

CRANFIELD UNIVERSITY

Matthew H. Robinson

**The Aerodynamics of Aero-Engine
Nacelles**

School of Aerospace, Transport and Manufacturing

Centre For Propulsion Engineering

Ph.D. Thesis

Academic Year: 2018

Supervisor: Prof. D. G. MacManus

February 2018

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

Abstract

This thesis deals with the aerodynamics of aero-engine nacelles with a focus on the influence of a short and slim nacelle design on the drag performance. As turbofan engines are designed with increasingly reduced specific thrusts in order to improve propulsive efficiency, the fan diameter tends to grow. With a larger fan, the engine weight and nacelle drag grow which may offset the benefit from the reduced specific thrust. It is imperative to determine if a reduced length and thickness nacelle, compared to a conventional design, will enable the viable use of these reduced specific thrust aero-engine designs. The research aims to answer this question with a focus on cruise drag, spillage drag, drag rise and windmill performance of isolated and installed short, and slim nacelles.

An innovative optimisation process was developed with a computational fluid dynamics process included as a means to evaluate nacelle drag. This was applied to different nacelle designs in a novel design space to optimise for cruise and off-design performance with a multi-objective genetic algorithm. The optimisation routine was extensively tested and verified against a number of analytical functions to ensure it could adequately approximate optimal Pareto sets. The optimisation of both axisymmetric and non-axisymmetric nacelles was carried out on drag, spillage and drag rise Mach number as well as on two metrics which control the pressure distribution of the nacelle. Optimal nacelles were then chosen to study the influence of nacelle incidence, the windmill condition and installation onto an aircraft on the drag performance and to provide a new quantification of these impacts.

The optimisation demonstrated that under cruise conditions it is possible to have compact nacelle designs that offer reductions in drag. For example, a nacelle with a 23% reduction in length resulted in a 22% reduction in nacelle drag. However, these compact designs are more sensitive to off design condition. Specifically the spillage drag at a required drag rise Mach number of 0.87 could be 9 times higher for the reduced length nacelle. Nonetheless, it is possible to create a nacelle at the shortest length tested which had spillage of less than 6% of the cruise drag and met all requirements on drag rise to cruise at a Mach number of 0.85. This was enabled by an increase in the trailing edge radius such that it was equal to the highlight radius which improved the wave drag characteristics. Whilst the shortened nacelle was viable at low incidence, the increased wave drag resulted in the drag benefit relative to the conventional design being negated by an incidence of 6 degrees. In addition, this reduced length nacelle experienced separation at the end of runway windmill

condition at 22 degrees, which is below the requirement of 30 degrees. Once installed on an aircraft the impact of reducing the nacelle length was a decrease in overall cruise aircraft drag of 3%. These studies demonstrate that there is a significant cruise benefit available from a short nacelle but that the off design conditions, most notably windmill requirements, will need to be addressed.

Acknowledgements

Firstly, I would like to thank Professor David MacManus for his extensive assistance throughout the time of this project. His guidance has been vital in the completion of this work. Christopher Sheaf and Dr. Nicholas Grech have also both provided invaluable contributions to the understanding and contextualisation of the work. The project benefited enormously from being completed in collaboration with a close team to whom I have enormous gratitude for their assistance, cooperation and patient explanations. In particular I would like to thank Dr. Robert Christie, Dr. Tomasz Stankowski, Dr. Alexander Heidebrecht, Santiago Ramirez, John Otter, and Dr. Ioannis Goulos. I would also like to thank the broader network within the department and the UTC for providing support throughout the project.

I have met a number of remarkable people during my time in Cranfield and to them I am very appreciative for making this process inordinately less arduous. Finally, I would like to thank my family for their support throughout the 3 years I spent doing this PhD. In particular, I would like to thank my sister and her entire family for welcoming me into their home as I was just getting started.

Contents

Acknowledgements	v
Abstract	v
List of Contents	xi
List of Figures	xix
List of Tables	xxi
Nomenclature	xxv
1 Introduction	1
1.1 Rationale	1
1.2 Aims and Objectives	4
1.2.1 Overall aim	4
1.2.2 Research objectives	4
1.2.3 Novelty Claims	4
1.3 Thesis structure	5
2 Literature Review	7
2.1 Nacelle design and drag performance	7
2.1.1 Spillage and windmill	9
2.1.2 Drag extraction	11
2.1.3 Geometrical definition	12
2.2 Engine Integration	13
2.3 Previous nacelle design space exploration	19
2.4 Optimisation	26
2.5 Gap in Knowledge	31
3 Methodology	33
3.1 Computational Fluid Dynamics Methods	33
3.1.1 Computational domain	35
3.1.2 Independence studies	37
3.2 Engine Models and Geometry Definition	37
3.2.1 Cycle Modelling	37
3.2.2 Nacelle Parameterisation	38
3.3 Installation positions	42
3.4 Thrust/Drag Accounting	43

3.4.1	Isolated Engine Thrust/Drag Bookkeeping	43
3.4.2	Wind Tunnel Corrections	45
3.4.3	Full aircraft Thrust/Drag Bookkeeping	46
3.4.4	Key metrics	50
3.4.4.1	Drag rise	50
3.4.4.2	Spillage	51
3.4.4.3	Pressure coefficient metrics	51
3.5	Optimisation Method	53
3.5.1	Key metrics	53
3.6	Conclusions	55
4	Validation and Verification	57
4.1	CFD Validation	57
4.1.1	Axisymmetric nacelle aerodynamic validation	60
4.1.2	3D nacelle aerodynamic validation	64
4.1.3	Intake aerodynamic validation	69
4.1.3.1	Geometry	69
4.1.3.2	Validation	69
4.1.3.3	Influence of turbulence model and discretisation order	72
4.2	Optimisation Verification	73
4.2.1	Convergence to optima	73
4.2.2	Mutation operator	77
4.2.3	Initial generation size	79
4.2.4	Initial generation sampling	83
4.2.5	Additional tests	86
4.3	Conclusions	86
5	Axisymmetric Engine Design Studies	89
5.1	Configuration Testing	89
5.2	Baseline Engine Performance	101
5.2.1	Influence of freestream Mach number	105
5.2.2	Influence of engine mass flow and power setting	108
5.3	Axisymmetric nacelle design by full-factorial approach	112
5.4	An improved parameterisation	115
5.5	Axisymmetric nacelle optimisation	123
5.5.1	Optimisation on existing design	123
5.5.2	Optimisation on Novel Nacelles	134
5.5.3	Optimisation with directly optimised pressure distributions	138
5.5.4	Influence of length and trailing edge offset on optimal designs	141
5.6	Analysis of design parameters	145
5.7	Influence of Incidence on Aerodynamic Performance	146

5.8	Influence of Windmill on Aerodynamic Performance	152
5.9	Influence of Installation on Aerodynamic Performance	159
5.9.1	Clean Wing	161
5.9.2	E2 Engine	163
5.9.2.1	Influence of Freestream Mach Number	166
5.9.3	E3 engine	172
5.9.4	Influence of Installation Angle	177
5.9.5	Net Propulsive Force	181
5.10	Conclusions	184
6	Non-axisymmetric Engine Design Studies	187
6.1	Baseline non-axisymmetric design	187
6.2	Fidelity reduction to enable further studies	189
6.3	Non-axisymmetric optimisation	194
6.4	Influence of Incidence	203
6.5	Influence of Installation	207
6.5.1	Baseline Position	207
6.5.2	Close-coupled Position	210
6.6	Conclusions	214
7	Conclusions and Future Work	217
7.1	Conclusions	217
7.1.1	Conclusions on drag evaluation	217
7.1.2	Conclusions on optimisation method	218
7.1.3	Conclusions on short nacelle performance	219
7.2	Future Work	221
	References	229
A	Summary of mesh and domain independence studies	231
A.1	Mesh independence	231
A.1.1	2D axisymmetric isolated nacelle studies	232
A.1.2	3D isolated nacelle studies	232
A.1.3	3D installed studies	233
A.1.4	3D isolated nacelle studies	233
A.2	Domain independence	234
A.2.1	2D axisymmetric isolated nacelle studies	234
A.2.2	3D isolated nacelle studies	234
B	Summary of additional validation studies	237
B.1	Effect of scheme order	237
B.2	Intake validation with different turbulence models	238

C	Prediction of sting drag coefficient	241
D	Response surface model based optimisation	247
D.1	Influence of initial sample size	248
D.2	Influence of constraints	250
D.3	Influence of weighting the objective functions	252
E	Additional non-axisymmetric optimisation	259

List of Figures

1.1	Demonstration of the trend in aero-engine fan size ^{1,2}	3
1.2	Difficulties with increased fan diameter ³	3
2.1	Thrust and drag split for an aero-engine	8
2.2	Nacelle geometry parameterisation based on ESDU ⁴ and pre-entry and post exit streamtubes	13
2.3	Common Research Model aircraft simulated at $\alpha_{a/c} = 2.5^\circ$, $M_\infty =$ 0.82 , $Re = 45 \times 10^6$ with (A) baseline engine, (B) 23% larger fan diameter engine ⁵	15
2.4	Methodology for the extraction of interference drag from the mea- sured data ⁶	17
2.5	Example of measured drag for cowls at lowest vertical height, showing the installation drag, isolated nacelle drag and interference drag ⁶ . .	18
2.6	Design space of current nacelle designs	20
2.7	Nacelle design in Howe and Wynosky ⁷	21
2.8	Influence of nacelle length and thickness on wave drag ⁷	22
2.9	Parametric design space with Dassonville data ⁸ in terms of nacelle drag	23
2.10	Parametric design space with Dassonville data ⁸ in terms of drag rise Mach number	23
2.11	Nacelle designs with some existing designs	25
2.12	Nacelle designs for Zimbrick study ⁹	25
2.13	Influence of BPR and nacelle design in Zimbrick study, based on Zimbrick ⁹	26
2.14	Influence of r_{TE}/r_{hi} on nacelle drag in Zimbrick study, based on Zim- brick ⁹	26
2.15	NSGA-II Evolutionary Ranking Algorithm ¹⁰	28
2.16	Different mutation methods	30
2.17	Different crossover methods	30
3.1	Example of isolated engine mesh	35
3.2	Example of installed engine mesh	36
3.3	Fan cowl parameterisation	41
3.4	Schematic of hybrid iCST parameterisation	41

3.5	Schematic of 3D nacelle geometric parameters	42
3.6	Engine installation positions relative to open source locations based on Stankowski ¹¹	43
3.7	Thrust and drag accounting system	44
3.8	Coordinate systems present in the installed configuration for aerodynamic forces, aircraft and engine ¹¹	47
3.9	Definition of installation drag and interference drag	49
3.10	Distribution of pressure coefficient and pressure coefficient gradient	52
3.11	Routine for Intake and Nacelle Genetic Optimisation flowchart	54
3.12	Schematic of hypervolume in two dimensions, based on Deb ¹²	55
4.1	Validation nacelles with some existing designs	59
4.2	Nacelle/sting configuration (NS)	61
4.3	Drag coefficient of nacelle and sting section ($C_{D'}$) as a function of freestream Mach number and MFCR for the V1 nacelle in the nacelle/sting configuration (NS) from CFD analysis and experimental data	62
4.4	Drag coefficient (C_{Dc}) as a function of freestream Mach number and MFCR for the V2 cowl in the nacelle/sting configuration (NS) from CFD analysis and experimental data	64
4.5	M_{DR} as a function of $MFCR$ for the V2 nacelle from both CFD and experimental data evaluated by two criteria for drag rise	65
4.6	M_{isen} as a function of x/L_{nac} for the V2 nacelle from both CFD and experimental data across a range of freestream Mach numbers (M_∞)	66
4.7	Drag coefficient (C_{Dc}) as a function of freestream Mach number V2 cowl in the nacelle/sting configuration (NS) from CFD analyses, in 2D and 3D, and experimental data	67
4.8	Isentropic Mach number distributions as a function of axial length for the V2 nacelle in the nacelle/sting configuration (NS) from CFD analysis, computed in 2D CFD and 3D CFD, and experimental data	68
4.9	Drag coefficient (C_{Dc}) as a function of freestream Mach number in the nacelle/sting configuration (NS) from CFD analyses and experimental data.	70
4.10	Isentropic Mach number as a function of axial length for the V2 cowl in the nacelle/sting configuration (NS) from CFD analysis and experimental data.	71
4.11	Isentropic Mach distributions from CFD method and measurement at $M_\infty = 0.25$, $WRR=2133 \text{ lbs}^{-1}$ at $\alpha = 33^\circ$ at three azimuthal positions	73
4.12	Isentropic Mach distributions from CFD method and measurement at $M_\infty = 0.25$, $WRR=2142 \text{ lbs}^{-1}$ at $\alpha = 33^\circ$ at three azimuthal positions	74
4.13	Isentropic Mach distributions from CFD method and measurement at $M_\infty = 0.25$, $WRR=2129 \text{ lbs}^{-1}$ at $\alpha = 33.25^\circ$ at three azimuthal positions	75

4.14	Highest, mean, and lowest objective values in the mating pool at given generations for the Booth function as calculated by the NSGA-II algorithm	76
4.15	Optimal Pareto sets for analytical optimisation problems	78
4.16	Comparison of non-uniform mutation and Gaussian mutation for the Booth function	80
4.17	Median hypervolume of zeroth and final generations with IQR and range for 40 optimisation analyses of the Kursawe function with (a) $i_{gen0} = 40$, $N=100$ and (b) $i_{gen0} = 400$, $N=91$	81
4.18	Influence of initial generation size, generation number and Gaussian mutation size on the ratio of the true hypervolume, relative to (2,2,2), determined across 50 independent optimisations of the DTLZ2 problem	82
4.19	Influence of initial generation size, generation number and sampling method on the ratio of the true hypervolume, relative to (2,2,2), determined across 50 independent optimisations of the DTLZ2 problem	85
5.1	Configurations tested for nacelle models. Green lines indicate inviscid walls.	90
5.2	Change in pressure coefficient caused by the presence of the sting at $M_\infty = 0.87$, $MFCR = 0.7$ relative to a nacelle with no sting (N) . .	93
5.3	Drag coefficient as a function of M_∞ for V2 at $MFCR = 0.7$ for CFD analysis with (NS) and without sting (N), with a cylindrical sting (NCS) and with an inviscid cylindrical sting (NICS)	93
5.4	The influence of (A) freestream Mach number and (B) MFCR on drag for the V1 nacelle modelled with the sting (NS) and without (N)	94
5.5	The influence of freestream Mach number on drag for the V1 nacelle modelled with the sting (NS) and without (N)	95
5.6	Drag decomposition for CFD results from five configurations of E2 engine at $M_\infty = 0.82$, $MFCR = 0.75$, $\alpha_{CL} = 0^\circ$	97
5.7	(A) Mach contours for E2 datum nacelle and (B)-(D) relative Mach number increase from the NE/FS/RE configuration, at $M_\infty = 0.82$, $MFCR=0.75$, $\alpha_{CL} = 0^\circ$	98
5.8	CFD results for E2 nacelle in different configurations evaluated in terms of nacelle drag coefficient (c_{Dnac}) as a function of M_∞ at constant $MFCR = 0.75$, $\alpha_{CL} = 0^\circ$	98
5.9	Three engine nacelle designs at two different aerodynamic conditions	104
5.10	Three baseline engine nacelle designs	105
5.11	Three engine nacelle designs at $MFCR = 0.75$ as a function of M_∞	107
5.12	D_{nac}/F_{Nref} as a function of M_∞ for the three baseline nacelles . . .	108
5.13	The isentropic Mach number distributions of three engine nacelle designs as a function of $MFCR$	110
5.14	Streamlines from the trailing edge of the nacelle	111
5.15	Drag performance as a function of $MFCR$ for the three baseline nacelles	112

5.16	2 dimensional full factorial design space with example produced engines and validation engines	113
5.17	Subset of designs from full factorial of engine nacelles	115
5.18	Cruise drag ($c_{Dcruise}$) as a function of non dimensional length (l_{nac}/r_{hi}) and thickness ratio (r_{max}/r_{hi}) for a full factorial of engine designs	116
5.19	Cruise drag ($c_{Dcruise}$) as a function of non dimensional length (l_{nac}/r_{hi}) and thickness ratio (r_{max}/r_{hi}) for a full factorial of engine designs	117
5.20	Subset of designs from full factorial of engine nacelles	118
5.21	Influence of β on geometrical inflections	119
5.22	Influence of κ_{TE} on geometrical inflections and curvature distributions	120
5.23	Influence of r_{if} and β on geometrical inflections	122
5.24	Influence of r_{if} and β on geometrical inflections with a hybrid iCST fan cowl	123
5.25	Evolution of the mating pool for the V2 nacelle optimisation	127
5.26	Evolution of the mating pool for the V2 nacelle optimisation	128
5.27	Hypervolume convergence for the V2 nacelle optimisation case	129
5.28	Pareto optimal set for optimisation of the validation nacelle 2 in objective space	129
5.29	Example optimal design cowl Mach number distribution	130
5.30	Pareto set in the design space coloured by $C_{Dcruise}$	131
5.31	Pareto set in the design space coloured by M_{DR}	132
5.32	Pareto set in the design space coloured by C_{Dspill}	133
5.33	Example optimal fan cowl Isentropic Mach number profiles at a length of $l_{nac}/r_{hi} = 3.8$ for different required M_{DR} and $C_{Dcruise}$	135
5.34	Pareto optimal sets for optimisations performed at two nacelle lengths, plotted as a function of the performance metrics	137
5.35	Example optimal Isentropic Mach number profiles at a length of $l_{nac}/r_{hi} = 3.1$ with (a) a required $M_{DR} \geq 0.86$ and (b) a required $M_{DR} \geq 0.87$	140
5.36	C_{Dnac} as a function of M_∞ for designs with different $C_{Dcruise}$ criteria for a nacelle length of $l_{nac}/r_{hi} = 3.1$ with and without pressure controls at a required $M_{DR} \geq 0.87$	141
5.37	Pareto optimal sets for optimisations performed at different lengths and trailing edge radii, plotted as a function of the performance metrics	142
5.38	Minimum achievable (a) $c_{Dcruise}$ and (b) c_{Dspill} for a given M_{DR}	144
5.39	Example optimal Isentropic Mach number profiles for all trailing edge radii at a length of (a) $l_{nac}/r_{hi} = 3.1$ and (b) $l_{nac}/r_{hi} = 3.8$	147
5.40	Pareto optimal sets for optimisations performed at different lengths and trailing edge radii, plotted as a function of the design metrics. Dashed lines indicate median value and dotted lines indicate the quartiles. Number of individuals are indicated by the relative thickness of the distributions	148

5.41 (a) Curvature distribution and (b) Rate of change of curvature distribution for six chosen optimal designs for different lengths and trailing edge radii	149
5.42 C_{Dnac}/F_{Nref} as a function of α_{CL} for three different nacelle designs	150
5.43 Isentropic Mach number distributions for three different engine designs as a function of incidence (α_{CL}) in the cruise condition, E2: $M_\infty = 0.82$, MFCR = 0.75; E3: $M_\infty = 0.852$, MFCR = 0.7	151
5.44 M_{peak} as a function of α_{CL} for three different nacelle designs	152
5.45 MFCR as a function of M_∞ for three engine cycles in the windmill condition based on ESDU data ¹³	155
5.46 Influence of M_∞ and h on semi-dimensional windmill drag ($c_D A$ for four different engine designs	156
5.47 Influence of M_∞ and h on semi-dimensional windmill drag ($c_D A$ for four different engine models	157
5.48 Isentropic Mach distributions for four engine designs under the windmill condition	158
5.49 Influence of α_{CL} on windmill drag (D_{nac}) for four different engine designs in an end of runway condition ($M_\infty = 0.25$, $h = 0m$)	159
5.50 Contours of X-Velocity (U_x) for three different engine designs as a function of incidence (α_{CL})	160
5.51 Drag polar for the CRM clean wing aircraft at $M_\infty = 0.82$ and $M_\infty = 0.85$	161
5.52 $C_L - \alpha$ curve for the CRM clean wing aircraft at $M_\infty = 0.82$ and $M_\infty = 0.85$	162
5.53 Isentropic Mach distributions on the CRM wing in the engine installation plane at two freestream Mach numbers across a range of incidences	163
5.54 Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E2 engine installed in the datum position	165
5.55 Stackup of drag forces acting on the isolated and installed E2 engine and airframe	165
5.56 Isentropic Mach distributions on the E2 nacelle at two incidences, in an isolated and installed (C3) configuration at $M_\infty = 0.85$	166
5.57 Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.82$ and $M_\infty = 0.85$ and for the CRM with the E2 engine installed in position C3	169
5.58 Stackup of drag forces acting on the isolated and installed E2 engine and airframe	169
5.59 Isentropic Mach distributions on the CRM wing in the engine installation plane at two freestream Mach numbers across a range of incidences with and without engine E2 installed in the baseline position (C3)	170

5.60	Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the clean wing (WBT0) and with E2 installed in C3 configuration	171
5.61	Isentropic Mach distributions on the E2 nacelle at two incidences, in an isolated and installed (C3) configuration at $M_\infty = 0.82$	172
5.62	Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.82$ and $M_\infty = 0.85$ and for the CRM with the E2 and E3 engines installed in position C3	174
5.63	Stackup of drag forces acting on the isolated and installed E2 and E3 engine and airframe in position C3	174
5.64	Isentropic Mach distributions on the CRM wing in the engine installation plane at two freestream Mach numbers across a range of incidences with and without engine E3 installed in the baseline position (C3)	175
5.65	Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the clean wing (WBT0) and with E3 installed in C3 configuration	176
5.66	Isentropic Mach distributions on the E3 nacelle at two incidences, in an isolated and installed in the datum position	177
5.67	Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E2 and E3 engines installed in position C3 with two installation angles (α_{inst})	178
5.68	Isentropic Mach distributions on the CRM wing in the engine installation plane for two installation angles with engine E3 installed in the baseline position (C3)	179
5.69	Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the clean wing (WBT0) and with E3 installed in C3 configuration	180
5.70	Isentropic Mach distributions on the E3 nacelle for two installation angles	181
5.71	Stackup of forces acting on the isolated and installed E2 engine and airframe in datum position	182
5.72	Stackup of forces acting on the isolated and installed E3 engine and airframe in datum position	183
5.73	Isolated, $\alpha_{CL} = 4^\circ$ and installed, $\alpha_{a/c} = 2^\circ$, flowfields for the E3 engine installed in position C3	184
6.1	Comparison of axisymmetric and sub-optimal non-axisymmetric nacelle designs at $M_\infty = 0.85$, $MFCR = 0.7$	188
6.2	Mesh adaptations for reduced fidelity strategy	190
6.3	Comparison of high fidelity and low fidelity meshes for a single nacelle design	192
6.4	Comparison of high fidelity and low fidelity meshes for a full generation in terms of cruise drag	193

