in Figure C.6, where the CFD was compared against the empirical analytical formula for different flight conditions. The top graph shows the actual drag quantities, and the difference between the two is shown as a percentage in the bottom plot. Figure C.7 illustrates the fuselage geometry representation with reference to a large business jet, of the type being studied in this work.

¹Only a small range of cruise was considered to minimise the number of CFD cases required.

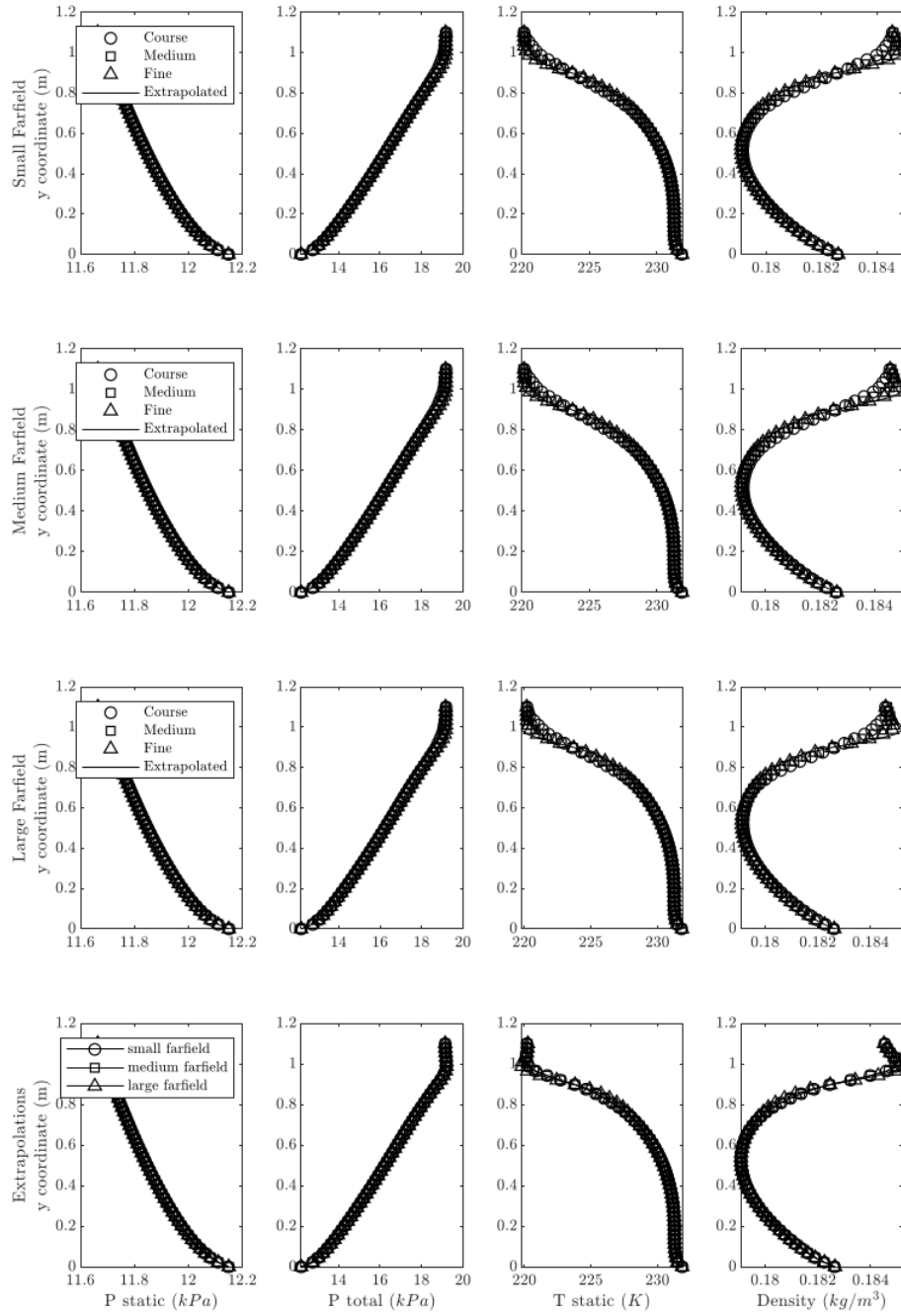


Figure C.4: Isolated fuselage GCI study: trailing edge wake properties with Richardson extrapolation

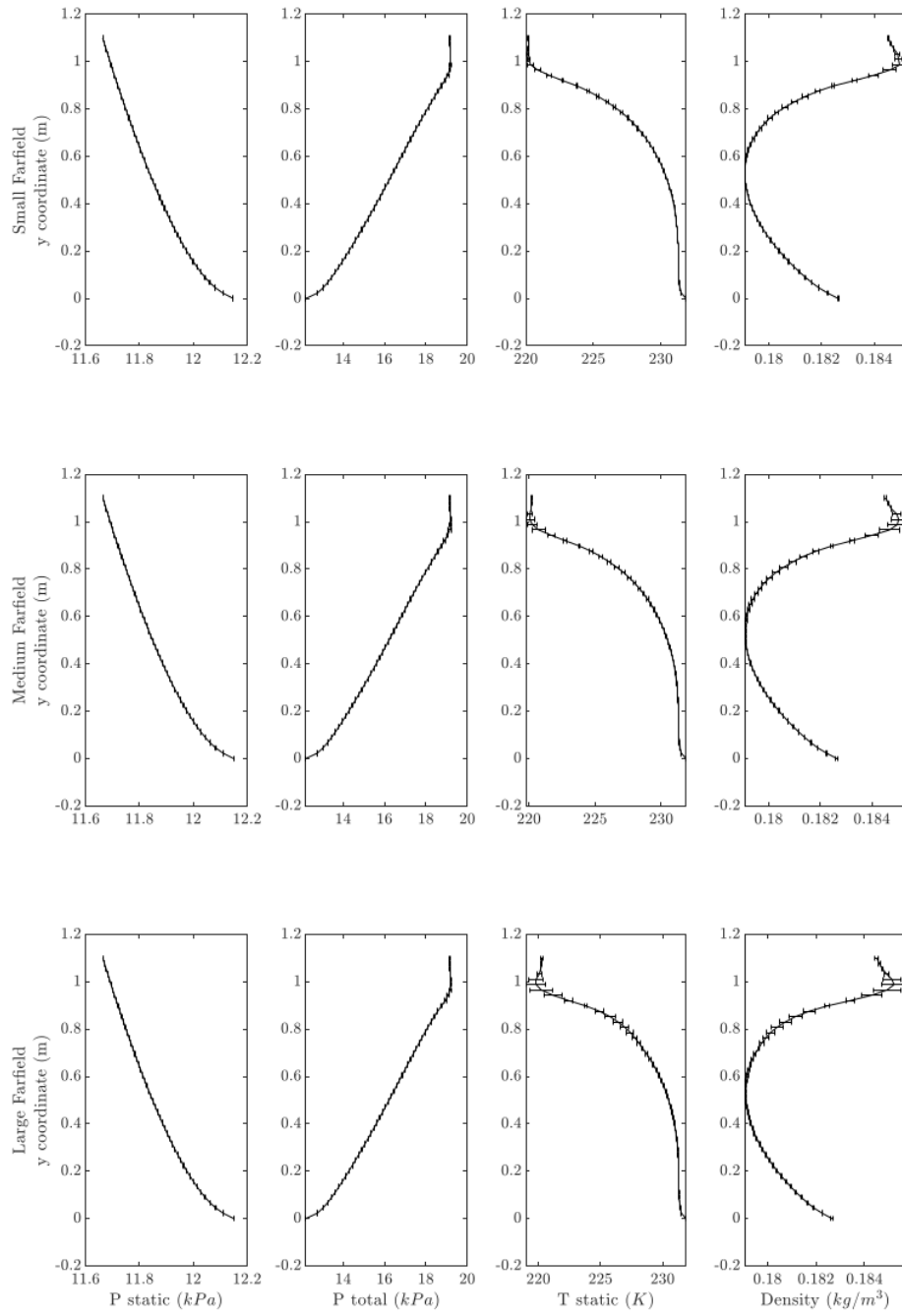
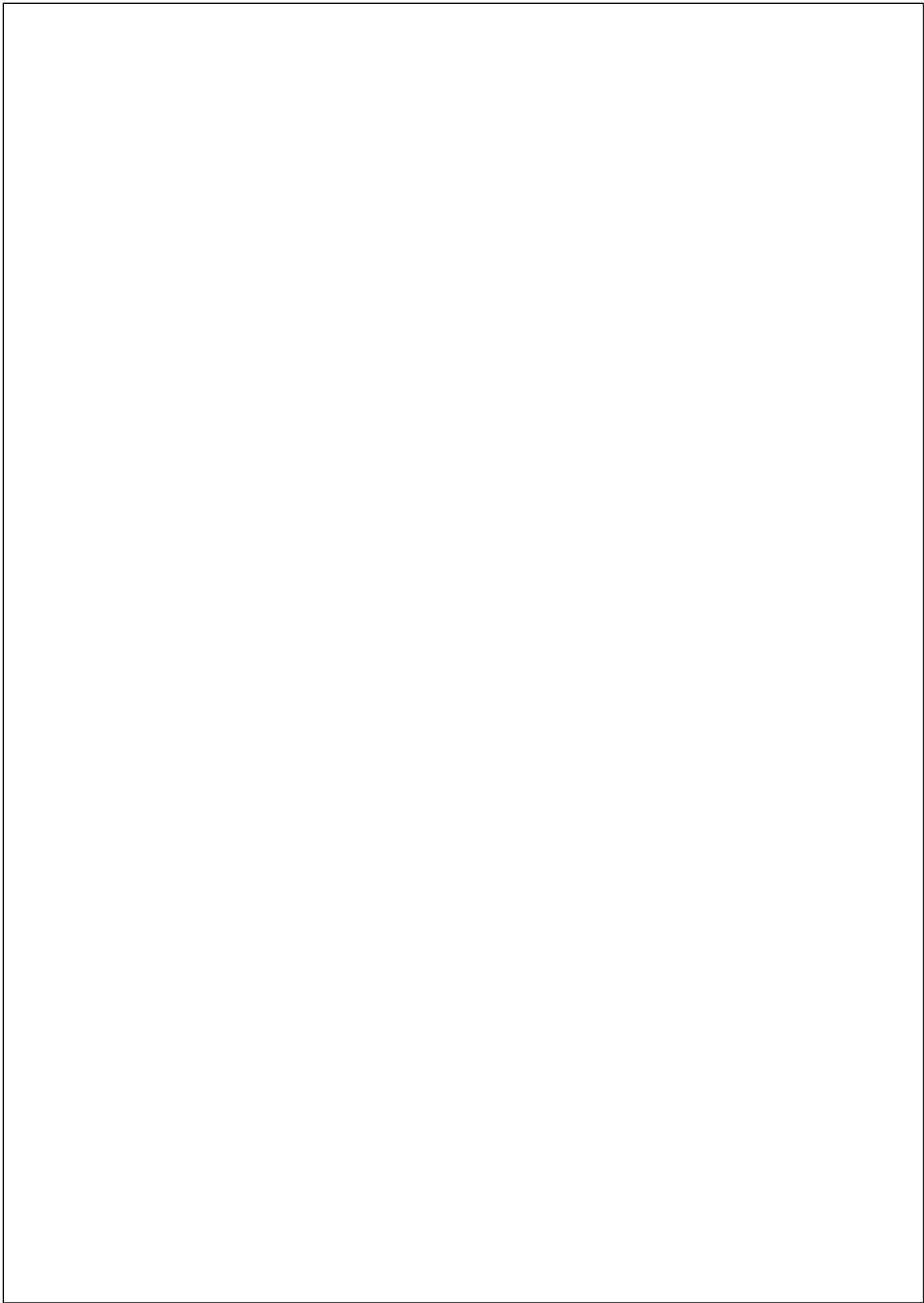


Figure C.5: Isolated fuselage GCI study: GCI uncertainties of trailing edge wake properties



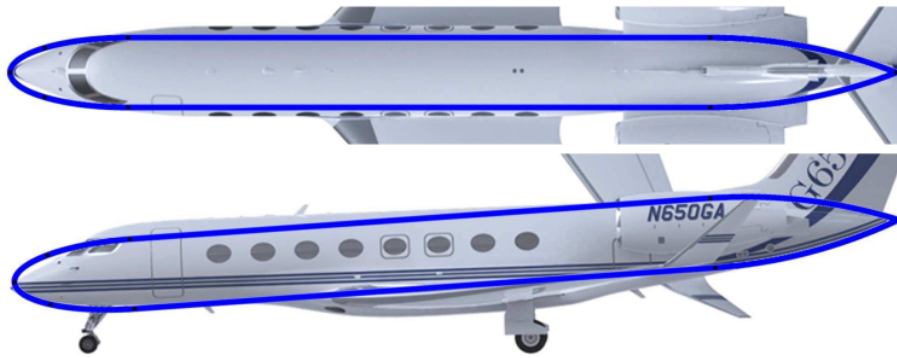


Figure C.7: Final fuselage geometry superimposed over a picture of a Gulfstream G650 [2] for reference as to how representative the geometry is.

Appendix D

Baseline Engine Model

Aircraft aerodynamic and engine performance data, representative of a state-of-the-art business jet, were supplied by Rolls-Royce. The engine data supplied included fuel flow for thrust ranges at various flight conditions and thrust ratings. A physics based model of the engine was created to best match the data supplied, i.e. SFC at different thrust requirements over various flight conditions. This model was necessary to capture the cycle performance affects of different power off-take and thrust requirement combinations resulting from various BLI thrust splits and corresponding BLI fan powers. Cranfield University's TURBOMATCH code [82] was used to create the 0D physics based model. The engine matching procedure is described briefly here, along with results showing how well the model was able to reproduce thrust SFC curves for different flight conditions.

The engine was assumed to be similar to the Rolls-Royce BR725, and publicly available information was used to guide the creation of the engine model to match the data [83]. As such, the engine was assumed to be a twin-spool turbofan comprised of a single stage fan, ten stage axial HPC, two stage axial HPT and three stage LPT. TURBOMATCH is a 0D code which uses component maps to calculate off-design performance of each of the components comprising the engine. A TURBOMATCH component flow diagram of the engine architecture is depicted in Figure D.1. A brief description of the TURBOMATCH components used in this engine are listed below, and the reader is referred to MacMillan [84] for additional information on the models used.

- INTAKE - uses the international standard atmosphere and calculates thermodynamic conditions based on altitude, Mach number and user-specified intake pressure recovery
- PREMAS - allows for user defined splits in flow gas path and is used here for bypass split and cooling flow bleeds.
- COMPRE - thermodynamic compressor model utilizing compressor maps for off-design calculations
- DUCTER - models ducts and is used here to include losses in total pressure between components

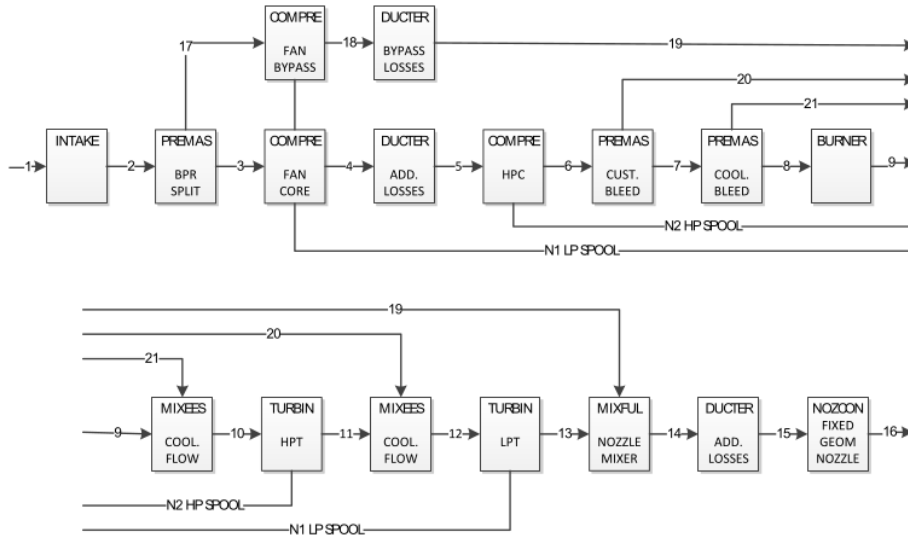


Figure D.1: TURBOMATCH flow chart of baseline engine

- BURNER - combustion model which calculates change in gas thermodynamic properties using combustion maps
- MIXEES - constant-area mixing of two flow streams used for cooling flow
- MIXFUL - mixing of larger flow streams, such as the bypass and core flows of this engine
- NOZCON - convergent nozzle performance
- TURBIN - thermodynamic turbine model, utilizing maps for off-design calculations.

Assuming that the engine is indeed similar to the BR725, the design-point was based on the top-of-climb condition. This allowed for a good estimation of the mass-flow-rate through the engine, knowing the fan radius from literature [83], and assuming a $MFR \approx 1$. This is a reasonable assumption, as the engine is generally sized for this operating condition. The BPR at TOC was assumed to be approximately the same as the value reported in Jane's Aero Engines [83]. Following Guha [85], it is known that there is an optimum FPR corresponding to a minimum SFC for a given turbine exit temperature TET and overall pressure ratio OPR. Also, Guha shows that the optimum for a mixed exhaust engine occurs where the total pressures of the bypass and core are equal, for optimum mixing. This point is usually at cruise, and therefore the design point at TOC required the bypass total pressure to be higher than the core. The ratio

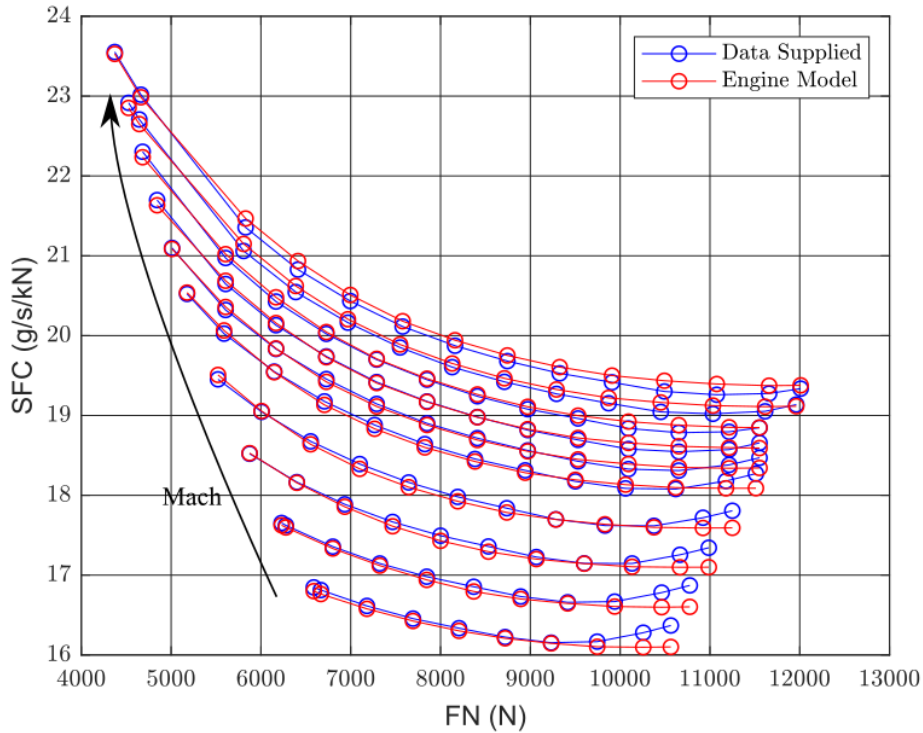


Figure D.2: Baseline engine model off-design performance versus data supplied. Showing SFC as a function of intrinsic net thrust at FL450 for M 0.6 - 0.925.

is normally around 1.1 at this condition [86]. Therefore, the TOC design-point could be determined for any combination of BPR, TET and OPR.

Subsequently, the next task was to find the TOC design point that allowed for the off-design performance of the engine to be matched. In doing so, a myriad of parameters, in addition to TET, OPR and BPR could be varied. These included component maps, component efficiencies, pressure ratios of the fan and HPC as well as pressure losses throughout the system. As such, a global optimisation tool was used to handle all these parameters and find the TOC design point that produced the closest match in performance. Figure D.2 depicts the quality of the match achieved in terms of the shapes of the trends for different Mach numbers. Figure D.3 gives an indication of the statistical accuracy achieved over all flight conditions, and shows the different accuracy at different mission segments. Of most importance is the accuracy of cruise at 45kft, and this graph clearly shows sufficient accuracy with a mean difference around 2% and standard deviation of 0.5% for the different flight Mach numbers.

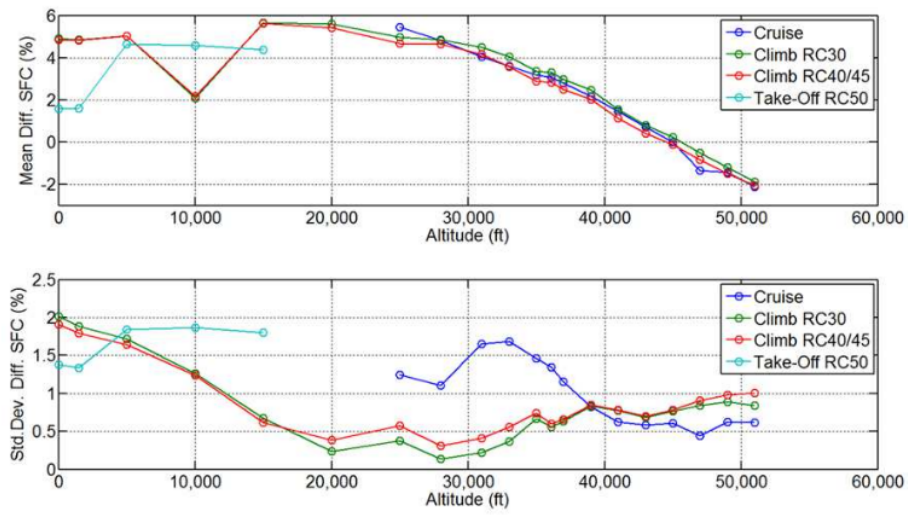


Figure D.3: Optimiser matched engine solution accuracy represented statistically for all flight conditions

Appendix E

Aircraft Performance

In order to assess the benefits of BLI at an aircraft mission level, an aircraft performance model is required. Various commercial codes exist, with Piano [78] amongst the most prominently used in industry. However, a licence to this software was not available to the author. The Department of Power and Propulsion at Cranfield University has its own in-house code HERMES [73, 87]. HERMES makes use of a drag build-up method to calculate aircraft aerodynamics, and is coupled to TURBOMATCH [82] (Cranfield University's in house gas turbine code) to calculate propulsion system performance. The initial intention was to utilize HERMES for the purposes of this research; however certain limitations of HERMES (similar to those of commercial codes) require significant work-arounds to incorporate unconventional highly integrated propulsion-airframe designs, and therefore it was decided to develop a new code to allow for the flexibility in the modelling required. The primary motivation was the need for a performance code that allows for various definitions of thrust-drag accounting, as this is of high importance when considering BLI performance. HERMES is structured around the definition of "standard net thrust" (see Ref. [36]) and does not conveniently allow for inclusion of additional throttle-dependent drag terms, such as spillage drag. For instance, it is the intention of this work to provide maps of drag terms relating to mass flow ratio from CFD studies, and HERMES is inflexible in terms of the reading in such maps. Additionally HERMES assumes conventional airframe and propulsion system geometries in its drag calculations. It thus becomes difficult to incorporate BLI propulsor drag without additional effort to transform this drag into a fictitious change in aircraft geometry. A secondary motivation was the absence of wave drag estimations, which is of high importance to business jet performance. Furthermore, HERMES is based around a set mission profile structure, which makes it difficult to model the NBAA IFR reserves profile [88] typically used to compare business jet performance in industry. The final motivation was to have a performance code within Matlab to allow for ease of integration with the other tools developed in this work. A new performance code has been written in order to circumvent the above flexibility issues, and is described in the sections that follow.