6.5	Comparison of high fidelity and low fidelity meshes for a full generation in terms of drag rise Mach number	193
6.6	Comparison of high fidelity and low fidelity meshes for a full generation in terms of spillage drag	194
6.7	Filtered design space for the 3D optimisation for topline and bottomline	196
6.8	Sketch of aerolines where red is fixed, blue is variable in the optimisation	197
6.9	Final Pareto set for the 3D non-axisymmetric optimisation in objective space	198
6.10	Comparison of sideline isentropic Mach number distributions for all designs in generation 0 and 19	199
6.11	Final Pareto set for the 3D non-axisymmetric optimisation in design variable space. Violin plots indicate the distribution of the population, as well as maximum, minimum and median values	200
6.12	Four Pareto optimal designs from the non-axisymmetric optimisation	202
6.13	Comparison of low and high fidelity solutions for the chosen solution	203
6.14	C_{Dnac}/F_{Nref} as a function of α_{CL} for four different nacelle designs .	205
6.15	Isentropic Mach number distributions for the optimal non-axisymmetric engine designs as a function of incidence (α_{CL}) in the cruise condition	206
6.16	M_{peak} as a function of α_{CL} for four different nacelle designs	207
6.17	Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E3 and E3NA engines installed in the datum position	208
6.18	Stackup of forces acting on the isolated and installed E3NA engine and airframe in the datum position	208
6.19	Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the CRM with the E3 and E3NA engines installed in the datum installation position	209
6.20	Isentropic Mach distributions on the E3NA nacelle in isolation and installed in datum position	210
6.21	Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E3 and E3NA engines installed in the close-coupled position	212
6.22	Stackup of forces acting on the isolated and installed E3NA engine and airframe in close-coupled position	212
6.23	Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the CRM with the E3 and E3NA engines installed in close-coupled configuration	213
6.24	Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the CRM with the E3 and E3NA engines installed in close-coupled configuration	214
B.1	Isentropic Mach distributions for datum mesh at $M=0.25$, $WRR=2133lb/s$ at 33 degrees at 2nd order discretisation and 3rd order discretisation	238

B.2	Isentropic Mach number distributions of Spalart-Allmaras results over lower intake lip at centreline for the I1 intake case, with $M_\infty = 0.25$, $\alpha_{CL} = 33^\circ$, WRR=2133 lb/s and with datum mesh (6.7 million cells)	239
B.3	Isentropic Mach number distributions of Transition-SST results over lower intake lip at centreline for the I1 intake case, with $M_\infty = 0.25$, $\alpha_{CL} = 33^\circ$, WRR=2133 lb/s and with datum mesh (6.7 million cells)	239
C.1	Skin friction drag extracted for sting attached to nacelle V2 CFD data and flat plate estimation	243
C.2	Drag coefficient for CFD analysis and experimental data for nacelle V2	244
C.3	Values of sting drag calculated from CFD and updated flat plate correlations	245
D.1	Population in objective space	248
D.2	Final Pareto optimal set extracted from the RSM plotted as a function of the 3 objective functions	249
D.3	Mean Hypervolume from 50 individual optimizations with the RSM as a function of zeroth generation size and generation number	251
D.4	Example final Pareto optimal set extracted from the RSM (maximum obtained M_{DR})	253
D.5	Example final Pareto optimal sets extracted from the RSM	256
E.1	Final Pareto set for the 3D non-axisymmetric optimisation in objective space	260
E.2	Optimal designs from (A) previous optimisation (Section 6.3) and (B) optimisation with added aeroline control at $\theta = 45^\circ$	261

List of Tables

4.1	Measured and CFD predicted drag rise Mach number for V1	61
4.2	Measured and CFD predicted drag rise Mach numbers for V2 and V3 cowl at incidences between $\alpha_{CL} = 0^\circ$ and $\alpha_{CL} = 6^\circ$	67
5.1	Drag rise Mach number for different configurations of E2 nacelle . .	99
5.2	Cruise design point for engine cycles	101
5.3	Summary of baseline engine nacelle design parameters	101
5.4	Parametric geometry bounds used for the optimisation of the V2 nacelle	124
5.5	Optimisation bounds for nacelle geometric parameters	134
A.1	Results from the CFD simulations of 2-dimensional axisymmetric nacelle meshes produced for independence studies	232
A.2	Results from the CFD simulations of 3-dimensional nacelle meshes produced for independence studies	232
A.3	Results from the CFD simulations of WBT0 configuration at $\alpha_{a/c} =$ 2.5°	233
A.4	Results from the CFD simulations of WBNP configuration at $\alpha_{a/c} =$ 2.5°	233
A.5	Results from the CFD simulations of 3-dimensional nacelle meshes produced for independence studies	234
A.6	2-dimensional axisymmetric nacelle meshes produced for indepen- dence studies	234
A.7	3-dimensional isolated nacelle domain study	235
D.1	Average minimum cruise drag and spillage drag at a given drag rise Mach number range for the different constrained optimizations . . .	252
D.2	Average minimum cruise drag and spillage drag at a given drag rise Mach number range for the different weighted optimisations	258
E.1	Parametric geometry bounds used for the optimisation of the non- axisymmetric nacelle	259

Nomenclature

Abbreviations

AGARD	Advisory Group for Aerospace Research and Development
AR	Aspect Ratio
ARA	Aircraft Research Association
BPR	Bypass Ratio
CFD	Computational Fluid Dynamics
CNPR	Core Nozzle Pressure Ratio
CR	Contraction Ratio
CRM	Common Research Model
CST	Class Shape Transformation
ESDU	Engineering Sciences Data Unit
FNPR	Fan Nozzle Pressure Ratio
FS	Full Scale
iCST	Intuitive Class Shape Transformation
IQR	Interquartile Range
LHS	Latin Hypercube Sampling
MFCR	Mass Flow Capture Ratio
MOO	Multi-Objective Optimisation

MS	Model Scale
N	Nacelle with cylindrical nozzle configuration
NACA	National Advisory Committee for Aeronautics
NASA	National Aeronautics and Space Administration
NE	Nacelle with representative dual stream Exhaust configuration
NPF	Net Propulsive Force
NS	Nacelle with experimental Sting configuration
NSGA-II	Non-dominated Sorting Genetic Algorithm II
OPR	Overall Pressure Ratio
PARSEC	Parameterisation method for airfoil sections
PM	Pressure Matched
RANS	Reynolds Averaged Navier Stokes
RE	Representative Efflux
RINGO	Routine for Intake and Nacelle Genetic Optimisation
RSM	Response Surface Model
SST	Shear Stress Transport
TFN	Through Flow Nacelle
TPS	Turbine Powered Simulator
TSFC	Thrust Specific Fuel Consumption
UHBPR	Ultra-High Bypass ratio
WBT0	Wing/Body/Tailplane configuration
WBT0N	Wing/Body/Tailplane/Nacelle configuration

Greek Symbols

α	Incidence	[°]
----------	-----------	-----

β	Boattail angle	[°]
δ_{ij}	Kronecker delta	[-]
γ	Ratio of specific heats	[-]
κ	Curvature	$[m^{-1}]$
μ	Dynamic viscosity	$[kgm^{-1}s^{-1}]$
μ	Mean	[-]
μ_t	Eddy viscosity	$[m^2s^{-1}]$
ω	Specific rate of dissipation	$[s^{-1}]$
ϕ	Integral force in the drag domain	[N]
ρ	Density	$[kgm^{-3}]$
σ	Standard deviation	[-]
τ	Shear stress	[Pa]
θ	Azimuthal angle	[°]
θ_{droop}	Droop angle	[°]
θ_{scarf}	Scarf angle	[°]

Roman Symbols

$\bar{Y}$	Pareto set	[-]
$\dot{m}$	Mass Flow	$[kgs^{-1}]$
$\tilde{u}$	Mass Averaged Velocity	$[ms^{-1}]$
A	Area	$[m^2]$
C	Coefficient	[-]
D	Drag	[N]
d	Diameter	[m]
f	Objective function	[-]

F_G	Gauge Stream Force	[N]
g	Constraint	[-]
i	Population size	[-]
k	Turbulent kinetic energy	[Jkg^{-1}]
L	Lift	[N]
l	Length	[m]
M	Mach Number	[-]
N	Total number of generations	[-]
n_x	Outward vector resolved in x-direction	
n_y	Outward vector resolved in y-direction	
n_z	Outward vector resolved in z-direction	
p	Static pressure	[Pa]
P_t	Parent generation	[-]
Q_t	Child generation	[-]
r	Radius	[m]
Re	Reynolds Number	[-]
S	Surface area	[m^2]
t	Generation	[-]
V	Velocity	[ms^{-1}]
x	Independent variable	
x_i	Individual in generation	[-]
Y'	Pareto set approximation	[-]

Subscripts

∞	Freestream
----------	------------

<i>a/c</i>	Aircraft
<i>a/f</i>	Airframe
<i>aft</i>	Fan Cowl Afterbody
<i>cowl</i>	Fan Cowl
<i>D</i>	Drag
<i>DR</i>	Drag Rise
<i>fore</i>	Fan Cowl Forebody
<i>hi</i>	Highlight
<i>i</i>	Integer
<i>if</i>	Initial Forebody
<i>inst</i>	Installation
<i>interf</i>	Interference
<i>max</i>	Maximum
<i>nac</i>	Nacelle
<i>peak</i>	Local Maximum Value
<i>post</i>	Post-exit
<i>pre</i>	Pre-entry
<i>th</i>	Throat

CHAPTER 1

Introduction

1.1 Rationale

Turbofan engines are the most prevalent propulsion system for large civil aviation applications. To achieve a given thrust an engine can accelerate a small mass of air to a high velocity or a large mass of air to a relatively lower velocity to achieve a momentum change. The replacement of turbojets and the development of turbofans has followed a trend to reduce the jet velocity. The reason for this is to improve the propulsive efficiency which varies as the inverse of the specific thrust. The propulsive efficiency can be minimised by reducing the ratio of exit velocity to freestream velocity V_e/V_∞ (Equation 1.1).¹⁴ The reduction in the exit velocity for a turbofan is dominated by the ability of the fan to deliver a greater volume of air at a lower velocity than previous engines.¹⁵ To pass more air through the fan it then becomes necessary to increase the fan diameter whilst reducing the fan pressure ratio. This increase in diameter follows the trend of increased bypass ratio which has been followed since turbofans were adopted in civil aviation.¹⁶ If the installation of the engine is not considered, the cycle should then require less fuel to deliver the thrust. A reduction in the fuel burn will benefit the aircraft operator as well as help meet international obligations to reduce emissions.¹⁷⁻²⁰

$$\eta_p = \frac{\text{thrust power}}{\text{propulsive power}} = \frac{TV_\infty}{W_{out}} = \frac{2}{V_e/V_\infty + 1} \quad (1.1)$$

A larger fan diameter causes several integration problems (Figure 1.1). Firstly, the physical problem of fitting a large engine under a wing leads to conflicts between

ground clearance and wing interference. Closer coupling of the engine to the wing is likely to be required in future designs and this will result in a greater level of interaction of the wing and engine aerodynamics.²¹ Secondly, a larger engine will necessitate a nacelle with more surface area and weight. This will result in more drag and reduced payload respectively. These problems may reverse the benefits of higher bypass ratio and make the expected increases in bypass ratio implausible (Figure 1.2). An increase in the drag entails a greater thrust requirement and a more inefficient aircraft. The direct impact of drag has been estimated²² at 0.07% of specific fuel consumption for a 1% reduction in nacelle only drag. Since the nacelle is estimated to contribute between 8-15% of the total aircraft drag^{11,23}, a reduction in full aircraft drag of 1% would be expected to be around an order of magnitude larger. One report estimates the impact of a 1% full aircraft drag reduction as a decrease in operating costs of \$12 million a year per aircraft.²⁴ Evidently, such an improvement in drag would be a major benefit to aircraft operators and would help to meet the planned reduction in harmful emissions.

A proposed solution to the drag and weight issue comes in the form of the 'open rotor' which consists of an unducted fan without a nacelle.¹⁹ Whilst this removes the problem of increased drag, there are a number of other problems which arise, including a noise penalty, integration issues associated with larger diameters, and a safety concern over uncontained fan blade separation.²⁵ Due to these current problems and the potential improvements from geared or variable pitch fans it is expected that open rotors are not an immediate solution.¹⁵ One mitigation to the nacelle drag and weight problems, which is expected to be incorporated in future designs, is that of a slimline nacelle.¹⁵ This name indicates a general departure from the current design space for nacelles and a push for shorter and slimmer designs. It is expected that a reduction in length and thickness will help mitigate the increased drag and weight of large fan diameters. The evaluation of low specific thrust engines will therefore require an understanding of the limits of the nacelle design space.

The reduction in nacelle size will include an expected reduction in the intake length.^{26,27} Whilst this reduction will include weight savings relative to a conventional length intake, there are associated problems which must be addressed to enable this technology. These issues typically arise at very off-design conditions where the intake is required to avoid separation whilst operating under high incidences or crosswind. In addition, a reduction in nacelle drag may be outweighed by a fan performance penalty due to inlet distortion²² and highlights the importance of understanding these trade offs for future engines. In addition to the intake, a

reduced length nacelle may have an impact on the exhaust system. This may be evident as a shorter bypass duct and a greater interaction between the exhaust flow and the aircraft. Since an improvement in the nozzle efficiency may have a much greater impact on the specific fuel consumption than a reduction in drag²⁸, it is also vital that the influence of the exhaust system is considered.

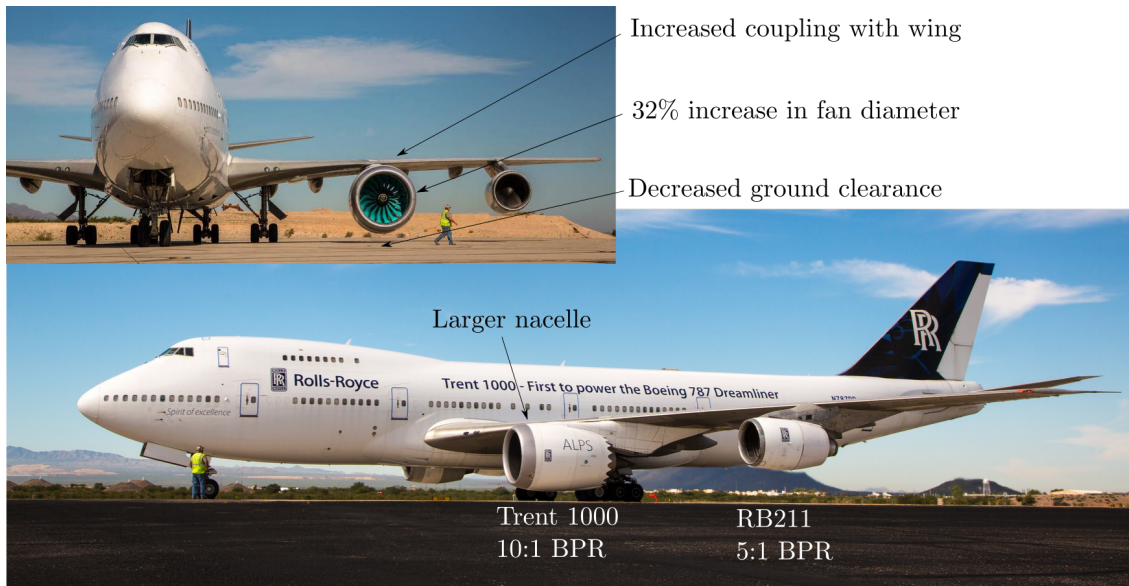


FIGURE 1.1: Demonstration of the trend in aero-engine fan size^{1,2}

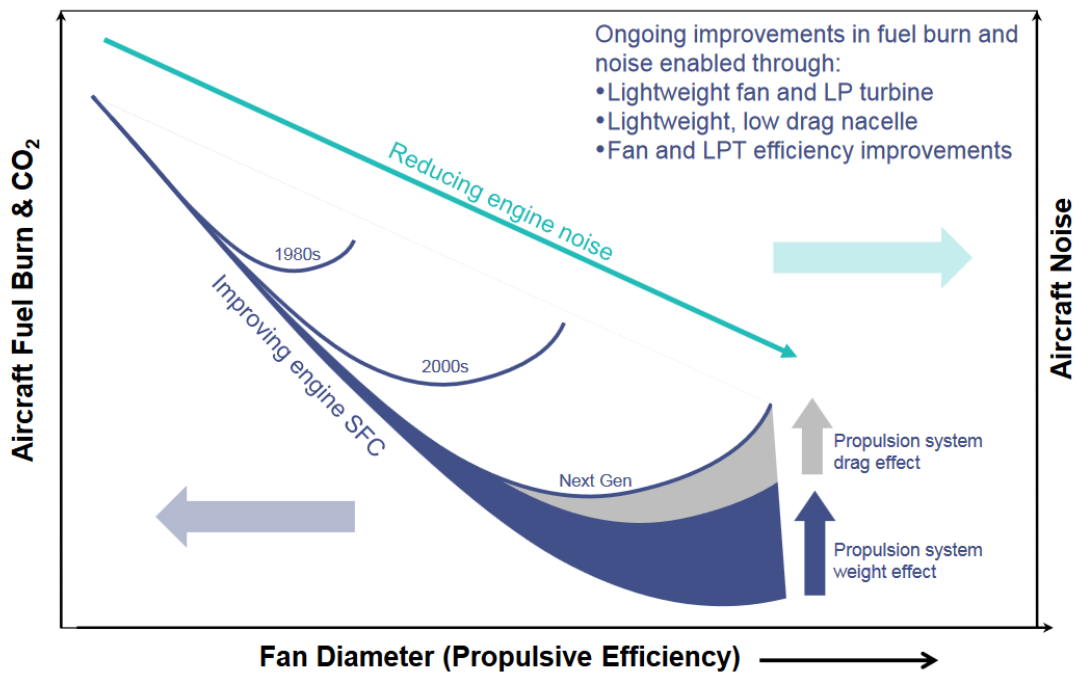


FIGURE 1.2: Difficulties with increased fan diameter³

1.2 Aims and Objectives

1.2.1 Overall aim

The aim of this project is to evaluate the novel nacelle design space in terms of short and slim nacelles and to assess the viability of nacelle size reduction.

1.2.2 Research objectives

The project will achieve the stated aim (section [1.2.1](#)) through the fulfilment of the following objectives:

1. To validate a computational method for engine aerodynamics and performance against experimental data.
2. To establish a method which allows for the design of a novel, short and slim nacelle to allow for aerodynamic analysis.
3. To populate a design space for novel aero-engine nacelles parametrically and compare the aerodynamic performance to a more conventional design in cruise and off design.
4. To determine computationally the effects of an installation for an ultra-high bypass ratio (UHBPR) engine on a transonic civil transport aircraft with a focus on the improvements available from a novel nacelle configuration.

1.2.3 Novelty Claims

Through the fulfilment of the aim and the completion of the objectives, the following novel contributions were made in the thesis:

- An optimisation method has been developed, demonstrated and tested for nacelle aerodynamic design. This process has been developed for 2D axisymmetric and for 3D based on a flexible parametric geometry definition, computational fluid dynamic analyses and a genetic algorithm.

- A novel design space for nacelles has been explored as a function of 7 design variables and 5 aerodynamic variables with an optimisation approach. This study demonstrated the minimum length possible and the trailing edge radius required for a cruise Mach number of 0.85.
- Studies on the sensitivity of nacelles in the novel design space to incidence, installation and windmilling have been performed. The expected influence of each of these for future UHBPR engine configurations is quantified.
- An approach has been proposed and implemented to limit the number of inflections in the pressure distributions produced by an optimisation of nacelle aerodynamics. This reduces the known problem of aerodynamic optimisation producing unconventional aerodynamic distributions.
- The NSGA-II algorithm has been tested on a suite of problems for a range of optimisation parameters. An alternative initialisation method is demonstrated using a Latin hypercube sample to initiate the process.
- An application of optimisation was demonstrated for a non-axisymmetric parametric nacelle. This produced designs with improved peak Mach numbers relative to the axisymmetric case and allowed a quantification of the improved incidence performance.
- The impact of short nacelles on an overall aircraft drag was demonstrated and compared to a conventional nacelle design. Sensitivity to installation position, installation angle and off design is quantified.

1.3 Thesis structure

The remainder of this thesis details the project undertaken to meet the stated aim. Firstly, a literature review was carried out which is the focus of chapter 2. This culminates in a description of the gap in the literature which it is expected that this project will fill. Chapter 3 describes the specific methods which have been used to achieve this and how they relate to current best practice. Novel methods of nacelle design are also detailed within this chapter, which meets objective 2. The methods which have been chosen were then validated against existing data and the implementation of the novel methods verified to ensure they are well calibrated; this is the focus of chapter 4. Chapter 4 therefore meets objective 1. Chapter 5 and

6 consist of the body of work undertaken to establish the design space available for novel nacelle designs. These chapters collectively complete objectives 3 and 4, as well as ensure that the method developed to meet objective 2 was appropriate. The final chapter summarises the conclusions which were reached through this project and their expected impact on engine design. In addition, this chapter summarises how the thesis has filled the gap determined in the literature and recommends some further avenues of work.

CHAPTER 2

Literature Review

A literature review has been carried out to determine the previous work undertaken in the assessment of nacelle drag prediction, installation effects and on novel nacelle concepts. In addition a review of the current work on nacelle optimisation and a background of the tools used in this thesis are presented.