E.1 The Vehicle Performance Deck

Based on the above argument, the underlining concept behind the new model was to decouple the aircraft drag calculations from the performance calculations to allow for flexibility in thrust-drag accounting definitions. To model an aircraft, a vehicle performance deck (VPD) needs to be generated. The VPD is an extension to the conventional engine deck in that it serves to provide look-up tables correlating throttle settings to net vehicle force, NVF ($NVF = \text{Thrust} - \text{Drag}$), instead of just standard net thrust. Therefore the VPD must contain thrust and drag calculations across the entire flight envelope (or at least those required for the mission), according to the user-defined thrust-drag accounting system. This also allows more easily for the throttle dependent drag terms to be incorporated into the mission level performance. From the mission performance code perspective, the VPD must have three generalised aspects:

1. Database of look-up tables
2. A function to interact with the database based on standardised form of inputs
3. A throttle settings file for the mission

The database must at least contain look-up tables of NVF and fuel flow rate (an additional throttle parameter, such as TET, is generally also required) corresponding to ranges in altitude, Mach and C_L appropriate for the mission. Additional look-up tables of other parameters may be included and the VPD function can return these values to be stored for later analysis. The VPD function must be of a format that allows for the input of the Matlab database, a control parameter specifier, a corresponding value for the control parameter, a mission segment specifier and the flight conditions (i.e. altitude, Mach and C_L). Typically the control parameter is a throttle setting (e.g. TET) or the required NVF. For example; at climb the user may wish to climb at a constant TET and calculate a corresponding NVF and hence rate-of-climb (ROC), alternatively the ROC may be set where the mission performance code then calculates the required NVF and the corresponding TET (and associated fuel flow rate). The function output must at least output the NVF and fuel flow rate, but may return other parameters in a generalised structure.

E.2 Mission Profile and Performance Calculations

With the aircraft performance model developed in this work, a wide range of mission profile formats may be simulated through various combinations of different segments. However, this work has focused on the NBAA Jet Range format [88]. This mission profile was selected as it is the profile typically used in aircraft brochures and forms a consistent basis for comparison between aircraft [89]. The range format is illustrated in Figure E.1, and details two possible range calculations: 1) NBAA Visual Flight Rules (VFR) Reserves and 2) NBAA Instrument Flight Rules (IFR) Reserves. The latter is that most often quoted and will form the basis of the mission level performance

calculations in this work, although the code does include the option to calculate VFR and IFR reserve missions. The overall calculation begins with first determining the amount of fuel required for the reserves mission profile, as this mass has to be carried throughout the main mission. The reserves profile is details in Figure E.1 between segments J and O. The reserve mission profile calculation procedure for a specified diversion cruise Mach number and altitude is illustrated in Figure E.2, and is comprised of three numerical loops. The aircraft weight, with payload, at the end of the mission is known and is denoted as O' . The segment calculation O-O' accounts for the fuel reserves required after the aircraft has landed. This fuel is assumed to be equivalent to the amount of fuel spent at loiter at 5,000 ft for a specified amount of time (30 min assuming VFR, or 45 min for IFR). Therefore the aircraft mass at point O must be calculated iteratively. Similarly, the mass at the diversion top-of-descent (TOD) is unknown and must be calculated numerically to ensure that there is enough fuel to descend. Finally, the initial mass at J must be determined iteratively to converge on the required range of 200 nautical miles. At a given altitude, this procedure is repeated to determine the Mach number that optimises for minimum fuel consumption. A further optimisation loop is then required to determine the altitude of the diversion cruise which yields the least fuel consumption (this step has not yet been incorporated into the current code). Having determined the amount of fuel required for the reserves profile, the masses at the start (A) and end (J) of the main mission are known. The main mission calculation procedure begins with the climb and descent calculations to determine the aircraft masses at the start (D) and end (I) of the main mission cruise. Thus, only the mass at TOD (point I) needs to be estimated in order to converge on the amount of fuel required for descent. The methodologies and calculations of the segments comprising Figure E.1 are described briefly in the sections that follow.

E.3 Description of Main and Reserve Mission Segments

The following subsections describe the assumptions and calculations used to determine the performance of each segment of the main and reserves mission as detailed in Figure E.1.

E.3.1 A-B: Taxi Out

The taxi is assumed to occur at idle throttle settings. The throttle setting is user defined and so is the amount of time spent taxiing. The amount of fuel burnt is then simply the fuel flow rate at the given throttle setting multiplied by the time, i.e. 10 min. The idle throttle settings used are difficult to determine. This work assumes that idle occurs at the lowest TET possible that allows for convergence of the propulsion system model and returns realistic outputs.

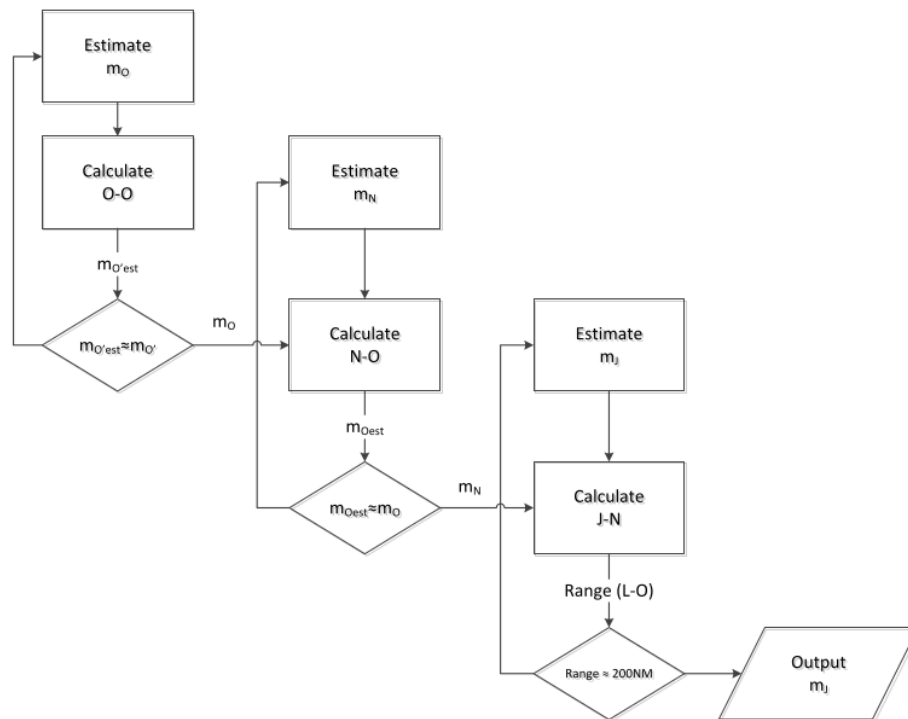


Figure E.2: Calculation procedure followed to calculate the amount of fuel required for the NBAA IFR reserves profile

E.3.2 B-C: Take-Off

In higher fidelity codes, the take-off calculations can become rather complex and require additional considerations of high lift devices and regulatory requirements. In complying with the NBAA Jet Range Profile, such high fidelity is not warranted and the necessary estimation simply assumes the amount of fuel burnt for the propulsion system operating at full power for 1 min. In the model developed the following is done to calculate the fuel burnt during take-off. The throttle setting is user-defined (this should correlate to a maximum take-off thrust rating, with derates applied if necessary), and the fuel flow rates corresponding to this throttle setting at points B and C are simply averaged and applied to the 1 min take-off time to determine the total fuel burnt. The differences in fuel flow rate at B and C are due to the different flight conditions. Point B is assumed to be at sea level static conditions and point C at an air traffic management altitude and speed of 1500 ft and 250 kCAS respectively [90].

E.3.3 C-D: Climb to First Cruise Altitude

Total Energy Model

The method for calculating the aircraft's climb performance at any instantaneous point along the segment is the total energy model, which is a balance of the changes in aircraft kinetic and potential energies corresponding to the work done by the forces acting on the aircraft. The reader is referred to the approach adopted in this work which is described in reference in Ref. [91]. For an analytical solution to the total energy model, two of three possible aircraft control parameters must be specified, resulting in three possible calculation options:

1. Calculation of ROC by controlling speed and throttle
2. Calculation of speed by controlling ROC and throttle
3. Calculation of throttle by controlling speed and ROC

The first option is most commonly used, where the throttle setting is typically defined by a maximum climb thrust rating and the aircraft speed schedule is determined to optimise climb performance within the limitations of the aircraft's performance capabilities. However the second is also incorporated to allow for limitations on ROC that the user may wish to impose. In that case the speed schedule is still the control parameter (and used to optimise the climb performance) and the corresponding throttle is calculated to meet with the imposed ROC restriction. Thus the total energy model may be expressed mathematically with ROC the dependent variable as given by Equation E.1. The third term of Equation E.1, is referred to as the energy share factor and accounts for the proportion of power used to climb versus acceleration. The calculation of the energy share factor differs according to the geopotential altitude H_p (i.e. altitude relative to the tropopause) and the speed "type" at which the aircraft climbs (i.e. at constant Mach or CAS). Thus Equation E.1 becomes a piecewise function of eight variations, which

have been incorporated within the performance model developed in this work. These variations have been omitted here for the sake of brevity and the reader is referred to Reference [91] for their definitions. The climb model, with its current capabilities, only allows for climb at a constant Mach or CAS.

$$\text{ROC} = \frac{T - \Delta T}{T} \frac{\text{NVF} \cdot V_{\text{TAS}}}{mg} \left[1 + \left(\frac{V_{\text{TAS}}}{g} \right) \left(\frac{dV_{\text{TAS}}}{dH_p} \right) \right]^{-1} \quad (\text{E.1})$$

The Climb Profile

Figure E.1 does not specify the details as to how the climb profile should be calculated, and thus information from Reference [90] has been used to guide the generation of an appropriate climb profile. The climb profile is ultimately defined by the speed schedule followed by the aircraft at various altitudes. The user-defined (or variables for optimisation, as will be discussed later) are those speeds not strictly regulated by the air traffic management (ATM), and are; (a) constant CAS to transition altitude and (b) constant Mach to first cruise altitude. The transition altitude is the altitude at which the TAS of (a) and (b) are equal, and is calculated analytically. The climb is assumed to begin at an initial altitude of 1500 ft, at which the aircraft accelerates at level flight to a speed of 250 kCAS (kts CAS) from its speed at end of take-off. Climb continues from 1,500 ft to 10,000 ft whilst maintaining its speed at 250 kCAS, as per ATM regulations. Thereafter the aircraft accelerates to the scheduled CAS (a), where it will begin its climb to the transition altitude (or first cruise altitude if the transition is higher than that of the cruise altitude). The climb then continues from the transition altitude to the first cruise altitude at a constant Mach (b). Finally, the aircraft accelerates at level flight to the first cruise Mach, should it greater than that at the transition altitude. It is at this point that the climb segment is assumed to end and the cruise segment is assumed to start.

Climb Calculations for a Given Speed Schedule

For calculating level flight acceleration, the thrust at the endpoints of the acceleration subsegment are obtained for the climb thrust rating, and by estimating the aircraft mass at the end of the segment. The average acceleration of the aircraft is then known across the acceleration subsegment. The amount of time, and thus distance covered, is calculated from the averaged acceleration. The change in aircraft mass is obtained from the averaged fuel flow rate and a new estimate for the mass at the end of the subsegment is obtained. This process is repeated until the end mass converges within a given tolerance.

The climb subsegments of the climb profile are discretized in terms of altitude intervals in order to calculate the climb performance. The size of the altitude interval steps is calculated iteratively, such that the entire climb range converges within a set tolerance. Reference [90] discusses appropriate interval sizes to obtain satisfactory accuracy in range, however this accuracy was found to be too relaxed when calculating an optimum speed schedule (i.e. a tighter constraint was required to allow for convergence

on an optimum speed schedule). Nonetheless, for each altitude interval the start mass is known but the end mass is not. Therefore, similarly to the acceleration calculation, the end mass requires estimation. Based on this estimation, the aircraft performance at the altitude interval end points are known from the VPD. The average ROC is then used to determine the time across the interval as well as the distance and mass change (it is noted that another iteration is used to account for climb angle when determining the lift coefficient). The mass change provides a new estimate, and the calculation is repeated until satisfactory convergence is obtained.

Climb Segment Optimisation

In Figure E.1 an "optimum ROC" is required to determine the climb segment performance. However this is ambiguous as it does not state what exactly should be optimised. In optimising climb performance there are typically three options [90]:

1. minimum climb duration
2. minimum climb range
3. least fuel burnt during climb

As the throttle setting is assumed fixed, according to a climb thrust rating, the climb is optimised by the speed schedule at which the aircraft climbs. For the first two optimums above, the speed of the aircraft should correspond to the maximum ROC and maximum climb angle, respectively, at each instance along the segment. However, this is not possible because the altitude and aircraft mass change throughout the climb requiring a unique speed solution at each instance [90]. Thus the climb speed schedule, as defined earlier, needs to be determined iteratively to obtain the averaged optimum ROC or climb angle across the whole climb segment. For the third optimum, the determination of the schedule becomes more complex.

As the maximum climb thrust rating control parameter is not generally at a constant fuel flow rate, minimising the time to climb does not correspond to a minimum fuel consumption. Furthermore, thrust reduces with increase in altitude and airspeed, and there exists an optimum speed to maximise L/D at each altitude. Flying at the averaged optimum speed to maximise L/D also does not necessarily result in the least fuel burn. This is because it may be more beneficial to climb at a less fuel efficient speed to get to the cruise altitude sooner, where the engine operates more efficiently. Thus, to minimise fuel consumption the concept of flying to a common point should be used [90]. The common point is a range that ends somewhere during the first cruise step. The climb speed schedule may be optimised to minimise fuel consumed to reach this common point.

The code currently includes capabilities to calculate optimum climb for the first two options, and the third option is yet to be included due to its complexity. Nonetheless, it is assumed that, in the case of business jets, it is most logical to optimise for minimum time to climb, as this is an important characteristic for comparison between business jets [89].

As mentioned earlier, the speed schedule is defined by a constant CAS between 10,000 ft and the transition altitude, and a constant Mach number between the transition altitude and first cruise altitude. Therefore the above procedures for all three options becomes complicated further by having to calculate what combinations of the two speeds yields the optimum. To circumvent this added complication, this work assumes the approximation that the optimum is more sensitive to the speed below the transition than the Mach number above [90], and therefore the optimisation process relies on a pre-defined value for climb above the transition altitude. The constant Mach portion of the climb should be calculated to ensure that the aircraft remains safely within its buffet margins. Piano's default determines the climb Mach to be a value just below the drag divergence value [77]. This has not yet been included in this work, and a simplifying assumption is made that the climb Mach above transition is equal to the first cruise Mach number.

E.3.4 D-I: Stepped Cruise

As an aircraft burns fuel and becomes lighter during cruise, there exists an altitude at which the fuel mileage is greatest for a given cruise Mach number. Additionally, there exists at each altitude and Mach number which optimises the fuel mileage. The fuel mileage is defined as the ratio of range flown to fuel burnt, and at an instantaneous point equates to a ratio of TAS over fuel flow rate. This work will refer to this parameter as the fuel specific air range (FSAR):

$$\text{FSAR} = \frac{V_{\text{TAS}}}{\dot{m}_f} \quad (\text{E.2})$$

As the weight of the aircraft changes so does the lift, drag and hence thrust requirement. For a given weight, changing the altitude and flight speed alters both the aircraft's aerodynamic efficiency and engine efficiency. Therefore, there is an altitude and Mach combination which trades off these effects to give a minimum fuel rate value. As the aircraft gets lighter the optimum altitude increases (for a given Mach) and the greatest range is thus achieved if the aircraft were able to steadily climb throughout the cruise to track this optimum. Unfortunately this is impractical for air traffic management reasons, but aircraft are permitted to follow a stepped cruise profile in an attempt to optimise range. To best make use of stepped cruise, it is necessary to calculate when it is optimum for the aircraft to climb to the next altitude. As an approximation, this model assumes that the aircraft should begin to climb to the next altitude as soon as the FSAR_{i+1} at the next altitude Alt_{i+1} and Mach M_{i+1} equals the FSAR_i at the current flight conditions, where both values are determined at the aircraft's current mass m_i . Therefore these climb points (E and G of Figure E.1) are purely dependent on mass and the VPD, and can be determined before the cruise calculation begins. Because the mass at top-of-climb (TOC) and top-of-descent (TOD) (as these calculations are performed before cruise) are known, the bisection method may be used to numerically calculate the mass at the points at which the aircraft should climb to the next altitude.

Having now determined the mass at the end of each cruise step, the range, time and fuel burnt during cruise may be integrated numerically by discretizing each step in terms of mass increments. The mass, and hence lift, at the start and end of each interval are known, and the fuel flow can be interpolated from the VPD for the given altitude, Mach, and C_L such that the NVF is zero. The fuel flow is averaged to and used to calculate the time take to burn the amount of fuel equivalent to the interval mass increment. The distance covered is then simply calculated from the product of the true airspeed and interval time. The accuracy of the calculation is obtained iteratively by changing the number of mass intervals (and hence interval size) until it converges on a range within a specified tolerance.

The step climbs are calculated for constant Mach to that of the cruise speed when the climb began. A user defined throttle setting for this climb is used, or taken to equal that used for the climb to initial cruise segment. The user may also define the minimum required residual ROC, otherwise a default of 1.5 m/s is assumed. The aircraft is then allowed to accelerate to the new Mach number if it has been specified to be different by the user.

E.3.5 I-J: Main Mission Descent

During descent, the throttle settings are set to idle and the aircraft follows a speed schedule until it reaches the approach altitude. The descent speed schedule has little impact on the overall mission performance because of operating at idle throttle settings, and therefore there is little need to optimise a descent speed schedule [90]. Instead it is reasonable to assume that the descent speed schedule is simply that of the climb schedule in reverse [77]. The only difference is that the altitude and Mach number at TOD may differ from that at TOC due to the user-defined stepped cruise requirements. To account for this, the model assumes that the aircraft first decelerates to the initial cruise Mach number before descending. The descend schedule is therefore as follows:

1. Deceleration to first cruise Mach number
2. Constant Mach descent from TOD to transition altitude
3. Constant CAS descend to 10,000ft
4. 250 (kts) CAS descend to approach altitude (assumed 1,500 ft)

The descent calculations are also based on the total energy model as described in the climb calculations. The rate-of-descent(ROD) is calculated directly as a function of the throttle and speed setting at each instance. However, as per Figure E.1, there is a restriction on descent rate of 3000 fpm. Therefore, the code monitors the throttle to ensure that this value is not exceeded.

E.3.6 J-K: Standard Instrument Approach

The calculations of a missed approach can be complex. The NBAA Jet Range Format, as per Figure E.1, simplifies the calculation by assuming that the fuel used during approach

is approximately equivalent to that which would be used to loiter at 5000 ft for 5 min. The distance covered is also assumed to be zero as it is in reality very small in comparison to the overall mission. The code developed assumes that the aircraft flies at steady level flight at a constant Mach number which optimises endurance during loiter. This Mach number corresponds to the maximum L/D at any instance during loiter, but changes as fuel is burnt and the mass of the aircraft becomes lighter. Therefore, the Mach number to optimise endurance is calculated numerically. During each Mach number estimate the steady level flight performance is calculated by discretizing the segment into time intervals. The accuracy of the loiter calculation is judged based on convergence on the amount of fuel burnt, which is performed by numerically altering the size of the time intervals.

E.3.7 K-L: Climb to Hold Altitude Following Missed Approach

Similarly to the main mission climb performance, there is ambiguity as to what the optimum ROC, required by Figure E.1, is meant to optimise. As this segment is within the reserves profile, it seems intuitive that the climb should optimise for minimum fuel consumption. However, as stated earlier in the main mission climb description, this optimisation procedure has not yet been incorporated into the performance model. Therefore the climb is calculated to correspond to the maximum climb thrust rating at a constant speed of 250 kCAS as the climb takes place below the ATM altitude of 10,000 ft.

E.3.8 L-L: Hold for Clearance

The aircraft is required to hold at 5,000 ft for an assumed 5 min to wait for clearance to proceed to a diversion airport. The loiter is once again calculated to maximise endurance, thereby minimising fuel flow. Therefore the same calculation procedure as that used for segment J-K is used, to iteratively determine the optimum loiter Mach number. Naturally this segment does not contribute to the overall range flown by the aircraft.

E.3.9 L-M: Climb from Hold to Diversion Cruise Altitude

Once again there is ambiguity as to what the optimum ROC refers to in Figure E.1. As this is part of the reserves mission profile it is assumed that the most fuel economic climb should be used. However, as stated previously, this capability has not yet been incorporated within the current code. Thus it is assumed that the climb occurs to optimise minimum climb duration. The climb calculation therefore follows the same as that for the main mission, except that it begins at a higher altitude of 5,000 ft and, more importantly, that the final Mach number and altitude is unknown. This is because the diversion cruise Mach number and altitude still need to be determined numerically to optimise endurance. This is dealt with in a numerical scheme that incorporates all of the reserves segments. Thus assuming the diversion altitude and Mach are known, the constant CAS portion of the speed schedule is determined numerically to optimise

for minimum climb duration. This follows closely the procedure developed for the main mission climb (segment C-D), which has been described previously.

E.3.10 M-N: Diversion Cruise

The NBAA jet Range Format requirements specify economy cruise for the diversion mission. Unlike the main mission cruise, the diversion cruise should be flown to maximise endurance and not range. There exists an altitude and Mach combination which maximises L/D and these need to be determined numerically. This has to be performed in conjunction with the other segments of the reserves mission, while maintaining the constraint of the 200 nautical mile diversion range. This is because the other segments dictate the start and end mass of the cruise altitude, which in turn affects the averaged optimum Mach number for a given altitude. Currently, the code only optimises endurance by numerically calculating Mach for a given diversion altitude. Based on estimation in this numerical procedure, the masses at the diversion cruise endpoints are known and the cruise is discretized into mass intervals as was done for the main mission cruise. For a estimated diversion cruise altitude and Mach, the mass interval size is calculated numerically to converge on range within a set tolerance.

E.3.11 N-O: Diversion Descent to Land

The descent calculations for the diversion mission follow the same procedure to that of the main mission, where the speed schedule is that of the diversion climb schedule in reverse (and excluding hold). The approach and land is calculated in a similar way to that of the take-off calculations, where the user defines an approach thrust rating by specifying an approach throttle setting.