2.1 Nacelle design and drag performance

A nacelle is designed with several key requirements in mind which must be fulfilled throughout all operating conditions. The intake of the nacelle must deliver a mass of air to the fan with a low amount of distortion and at an appropriate Mach number which will not result in detrimental separation of the flow on the fan blades. The design of the external cowl is typically dictated by the desired drag performance of the engine. A conventionally designed supercritical nacelle is characterised by a curvature driven acceleration over the forebody followed by constant deceleration.²⁹ For nacelles designed in this way, the peak Mach number reached in the initial acceleration is minimised. A nacelle in a transonic airflow will experience local regions of supersonic flow which typically terminate in shock waves and contribute to the nacelle drag. The assessment of the drag is subject to bookkeeping between thrust contributions and drag contributions to the overall net propulsive force (NPF). The specifics of this are considered in a later section of this thesis (section 3.4), but the demarcation between thrust and drag is typically made at the incoming and outgoing streamtubes, i.e. changes to the internal flow constitute thrust and changes made to the external flow constitute drag (Figure 2.1).^{30–33}

As the freestream Mach number (M_∞) is increased, the peak local Mach number (M_{peak}) on the nacelle increases, which results in stronger shocks and worsens the associated wave drag penalties. As a result of these sensitivities, the off-design aspects are of great importance, such as the nacelle performance at higher cruise Mach numbers. The Mach number at which a rapid increase in drag with freestream Mach number arises is termed the drag rise Mach number and is often defined based on a drag gradient criterion as a function of the freestream Mach number. Though criteria for nacelles are not found in the published literature, several cases for full aircraft drag rise are available. For example, Roskam³⁴ defines the drag rise Mach number (M_{DR}) based on a drag gradient (equation 2.1). The level of the criterion varies and Shevell³⁵ reports a value of 0.05 based on the same gradient. Roskam also discusses an alternative to the gradient based criteria as an increase of 0.002 in drag coefficient above the incompressible level. M_{DR} is a key parameter for the assessment of nacelle configurations as a designer will tend to avoid being within 0.02 of the drag rise during cruise³⁶. Each of these criteria have potential downsides. Where a gradient is considered, the values will be dependent on the area used to non-dimensionalise the drag. Conversely, an offset requires a level from which it can be measured and the incompressible level may be impractical or inconsequential for a transonic nacelle design. Ultimately, the influence of wave drag would need to be considered across a full flight envelope and trade-offs balanced, however at a preliminary design stage a measure of the drag rise Mach number is far more expedient.

$$\left. \frac{\partial C_D}{\partial M_\infty} \right|_{M_{DR}} = 0.1 \quad (2.1)$$

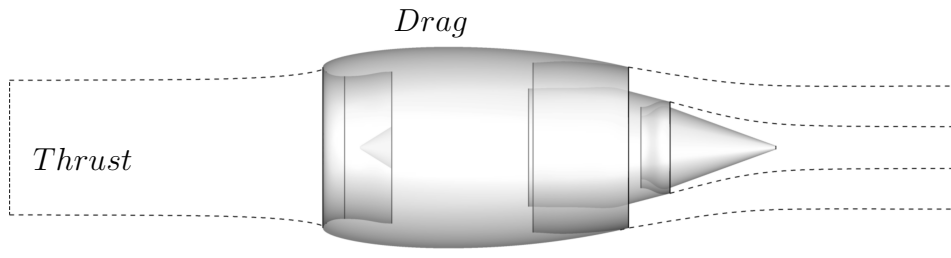


FIGURE 2.1: Thrust and drag split for an aero-engine

2.1.1 Spillage and windmill

The streamtube which enters an aero-engine varies with the aerodynamic conditions and the engine mass flow requirement. Mass flow capture ratio (MFCR) is the ratio of the cross sectional area of this streamtube at upstream infinity divided by the highlight plane area (equation 2.2). An engine intake is sized for the maximum required mass flow so usually operates at a $MFCR < 1$ during the cruise.³⁷

$$MFCR = A_{\infty}/A_{hi} \quad (2.2)$$

Spillage can be understood as a drag incurred by the influence of the MFCR. In potential flow, the drag of the nacelle from the upstream infinity plane integrated to the downstream infinity plane is zero because it is an infinite body.³⁸ This indicates that when the $MFCR < 1$, in potential flow, the rearwards force acting on the pre-entry streamtube (ϕ_{pre}) must be balanced by a forwards acting force on the fan cowl forebody (ϕ_{fore}) between the stagnation point and the maximum radius, $\phi_{pre} - \phi_{fore} = 0$. Similarly, the forwards acting post exit streamtube force (ϕ_{post}) must be balanced by a rearwards acting force on the fan cowl afterbody, $\phi_{aft} - \phi_{post} = 0$. In a real flow however this is not the case and $\phi_{pre} - \phi_{fore} > 0$ resulting in a positive drag term. This drag component can be referred to as spillage drag.³⁹ This definition is useful as the amount of spillage can be determined by comparing the drag at a $MFCR < 1$ with the drag at $MFCR = 1$.

An alternative method for evaluating the spillage drag is proposed by ESDU⁴ which starts by the estimation of a critical area ratio. This is defined as the MFCR at which separation occurs on the fan cowl and below which spillage drag is encountered. In this definition, it is assumed that the drag increase as a response to MFCR above the critical MFCR is negligible as was found for early tests on NACA cowls.⁴⁰ The critical area ratio can be determined for a NACA 1-series fan cowl based on experimental correlations^{4,41}. This definition is less useful as in a transonic flow regime the cowl will experience increased wave drag due to the change in MFCR as the local incidence of the nacelle highlight changes. This will negatively impact the performance of the nacelle. Conversely, throughout the flight a well designed nacelle should not separate and so the spillage encountered below the critical MFCR is less pertinent to overall performance. For example a typical mid-cruise MFCR for a subsonic aircraft is in the region of 0.7³⁷ which may be reduced throughout the cruise due to the mass loss of the overall aircraft.

In the event of an engine failure, the engine mass flow will be greatly reduced. In the windmill condition, the fan rotor will freely rotate and large losses will be incurred by the bypass outlet guide vane which operate far off design. This results in an internal drag force for which there are several methods of predicting.^{13,42-44} These are generally based around either empirical correlations¹³, underlying physics⁴² or computational simulations.^{43,44} The spillage drag in this condition will be much larger than in cruise as the MFCR can reduce as low as $MFCR = 0.3$.⁴⁵ Few studies have considered external nacelle aerodynamics in this region of MFCR. Whilst a study⁴⁶ that looked at this low MFCR noted an increase in drag due to MFCR of the order of between 25% and 200% relative to a cruise MFCR, none focussed on the impact this would have for a windmill event at a range of Mach numbers or with different length nacelles. The difference in forebody design resulted in the range between 25% and 200% which indicates the importance of having a good pressure distribution.⁴⁶ In addition, the influence of shortened nacelles on this condition has yet to be considered, which will likely have a notable impact if worsened spillage performance is a general characteristic of these novel nacelles. It is important for engine designers to reduce the drag penalty associated with these operating regimes. For engines of large diameters the windmill drag can create an excessive yaw moment resulting in the need for a larger vertical stabiliser. For instance, one study⁴⁷ identified that an increase in fan diameter from 2.84m to 3.78m on the B777-200ER at $M_\infty = 0.2$ resulted in an increase of wind-mill drag of over 150% and an increase in the tail size of 15% was required to balance this yaw moment. The critical points of the flight envelope for nacelle design are the start of descent when the engine is near idle condition, the wind-milling condition, representing a shaft failure and the top of climb.⁴⁸ The former two of these conditions are characterised by a low MFCR, leading to high spillage drag. The latter is characterised by low speed and high MFCR and is of most importance for the intake flow. One paper notes that the flight envelope in an emergency operation extends up to 30° incidence up to $M_\infty = 0.25$.⁴⁹ These studies all highlight the importance of the spillage drag on the performance of the nacelle, whilst it has also been noted that this worsens for slim nacelles.⁴⁸

2.1.2 Drag extraction

There are many methods of extracting drag from a computational fluid dynamics (CFD) solution which are split broadly into far field methods and near field methods.⁵⁰ Both of these approaches aim to exclude spurious drag as far as possible. Near field drag extraction methods are based on the integration of the forces acting on an object tangential to the flow direction and can formally be defined by equation 2.3.⁵⁰ The first term in the brackets (equation 2.3) describes the pressure force acting on the nacelle whilst the other three terms are the viscous forces. The pressure term includes drag contributions from shock waves and boundary layer displacement effects making it impossible to separate these influences in a nearfield method. The other problem with a near field formulation is the reliance on mesh density and turbulence model to ensure adequate resolution in the pressure integration and to avoid numerical dissipation which leads to spurious drag.⁵¹ Extraction of the streamtubes to define the boundaries of thrust and drag is very difficult in 3D CFD especially when interference from a wing pressure field is added. The difficulty for the pre-entry streamtube can be overcome through the use of the modified nearfield method.⁵² The method avoids the need to extract the nacelle leading edge stagnation points through the integration of all wall forces up to the fan face. The internal forces are then subtracted through the subtraction of the gauge stream forces between the fan and the upstream infinity plane.

Conversely, far field methods determine drag through surface integration of a control volume surrounding the object and flow of interest. It is assumed for these methods that the control volume contains all viscous and wave drag effects and that the momentum loss measured at the surfaces is caused by these. The wake integral method is a form of far field drag extraction where the drag is calculated based on the losses measured in a downstream wake. The integral (equation 2.4) is evaluated at a nominal downstream infinity plane, which for practical reasons is taken at a point at which there are no further total pressure losses. This method requires for the wake to be split into regions of area (A) where the local mass flow rate ($\dot{m}$) is known and the local flow velocity (V) can be measured.

$$D = \int \int_{S_{Body}} (-pn_x + \tau_{xx}n_x + \tau_{xy}n_y + \tau_{xz}n_z) dS \quad (2.3)$$

$$D = \int \dot{m} (V_\infty - V) dA \quad (2.4)$$

The wake integral method is a commonly used far-field drag extraction method for wind tunnel calculations given how it can be computed from pressure readings in the wake of an object. However the drag result is determined by extracting a boundary between the readings in the wake and the readings in the freestream, chosen by a parameter determined for a particular set of tests. Due to the reliance on the number of pitots which fall within each area, the drag estimates from this method can be inaccurate.

2.1.3 Geometrical definition

A nacelle is typically parameterised by a number of key variables as defined in the established ESDU approach for nacelle design (Figure 2.2).⁴ Key parameters in nacelle design include the highlight diameter (d_{hi}), which directly determines the highlight area and thus the MFCR for a given intake mass flow and freestream aerodynamic condition. The maximum diameter (d_{max}) determines the frontal facing area of the nacelle. l_{fore} is the forebody length and the combination of d_{max} and l_f are chosen to allow a favourable curvature distribution and thus C_p distribution over the forebody. This requires a certain amount of initial forebody curvature, $1/r_{if}$, to control the acceleration over the forebody as much as possible. Some older nacelles have a cylindrical centrebody of length l_c , which allows a separation distance between the forebody and afterbody geometries and reduces the interactions between the aerodynamics of these two sections. Exit diameter (d_{TE}) is dictated by the requirement of the engine design to achieve the cycle design point and optimised for efficiency and operability. The distance l_{aft} gives the distance from the maximum diameter to the trailing edge and must be of sufficient length to allow the gradient to transition smoothly from zero at the maximum radius to the required boattail angle (β). l_{nac} is then the overall length of the nacelle and, to reduce skin friction drag, should be minimised as far as possible.

The development of nacelle forebodies began with the requirement for an aerodynamic shape for use as a cowl which did not generate excessive drag. Initial wind tunnel experiments investigated the NACA A, B and C cowls which were designed to give a high critical Mach number.⁵³ The NACA 1-series ordinates were developed from these initial tests⁴⁰ to provide higher critical Mach numbers and to allow operation at higher freestream Mach numbers for a greater range of cowl dimensions. These ordinates can then be scaled by the required nacelle proportions, d_{max} , d_{hi} and l_{fore} (Figure 2.2) to provide suitable supercritical cowl designs. As the

required dimensions and operating conditions expanded for turbofan engines more complex designs have been developed.⁵⁴ Many subsequent designs were focused on improving the off design characteristics of nacelles by increasing the lip radius and thereby decreasing curvatures and accelerations over the forebody.⁴⁶ An alternative approach to nacelle design has been to maintain laminar flow over as much of the nacelle as possible, resulting in a lower skin friction drag. This approach has led to, for example, the European Laminar Flow Investigation II (ELFIN II) and Laminar flow Research Action (LARA) projects to develop laminar flow nacelles with transition positions at positions further downstream on the nacelle.⁵⁵ For example, an analysis of natural laminar flow nacelles, with transition at 40% chord, used to power a twin-engine aircraft with a 2000nm mission demonstrated potential fuel burn reductions of 1.7%.⁵⁶

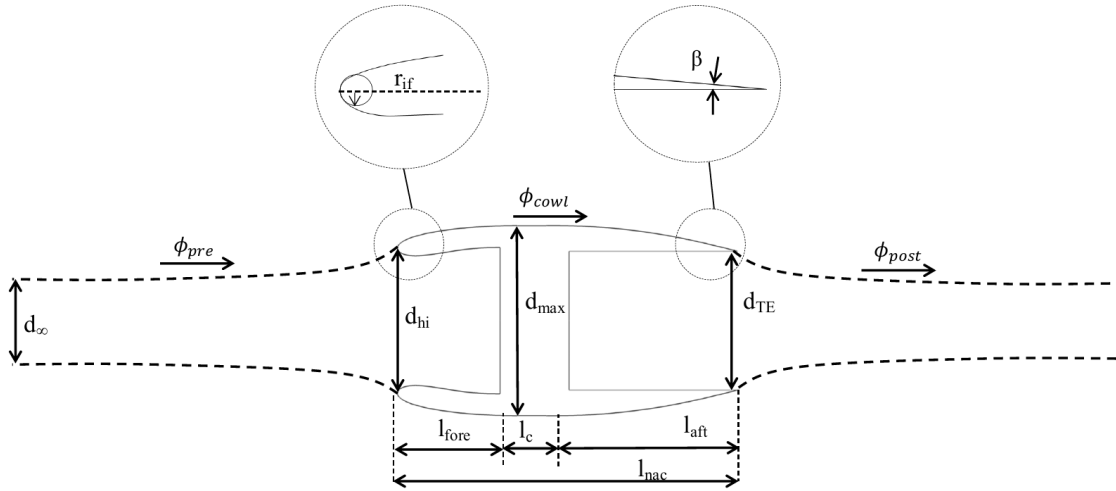


FIGURE 2.2: Nacelle geometry parameterisation based on ESDU⁴ and pre-entry and post exit streamtubes

2.2 Engine Integration

This section is intended to provide a background as to the methods and results of previous work carried out to assess the installation effects of engines. Historically, this effort has been focused on wind tunnel analysis and measurements of drag and lift at model scales using the principle of dynamic similarity to make judgements about full scale aircraft. However in recent years, as computational resources have become more powerful and more widely available, an increase in the amount of CFD analysis has also been used.

A notable contributor to the drag of a modern aero-engine comes from the nacelle.^{5,23} The purpose of the nacelle is to provide uniform undistorted flow to the fan via the intake with as little total pressure loss as possible and simultaneously to contribute as little drag as possible.⁵⁷ In a wind tunnel, an isolated engine nacelle is typically modelled using a sting mounted model.⁴⁶ There are two main approaches to modelling the presence of an engine for the purpose of installation. The first is to use a simple through flow nacelle (TFN) which consists of a model representation of an engine nacelle through which the air is allowed to freely flow. The TFN therefore does not model the influence or presence of a jet. An alternative to the TFN is the turbine powered simulator (TPS) which simulates the jet flow with a model fan. This models the engine more accurately than the TFN by adding the jet effect, but does not incorporate the high temperatures seen in an actual engine jet. It has been demonstrated that simulation with a TPS is generally a more accurate representation of an actual engine installation.⁵⁸ As jet engines have increased in size, the nacelle design has become more critical and therefore so has the accurate drag prediction of jet engine nacelles. A typical underwing podded nacelle installation of two engines can contribute around 35-40 drag counts to an aircraft²³, or around 14% of the drag, and an increase in diameter will cause greater interference with the wing (Figure 2.3).⁵

The understanding of aerodynamic interference has been studied extensively to allow for more optimised designs. This has led to the introduction of various fairings on modern aircraft to mitigate the interference effects. These can be seen at the interface between lifting surfaces and the fuselage as well as around the nacelle and pylon. A summary of the primary locations of interference drag caused by an underwing engine installation were provided by Haines²¹ and are summarised below:

- Changes in the spillage performance of the intake under the influence of upwash and sidewash of the wing.
- Impact on the pylon leading edge due to wing upwash and downwash.
- Change in flow over the cowl afterbody due to the impact of the wing stagnation field.
- Presence of the gully flow with overspeeds possibly resulting in shock waves and separation.

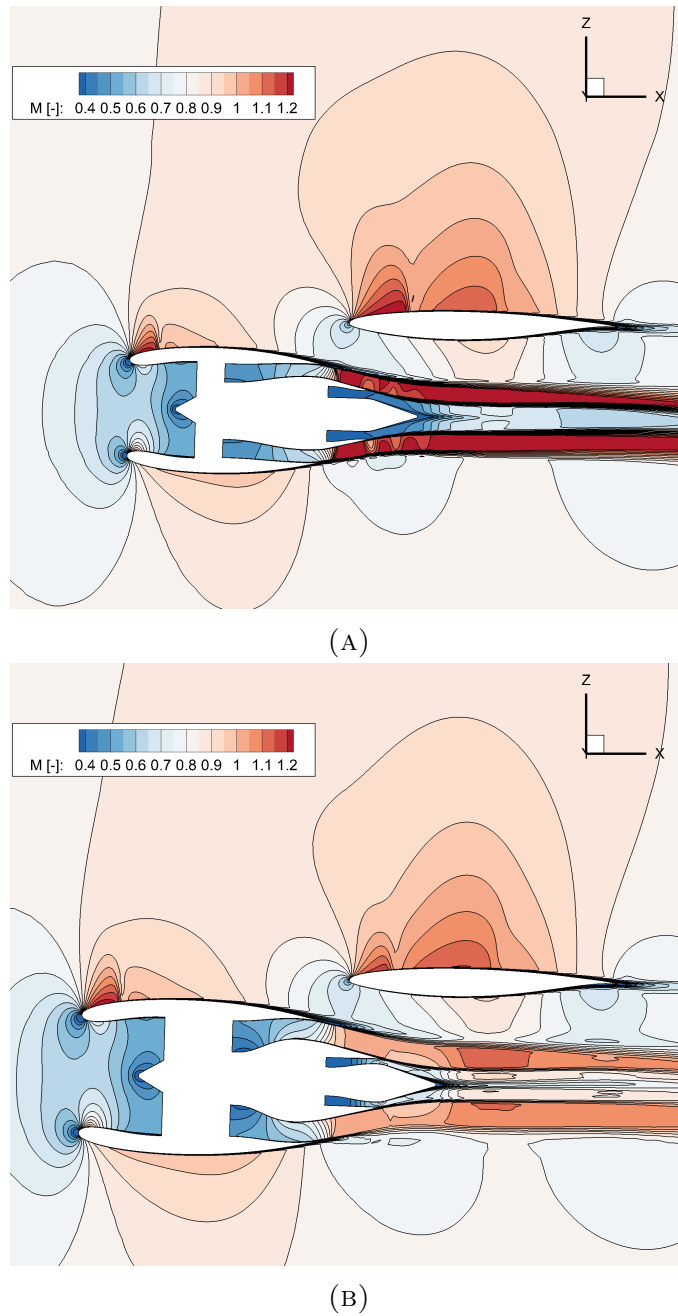


FIGURE 2.3: Common Research Model aircraft simulated at $\alpha_{a/c} = 2.5^\circ$, $M_\infty = 0.82$, $Re = 45 \times 10^6$ with (A) baseline engine, (B) 23% larger fan diameter engine⁵

- Non-planar lifting surfaces impacting the drag through nacelle lift, pylon side lift and wing lift.
- Changes in the supercritical flow over the wing surface due to the gross presence of the nacelle.

- Regions of overexpanded fan jet flow over the core afterbody with increased viscous drag and strong terminal shocks.

As engines increase in size, these influences on interference are expected to become more important and nacelle design will become of increasing importance. Significant work has been carried out to try to understand and mitigate the impact of large engines on the overall performance of aircraft. A study into the overall system performance of high BPR engine designs up to a BPR of 21.5 looked at improving three baseline engines and found that increases in fan diameter from 2.85m to 3.25m, with a BPR increase from 6.2 to 14.3, could provide improvements in overall fuel burn of 15-16% for an engine with a thrust class in the range of 350-390 kN.⁴⁷ However this study did not look at the detailed aerodynamic design of the nacelles or the interaction with the wing. Another assessment⁷ of 'advanced' engine and nacelle concepts looked to apply slimmer and shorter nacelles to engines in the thrust class range of 107-267kN. These engines incorporated novel nacelles for engines with geared cycles and demonstrated that with an optimal nacelle design, savings of 3% in fuel burn were possible from the advanced nacelle design for a large twin engine aircraft.⁷ Grieb and Eckardt⁵⁹ considered the requirement for reduced weight nacelles for a shrouded propfan relative to a turbofan engine. In this study they determined that for an increase in fan diameter from 1.6m to 2.5m the nacelle length had to be reduced from $l_{nac}/r_{fan} = 4.0$ to $l_{nac}/r_{fan} = 2.0$ to retain the same value of drag. There are however many degrees of freedom which can affect the actual drag and lift penalty and the local flow field. These include the engine position on the wing, spanwise, vertically and upstream or downstream, the engine size and the engine nacelle design.