E.4 Verification

Piano has provided a sample model of the Gulfstream G550, detailing the results of the performance calculations reference. This sample model was used to verify the performance code presented in this chapter. A comparison of the calculated payload-range diagram against that of the sample model (as well as against data from the Purchase Planning Handbook [89]) is summarised in Figure E.3.

E.5 Baseline Aircraft Payload Range Diagram

The tool developed in this appendix has been used to model the mission-level performance of the baseline aircraft, given the drag polar and baseline engine models discussed in Appendices B and D. Figure E.4 depicts the aircraft's payload range diagram, which has been plotted alongside values for the Gulfstream G650, for comparison against a current state-of-the-art aircraft of similar size. As can be seen, the two payload diagrams are very close to one another, and this gives confidence that the drag, engine and

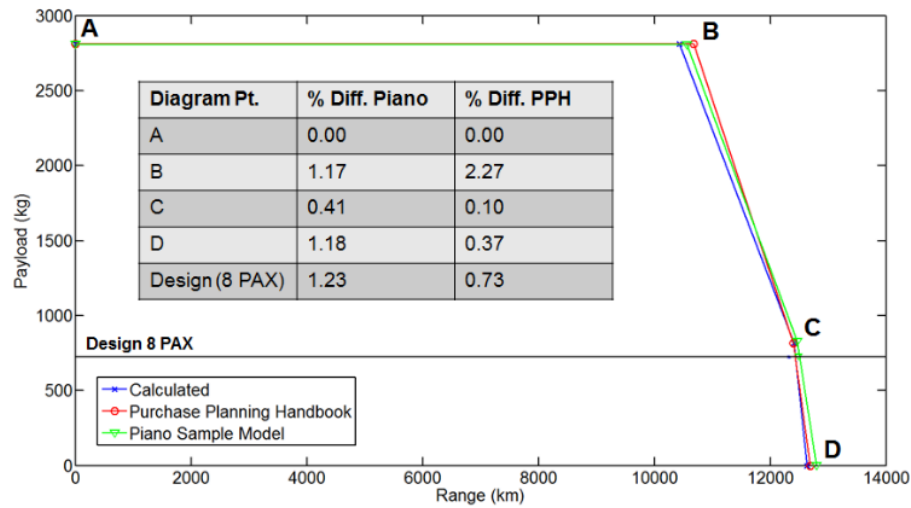
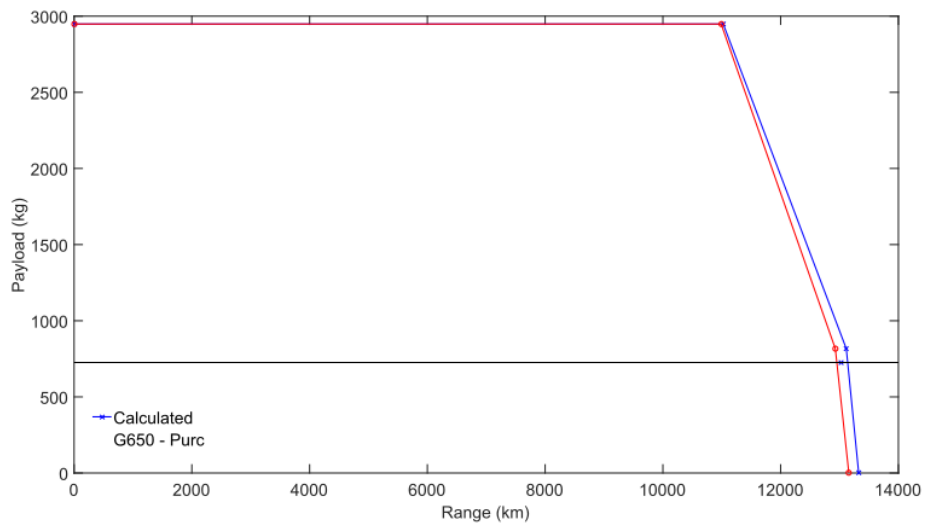


Figure E.3: Aircraft Mission Performance Code Verification. Comparison against Piano sample model [78] and Purchase Planning Handbook [89]

mission profile models are correct. It can be seen that the baseline aircraft is a large, long range business jet and deemed appropriate for BLI applications. The purpose of modelling the payload diagram is to allow for comparisons to be made in the potential gains achievable in range using BLI.





the way for higher fidelity CFD assessments. This appendix documents in brief the parametric studies used to identify potential regions of interest in the design exploration. The subsequent results have been used to guide the CFD performance assessments of Chapter 6

F.1 Baseline Aircraft

To begin the study, it was first necessary to model the baseline aircraft in terms of its drag polars, engine performance and mission-level performance, which have been documented in Appendices B, D and E, respectively. The reasons for this was to: (1) provide a reference for evaluating benefits in SFC, (2) develop an engine model capable of capturing the effects of off-design cycle performance in response to different thrust and power off-take demands, (3) provide fuselage overall geometric constraints which govern the geometry of the BLI propelled fuselage. The author was provided with some drag polar and engine performance data by Rolls-Royce, which was deemed to be representative of a state-of-the-art business jet. In order to have workable physical models for analysing the aerodynamic and engine cycle performance in response to BLI retrofitting, the airframe geometries and engine model had to be reverse engineered using a bespoke drag build-up code (developed in Appendix B) and an in-house gas turbine modelling tool [82]. These tools were incorporated within a global optimisation code, in order to best match the data provided. These matching procedures have been documented in Appendices B and D, respectively. Finally, a bespoke aircraft mission performance tool, presented in Appendix E, was developed to allow for the calculation of payload range diagrams and block fuel burns. The purpose of this tool was two fold. Firstly, it allowed for comparison of the mission level performance of the baseline against other known state-of-the-art business jets, to give confidence that the drag build-up code and engine model produced a reasonable combined result. Secondly, it allows for the impact of the BLI performance to be analysed at a mission level within the parametric studies, to give a better sense of how SFC benefits translate to improvements in range. The mission level performance code has been verified in Appendix E, and has also been used to compare the baseline aircraft's performance against the Gulfstream G650 for reference.

F.2 Methodology

A simplified method was required for the preliminary parametric studies to quickly estimate the BLI propulsor performance without having to rely on CFD solutions. In keeping consistent with the Reference Net Vehicle Thrust accounting definition (see Section 3.6) it has been assumed that the total drag of the aircraft does not differ from the baseline. This discounts the aerodynamic coupling between the BLI propulsor and airframe, which can only be accounted for by CFD solutions, as well as the profile drag added by the BLI nacelle. Furthermore any changes in induced drag, due to additional propulsion system weight, are ignored. Finally, assuming $MFR = 1$ makes it possible

to equate the Reference Net Vehicle Thrust to the Intrinsic Net Thrust (i.e. $F_{N,ref} = F_{N,int}$). This is useful, because only the flow conditions at the highlight surface S_f are required to calculate $F_{N,ref}$ for a given FPR. In addition, because $MFR = 1$, there is no pre-diffusion and the ingested BL's profile is not affected by the presence of the propulsor. The BL profile along the fuselage is calculated from the 7th power law using flat plate approximation for turbulent flow:

$$u(r) = \begin{cases} 0.99 u_{\infty} \left(\frac{r-R}{\delta} \right)^{\frac{1}{7}} & 0 \leq r - R \leq \delta \\ \infty & r - R > \delta \end{cases}$$

where:

$$\delta = \frac{0.37 x}{Re_x^{1/5}}$$

and is applied around the fuselage surface, approximated to be a cylinder, as depicted in Figure F.1. As suggested by this figure, the BLR is varied by the amount of circumference (θ) and radius ($r - R$) ingested. These two amounts are varied parametrically to investigate the optimum amounts of BLR. An equivalent uniform flow is required for use in simplified 0D propulsor calculations. Assuming:

$$(\Delta r)^2 \approx 0, \quad \Delta r \rightarrow 0$$

the elemental area is given as:

$$\Delta A = \frac{\Delta \theta}{2} [(r + \Delta r)^2 - r^2] = r \Delta r \Delta \theta$$

and therefore:

$$dA = r dr d\theta$$

This allows for mass-flow-rate, momentum and kinetic energy to be obtained from the following set of integrals, for a given ingestion radius and angle:

$$\begin{aligned} \dot{m} &= \iint_{0,R}^{\theta,r} (\rho u r) dr d\theta \\ \dot{m}u &= \iint_{0,R}^{\theta,r} (\rho u^2 r) dr d\theta \\ E_k &= \iint_{0,R}^{\theta,r} \left(\frac{1}{2} \rho u^3 r \right) dr d\theta \end{aligned}$$

The equivalent uniform flow's velocity and density are iteratively calculated such that:

$$\begin{aligned} \dot{m} &= \dot{m}_{eq} = \left(\frac{\theta \rho_{eq} u_{eq}}{2} \right) (r^2 - R^2) \\ \dot{m}u &= (\dot{m}u)_{eq} = \left(\frac{\theta \rho_{eq} u_{eq}^2}{2} \right) (r^2 - R^2) \\ E_k &= E_{k,eq} = \left(\frac{\theta \rho_{eq} u_{eq}^3}{4} \right) (r^2 - R^2) \end{aligned}$$

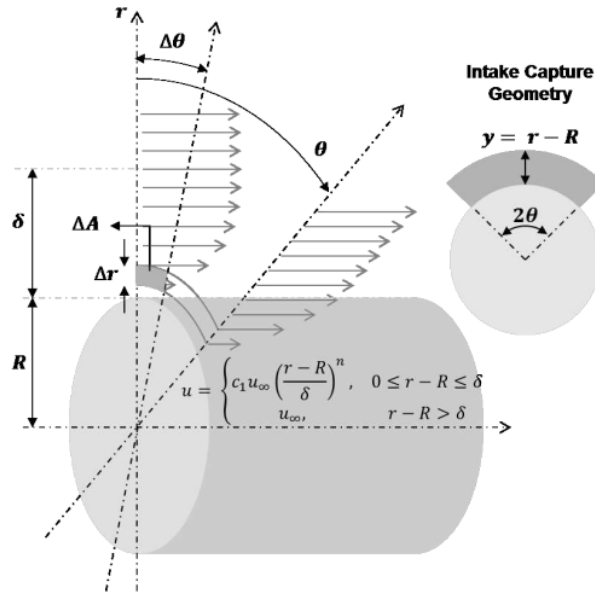


Figure F.1: Flat plate analogy applied to a cylinder, to approximate the BL profile over the fuselage.

Given this method of inlet flow averaging, the procedure for determining SFC benefits at cruise is depicted in Figure F.2. The method begins with the cruise flight conditions and the drag build-up method, of Appendix B, is used to determine the overall aircraft drag and the corresponding thrust required. The primary governing variable of the parametric study, is thrust-split, which determines how much thrust must be produced by the BLI propulsor(s) and main engines, respectively. The equivalent uniform flow conditions must now be determined before the BLI propulsor performance can be calculated. The calculation of the equivalent uniform flow has already been discussed above. The FPR must be calculated iteratively, such that the BLI thrust requirement is met. A bisection method is implemented here, as the FPR limits are known. The calculation of FPR takes into account estimates of losses in the pre-diffusion streamtube, intake duct and exhaust duct, respectively. The calculation also takes into account fan isentropic efficiency, and nozzle calculation are also based on the isentropic efficiency procedure detailed in [58]. Once the FPR has been calculated, the BLI shaft power is determined according to the change in enthalpy across the fan (this takes into account losses due to mechanical inefficiencies). The off-design performance of the baseline engine (see Appendix D), is calculated to provide the remaining thrust, and power to the BLI fans (from the low pressure spool). This gives a change in fuel flow from the baseline's performance at the same cruise conditions.