Interference effects have been quantified for aircraft models mounted with a TFN⁶⁰ which showed that the impact of moving to larger bypass ratios (up to 18) and larger diameters may not cause excessive interference drag. This testing reported that by appropriate placement and design, the drag penalty from larger engines could be approximately the same as smaller engines. This implies that whilst the larger engines can have a large penalty in terms of drag, by suitable design and positioning the losses can be reduced to acceptable levels. However as has been stated, a more accurate representation of an engine in a wind tunnel analysis can be made using the TPS model. Testing has also been carried out to assess the interference effects of a TPS on an aircraft model. There are some examples of this in the open literature^{61 62}. The intention of one wind tunnel analysis⁶ was

to understand the magnitude and causes of interference effects for closely coupled underwing engines. The testing comprised a half aircraft model fitted with a TPS. This gave it a d_{fan}/c_{ref} value of 0.328, which is representative of a significantly older engine than those considered in this project for aircraft installation. The other variables for the test schedule were axial and vertical position of the TPS, Mach number, angle of attack and a short and long fan cowl. The methodology used for the assessment of interference drag is summarised in figure (Figure 2.4). The method involves establishing thrust and mass flow from testing in a Mach simulation tank and then taking differences between the installed and isolated tests to get the appropriate interference terms. The main findings of this report were that there was an installation penalty in terms of drag for the installation of an engine which varied as a function of the lift coefficient compared to the clean wing configuration. The interference drag was calculated as seen in Figure 2.5. Here Build 1 refers to a long cowl with mixed jets, build 2 refers to a short cowl with separate jets with the fan cowl trailing edge at the same axial position as build 1, and build 3 refers to the short cowl moved axially forwards.

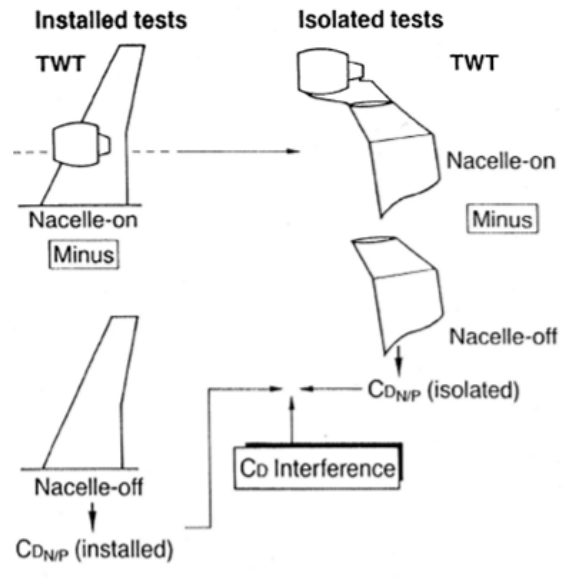


FIGURE 2.4: Methodology for the extraction of interference drag from the measured data⁶

The effect of installation can be divided into axial effect and vertical effect. In general there was no more than 2-3 drag counts difference between the installation drag at different vertical positions. The axial position had more influence with the installation with the least impact on drag being the furthest forward. The reduction in drag between axial positions was mostly at the low C_L values. The

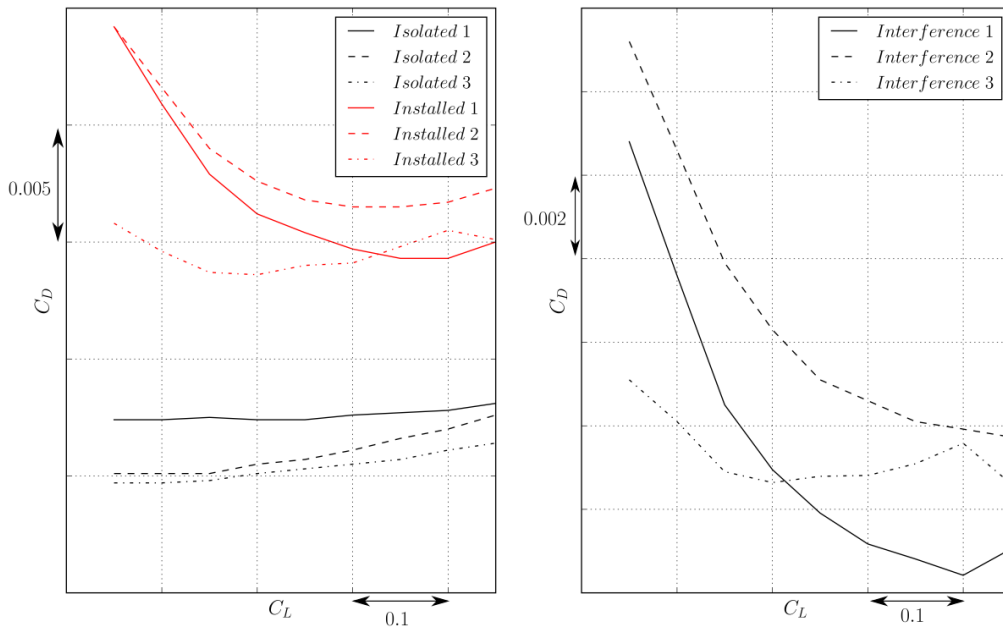


FIGURE 2.5: Example of measured drag for cowls at lowest vertical height, showing the installation drag, isolated nacelle drag and interference drag⁶

work carried out by Sibilli in UHBPR engine integration also confirmed this trend of axial position having the greatest influence⁶³.

In order to simulate the installation effects and quantify the interference terms of a UHBPR installation an appropriate aircraft model must be chosen. Appropriate and realistic aircraft geometries are typically unavailable in the public literature due to the cost which is involved in generating them. The Common Research Model (CRM) is an aircraft configuration comprising wing/body/nacelle/pylon/horizontal-tail⁶⁴. It was developed as a model for wind tunnel and CFD analysis to be used to predict drag characteristics of modern wide body civil aircraft⁶⁴. The CRM model is intended to represent a twin engine modern passenger jet. Principal experimental data for the CRM comes from the NASA Ames 11-ft wind tunnel and the NASA Langley National Transonic Facility⁶⁵. This analysis was carried out at a chord Reynolds number of 5 million for a variety of configurations including wing/body and wing/body/nacelle/pylon. The impact of the through-flow nacelle and pylon were seen to cause an increase in drag of around 20 drag counts at the cruise Mach number of 0.85 and lift coefficient of 0.5. Work has also been carried out with the CRM using CFD. Good agreement has been seen between the experimental results and CFD analyses^{11,66} carried out with the drag prediction workshop (DPW) gridding guidelines.⁶⁷

Stańkowski¹¹ performed a study which used the CRM aircraft to establish the installation effects for large civil underwing engines. Unlike in the method above (Figure 2.4) which considered the interference and installation effects at a constant incidence, Stańkowski used a retrimmed aircraft to ensure constant lift between the different configurations. In this study, three engine fan diameters were considered with a constant nacelle $l_{nac}/r_{hi} = 4.3$. This study demonstrated that a 23% larger engine with a geometrically scaled nacelle would have a much greater influence on the wing aerodynamics than the smaller engine (Figure 2.3).¹¹ The study demonstrated that whilst the smaller engine resulted in an installed drag increase relative to a clean wing of 27 drag counts, the larger engine had a penalty of 36 drag counts. This study was considered for a relatively low freestream Mach number of $M_\infty = 0.82$ and it is expected that this drag penalty would worsen at higher Mach numbers as wave drag increased.

Whilst some of these studies have demonstrated installation penalties associated with an UHBPR engine size, none have attempted to determine both design point and off design performance of the nacelle with a focus on the aerodynamics. Consequently, there is a need to establish the required experimental and computational tools to enable the reliable evaluation of the nacelle performance. This includes aspects such as drag, critical drag-rise Mach number as well as spillage drag.

2.3 Previous nacelle design space exploration

Due to the growing importance of nacelle design with increasing engine sizes, novel design concepts are being considered. The reduction of size of the nacelle in proportion to the engine will require some consideration of changes in the mounting of accessories or the removal of the thrust reversal unit⁶⁸. A novel configuration being planned for UHBPR engines is that of a slimline nacelle, with a lower nacelle thickness relative to fan diameter. Future turbofans may provide weight and diameter savings through removal of the thrust reverser unit which is enabled by the use of a variable pitch fan.¹⁵ This will allow for a much thinner nacelle in which to house the engine. A slimmer design is also of general interest and the extent to which weight and drag can be saved of importance. There is no formal definition of the term slimline nacelle, however nacelles described as slimline have certain qualities. The trend towards slimmer nacelles is characterised by a decrease in maximum nacelle radius (r_{max}). More aggressive nacelle designs are also expected to incorporate a

shorter overall nacelle length (l_{nac}) and also a shorter intake length (l_{int}). These features help to reduce the overall weight of the nacelle as well as the wetted area. To better predict the future trends and realistic design space for nacelles, current aero-engine nacelles have been examined in terms of their length (l_{nac}), highlight radius (r_{hi}) and trailing edge radius (r_{TE}) (Figure 2.6).⁶⁹⁻⁷¹

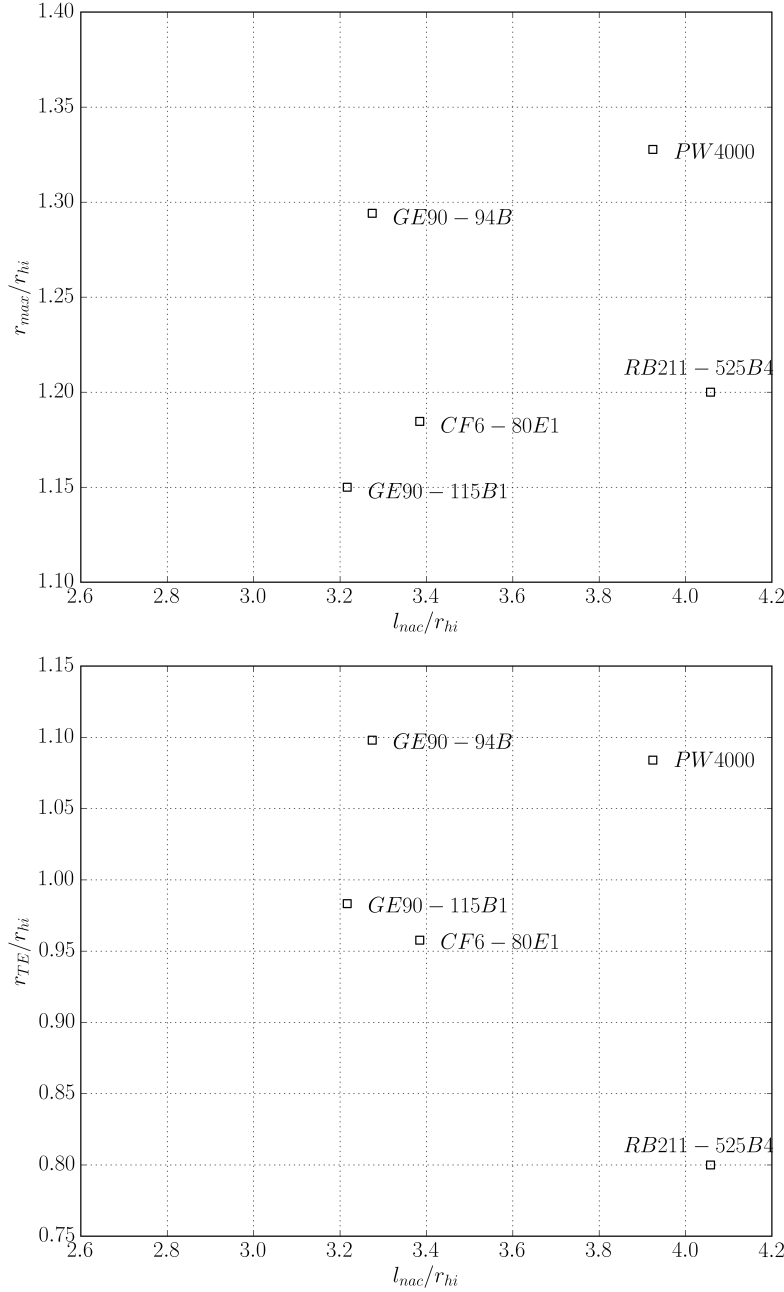


FIGURE 2.6: Design space of current nacelle designs

Howe and Wynosky⁷ undertook a wide ranging study to establish an advanced nacelle design for a high BPR turbofan engine. This included a consideration of

the intake, fan cowl and exhaust aerodynamics. As part of this study the authors produced a fan cowl design which reduced the intake length and the bypass duct length by 50% each relative to a conventional technology, which resulted in a shorter nacelle (Figure 2.7). As part of this study, the influence of nacelle length and thickness on wave drag was determined (Figure 2.8) at $M_\infty = 0.8$. This carpet plot demonstrates a key challenge in the design of short nacelles. For the longer nacelles wave drag occurs at the high values of d_{hi}/d_{max} (or low r_{max}/r_{hi}) which is as a result of higher curvature required at the lip of the nacelle to achieve the lower r_{max}/r_{hi} . At a particular value of r_{max}/r_{hi} however it is possible to achieve a design with no wave drag. For the shorter nacelles this is not the case and wave drag is encountered for most values of r_{max}/r_{hi} . The highest values of wave drag occur at the lowest length and lowest d_{hi}/d_{max} . This demonstrates the possibility of reducing wave drag by lowering d_{hi}/d_{max} which was available for longer nacelles is no longer possible for short nacelles as the wave drag incurred increases dramatically.

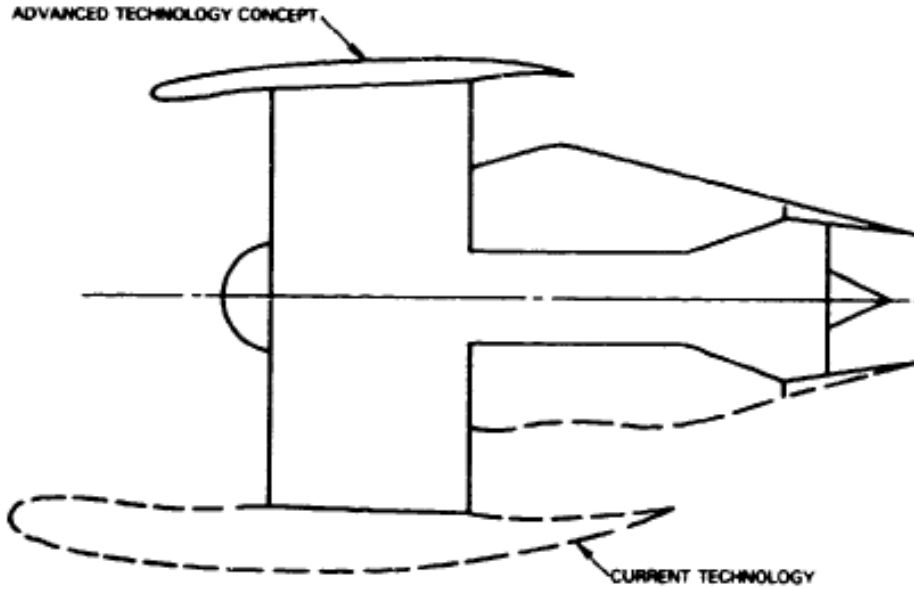


FIGURE 2.7: Nacelle design in Howe and Wynosky⁷

Previous work has been performed to give information about the performance of nacelles within a parametric design space.⁸ This study performed 13440 separate CFD computations for a Latin hypercube sampling of 60 different nacelle designs with a simplified cylindrical nozzle design. These designs covered a range of nacelle designs with four design variables: maximum radius (r_{max}), nacelle length (l_{nac}), forebody length (l_{fb}), and initial forebody radius of curvature (r_{if}). In terms of

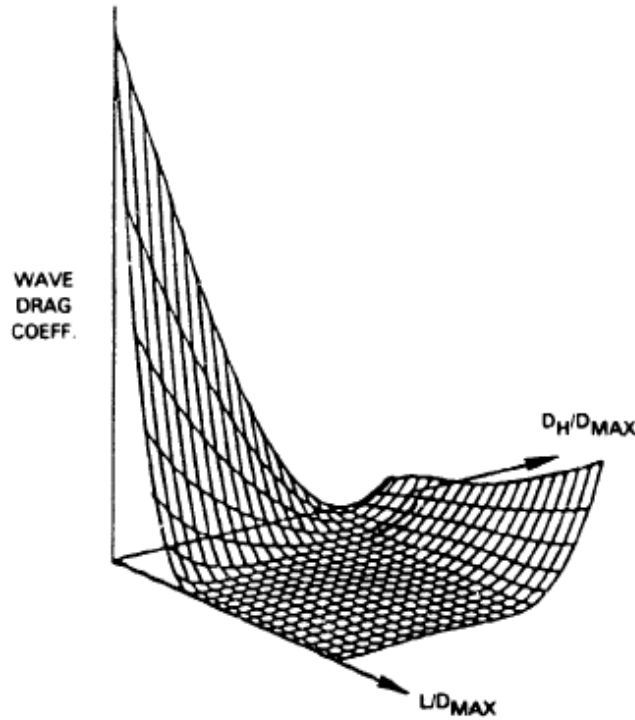


FIGURE 2.8: Influence of nacelle length and thickness on wave drag⁷

nacelle profile drag at $M=0.85$ and $MFCR=0.7$ (c_{Dref}) and drag rise Mach number (M_{DR}), these data can be viewed in the design space shown (Figures 2.9, 2.10). This work demonstrated that reduced nacelle length and thickness could reduce the drag incurred, but a balance must be found for the off design conditions. Amongst other trends in the specific designs it was demonstrated that the reduction in the nacelle length did not necessarily reduce the baseline drag of the nacelle (Figure 2.9). In addition, these plots demonstrate that a problem area may exist where a long nacelle is designed with a high maximum radius as this can produce both poor baseline drag and poor drag rise performance (Figure 2.10).

Slim nacelles have been tested in the past to ascertain whether they can provide an improvement in interference drag. One study demonstrated that minimisation of the maximum nacelle diameter could provide a fuel consumption reduction of 2.5%⁶⁸. Wind tunnel experiments have also shown that uninstalled slim nacelle designs ($r_{max}/r_{hi} = 1.09$) can have a drag rise around 0.8, however when placed in a forward installation position this was increased to beyond 0.85.⁴⁸ This change was seen as a beneficial wing interference effect which caused a movement and weakening of the nacelle shock. No tests have been found for a slim line nacelle in further off-design conditions and given the need for a thicker inlet lip at high angles of attack

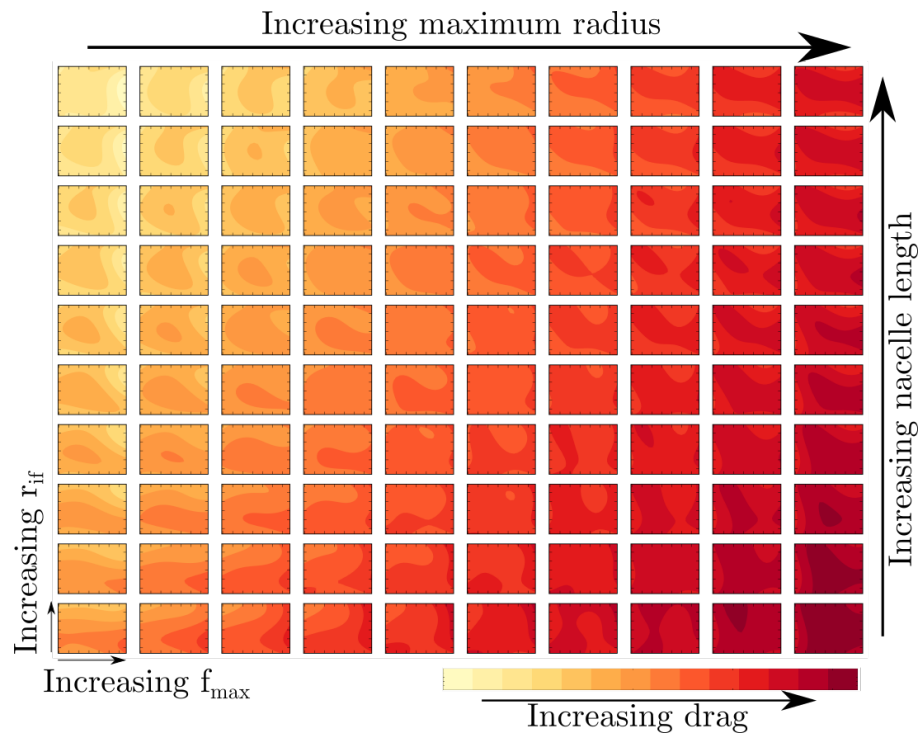


FIGURE 2.9: Parametric design space with Dassonville data⁸ in terms of nacelle drag

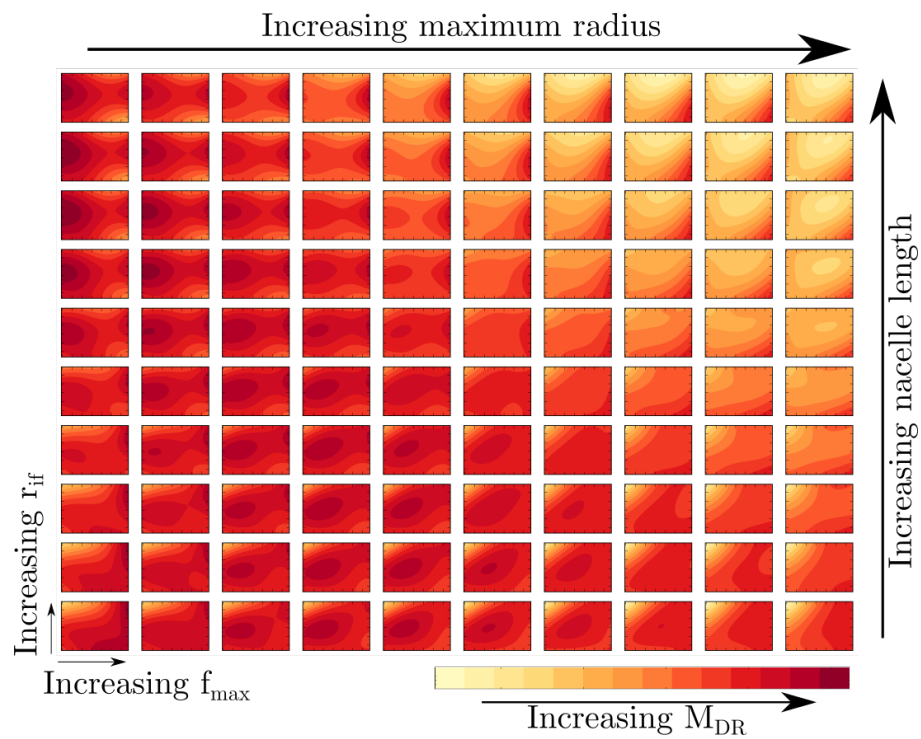


FIGURE 2.10: Parametric design space with Dassonville data⁸ in terms of drag rise Mach number

to mitigate flow separation; it is possible that this could be the critical part of the design for the external lines of slim line nacelles.