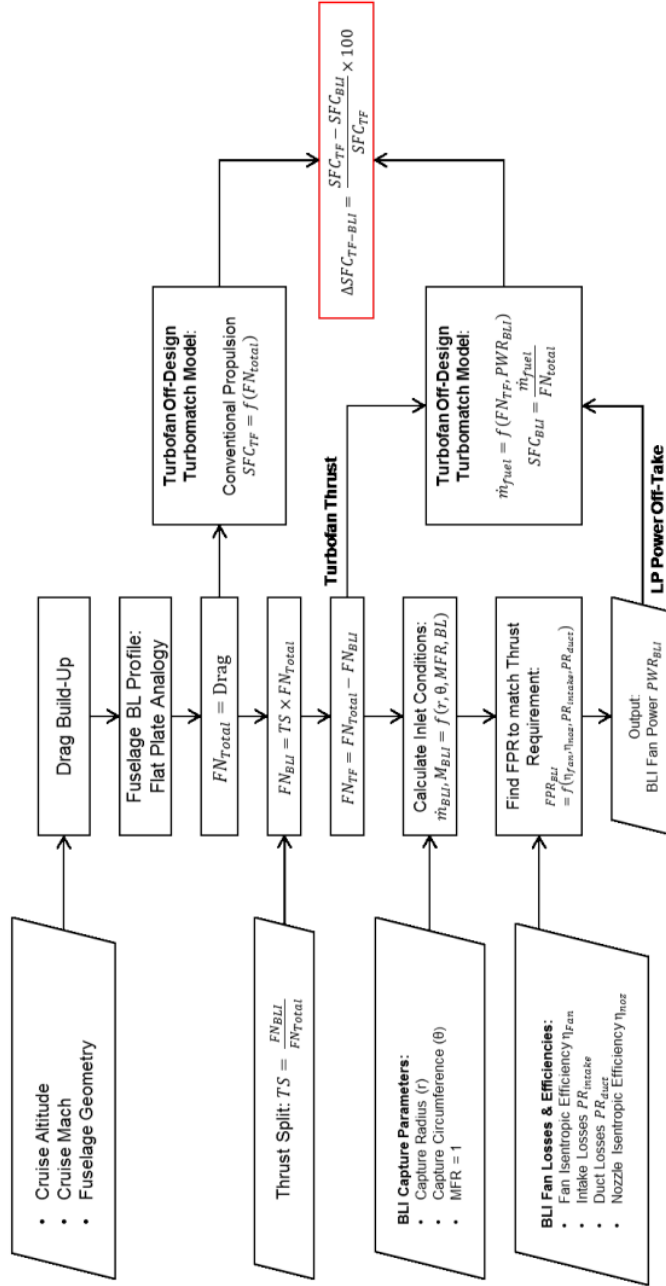


Figure F.2: Summary of methodology used to calculate SFC benefits for the preliminary parametric study.

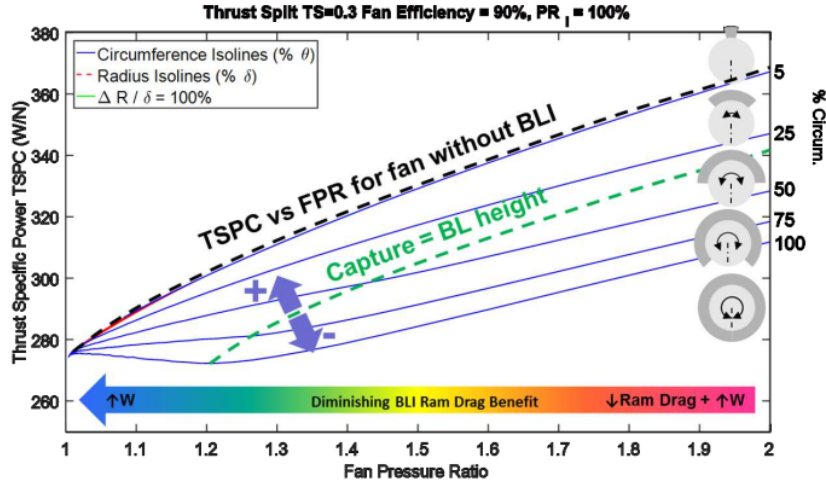


Figure F.3: Diagram of decoupled propulsor performance trends for an arbitrary thrust-split and fan efficiency, but no intake pressure recovery losses.

F.3 Decoupled BLI Propulsor Maps

The method presented above, may be used to obtain the decoupled BLI propulsor performance. The decoupled BLI propulsor performance map, helps to explain the source of benefits obtained from BLI, given the assumptions made in this appendix. Figure F.3 depicts such a map, where an arbitrary thrust-split of 30% and fan efficiency of 90% was selected for demonstrative purposes. To conceptualise the ideal performance, the map shown excludes intake pressure recovery losses. The map shows TSPC on the LHS vertical axis. This is also representative of the propulsive efficiency (as defined in Section 3.7.2), where a TSPC of 250 corresponds to a 100% in propulsive efficiency. The horizontal axis gives the BLI FPR, and the RHS vertical axis shows the different amounts of circumference ingested. The amount of circumference ingested is also indicated by the blue iso-lines in the figure. Variations in ingestion radius fall either side of the green dashed line, below which radius' less than the BL thickness are ingested.

The trends in this figure are defined by a trade-off between ram drag benefits and reduced specific thrust (due to increases in mass-flow-rate). At very small ingestion circumference angles, θ , the ingestion radius, r , must be increased well above the BL thickness, δ , to provide the required thrust at a given FPR. In doing so, the equivalent uniform flow approaches free-stream conditions, and no benefit is realised due to reductions in ram drag. Therefore, as $\theta \rightarrow 0$, the propulsive efficiency coincides with that of a free-stream ingesting podded propulsor (which is indicated by the dashed black line). This explains why the best propulsive efficiencies are realised when the entire circumference is ingested. Assuming $\theta = 2\pi$, it may be observed that there is an optimum FPR

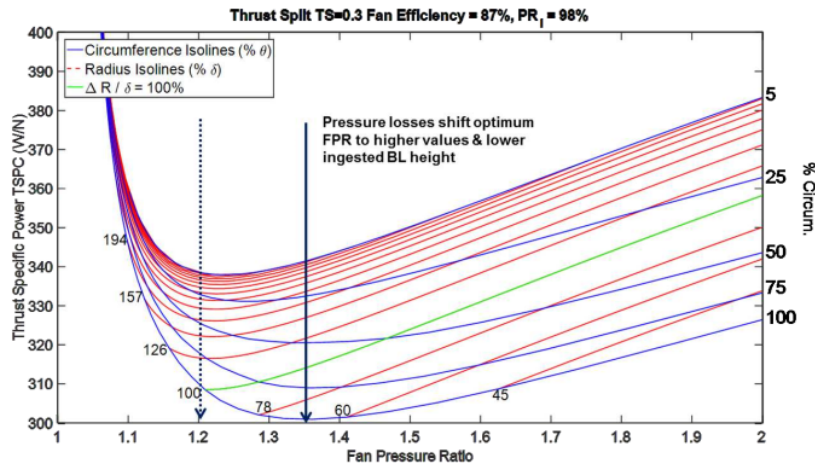


Figure F.4: Change in trends from Figure F.3 after introducing penalties in fan efficiency (to account for distortion) and losses in intake pressure recovery.

that corresponds to a minimum TSPC (or maximum propulsive efficiency). This occurs where $r \approx \delta$, and is the optimal balance between benefits from: (1) the reduced ram drag associated with the lower momentum flow at smaller ingestion radius' (this is due to the shape of the BL profile) and (2) reducing specific thrust through ingesting larger quantities of mass-flow-rate.

Figure F.4 shows how the trends of Figure F.3 change, when fan efficiency penalties and intake pressure recovery losses are introduced. Flow distortion at the fan face is likely to negatively affect the BLI propulsor performance, and this may be accounted for by parametrically varying the fan efficiency. The fan efficiency results in a shift of the decoupled map to regions of higher TSPC (or lower propulsive efficiency). Intake pressure recovery losses will be likely in BLI, due to the ducting required and also by subjecting the BL to adverse pressure gradients. These pressure losses also contribute to the map's shift to higher TSPCs, but, more importantly, result in a change in shape of the trends. At very low FPRs, the influence of pressure losses becomes dominant, and this causes a steep rise in TSPC. As such, the optimum FPR shifts to higher values to compensate for the pressure losses. This optimum corresponds to a re-balance in the trade-off between ram drag benefits and reduced specific thrust. It is now beneficial to ingest less than the BL thickness, so as to take greater advantage of the reduced ram drag. This particular example shows an optimum point ingesting approximately 65% of the BL thickness. The key message here, is that there is an FPR which optimises propulsive efficiency for a given thrust-split requirement (and for a given set of fan efficiency penalties and pressure recovery losses). The next step is to translate these optimum propulsive efficiencies into SFC benefits at cruise. This is done by including

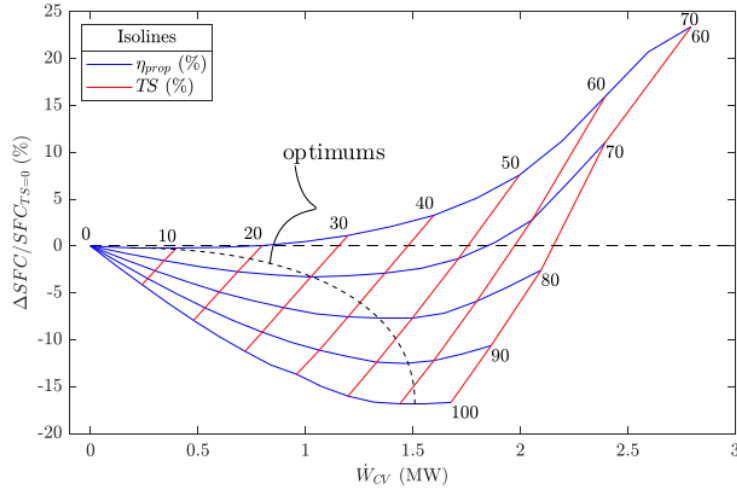


Figure F.5: Decoupled turbofan engine performance map corresponding to BLI propulsive efficiency and thrust-split combinations

the turbofan off-design cycle performance for given combinations of power off-take and remaining thrust.

F.4 Decoupled Turbofan Performance Map

The turbofan's off-design performance, corresponding to different thrust-splits and propulsive efficiencies, has been presented already in Chapter 6. The map of the turbofan performance is repeated here, in Figure F.5, for convenience. The map is obtained from the baseline engine model (developed in Appendix D), by calculating an off-design operating point, which matches the power off-take demanded by the BLI propulsor (taken off the low pressure spool), whilst producing the remaining thrust dictated by the thrust-split. It can be seen that there is an ideal optimum benefit of 17.5% at a propulsive efficiency (of the BLI propulsors) of 100% and thrust-split of around 62%.