Motycka et al⁷² combined a TPS model with 16 separate nacelle designs which covered a matrix of nacelle forebody lengths (l_{int}), maximum radius to highlight radius ratios (r_{max}/r_{hi}), and nacelle and core cowl boat-tail angles. The study covered a range of forebody lengths from $l_{int}/r_{hi} = 0.89$ to $l_{int}/r_{hi} = 2.05$ and a range of radius ratios from $r_{max}/r_{hi} = 1.11$ to $r_{max}/r_{hi} = 1.28$ (Figure 2.11). The fan cowl boat-tail angles were relatively low at $\beta = 3^\circ$ and $\beta = 6^\circ$. Three freestream Mach numbers, $M_\infty = 0.7$, $M_\infty = 0.85$ and $M_\infty = 0.9$ as well as a range of MFCRs between $MFCR = 0.4$ and $MFCR = 0.7$ were tested. At $MFCR=0.525$ and at all three M_∞ there was an optimum value of r_{max}/r_{hi} for minimal drag. This optimal r_{max}/r_{hi} was at $r_{max}/r_{hi} = 1.22$ for the three values of M_∞ . The highest drags were observed at the lowest values of r_{max}/r_{hi} . Another key conclusion from this study was that the shortest nacelles tested had lower profile drag than the long nacelles up to $M_\infty = 0.85$. Above this freestream Mach number however the longer nacelles had lower drag than the short nacelles. The report indicates that a low pressure region is favourable on the forebody as it produces a forward force, until a shock appears due to this acceleration and then this begins to increase the drag. Whilst this study gave an indication of the trade-offs which exist between these nacelle dimensions, it was based around geometries which are unlikely to be representative of the next generation UHBPR engine nacelles. Specifically, the forebody geometries were all based on NACA 1-series and the afterbodies were separated by a cylindrical section from the forebody. Neither of these are expected in future designs which will need to be more compact and continuously curved.

Zimbrick and Colehour⁹ conducted a study of the effect of BPR and nacelle length on the overall engine performance. As part of this study, several short nacelles were designed (Figure 2.12). These designs were much shorter than any of the current designs (Figure 2.6) with length ratios in the range of $l_{nac}/r_{hi} = 1.2$ to $l_{nac}/r_{hi} = 2.4$. This study however was based around an operating Mach number of $M_\infty = 0.8$ and the aerodynamics of the nacelles were not published. However, the study determined that an increase in the BPR from $BPR=10.6$ to $BPR=17.5$ could reduce the thrust specific fuel consumption (TSFC) by around 5%. The penalty from the longer nacelle increased from 5% TSFC to 6% TSFC as a result of the increase in BPR which indicates it is not overly penalising (Figure 2.13). In addition, the short nacelle contributed even less penalty to the TSFC even when designed to meet the same intake separation criteria (Figure 2.13). Another useful aspect of nacelle

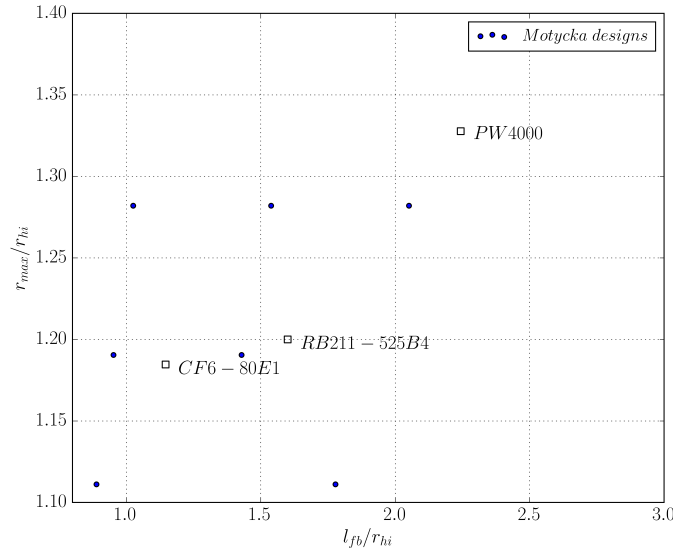
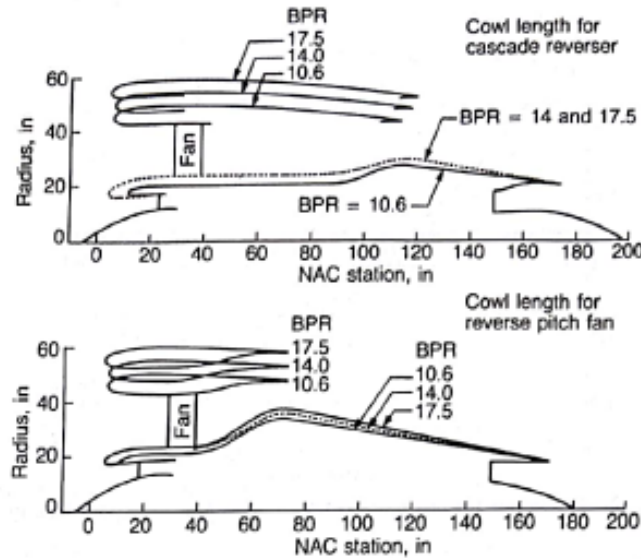


FIGURE 2.11: Nacelle designs with some existing designs

design which is detailed in this study is the requirement to increase the r_{TE}/r_{hi} as the nacelle length is reduced to limit the wave drag (Figure 2.14). The study showed that a reduction from $r_{TE}/r_{hi} = 1.0$ to $r_{TE}/r_{hi} = 0.98$ caused wave drag of the order of 50% of the drag at $r_{TE}/r_{hi} = 1.0$. Whilst this is for a very short nacelle of around $l_{nac}/r_{hi} = 1.2$, it is likely a similar effect will be present for longer nacelles at higher Mach numbers.

FIGURE 2.12: Nacelle designs for Zimbrick study⁹

Petrusson⁷³ further demonstrates some of the problems which are expected in the design of short nacelles for ultra-high bypass ratio engines. This study demonstrated

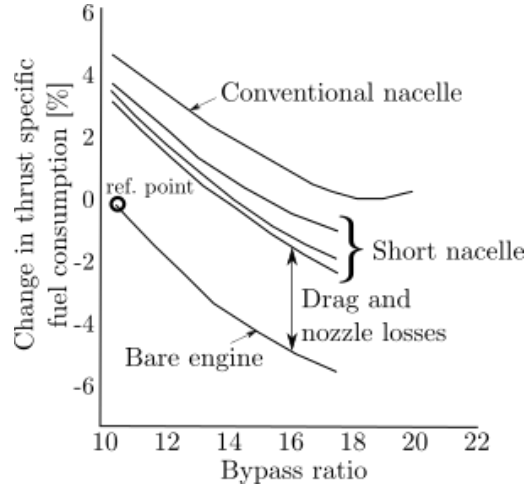


FIGURE 2.13: Influence of BPR and nacelle design in Zimbrick study, based on Zimbrick⁹

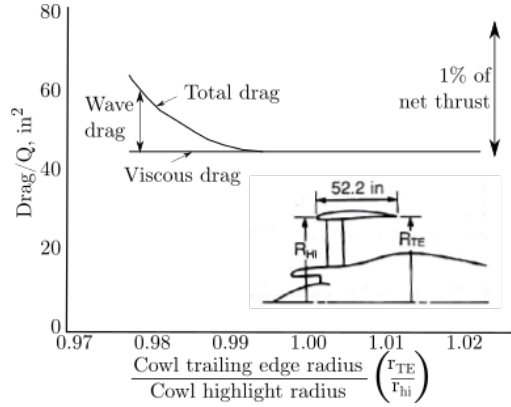


FIGURE 2.14: Influence of r_{TE}/r_{hi} on nacelle drag in Zimbrick study, based on Zimbrick⁹

that shortened nacelles may exhibit non-monotonic velocity profiles and high peak Mach numbers in cruise flight. Furthermore, the same study demonstrated that a reduction in length from $l_{nac}/d_{fan} = 2.5$ to $l_{nac}/d_{fan} = 0.6$ resulted in a threefold increase in the drag at freestream Mach $M_\infty = 0.8$ and at 10668m altitude.

2.4 Optimisation

Optimisation algorithms have previously been applied to a number of aerodynamic problems^{74,75} and have shown a capability to simultaneously optimise at several aerodynamic points. Previous studies have also been carried out specifically on the problem of nacelle external design through optimisation.^{76–79} Two of these studies^{77,78} focussed on the use of a surrogate model to provide a quicker optimisation

approach. Both of these studies used a CFD method to obtain high fidelity simulations of the nacelles which were used to improve a surrogate model. This resulted in improvements from a baseline design but since they did not simulate multiple points in the CFD model they could not directly assess the suitability of the designs in off design. Since this is expected to be a major challenge for future high BPR engines, this is a gap which the present study aims to fill. The studies also did not focus on high transonic Mach number operations and as such the expected problems of off design would have been lessened. As CFD approaches become more computationally efficient and resource more readily available a direct CFD optimisation approach becomes possible. One study which used direct CFD simulation⁷⁶ demonstrated the applicability of multi-objective optimisation to isolated nacelles. In this study the focus was on cruise and cross-wind performance and produced a Pareto front of pressure drag and intake distortion. The authors concluded that whilst the Pareto front was established further constraints were required to avoid unrealistic shapes. For example, the optimal fan cowl produced for the two objectives demonstrated two peaks in negative pressure coefficient which were more severe than the baseline. This would result in increased spillage as it strengthens the shock wave at lower MFCRs. Another multi-objective optimisation study of nacelles⁷⁹ minimised fan cowl peak Mach number during cruise and intake peak Mach number in a take-off condition and achieved a 14% reduction in the external peak Mach number relative to the baseline. This study did not directly consider the drag performance in cruise however and also did not look at spillage or drag rise. These previous studies which used optimisation methods on nacelle aerodynamics indicated that this could be a powerful tool to quickly design a nacelle which performs well.

Multi-objective optimisation (MOO) deals with a certain class of problems in which an optimal solution is sought with greater than one objective functions to minimise. In contrast to a single objective optimisation, MOO can be used to assess the trade-offs between objectives through the generation of non-dominated sets, or Pareto fronts, of the possible solutions to the problem. Members of a given Pareto front do not dominate any other member of the same front. A solution, $x^{(1)}$, is said to dominate another solution, $x^{(2)}$, if equation 2.5 is satisfied for all objective functions and the strict inequality is true for at least one objective function⁸⁰.

$$x^{(1)} \leq x^{(2)} \quad (2.5)$$

One group of optimisation algorithms employ evolutionary mechanisms to rank solutions by fitness and preserve beneficial traits over a series of generations. NSGA-II was proposed by Deb as an appropriate evolutionary algorithm for a wide variety of multi-objective problems.⁸¹ The algorithm is a multi-step process (Figure 2.15) which begins with a parent population P_t and a child population Q_t created from crossover and mutation. These are then sorted into non-dominated sets and given a crowd distance value based on their proximity to other solutions in the combined population. Tournaments are then carried out between individuals to determine the subsequent mating pool from which later generations are produced. This algorithm has been developed to preserve diversity, whilst quickly converging on a Pareto optimal front and has been demonstrated as appropriate for several non-convex problems.⁸²

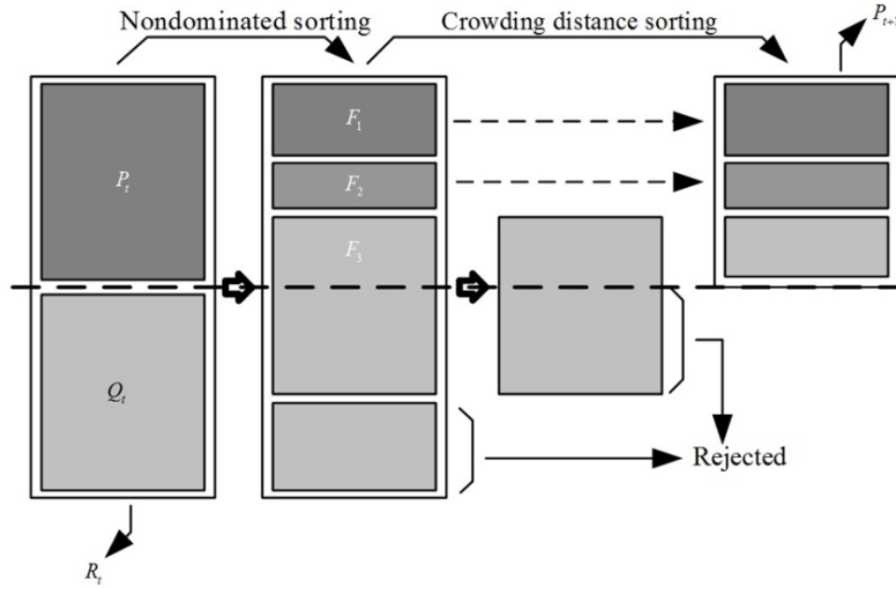


FIGURE 2.15: NSGA-II Evolutionary Ranking Algorithm¹⁰

Genetic algorithms include the use of mutation and crossover operators and the selection of these can have a direct impact on how well the optimisation is performed. Mutation is typically applied to a subset of the mating pool at each generation and a commonly used number of mutations is 10%.⁸¹ Three common operators for this are a random mutation (Figure 2.16a), non-uniform mutation (Figure 2.16b) and Gaussian mutation (Figure 2.16c). The first of these produces a mutated individual (x_{i2}) by applying a random amount of mutation to a member of the mating pool (x_i) (Equation 2.6).¹⁰ A mutation size is chosen by the user, Δ_i and randomly a value is chosen from within this bound by the randomly determined factor r_i . A benefit of this method is the relative simplicity with which it can be implemented however it

has no weighting towards the original parameter x_i^t and as such too high a value of Δ_i will result in the optimisation potentially moving away from the optimal. The non-uniform mutation operator⁸³ produces the mutated individual (x_{i2}) from the non-mutated individual (x_i) by the application of a boolean parameter (τ) which is either -1 or 1 and determines which direction in the bounds of the parameter between $x_i^{(L)}$ and $x_i^{(U)}$ the mutation will be applied (Equation 2.7). The amount of mutation applied is determined by a random value (r_i), a user-specified parameter (b), and the ratio of the current generation (t) and the total generations to be performed (t_{max}). This ensures that the mutated individual is statistically more likely to be closer to the original individual than far away and that as the generations progress the likelihood increases of a similar individual. A potential downside of this method is that it requires the tuning of the value b and the total number of generations to ensure that sufficient mutation is performed early on in the process. Gaussian mutation¹⁰ employs a similar method to ensure a mutated individual close to the non-mutated individual but is based around a Gaussian distribution to determine the mutation size (Equation 2.8). This requires a user input of simply a standard deviation (σ_i) which is used to randomly produce a value within a certain probability distribution and is then applied to the original individual.

$$x_{i2} = x_i + (r_i - 0.5) \Delta_i \quad (2.6)$$

$$x_{i2} = x_i + \tau \left(x_i^{(U)} - x_i^{(L)} \right) \left(1 - r_i^{\left(1 - \frac{t}{t_{max}} \right)^b} \right) \quad (2.7)$$

$$x_{i2} = x_i + N(0, \sigma_i) \quad (2.8)$$

Several crossover methods also exist which can alter the efficiency of the optimisation routine. These crossover routines begin from two parent individuals ($X^{1,t}$, $X^{2,t}$) and create a pair of offspring individuals ($X^{1,t+1}$, $X^{2,t+1}$). Single point crossover (Figure 2.17a) splits the input variables of the two individuals at the same position, k . The two offspring individuals are then produced from an alternate combination of these two series of variables. Two point crossover is similar except that two random splits in the variables are made in each parent individual (Figure 2.17b). These crossover methods have the downside that they do not vary the specific parameters and thus any variation must come from the mutation operator. Alternatives to these standard

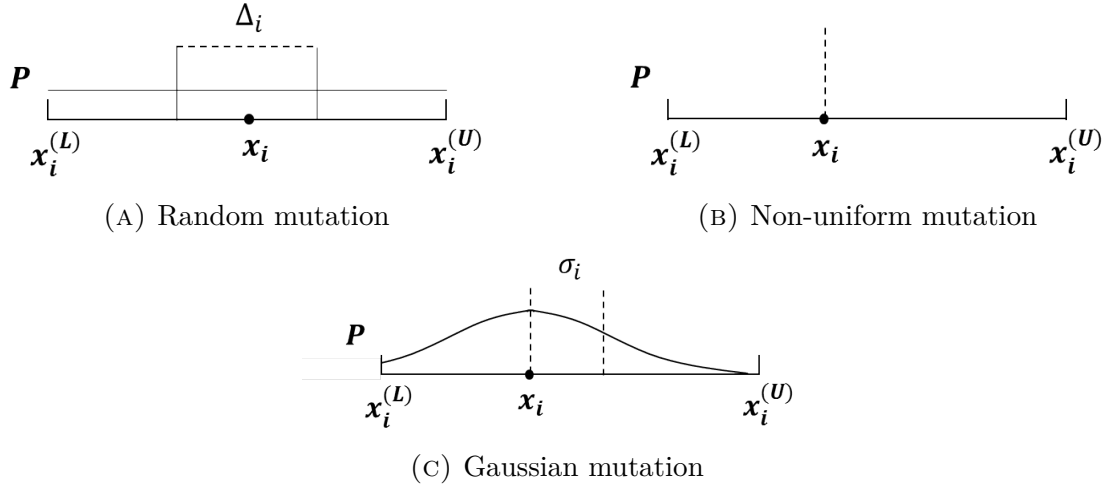


FIGURE 2.16: Different mutation methods

crossover methods include a flat crossover (Figure 2.17c) and a $BLX - \alpha$ crossover (Figure 2.17d).¹⁰ The former of these methods produces offspring individuals whose variables are randomly placed between the variables of the two parent individuals. $BLX - \alpha$ crossover expands this concept by adding a region either side of the two parent individual parameters which the offspring can be placed (Figure 2.17d). The size of this added region is determined by $\alpha (X^{2,t} - X^{1,t})$ where α is a user defined parameter.

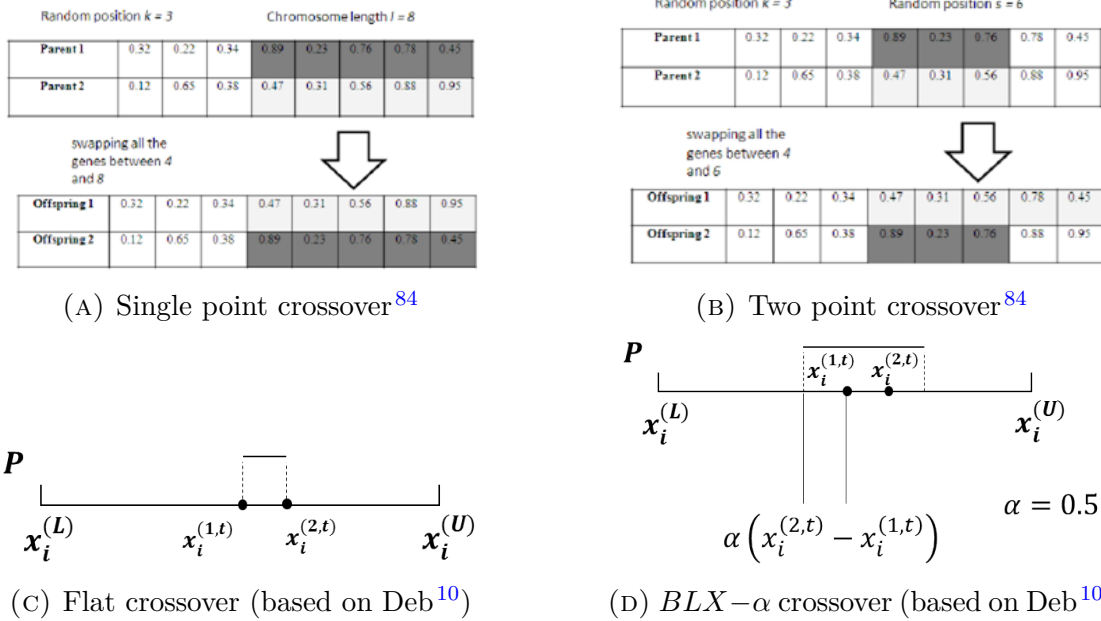


FIGURE 2.17: Different crossover methods

2.5 Gap in Knowledge

After the literature review a gap in the knowledge has been determined which this thesis addresses. Specifically it was decided to use the multi-objective capability of genetic algorithms to design novel nacelles for cruise and off design requirements. In the loop CFD will be combined with this method as this has not been fully realised for a MOO problem, but could allow more direct optimisation of the aerodynamics. This will enable representative UHBPR engine nacelles for studies on installation and windmill conditions which are not present in the current literature.

CHAPTER 3

Methodology

This chapter details the methods used within the project to assess the aerodynamic performance of aero-engines and aircraft. In addition, the methods used to optimise nacelle designs are explained.

3.1 Computational Fluid Dynamics Methods

Analysis of the aerodynamic performance of engine integration was carried out with computational fluid dynamics (CFD). Specific tools used were ANSYS ICEM v.15.0 as a meshing tool, Fluent v.15.0 as a pre-processor⁸⁵, solver and post-processor. Most of the meshing and CFD solutions were carried out in the batch modes of the respective software to allow for multi-processing. As such, ICEM was run non-interactively with .rpl macro files whilst Fluent was run non-interactively with .jou journal files. This also ensured consistency between meshes and CFD solutions in this project.

The motion of fluids is governed by the Navier-Stokes equations and can be derived from the application of Newton's second law to an infinitesimally small fluid element.⁸⁶ These equations provide instantaneous information about a fluid flow. An alternative formulation of these equations, which removes the time dependant terms, is called the Reynolds-averaged Navier-Stokes equations (Equations 3.1, 3.2). The averaging of the time dependant terms introduces additional terms to the equation, $-\rho \overline{u'_i u'_j}$, which represent the effect of turbulence on the mean flow and are called the Reynolds stresses. Since these new terms introduce more unknowns to the equations they must be modelled or substituted to close the equations. One approach to this

is to directly compute the Reynolds stress tensor components through transport equations for each term.⁸⁷ Since this is a costly procedure however, an alternative is to compute an eddy-viscosity which is related to the Reynolds stresses under the Boussinesq hypothesis (Equation 3.3).⁸⁸ This approximation removes the need to model the Reynolds stresses directly and allows a simpler model of the eddy viscosity (μ_t) and turbulent kinetic energy (k). There are a number of eddy viscosity models which provide solutions for these which include the one equation Spalart-Allmaras model⁸⁹, the two equation $k - \epsilon$ model⁹⁰ and the two equation $k - \omega$ model.⁹¹

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{\rho} \tilde{u}_i) = 0 \quad (3.1)$$

$$\frac{\partial}{\partial t} (\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_i \tilde{u}_j) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \tilde{u}_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\bar{\rho} u'_i u'_j) \quad (3.2)$$

$$-\bar{\rho} u'_i u'_j = \mu_t \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial \tilde{u}_k}{\partial x_k} \right) \delta_{ij} \quad (3.3)$$

The CFD process involved the solution of the steady-state Reynolds-Averaged Navier Stokes (RANS) equations (Equations 3.1, 3.2). In order to model the turbulence parameters a closure model is required which can provide the eddy viscosity. The closure model used for the majority of this project was the $k - \omega SST$ model. This model utilises the $k - \omega$ formulation of Menter^{92,93} which switches from $k - \omega$ in the near wall region to $k - \epsilon$ in the free stream to fully realise the benefits of both models.⁹² This model also gives improved prediction of separated flows due to adverse pressure gradients in comparison to other two-equation and one-equation models.⁹⁴ Some studies were carried out to determine the effect of turbulence model on the result obtained (Chapter 4). The solution was obtained through a density-based, implicit solver. A Roe-FDS scheme was used with Green-Gauss node-based discretisation. Spatial discretisation was converged to a second-order level. Air was assumed to be an ideal gas and viscosity was modelled using Sutherland's Law whilst the specific heat C_p was varied with a piecewise polynomial.

3.1.1 Computational domain

Over the course of the project a number of computational meshes have been created and computed which represented a variety of configurations and models (Figures 3.1, 3.2). For any 2D axisymmetric simulations the domain of the mesh was a semi-circle and except where described the domain for the 3D work was a sphere. These domains represented the free stream and a farfield boundary condition was therefore used. At this boundary a free stream Mach number, static pressure and static temperature were imposed to represent flight conditions. At the entry to the fan, i.e. exit of the intake, a pressure outlet boundary condition is imposed. At this boundary, a mass flow is set which within Fluent iterates on a static pressure to achieve the mass required. The entry planes to the nozzle ducts were set to pressure inlets where a total pressure and total temperature were imposed. For the values imposed at all of these boundaries, a model of an engine was required to give representative values, which is the subject of section 3.2. The meshes, except where described as intentionally low-fidelity, were created in adherence to the drag prediction workshop gridding guidelines.⁶⁷

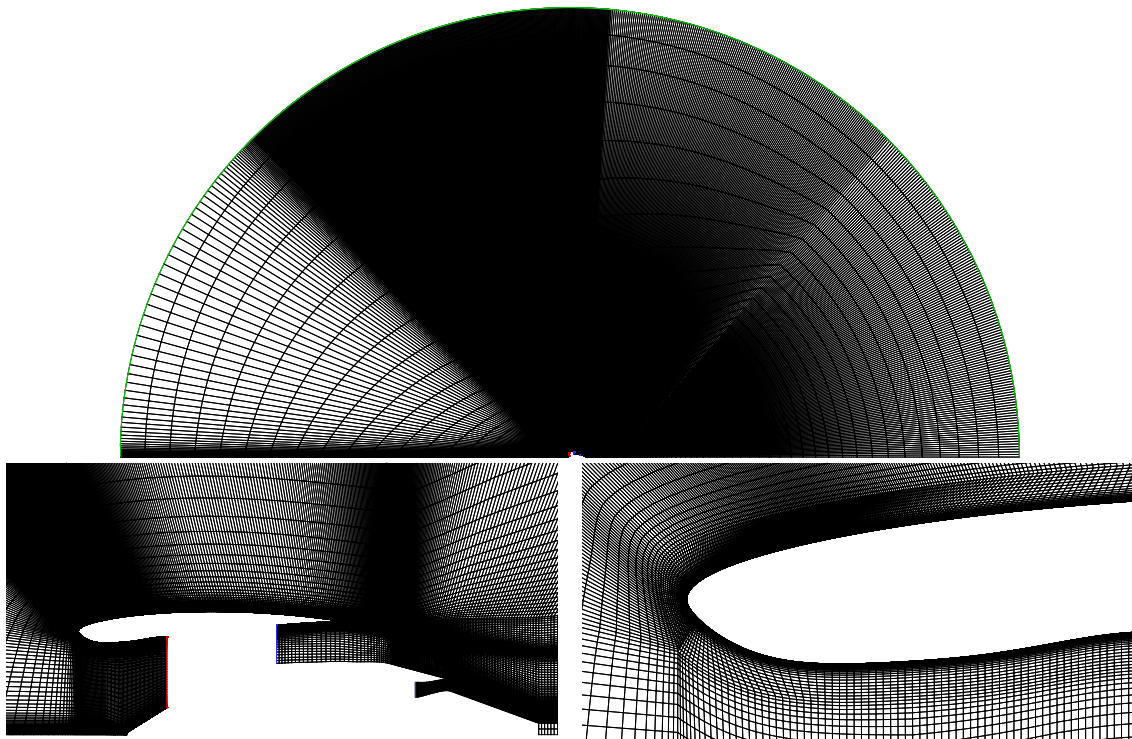


FIGURE 3.1: Example of isolated engine mesh

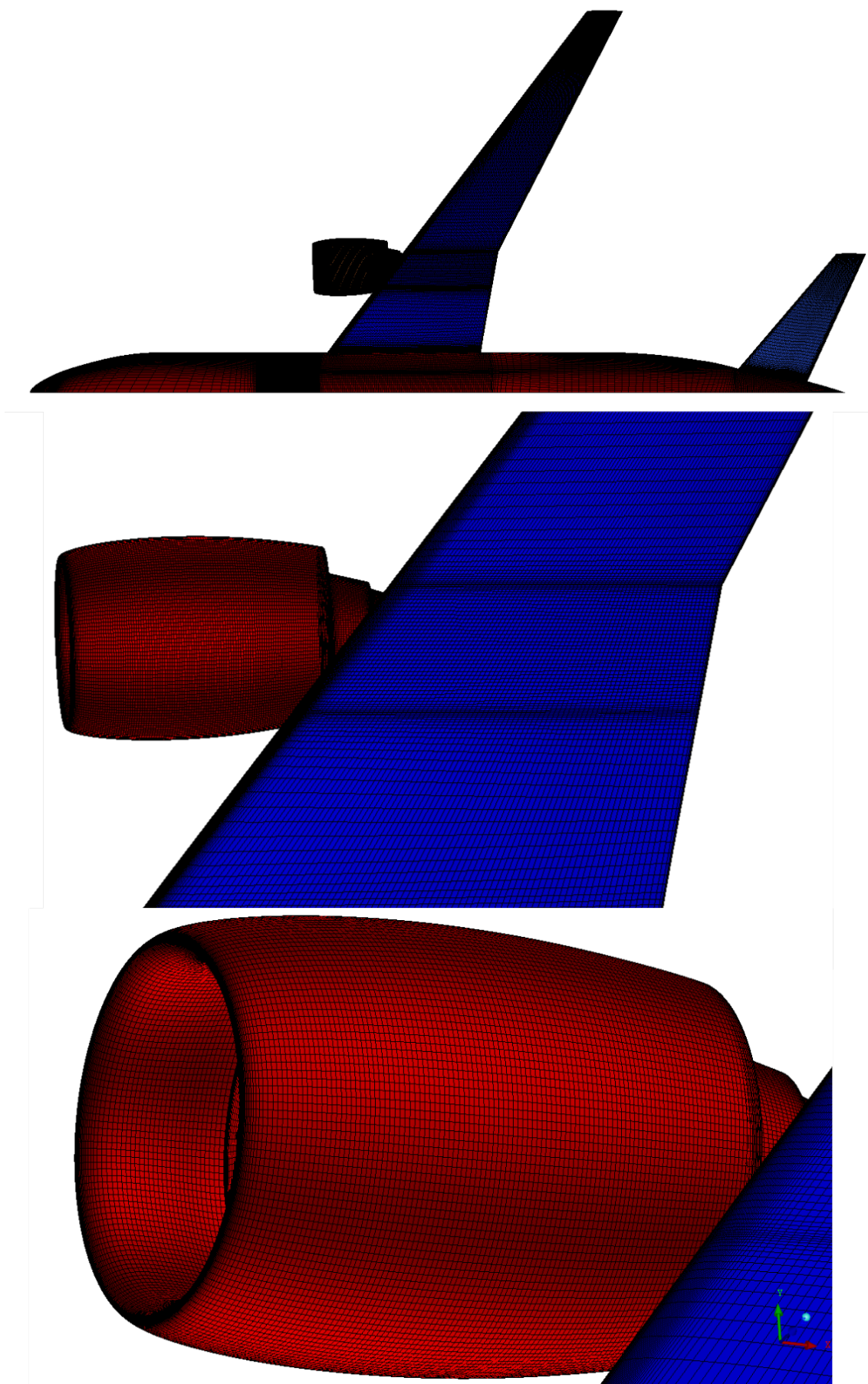


FIGURE 3.2: Example of installed engine mesh

3.1.2 Independence studies

Iterative convergence was monitored for all cases in terms of normalized residuals, as well as integral and peak values within the flow solution. Typically a converged solution had normalized residuals which had reduced by five orders of magnitude from the farfield initialisation. In addition it was ensured that the integral and peak values monitored in the solution had ceased to vary or were oscillating within 0.1% of the predicted value.

Multiple mesh and domain independence studies were carried out (Appendix A). Based on the studies detailed in the appendix, meshes and domain sizes were chosen for the 2D axisymmetric engine and 3D isolated engine domains. The installed domain and mesh size was set in accordance with previous work.¹¹ It was established that the solution was insensitive to the domain size for the 2D work when set to be $r_{Domain} = 80r_{nac}$. For the 3D isolated case, this was found to be $r_{Domain} = 100r_{nac}$.

3.2 Engine Models and Geometry Definition

3.2.1 Cycle Modelling

Before an assessment of nacelle aerodynamics could be made, boundary conditions for the CFD simulations were required which represented the influx and efflux of the engine. To acquire typical values for such ultra-high bypass ratio engines two zero-dimensional cycles were generated. To generate these cycles, a modular existent tool was used, called Turbomatch.⁹⁵ Turbomatch is a scheme used within Cranfield University for engine cycle modelling. It comprises a series of blocks which represent individual gas turbine components, which can be added or removed to match different engine designs. Turbomatch uses mass and energy balance through an iterative method based on scaling experimental component maps to match the requirements input. Starting from the intake, the individual blocks solve for a given inlet condition and produce output parameters. The input and output quantities of all the bricks are a fuel-air ratio, a mass flow, static pressure, total pressure, static temperature, total temperature, velocity and area. The output from one brick is then passed as the input to the following brick, e.g. from intake to compressor. Other data required to calculate the performance of the engine is provided within

the code. The key components modelled for a turbofan engine are an intake, compressor, combustion chamber, turbine and nozzle. The intake block simply applies a total pressure loss to the freestream flow. The compressor efficiency is determined from a series of characteristics and the required pressure ratio given as input. This then determines a work requirement for the turbine block, where an efficiency is again determined from a set of characteristics within the library of Turbomatch. For the models in this thesis, a turbine entry temperature was provided to Turbomatch and the fuel flow rate determined within the code. The convergent nozzle block expands the flow from the turbine to the freestream and determines an exit area which is required for the geometry definition.

Turbomatch was used to create engine cycles based on the required thrust. This required thrust was estimated from the CRM aircraft and the cycle iterated on until the cruise thrust was attained. The fan diameters were determined first for the engine sizes it was chosen to investigate, in order to represent an expected large UHBPR engine. The BPR and overall pressure ratio (OPR) were constrained by the expected limits on the cycle for these future large engines. Once a cycle was determined which produced the required thrust, a final iterative process was carried out to optimise the fan pressure ratio and turbine entry temperature to provide minimum specific fuel consumption.

3.2.2 Nacelle Parameterisation

An existing parameterisation based on intuitive class shape transformation (iCST) curves to define the engine aerolines was used.²⁷ For the purposes of this thesis, the entire engine housing will be termed the nacelle. The nacelle then consists of an intake, a fan cowl, bypass and core nozzles, a core cowl and a plug. A series of iCST curves were used to define each of these separately. iCST curves are an extension of the class shape transformation (CST) method created for aerofoil design⁹⁶. A CST curve is defined by the product of a class function ($C(\psi)$) and a shape function ($S(\psi)$) and the addition of a trailing edge term ($\psi\Delta\xi_{TE}$) (Equation 3.4). The class function (Equation 3.5) determines the fundamental shape of the curve which are determined by the N_1 and N_2 terms. Through the modification of these values various shapes can be determined including a round nosed aerofoil, a straight line, an elliptic aerofoil, or a Sears-Haack body⁹⁶. The shape function is defined by Bernstein polynomials of order n (Equation 3.6), chosen for their inherent smoothness and partition of unity property. This property ensures that

any order of Bernstein polynomial can be used to provide the perturbations to the class function. The use of Bernstein polynomials requires setting the value of a number of coefficients which are non-intuitive in the sense of aerodynamic design. The iCST approach helps to resolve this problem through the combination of the parameterisation method for airfoil sections (PARSEC) and the CST method⁹⁷.

$$\xi(\psi) = S(\psi)C(\psi) + \psi\Delta\xi_{TE} ; \quad \xi = \frac{y}{c} , \quad \psi = \frac{x}{c} \quad (3.4)$$

$$C_{N_2}^{N_1}(\psi) = \psi^{N_1}[1 - \psi]^{N_2} \text{ for } 0 \leq \psi \leq 1 \quad (3.5)$$

$$BPn = \sum_{r=0}^n K_{r,n} x^r (1 - x)^{n-r} \quad (3.6)$$

For the intake and fan cowl an aerofoil class function was used. For the remainder of the aerolines a straight line class function was used. These provided the basic shapes to which the shape function was added. The shape function for all aerolines consisted of Bernstein polynomials. To determine the coefficients of the Bernstein polynomials a series of intuitive design parameters were defined.^{27,98} The purpose of the intake is to deliver a sufficient mass flow with as little distortion as possible to the fan. It must also incur as few losses in doing so as possible to avoid efficiency loss of the entire system. The fan requires a slower inlet Mach number than a typical aircraft cruises so during this section of the flight the flow must also be decelerated. The throat is typically sized to allow a deceleration even at maximum power setting and flight Mach number. This throat area gives a specific radius (r_{th}) which is then positioned sufficiently far upstream of the fan to avoid diffusive separation under any of the flight conditions. The axial and radial location of the highlight relative to the throat then give the intake a particular contraction ratio ($CR = \frac{A_{hi}}{A_{th}}$) and aspect ratio (AR). The contraction ratio determines the amount of acceleration experienced by the flow relative to the highlight Mach number. The highlight area must also be sized to prevent too low a mass flow capture ratio MFCR in cruise operation. Too high a r_{hi} would result in a low MFCR for a given engine design and would be detrimental to the performance of the fan cowl. The design of the fan cowl relies on eight design parameters in this study (r_{hi} , l_{nac} , r_{max} , l_{fb} , r_{if} , r_{TE} , β , κ_{TE}) (Figure 3.3). The highlight radius (r_{hi}) is common with the intake and as discussed determines the mass flow capture ratio for the fan cowl. The initial

forebody radius (r_{if}) is the inverse of the curvature at the cowl lip. A high value will therefore indicate less curvature and a less sharp fan cowl. The maximum radius (r_{max}) and length (l_{nac}) are usually determined together to ensure space within the nacelle for all the engine peripheries and the thrust reverse unit. In addition these two parameters have a direct importance to the aerodynamics since they determine the fineness of the fan cowl. In general, it is desired to minimise the length and maximum radius to reduce the profile drag of the fan cowl. The r_{max} chosen for a design also has fundamental implications to the installation interference on the aircraft wing due to close coupling and ground clearance. The axial position of the maximum radius (l_{fb}) determines the split between forebody and afterbody. Historically a cylindrical centerbody has also been incorporated which provided separation between the acceleration of the forebody and the deceleration of the afterbody whilst providing additional volume for internal peripheries. The fan cowls in this study do not include this and are continually curved towards the engine axis. The trailing edge radius (r_{TE}) will be largely fixed by the requirements of the engine nozzle exit area and the core cowl size. The fan cowl boattail angle (β) will also need to be compatible with the angles of the core cowl and plug whilst being low enough to avoid a diffusive separation or excessive boundary layer growth on the afterbody of the fan cowl.

A further addition was made to the parameterisation which involved splitting the fan cowl curve into two separate iCST lines. This involved defining two iCST curves which both started at the highlight and ended at the trailing edge of the fan cowl. A single curve was created by sampling the forebody section of one curve and concatenating the afterbody section of the other curve. The two curves also had the same maximum radius (r_{max} , f_{max}), boattail angle (β) and initial forebody radius (r_{if}) however the afterbody curve was additionally constrained by a specified trailing edge curvature (κ_{TE}) whereas the forebody curve did not have this constraint. The afterbody curve was also constrained to match the first, second and third derivatives of the forebody curve at the position of maximum radius (Figure 3.4). This hybrid iCST approach allows the afterbody requirements to be decoupled to some extent from the forebody constraints. Demonstration of this adaptation is further detailed in section 5.4.

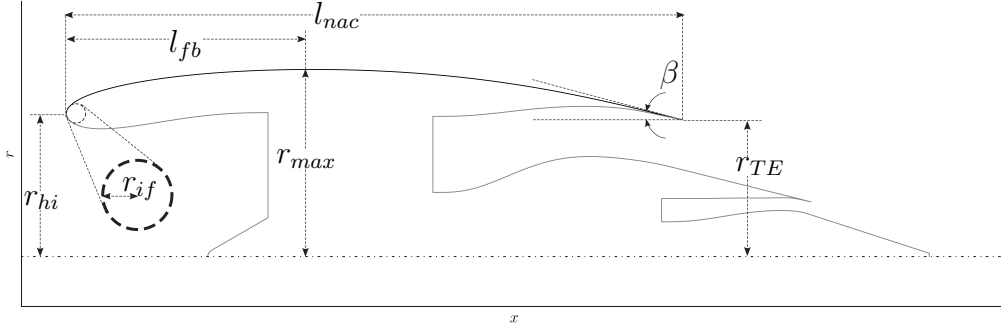


FIGURE 3.3: Fan cowl parameterisation

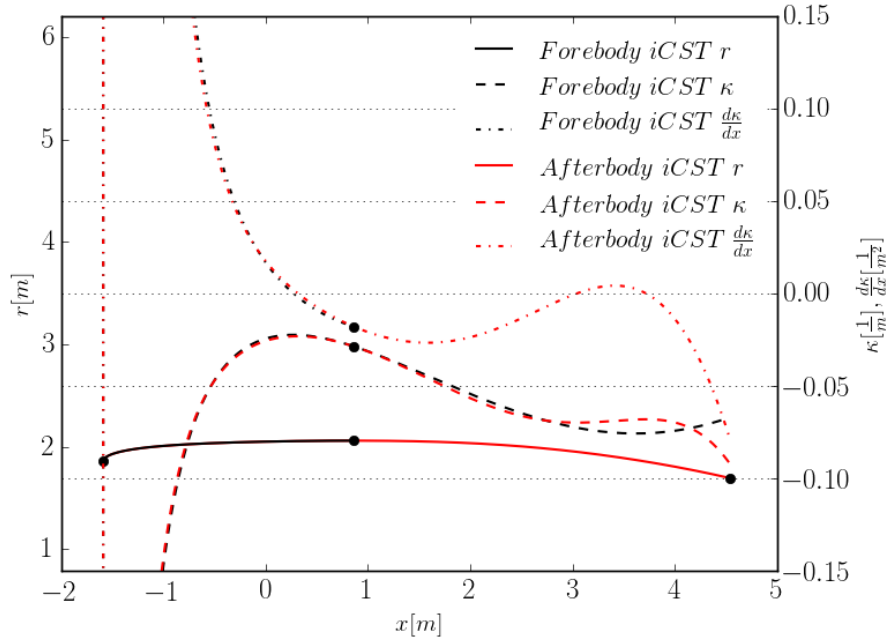


FIGURE 3.4: Schematic of hybrid iCST parameterisation

For the 3D axisymmetric nacelle geometries, the 2D section was simply revolved around the axis to create the surface. For the non-axisymmetric designs however a 3D parameterisation was needed.⁹⁹ Two additional parameters which define a non-axisymmetric nacelle are droop, defined by the droop angle (θ_{droop}), and the scarf, defined by the scarf angle (θ_{scarf}) (Figure 3.5). These two parameters define the offset required for the highlight plane to re-orient the intake axis. In addition to these two parameters the 2D parameterisation was utilised on individual azimuthal lines ($\theta = n$) where n is an angle between 0° and 360° . The topline defines the aeroline which would be closest to the wing in an underwing installation, i.e. the highest vertical aeroline. The topline is then defined as $\theta = 0^\circ$ and θ increases in the clockwise direction. Each of the 2D parameters (Figure 3.3) can then be

defined at each of the azimuthal aeroline. The method used⁹⁹ to construct the non-axisymmetric nacelles constructs an azimuthal iCST for each of these parameters to ensure smooth behaviour in the intermediate azimuthal aerolines which are not specified. The result was an iCST based analytical definition of a nacelle in both the $x - r$ plane and $r - \theta$ which could be sampled accordingly. For this project the geometries were then sampled in 256 azimuthal sections for use in the CFD computations.