F.5 System-Level Performance Results

SFC benefits, for a particular cruise condition, are obtained by combining the decoupled BLI and turbofan performance maps. The outcomes of the preliminary parametric study, are summarised in Figures F.6 to F.8, which depict SFC benefits and corresponding BLR, FPR and shaft power, respectively. This set of results presented is limited to full circumference ingestion and one fan efficiency, for brevity. It can be seen from these figures, that the parametric study examined the implications of different pressure

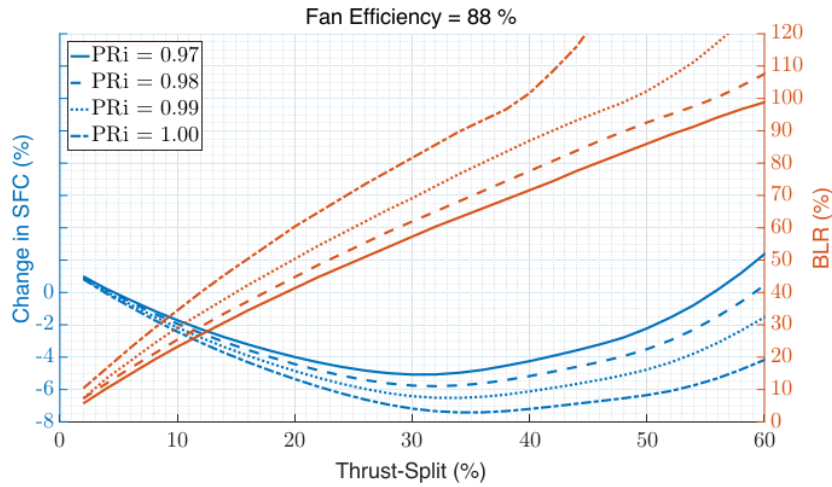


Figure F.6: Preliminary parametric study SFC benefits, with corresponding BLR values.

recovery losses, upto 3%. These results correspond to the optimum FPRs (i.e. maximum propulsive efficiencies) for each thrust-split and pressure recovery combination.

It is evident that thrust-split is the governing parameter, which manages the optimum SFC achieved. It is essentially responsible for the trade-off between optimal propulsive efficiency and optimal cycle performance. Intuitively, larger intake pressure losses shift the optimum to lower thrust-splits. This is because of the detriment in propulsive efficiency, which means that it becomes more beneficial to produce a little more thrust from the main engines (which are not experiencing that loss in pressure recovery). This also corresponds to a reduction in BLR (see Figure F.6), in order to take advantage of further reductions in ram drag, due to the lower momentum flow of the BL near the fuselage wall. Larger pressure losses results in an increased FPR, which has already been explained and is also made evident in Figure F.7. Finally, it may be seen from Figure F.8, that there is a fairly linear proportional relationship between thrust-split and shaft power.

To summarise, the system-level parametric results predict that an optimum SFC benefit between 5-7.5% is achievable at thrust-splits between 30-35%, when ingesting 65-90% of the BL thickness.

F.6 Mission Level Performance

The next goal of this preliminary parametric assessment, is to examine the impact of BLI at a mission level. A bespoke mission performance code was developed in Appendix E. The SFC benefits of the previous section, were applied to the cruise segment performance of the baseline aircraft. It is noted that changes in drag, due to stepped cruise

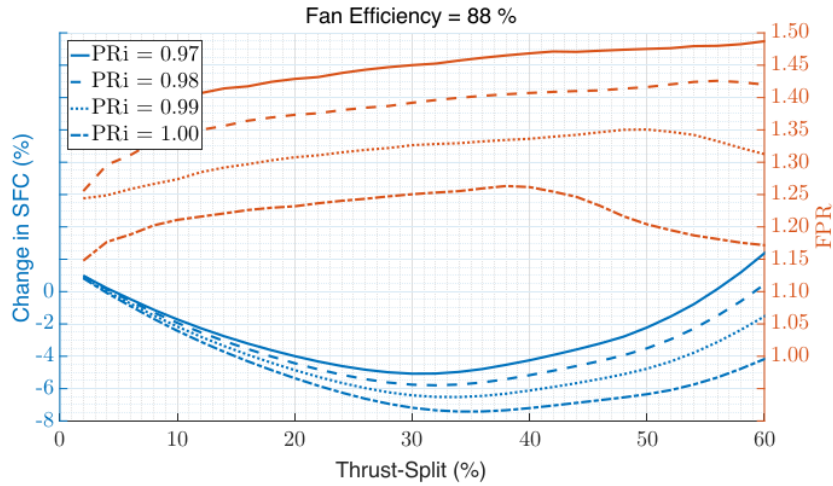


Figure F.7: Preliminary parametric study SFC benefits, with corresponding BLR values.

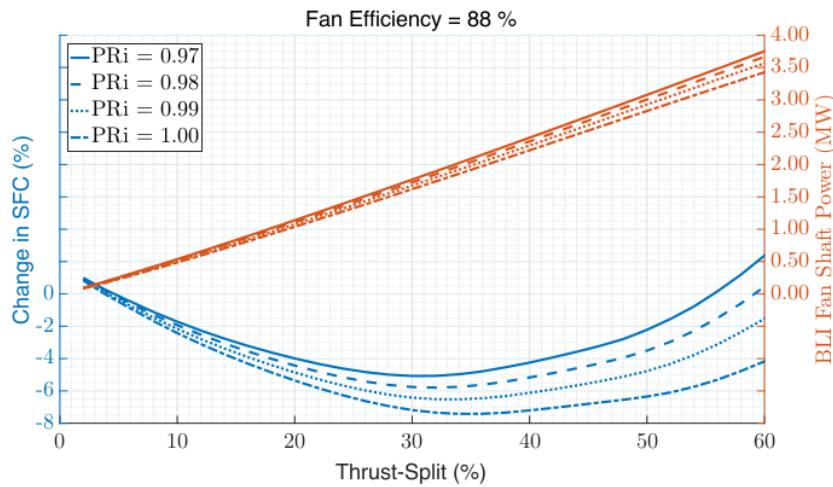


Figure F.8: Preliminary parametric study SFC benefits, with corresponding BLR values.

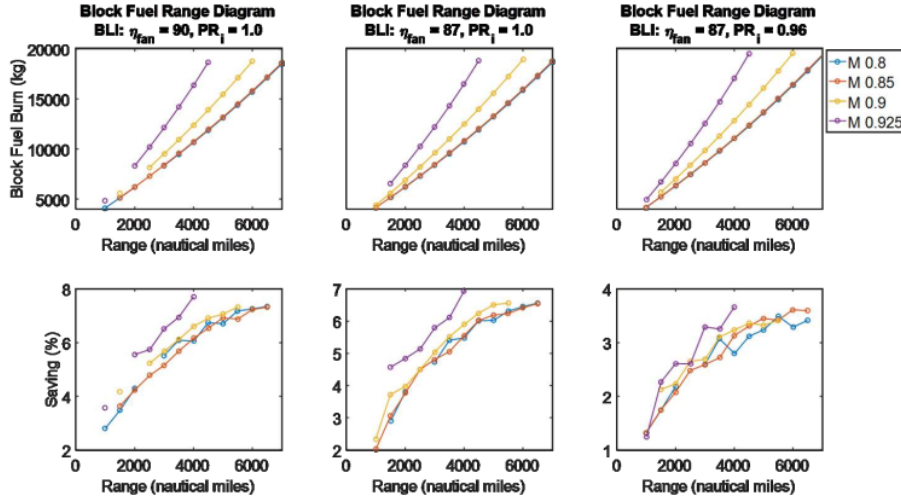


Figure F.9: Block Fuel Burn (and fuel savings) vs. Range diagrams, for a BLI propelled business jet flying at different cruise Mach numbers.

and changes mass (from fuel burn), were accounted for, and this altered the thrust requirements on the BLI propulsor and main engines. Figure F.9 illustrates the block fuel burn (and fuel savings) for different mission ranges at different cruise Mach numbers. The mission-level parametric studies also included different fan efficiencies and pressure recovery losses. As expected, flying at higher Mach numbers reduces the range achievable for a given amount of fuel. The range decreases further with larger BLI fan efficiency penalties and reduced BLI intake pressure recoveries. Counter intuitively, larger fuel savings are found at higher Mach numbers. This may be explained by the additional ram drag benefit offered by the BL profile at higher flight velocities. However, this fails to take into account the external aerodynamics of the BLI propulsor's nacelle. The inclusion of the BLI nacelle would reduce the drag rise Mach number, and it is likely that the dominating external aerodynamics would reverse this trend.

Nonetheless, the mission-level analysis shows that block fuel savings that correspond closely to the benefits in SFC at a system-level. This is due to the long ranges flown by this business jet, which spends the vast majority of its time at cruise. Following this reasoning, these parametric studies have not included the BL performance in climb and descent segments. The business jet customer is typically not concerned by fuel economy, but rather range capabilities. Therefore the benefits of BLI may be better presented as the corresponding increase in range. Moreover, the business jet customer is not interested in range benefits quoted in nautical miles, but rather in city-to-city trips. This is depicted in the global map of Figure F.10, which shows the furthest cities that may be reached from New York with the benefits of BLI. This figure shows that the



Image Produced using *Great Circle Mapper*, <http://www.gcmap.com/>

Figure F.10: Global map of increased range due to BLI.

aircraft may now also comfortably reach Hong-Kong, Manila, Cape Town, Johannesburg and Madurai, for a BLI business jet flying at a design cruise speed of M 0.85.

References

- [1] J. Kurzke. Fundamental Differences Between Conventional and Geared Turbofans. *ASME Turbo Expo: Power for Land, Sea, and Air*, 1, June 2009.
- [2] Gulfstream Aircraft Company. Gulfstream G650 Specifications, <http://www.gulfstream.com/aircraft/gulfstream-g650>, Last Accessed April 2016.
- [3] G. V. Brown. Weights and Efficiencies of Electric Components of a Turboelectric Aircraft Propulsion System . *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, January 2011.
- [4] N. Renard and S. Deck. A Theoretical Decomposition of Mean Skin Friction Generation into Physical Phenomena Across the Boundary Layer. *Journal of Fluid Mechanics*, 790:339–367, February 2016.
- [5] M. Drela. Power Balance in Aerodynamic Flows. *AIAA Journal*, 47(7):1761 – 1771, July 2009.
- [6] A. Arntz, O. Atinault, and A. Merlen. Exergy-Based Formulation for Aircraft Aeropropulsive Performance Assessment: Theoretical Development. *AIAA Journal*, 53(6):1627–1639, June 2014.
- [7] L. Paparone and R. Tognaccini. Computational Fluid Dynamics-Based Drag Prediction and Decomposition. *AIAA Journal*, 41(9):1647–1657, September 2003.
- [8] A. M. O. Smith and H. E. Roberts. The Jet Airplane Utilizing Boundary Layer Air for Propulsion. *Journal of the Aeronautical Sciences*, 14(2):97–109, Feb 1947.
- [9] F. T. Lynch. A Theoretical Investigation of the Effect of Ingesting Boundary Layer Air on Turbofan Engine Fuel Consumption. Report SM23981, Douglas Aircraft Co., 1960.
- [10] W. M. Douglass. Propulsive Efficiency with Boundary Layer Ingestion. Report MDC J0860, McDonnell Douglas Corp., 1970.
- [11] L. H. Smith. Wake Ingestion Propulsion Benefit. *Smith, L. H.*, 9(1):74–82, 1993.
- [12] A. Seitz and C. Gologan. Parametric Design Studies for Propulsive Fuselage Aircraft Concepts. In *4th CEAS Air & Space Conference*, 2013.