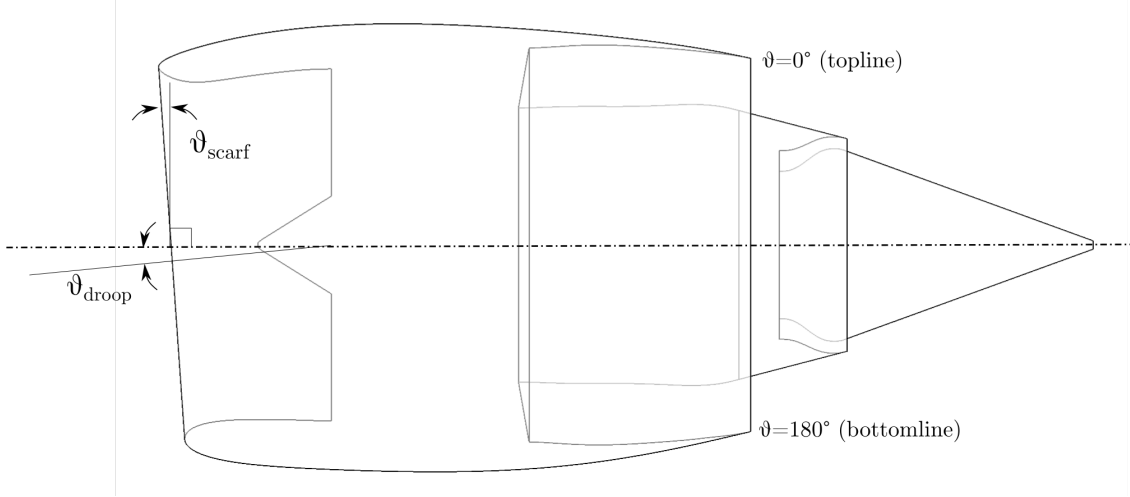


FIGURE 3.5: Schematic of 3D nacelle geometric parameters

3.3 Installation positions

In this project, the aero-engines are installed on the CRM aircraft in two positions relative to the wing. These are based on engine locations as defined and studied in previous work.¹¹ These positions were also contextualised with a number of production engines found in the open literature^{11 100} (Figure 3.6). The two positions considered in this project are equivalent to positions C3 and C1 in the previous project¹¹ but will simply be referred to as the datum and the close-coupled position throughout this thesis. Both positions define the location of the engine relative to the wing as a percentage of the local wing chord c_{local} . The spanwise location is constant and equivalent to the original CRM engine position. The datum position (C3) locates the fan cowl topline trailing edge $0.05c_{local}$ horizontally upstream of the wing leading edge and $0.1685c_{local}$ vertically lower than the wing leading edge (Figure 3.6). The close-coupled position (C1) locates the fan cowl topline trailing edge

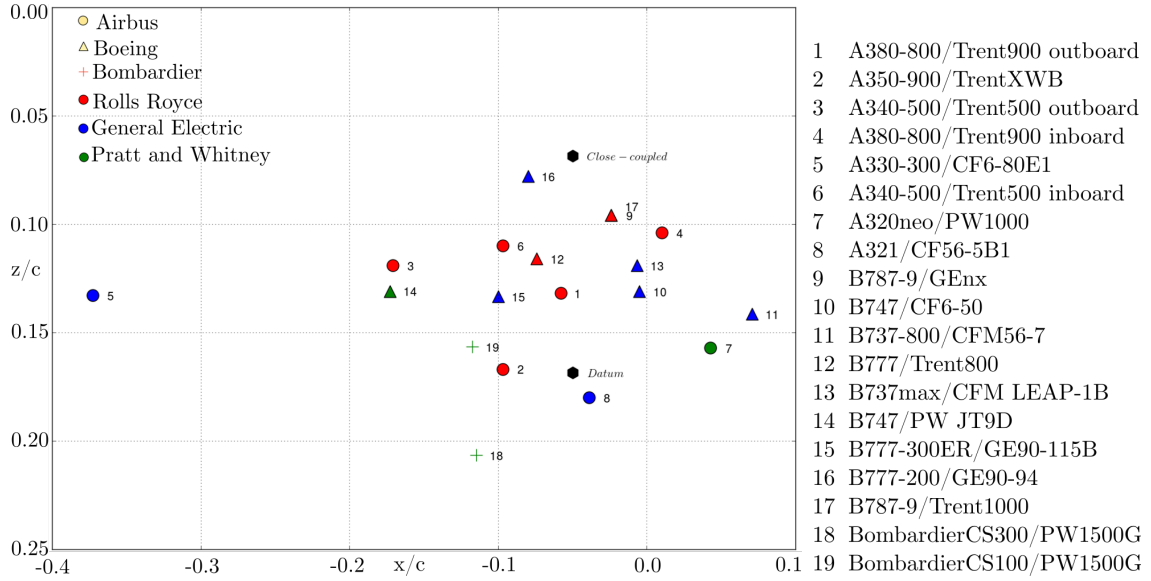


FIGURE 3.6: Engine installation positions relative to open source locations based on Stankowski¹¹

$0.05c_{local}$ horizontally upstream of the wing leading edge and $0.0685c_{local}$ vertically lower than the wing leading edge (Figure 3.6).

3.4 Thrust/Drag Accounting

3.4.1 Isolated Engine Thrust/Drag Bookkeeping

The modified near field method⁵² involves direct integration of the forces which act on the surfaces of the nacelle. It is a near field approach to drag extraction based on integration of the pressure and viscous forces which act on the outer nacelle and a momentum integration of the pre-entry stream tube⁵². Drag is evaluated either as the nacelle drag, or a modified nacelle drag. Figure 3.7 shows the constituent terms of both these drag values. The modified nacelle drag (D_{nac}^*) is defined as the sum of the integrated forces acting on the fan cowl ϕ_{nac} and pre-entry streamtube ϕ_{pre} (Equation 3.7). This was reported as a modified drag coefficient (C_{D^*}), relative to the maximum nacelle area and free stream conditions (Equation 3.8). For configurations with an exit streamtube, i.e. not sting-mounted, there is an additional drag component: the pressure force acting on the exit streamtube (ϕ_{post}). For the cases with this additional force, the pressure was integrated along a bounding streamline extended from the nacelle trailing edge. This post exit force (ϕ_{post}) was then

incorporated into the standard nacelle drag (D_{nac}) (Equation 3.9) and reported as a coefficient based on freestream conditions and maximum nacelle area (Equation 3.10).

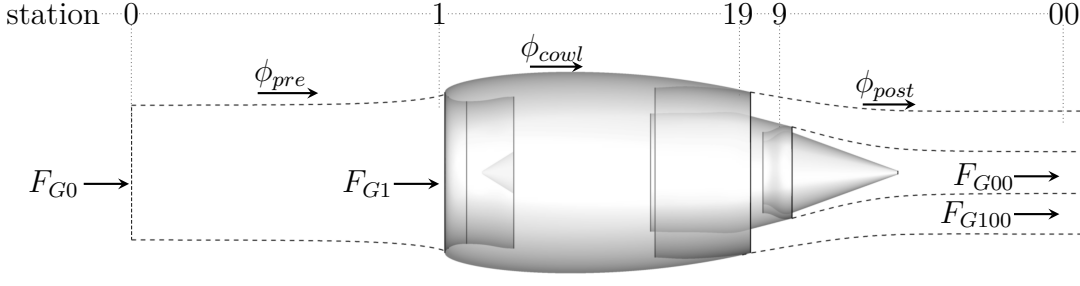


FIGURE 3.7: Thrust and drag accounting system

$$D_{nac}^* = \phi_{nac} + \phi_{pre} \quad (3.7)$$

$$C_{D^*nac} = \frac{D_{nac}^*}{\frac{1}{2}\rho_{\infty}V_{\infty}^2 A_m} \quad (3.8)$$

$$D_{nac} = \phi_{nac} + \phi_{pre} + \phi_{post} \quad (3.9)$$

$$C_{D,nac} = \frac{D_{nac}}{\frac{1}{2}\rho_{\infty}V_{\infty}^2 A_m} \quad (3.10)$$

The engine coordinate system (x,y,z) is defined such that the x-axis is parallel to the engine shaft. The z-axis is orthogonal to this and the positive direction indicates the topline of the nacelle. To simulate engines operating at an angle of attack, the incidence (α_{CL}) was defined as the angle between the engine centreline and the freestream flow. At incidence, the drag axis is not equivalent to the engine axis, and a transformed Cartesian coordinate system (Equations 3.11,3.12,3.13) is used to evaluate the drag.

The net propulsive force (NPF) of the engine is the overall resultant force which acts on the engine surfaces in the x' direction. The NPF can be broken down into thrust and drag based on either of the aforementioned drag definitions (Equation 3.14). The corresponding thrust values can be evaluated either as modified net thrust (F_{N*}) or standard net thrust (F_N). The standard net thrust is defined as the difference in gauge forces between the downstream infinity streamtube cross section and the upstream infinity streamtube cross section (Equation 3.15). For a dual

stream engine the modified net thrust is the difference between the gauge forces which exit the bypass and core nozzles and the gauge force at the upstream infinity streamtube cross section, with the integral force on the exhaust system subtracted (Equation 3.16).³⁰

$$x' = x \cos(\alpha_{CL}) \quad (3.11)$$

$$y' = y \quad (3.12)$$

$$z' = z \sin(\alpha_{CL}) \quad (3.13)$$

$$NPF = F_{N*} - D_{nac}^* = F_N - D_{nac} \quad (3.14)$$

$$F_N = F_{G00} - F_{G0} \quad (3.15)$$

$$F_N = F_{G19} + F_{G9} - F_{G0} - \theta_{exhaust} \quad (3.16)$$

3.4.2 Wind Tunnel Corrections

The validation cases studied for this project were simulated attached to an experimental sting. The nacelle therefore does not produce any post exit force (ϕ_{post}) and the modified nacelle drag is calculated from a downstream measurement rake. However, as the experimental rake is located between 0.12 and 0.7 nacelle lengths downstream of the trailing edge, depending on the rig, some of the measured drag is due to viscous losses incurred by the sting. In these cases an additional drag term acting on the sting ϕ_{sting} must be taken into account. An adjusted drag (D'_{nac}) is therefore defined which gives the value assessed at the location of the experimental rake, equation 3.17, or as a drag coefficient in equation 3.18. For the V1 nacelle test case this drag (D'_{nac}) is how the experimental data was reported.⁴⁶ As part of the experimental programme for validation nacelles V2 and V3 however this additional drag term (ϕ_{sting}) was estimated in the analyses of nacelles V2 and V3 based on flat plate aerodynamic characteristics and subtracted from the measured drag (D'_{nac}) to produce an estimate of the modified nacelle drag (D_{nac}^*).¹⁰¹ For the CFD validation of nacelles V2 and V3, this flat plate method was reproduced to give a corrected drag coefficient (C_{Dc}) (Equations 3.19, 3.20, 3.21, 3.22, 3.23).

$$D'_{nac} = \phi_{nac} + \phi_{pre} + \phi_{sting} \quad (3.17)$$

$$C_{D'} = \frac{D'_{nac}}{\frac{1}{2}\rho_{\infty}V_{\infty}^2 A_m} \quad (3.18)$$

$$c_f = \frac{0.455}{(\log_{10} Re)^{2.58} (1 + 0.144 M^2)^{0.65}} \quad (3.19)$$

$$Re_{nac} = Re_x L_t \quad (3.20)$$

$$Re_{combined} = Re_x (L_t + L_{sting}) \quad (3.21)$$

$$C_{D,sting} = \left[\frac{A_{wc} + A_{ws}}{A_{max}} \right] c_{f,combined} - \frac{A_{wc}}{A_{max}} c_{f,t} \quad (3.22)$$

$$C_{Dc} = C_{D'} - C_{D,sting} \quad (3.23)$$

3.4.3 Full aircraft Thrust/Drag Bookkeeping

Since this project is also to look at installation effects of aero-engine nacelles it is required to have a bookkeeping method for the full aircraft configuration. This method was largely adopted from a previous study¹¹ whilst ensuring compatibility with the AGARD definitions³⁰ through the addition of the post exit terms (ϕ_{post}) into the nacelle drag term (Equation 3.9). Prior to exposition of the force accounting it is important to clearly define the coordinate systems in use to define the resolved forces. An aircraft axis is defined (x, y, z) around the symmetry plane of the aircraft (Figure 3.8). The origin exists at the most forward point of the fuselage nose cone. The x-axis lies parallel to the axis of the cylinder which defines the fuselage mid-body, with the positive direction being from fore to aft of the aircraft. The y-axis runs normal to the symmetry plane, with the positive direction moving outwards from the symmetry plane. The z-axis is orthogonal to the x and y-axes and defined positive in the direction from the lower to the upper surface of the wing. An alternative axis is defined for convenience for the installed engine (ξ, η, ζ) as it does not sit parallel to the fuselage (Figure 3.8). To account for toe-in angle, which is in the x-y plane, and pitch angle, which is in the y-z plane, a transformed Cartesian coordinate system is defined for the engines. This is the same as the above defined system for the isolated engine in relation to the engine axis and can be transformed back to the aircraft coordinate system by a rotation by the toe-in and pitch angles. Finally, the aerodynamic axis (x', y', z') is defined relative to the aircraft axis (Figure 3.8) (Equations 3.24, 3.25, 3.26). Since this project does not consider aircraft in yaw, the y-axis is equivalent between aircraft and aerodynamic coordinate systems.

$$x' = x \cos(\alpha) \quad (3.24)$$

$$y' = y \quad (3.25)$$

$$z' = z \sin(\alpha) \quad (3.26)$$

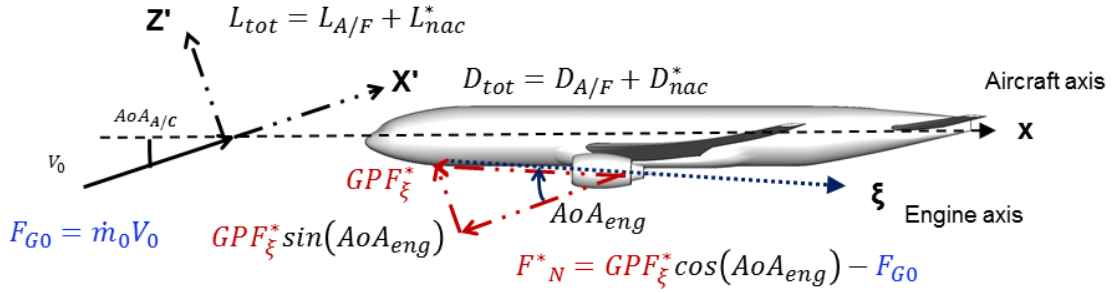


FIGURE 3.8: Coordinate systems present in the installed configuration for aerodynamic forces, aircraft and engine¹¹

As with the nacelle, the drag is defined in the aerodynamic axis, as being positive drag when a positive resultant force exists in the positive x' direction. The airframe drag is defined as the surface integral of all the pressure and viscous forces acting on the surface of the airframe, which includes fuselage, wing surfaces and horizontal tailplane surfaces (Equation 3.27). This is non-dimensionalised as a coefficient by freestream aerodynamic properties and the CRM wing planform (Equation 3.28). In the presence of an engine installation the drag domain is extended to include all the metal and streamtube surfaces used for the isolated engine above. This results in an overall aircraft drag ($D_{a/c}$) which is the sum of the airframe drag ($D_{a/f}$) and the fan cowl drag (ϕ_{nac}), pre-entry drag (ϕ_{pre}) and post-exit drag (ϕ_{post}) (Equation 3.29). Again this is non-dimensionalised by the freestream conditions and the wing planform area to obtain a drag coefficient (Equation 3.30). The overall force acting in the x' axis is termed the net propulsive force (NPF). The NPF is the difference between the standard net thrust and the overall aircraft drag (Equation 3.31). Lift is defined in a similar way however it is defined as positive in the positive z' axis. Airframe lift consists of a similar integration of all forces over the surface of the airframe which act in the z' direction (Equation 3.32). This is non-dimensionalised as a coefficient by freestream aerodynamic properties and the CRM wing planform (Equation 3.34). In addition, this same process is followed for the nacelle since no split is required in this axis for the nacelle (Equation 3.33). The summation of these two lift forces results in the overall lift force of the aircraft (Equation 3.35) which

is again non-dimensionalised by the freestream conditions and the wing planform area (Equation 3.36). For a configuration in which there is no engine installed then $D_{a/c} = D_{a/f}$ and $L_{a/c} = L_{a/f}$.

The installation penalty can be considered at a constant incidence ($\alpha_{a/c}$) in which case there would be a lift loss and a drag penalty from the addition of an engine to the configuration. Alternatively, it can be considered for a re-trimmed aircraft in which case the lift would be the same between the clean wing aircraft with an isolated engine and the aircraft with an engine installed. This is termed the superposition model.¹¹ The installation drag is then defined as the difference between the installed configuration (WBT0N) and the clean wing configuration (WBT0) at constant lift (Equation 3.37). To superimpose the isolated engine drag to the clean aircraft drag an equivalent engine incidence must be established which the engine would operate at were it installed. This equivalent incidence (Equation 3.38) is defined as the sum of the airframe incidence, the incidence at which the engine would be installed at, and the upwash which the engine would experience. This incidence can then be used to determine the isolated engine drag to superimpose on the airframe drag at each value of $\alpha_{a/f}$. The upwash was determined from the clean wing CFD computations at the position of the centre of the highlight for the engines. Similarly a lift can be evaluated at every airframe incidence of the superposition model (Equation 3.40). These two values were computed for each clean aircraft incidence and for each engine in each position considered. This allowed the production of a superposition polar (Figure 3.9). Interference drag (Equation 3.41) can then be defined as the difference between the drag of a full aircraft and the drag of an airframe superimposed with an engine (Figure 3.9). Since the CFD has been computed at given values of $\alpha_{a/c}$, this re-trimmed approach requires some interpolation between simulations to ensure equality in the lift coefficients. A second order approach was adopted for this process in accordance with previous analysis.¹¹ A thrust interference is similarly defined where the thrust in the superposition model is subtracted from the thrust in the installed configuration (Equation 3.42).

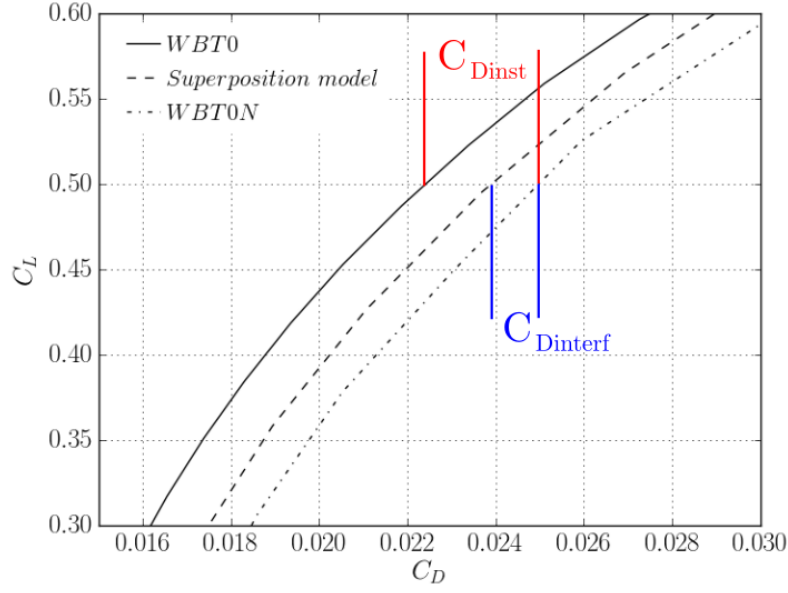


FIGURE 3.9: Definition of installation drag and interference drag

$$D_{a/f} = \int \int_{S_{a/f}} (-pn_{x'} + \tau_{x'x'}n_{x'} + \tau_{x'y'}n_{y'} + \tau_{x'z'}n_{z'}) dS \quad (3.27)$$

$$C_{D,a/f} = \frac{D_{a/f}}{0.5\rho_{\infty} * A_{planform} * V_{\infty}^2} \quad (3.28)$$

$$D_{a/c} = D_{a/f} + \phi_{pre} + \phi_{nac} + \phi_{post} \quad (3.29)$$

$$C_{D,a/c} = \frac{D_{a/c}}{0.5\rho_{\infty} * A_{planform} * V_{\infty}^2} \quad (3.30)$$

$$NPF = F_N - D_{a/c} \quad (3.31)$$

$$L_{a/f} = \int \int_{S_{a/f}} (-pn_{z'} + \tau_{z'z'}n_{z'} + \tau_{x'z'}n_{x'} + \tau_{y'z'}n_{y'}) dS \quad (3.32)$$

$$L_{nac} = \int \int_{S_{nac}} (-pn_{z'} + \tau_{z'z'}n_{z'} + \tau_{x'z'}n_{x'} + \tau_{y'z'}n_{y'}) dS \quad (3.33)$$

$$C_{L,a/f} = \frac{L_{a/f}}{0.5\rho_{\infty} * A_{planform} * V_{\infty}^2} \quad (3.34)$$

$$L_{a/c} = L_{a/f} + L_{nac} \quad (3.35)$$

$$C_{L,a/c} = \frac{L_{a/c}}{0.5\rho_{\infty} * A_{planform} * V_{\infty}^2} \quad (3.36)$$

$$C_{D,inst} = (C_{D,a/c} - C_{D,a/f}) \Big|_{C_L=const.} \quad (3.37)$$

$$\alpha_{equivalent} = \alpha_{a/f} + \alpha_{inst} + \alpha_{upwash} \quad (3.38)$$

$$D_{SM} = D_{a/f} + D_{nac} \Big|_{\alpha_{equivalent}} \quad (3.39)$$

$$L_{SM} = L_{a/f} + L_{nac} \Big|_{\alpha_{equivalent}} \quad (3.40)$$

$$D_{interf} = (D_{a/c} - D_{SM}) \Big|_{C_L=const.} \quad (3.41)$$

$$F_{N*,interf} = (F_{N*,inst} - F_{N*,SM}) \Big|_{C_L=const.} \quad (3.42)$$

3.4.4 Key metrics

Several aerodynamic metrics are considered throughout this project. The main consideration is drag, which as discussed (section 3.4) has varied definitions. For the majority of this report drag is considered as in equation 3.10. However, this definition involves non-dimensionalisation by the maximum nacelle area. Whilst this is a common definition and one used for the validation, it is not intuitive for comparisons where A_{max} is a variable. As such, for the optimisation procedures carried out in this project an additional coefficient is defined based on a common area (Equation 3.43)

$$c_{Dhi} = \frac{D_{nac}}{\frac{1}{2}\rho_{\infty}V_{\infty}^2 A_{hi}} \quad (3.43)$$