- [13] E. et al Greitzer. N+3 Aircraft Concept Designs and Trade Studies. Technical report, NASA, 2010.
- [14] A. Uranga. Preliminary Experimental Assessment of the Boundary Layer Ingestion Benefit for the D8 Aircraft. In *52nd Aerospace Sciences Meeting*, 2014.
- [15] H. J. Steiner, A. Seitz, K. Wieczorek, K. Plötner, A. T. Isikveren, and M. Hornung. Multi-Disciplinary Design and Feasibility Study of Distributed Propulsion Systems. In *28th International Congress of the Aeronautical Sciences*. Bauhaus Luftfahrt e.V., 2012.
- [16] A. P. Plas, M. A. Sargeant, V. Madani, D. Crichton, E. M. Greitzer, T. P. Hynes, and C. A. Hall. Performance of a Boundary Layer Ingesting (BLI) Propulsion System. *American Institute of Aeronautics and Astronautics*, Paper AIAA 2007-450, 2007.
- [17] Boeing Commercial Airplanes. Current Market Outlook 2014 - 2033. <http://www.boeing.com/boeing/commercial/cmo/index.page?>, Accessed October 2014.
- [18] Bombardier Commercial Aircraft. Market Forecast 2014 - 2033. <http://www.bombardier.com/en/media-centre.html>, Accessed October 2014.
- [19] S Sato. *The Power Balance Method for Aerodynamic Performance Assessment*. PhD Thesis, Massachusetts Institute of Technology, June 2012.
- [20] D. K. Hall. *Analysis of Civil Aircraft Propulsors with Boundary Layer Ingestion*. PhD Thesis, Massachusetts Institute of Technology, January 2015.
- [21] J. D. Anderson. *Fundamentals of Aerodynamics*. McGrawHill, 5 edition, 2011.
- [22] M. Drela. *Flight Vehicle Aerodynamics*. The MIT Press, 2014.
- [23] P. R. Spalart. On the Far Wake and Induced Drag of Aircraft. *Journal of Fluid Mechanics*, 603:413 – 430, February 2008.
- [24] D. Hayes, M. Lone, J. F. Whidborne, J. Camberos, and E. Coetzee. Adopting Exergy Analysis for use in Aerospace. In *Progress in Aerospace Sciences*, number <http://dx.doi.org/10.1016/j.paerosci.2017.07.004>, 2017.
- [25] D. Desterac and J. van der Vooren. Drag/thrust analysis of jet-propelled transonic transport aircraft; Definition of physical drag components. *Aerospace Science and Technology*, 8(6):545–556, September 2004.
- [26] M. B. Giles and R. M. Cummings. Wake Integration for Three-Dimensional Flow-field Computations: Theoretical Development. *Journal of Aircraft*, 36(2):357 – 365, March-April 1999.



- [40] D. Lovely. Boundary Layer and Shock Detection in CFD Solutions. MSc Thesis, Massachusetts Institute of Technology, February 2000.
- [41] M. Ueno, K. Yamamoto, K. Tanaka, M. Murayama, and R. Tognaccini. Far-Field Drag Analysis of NASA Common Research Model Simulation. *Journal of Aircraft*, 50(2):388–397, March-April 2013.
- [42] ANSYS. ANSYS Fluent Theory Guide: Version 17.1, 2017.
- [43] R. W. Luidens, N. O. Stockman, and J.H. Diedrich. An Approach to Optimum Subsonic Inlet Design. *ASME 1979 International Gas Turbine Conference and Exhibit and Solar Energy Conference*, Volume 1A: Gas Turbines(79-GT-51), 1979.
- [44] B. L. Berrier, M. B. Carter, and B. G. Allan. High Reynolds Number Investigation of a Flush-Mounted, S-Duct Inlet With Large Amounts of Boundary Layer Ingestion. Technical Report NASA/TP-2005-213766, NASA, Langley Research Center, Hampton, Virginia, September 2005.
- [45] M. J. Langley. The Design of Axisymmetric Cowls for Padded Nacelles for High By-Pass Ratio Turbofan Engines. Report 3846, NACA, 1979.
- [46] R. J. Re. Experimental Pressure Distributions on Axisymmetric Cowls at Mach Numbers From 0.60 to 0.92. Technical Memorandum NASA/TM-2006-214313, NASA, Langley Research Center, Hampton Virginia, August 2006.
- [47] John Seddon and Eustace Laurence Goldsmith. *Intake Aerodynamics*. Number 1563473615, 9781563473616 in AIAA education series. American Institute of Aeronautics and Astronautics, 1999, second edition, August 1999.
- [48] L. R. Owens, B. G. Allan, and S. A. Gorton. Boundary-Layer-Ingesting Inlet Flow Control. *Journal of Aircraft*, 45(4):1431–1440, August 2008.
- [49] B. J. Lee, M. Liou, and C. Kim. Optimizing a Boundary-Layer-Ingestion Offset Inlet by Discrete Adjoint Approach. *AIAA Journal*, 48(9):2008–2016, September 2010.
- [50] M. Rein and S. Koch. Experimental Study of Boundary-Layer Ingestion into a Diverterless S-Duct Intake. *AIAA Journal*, 53(11):3487– 3491, November 2015.
- [51] H. Kim and M. Liou. Shape design optimization of embedded engine inlets for N2B hybrid wing-body configuration. *Aerospace Science and Technology*, 30:128–149, 2013.
- [52] A. J. Anabtawi, R. F. Blackwater, P. B. S. Lissaman, and R. H. Liebeck. An Experimental Investigation of Boundary Layer Ingestion in a Diffusing S-Duct With and Without Passive Flow Control . *37th AIAA Aerospace Sciences Meeting and Exhibit* , January 1999.

- [53] BS Stratford. The prediction of separation of the turbulent boundary layer. *Journal of fluid mechanics*, 5(1):1–16, 1959.
- [54] J. H. Bell and R. D. Mehta. Contraction Design for Small Low-Speed Wind Tunnels. Technical Report JIAA TR - 84, Stanford University, Stanford University, Department of Aeronautics and Astronautics, Stanford, CA 94305, April 1988.
- [55] I. Goulos, T. Stankowski, D. MacManus, P. Woodrow, and C. Sheaf. Civil Turbofan Engine Exhaust Aerodynamics: Impact of Bypass Nozzle After-body Design. *Aerospace Science and Technology*, September 2017.
- [56] P. P. Walsh and P. Fletcher. *Gas Turbine Performance*. Number ISBN 0-632-06434. Blackwell Science, second edition, 2004.
- [57] B. J. McBride, M. J. Zehe, and S. Gordon. NASA Glenn Coefficients for Calculating Thermodynamic Properties of Individual Species. Technical Report NASA/TP-2002-211556, NASA, Glenn Research Center, Cleveland, Ohio, September 2002.
- [58] H. I. H. Saravanamuttoo, C. F. C. Rogers, H. Cohen, and P. V. Straznicky. *Gas Turbine Theory*. Pearson Education Ltd, 6 edition, 2009.
- [59] Hermann Schlichting, Klaus Gersten, Egon Krause, Herbert Oertel, and Katherine Mayes. *Boundary-layer theory*, volume 7. Springer, 1955.
- [60] Aerospace Recommended Practice 1420. Gas Turbine Inlet Flow Distortion Guidelines. *Society of Automotive Engineers*, (ARP-1420), 1978 (Revised in 2002).
- [61] W. T. Cousins and M. W. Davis, Jr. Evaluating Complex Inlet Distortion with a Parallel Compressor Model: Part 1 - Applications to Complex Patterns. *Proceedings of ASME Turbo Expo 2011*, June 2011.
- [62] W. T. Cousins and M. W. Davis, Jr. Evaluating Complex Inlet Distortion with a Parallel Compressor Model: Part 1 - Concepts, Theory, Extensions and Limitations. *Proceedings of ASME Turbo Expo 2011*, June 2011.
- [63] E. J. Gunn and C. A. Hall. Aerodynamics of Boundary Layer Ingesting Fans. *Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, Düsseldorf, Germany*, June 2014.
- [64] D. K. Hall, E. M. Greitzer, and C. S. Tan. Analysis of Fan Stage Design Attributes for Boundary Layer Ingestion. *Proceedings of ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition, Seoul, South Korea*, June 2016.
- [65] D. E. Harrington. Performance of Convergent and Plug Nozzles and Mach Numbers from 0 to 1.97. Technical Memorandum NASA TM X-2112, NASA, Lewis Research Center, NASA, Cleveland, Ohio 44135, October 1970.

- [66] C. N. Kaul. Anisentropic Gas Flow. *AIAA Journal*, 2(6):1145–1145, 1964.
- [67] G. K. Batchelor. *An Introduction to FLuid Dynamics*. Cambridge Mathematical Library. Cambridge University Press, 2000.
- [68] C. B. da Silva and J. C. F. Pereira. Invariants of the velocity-gradient, rate-of-strain, and rate-of-rotation tensors across the turbulent/nonturbulent interface in jets. *Physics of Fluids*, 20(055101):1 – 18, May 2008.
- [69] M. Gad-el Hak. Stoke’s Hypothesis for a Newtonian, Isotropic Fluid. *Journal of Fluids Engineering*, 117(1):3–5, 1995.
- [70] François G Schmitt. About Boussinesq’s turbulent viscosity hypothesis: historical remarks and a direct evaluation of its validity. *Comptes Rendus Mécanique*, 335(9-10):617–627, 2007.
- [71] P Sforza. Chapter 9 - Drag Estimation. In *Commercial Airplane Design Principles*, pages 349 – 404. Butterworth-Heinemann, Boston, 2014.
- [72] L. R. Jenkinson, P. Simpkin, and D. Rhodes. *Civil Jet Aircraft Design*, volume 7. Butterworth-Heinemann, 1999.
- [73] Cranfield University. HERMES and TmatchCalls (Versions V5 and V3) [Computer Software], September 2009.
- [74] D. P. Raymer. *Aircraft Design: A Conceptual Approach*. AIAA Education Series, 4 edition, 2006.
- [75] M. N. Sholz. Estimating the Oswald Factor from Basic Aircraft Geometrical Parameters. *Deutscher Luft- und Raumfahrtkongress*, 2012.
- [76] O. Gur, H. Mason, and J. A Schetz. Full-Configuration Drag Estimation. *Journal of Aircraft*, 47(4):1356–1367, July-August 2010.
- [77] D. Simos. *Piano User’s Guide*. Lissys Ltd, 2008.
- [78] D. Simos. Piano 5 for Windows (Version V.5.3) [Computer Software]. Lissys Ltd. <http://www.lissys.demon.co.uk/>, December 2014.
- [79] I. Celik. Procedure for Estimation and Reporting of Dcretization Error in CFD Applications. *Journal of Fluids Engineering: Editorial Policy Statement on the Control of Numerical Accuracy*, 2007.
- [80] P. J. Roache. Quantification of Uncertainty in Computational Fluid Dynamics. *Annu. Rev. Fluid. Mech.*, 29:123–160, 1997.
- [81] ANSYS. ANSYS ICEM and ICEM Theory Guide, 2017.
- [82] Cranfield University. TURBOMATCH (Version V.2.0) [Computer Software], March 2015.

- [83] Jane's Aero Engines. Rolls-Royce BR725 , April 2018.
- [84] W. L. MacMillan. *Develoment of a Modular Type Program for the Calculation of Gas Turbine Off-design Performancs*. PhD thesis, Cranfield University, 1974.
- [85] A. Guha. Optimum Fan Pressure Ratio for Bypass Engines with Separate or Mixed Exhausts. *Journal of Propulsion and Power*, 17(5), September-October 2001.
- [86] R. Hill, Rolls-Royce, Dahlewitz. Personal Communication. June 2016.
- [87] P. Laskaridis and P. Pilidis. An integrated Engine-Aircraft Performance Platform for Assessing New Technologies in Aeronautics. In *International Society for Air Breathing Engines (ISABE)*, 2005.
- [88] National Business Aviation Association. NBAA Management Guide. <https://www.nbaa.org/admin/management-guide/>, 2011.
- [89] Business and Commerical Aviation. Purchase Planning Handbook. Aviation Week <http://bca.aviationweek.com/>, 2015.
- [90] W. Blake. Jet Transport Performance Methods. The Performance Training Group, Flight Operations Engineering, Boeing Commercial Airplanes, March 2009.
- [91] A. Nuic. User Manual for the Base of Aircraft Data (BADA). Technical/Scientific Report 11/03/08-08, Eurocontrol, April 2011.