3.4.4.1 Drag rise

The second metric considered was the drag rise Mach number (M_{DR}) (Equation 3.44). The drag rise Mach number determines the freestream Mach number above which wave drag increases rapidly. Nacelles are typically designed to operate below this Mach number and a typical margin of safety used is $\Delta M = 0.02$ ³⁶. Since the desired cruise Mach number for the optimisation designs is $M_{\infty} = 0.85$ the target drag rise Mach number is M_{DR} .

$$\left. \frac{\partial C_D}{\partial M_\infty} \right|_{M_{DR}} = 0.1 \quad (3.44)$$

3.4.4.2 Spillage

As discussed in chapter 2, the spillage of an aero-engine nacelle can be defined in several different ways. The most relevant for the design however is typically the difference in drag encountered between a cruise condition and a lower MFCR condition. The drag of a nacelle can also be decomposed into profile, wave and spillage drag (Equation 3.45).¹⁰² This indicates that in addition to the baseline profile drag of a nacelle two other influences are present. The wave drag term is not present in a purely subsonic regime and grows larger as the freestream Mach number exceeds the critical Mach number. The spillage term occurs due to the influence of the mass flow capture ratio (MFCR), whereby a reduction in MFCR results in an increase in spillage on the forebody of the nacelle. Spillage in this project will be considered as a difference between a low MFCR condition and a nominal cruise condition.

$$C_{D,nac} = C_{D,prof} + C_{D,wave} + C_{D,spill} \quad (3.45)$$

3.4.4.3 Pressure coefficient metrics

In the course of the project, it was required to define two metrics based on the pressure coefficients on the fan cowl. The first metric of interest (Equation 3.46) was the sum of the peak pressure coefficient upstream of the shock across the number of shocks present on the cowl. This should provide an additional penalty to designs with high wave drag and also exclude designs which have several sonic regions. Shock location was determined by a local peak in pressure coefficient gradient which exceeded a value of 0.65, which was found to be a good indication of a shock (Figure 3.10). The second metric of interest (Equation 3.47) quantified the number of points at which the gradient changed polarity in the pressure distribution since a conventional design should ideally only do this once²⁹. Specifically, it calculated the total number of gradient ($\delta C_p / \delta x$) polarity changes and summed the magnitude of the second derivative at this polarity change (λ). When a constraint was applied to the former metric (C_{shock}) it limited the shock strength allowed

on the nacelle. Minimisation of the latter metric ($C_{inflect}$) results in designs with increasingly monotonic pressure distributions.

$$C_{shock} = \sum_m |c_p| \quad (3.46)$$

$$C_{inflect} = \sum_n |\lambda| \quad (3.47)$$

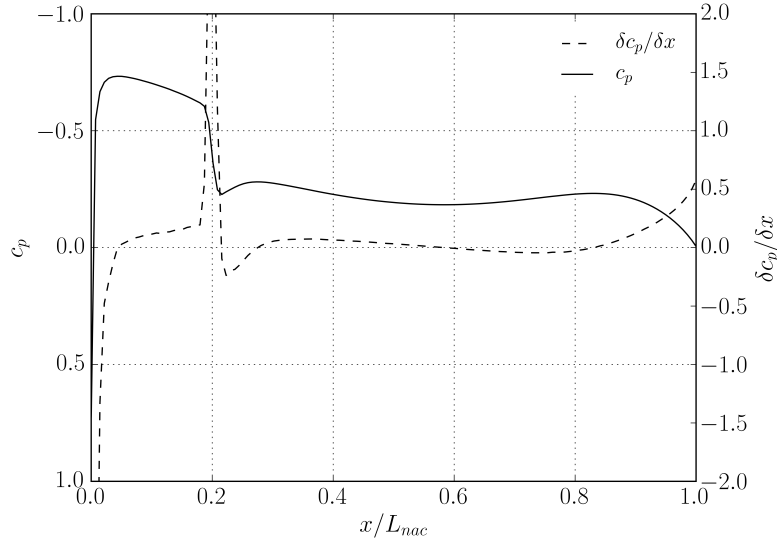


FIGURE 3.10: Distribution of pressure coefficient and pressure coefficient gradient

For the non-axisymmetric nacelles (Chapter 6) a weighted version of these two metrics was defined (Equations 3.48, 3.49). This allowed for a single metric to be defined based on the pressure distributions of all aerolines which were considered.

$$C_{shock,weighted} = 0.25C_{shock,topline} + 0.375C_{shock,sideline} + 0.25C_{shock,controlline} + 0.125C_{shock,bottomline} \quad (3.48)$$

$$C_{inflect,weighted} = 0.25C_{inflect,topline} + 0.375C_{inflect,sideline} + 0.25C_{inflect,controlline} + 0.125C_{inflect,bottomline} \quad (3.49)$$

3.5 Optimisation Method

A routine for intake and nacelle genetic optimisation (RINGO) has been developed within this project to create optimal nacelle designs. The routine is based on a real-parameter coded implementation of the NSGA-II algorithm⁸¹ coded in python. Real-parameter coding was used as opposed to binary coding to avoid problems such as Hamming cliffs which can exist when the fitness values are treated as binary strings.⁸¹ The generational optimisation method ranks a set of designs based on their associated objective function values $f(x)$ and constraints $g(x)$. In a generic optimisation these objectives and constraints could be evaluated from an analytical equation to link the input variables to the output variables. The implementation was thoroughly tested with analytical functions in chapter 4. Within RINGO, the objectives and constraints are evaluated from a computational fluid dynamics (CFD) simulation of the parametric geometry. This in-the-loop CFD process requires more computational resource than the use of a surrogate model based on integral parameters. However, it also provides a higher fidelity drag estimation and allows for optimisation of the aerodynamic characteristics. The overall routine of RINGO is described in Figure 3.11. This optimisation routine has subsequently been adapted and has been used in further work to optimise nacelles^{103,104} and intakes.¹⁰⁵

The optimisation was carried out with a Gaussian mutation operator with a mutation rate of 0.2 and mutation size of $\sigma = 0.0005$. The crossover was carried out using the $BLX - \alpha$ operator with $\alpha = 0.5$ which results in an even split between the two parent designs.¹⁰

For each design produced within the optimisation process a multi-grid mesh was produced and CFD simulations performed. To enable the calculation of the off design performance, the designs were simulated at a series of different freestream Mach numbers and two MFCRs. This resulted in ten separate solutions computed for each design. Each optimisation process consisted of 1560 different designs, so in total to optimise the fan cowl, for a specified length (l_{nac}/r_{hi}) and radius ratio (r_{TE}/r_{hi}), 15600 CFD computations were performed.

3.5.1 Key metrics

For a single objective optimisation problem the performance of an algorithm can simply be quoted as how accurately the optimal value produced by the algorithm

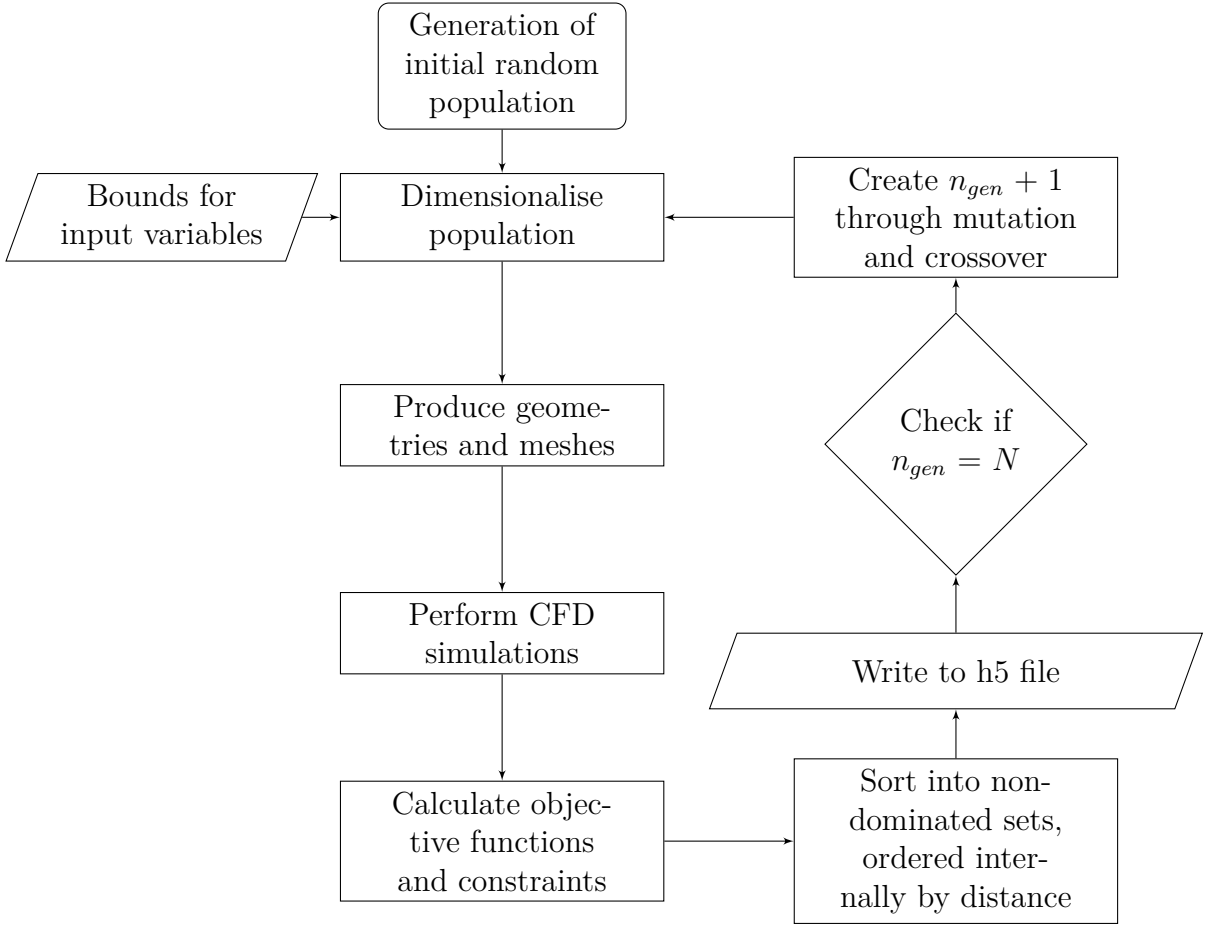


FIGURE 3.11: Routine for Intake and Nacelle Genetic Optimisation flowchart

approximates the true optimum. For a MOO problem however the optimum is an n -dimensional Pareto set. The performance of a MOO algorithm can be measured in terms of how well the Pareto set it produces (Y') approximates the analytically optimal Pareto set of solutions ($\bar{Y}$). Two aspects of this approximation are important: the convergence level and the distribution or diversity within the set. These requirements are orthogonal to each other within the objective space which means that a set Y' which converges closely to the Pareto set $\bar{Y}$ may not necessarily be well distributed along it. A genetic algorithm generally achieves both of these requirements through the application of dominance to ensure that more optimal solutions are favoured whilst ensuring diversity through a distance or spacing operator. There are metrics which can be used to assess these two requirements (convergence and diversity) separately, however in this project a metric which combines the two is used. This is formally defined as the hypervolume (Equation 3.50) and is most easily explained in a 2-objective optimisation problem. In this case, the hypervolume (HV) is the area bounded by the Pareto set and a reference coordinate

(Figure 3.12). In n-dimensions this concept is simply extended to a n-dimensional hypervolume. For a given Pareto set ($\bar{Y}$) and reference coordinate there is therefore a specific hypervolume which represents the maximum achievable value. Any less optimal set, whether due to poorer convergence or failure to adequately distribute solutions in the objective space, will have a lower associated hypervolume. For a known Pareto set ($\bar{Y}$) the hypervolume from the approximated set can be given as a percentage of the true hypervolume ($\bar{HV}$). It is important to note however that this percentage will vary depending on the reference point used to calculate the hypervolume so a reference value close to the Pareto set should be chosen to avoid overly optimistic results. Within this project, the open source software WFG¹⁰⁶ was used for calculating the hypervolume for the optimisations carried out.

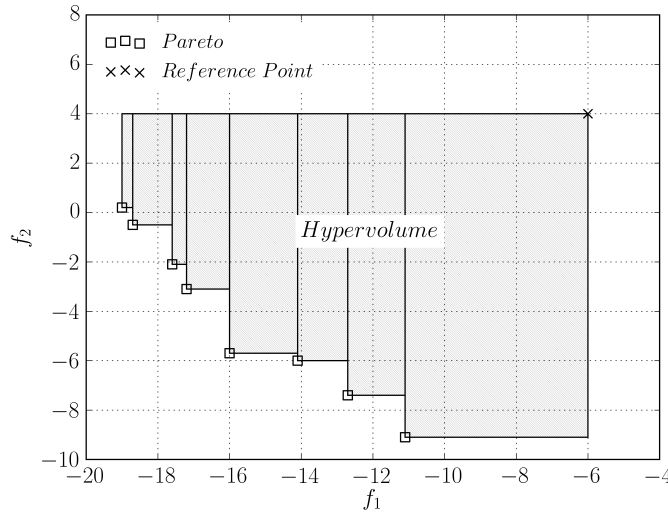


FIGURE 3.12: Schematic of hypervolume in two dimensions, based on Deb¹²

$$HV = volume \left(\bigcup_{i=1}^{|Y'|} v_i \right) \quad (3.50)$$

3.6 Conclusions

This chapter has outlined a method which will be followed through the work, the following summary points can be made:

- Computational Fluid Dynamics (CFD) will be the method used to establish nacelle performance. Multiple computations will allow the calculation of cruise drag, drag rise Mach number and spillage drag.

- Nacelles will be produced through a parameteric method based on intuitive class shape transformation (iCST) curves.
- Configurations which will be considered will include isolated and underwing engines. A drag accounting method has been detailed to fully capture relative performance of different engine designs.
- An optimisation method based on NSGA-II will be used to explore the design space and produce nacelles which are aerodynamically well designed for further study.
- An existing code, Turbomatch, will be used to provide boundary conditions for CFD of engine models.

CHAPTER 4

Validation and Verification

This chapter details the efforts which were made to validate the CFD method used throughout the project. Separate validation activities were carried out on the fan cowl and intake flows for a number of different geometries. In addition, this chapter includes details of a verification and calibration exercise which was undertaken to ensure the optimisation method was appropriately configured for the problems posed later on in the project.

4.1 CFD Validation

The computational fluid dynamics (CFD) method was tested on a series of different geometries and configurations to determine the accuracy of the simulations. These can be subdivided into simulations in which both the flow and geometry had rotational symmetry around the x-axis and could therefore be simplified into a 2-dimensional (2D) axisymmetric domain, and those which were not axisymmetric. The non-axisymmetric validation includes both simulations in which the aerodynamics were not axisymmetric, i.e. to simulate cases at incidence, and cases where the geometry is non-axisymmetric.

The first validation nacelle (V1) was the open source ARA cowl 1.⁴⁶ This was designed to assess alternatives to the conventional NACA series 1 type forebody designs. It is an axisymmetric design with a cylindrical centrebody, and a simple circular arc and straight line for an afterbody. The second validation nacelle (V2) was designed to provide a specific Mach number distribution on the nacelle.⁶⁹ This approach can allow an acceptable aerodynamic performance for required dimensions

through prescribed curvatures. V2 is also an axisymmetric design with no centrebody section and no straight line sections from the leading edge to the trailing edge, i.e. continuously curved. The absence of the centrebody causes the forebody flow to interact more strongly with the afterbody flow. The final nacelle considered (V3) is a three-dimensional design which incorporates both a conventional cylindrical centrebody as well as scarf and offset on the intake and nozzle.¹⁰⁷ The experimental data available was extensive and covered a range of different aerodynamic points and the nacelle drag extracted at each of these points. This allowed the nacelle drag as well as the drag as a function of the M_∞ and MFCR to be validated for. Nacelle V1 is significantly longer but has proportionally less frontal facing area (Figure 4.1). Aerodynamic assessment of this wide range of geometric parameters allows a performance estimation for both conventional and newer designs.

In the wind tunnel experiments^{46,69,107}, a rake was used to measure a plane downstream of the nacelle to assess the momentum loss throughout the wake plane, providing discrete measurements of total and static pressures. Values for experimental uncertainty were not provided in the experimental reports however. The experimental equipment provided 26 pressure readings at 4 azimuthal positions for the V1 nacelle to assess the wake losses.⁴⁶ From the CFD solution it was estimated that this accounted for between three and four measurements within the boundary layer for the range of Mach numbers tested. For the V2 and the V3 nacelles, the rake had an improved spatial resolution of probes near the surface of the sting so this number of measurement points was estimated to be between five and eight probes.¹⁰¹ To demonstrate the discretisation accuracy of this method, the wake integral method was performed on a CFD solution at both CFD resolution and rake resolution. The CFD resolution had between 38 and 40 finite volume cells in the wake boundary layer of the nacelle compared to the range of five to eight of the rake in the experiment. The assessment of the CFD based on these different resolutions was then converted into a simulated estimate of the uncertainty based on the original wind tunnel measurements to demonstrate the sensitivity of the wake integral method to measurement resolution. The simulated uncertainty in C_{Dc} increased as freestream Mach number increased, from $\pm 2 \times 10^{-5}$ at $M_\infty = 0.8$ to $\pm 5 \times 10^{-4}$ at $M_\infty = 0.89$. The uncertainty also depended on the MFCR and varied from $\pm 2 \times 10^{-5}$ at $MFCR = 0.7$ to $\pm 3 \times 10^{-5}$ at $MFCR = 0.45$. This growing error with Mach number has an impact on the ability to correctly determine the drag rise Mach number, particularly when determined by a gradient criterion. The simulated uncertainty on the drag rise Mach number based on equation 3.44 is $\pm 5 \times 10^{-4}$. In

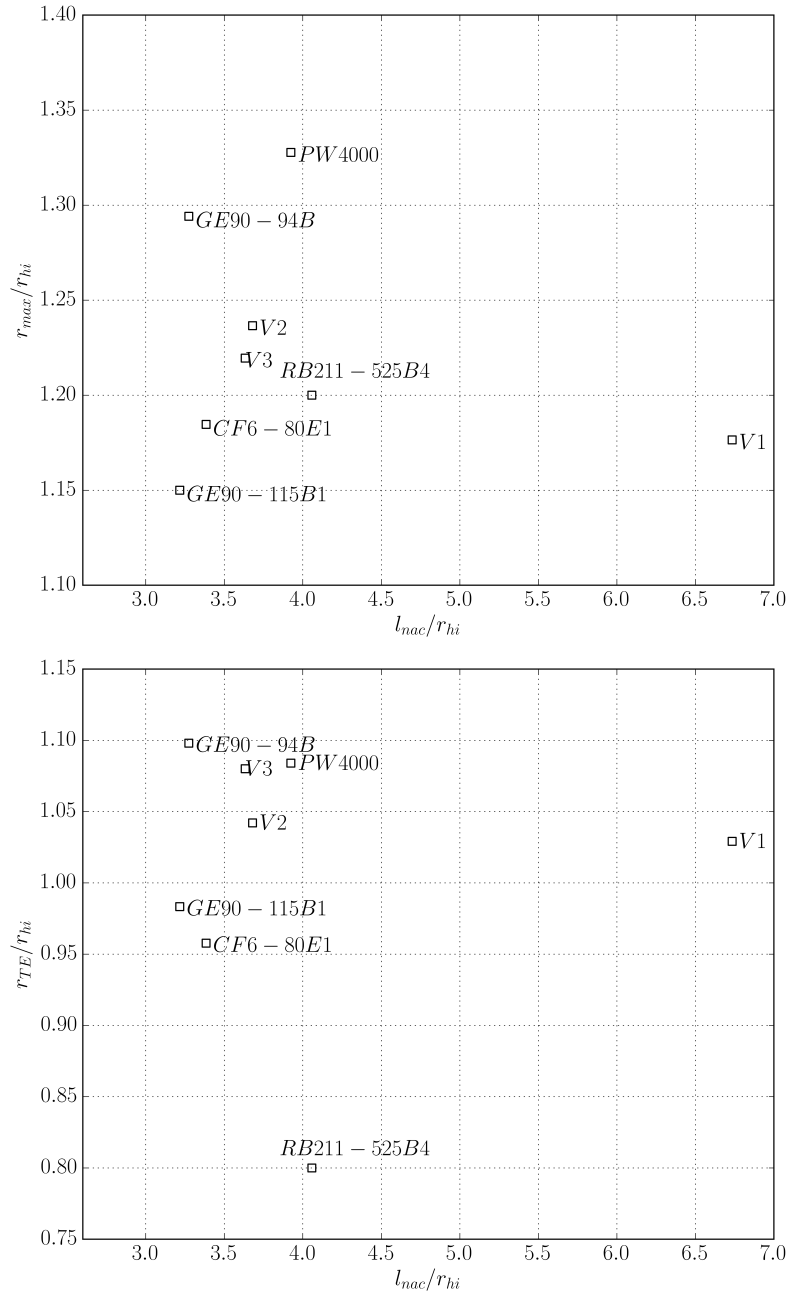


FIGURE 4.1: Validation nacelles with some existing designs

addition to this, small errors in the estimated free stream velocity can create large errors in drag, due to the summation of the differences in velocity over potentially relatively large areas (Equation 2.4). Therefore, for the CFD analyses the drag characteristics were determined using the modified near field method⁵².

Due to the placement of the measurement rake on the sting mounting, an additional drag term was measured in the experiment which was not part of the nacelle drag. In the wind tunnel analysis a correlation (Equation 3.19)¹⁰⁸ for flat plate skin friction was used to estimate the average skin friction coefficient of the sting section. Two values of Reynolds number (Re) were calculated, for the cowl length L_t (Equation 3.20) and the cowl and sting length L_{sting} combined (equation 3.21) from the Reynolds number per unit length (Re_x). Equation 3.19 then gives the average $C_{f,t}$ for the cowl and $C_{f,combined}$ for the sting and cowl combined. A calculation of the skin friction drag incurred by the sting (C_{Dsting}) is then estimated from the difference of the area weighted skin friction for $C_{f,combined}$ and $C_{f,t}$ (Equation 3.22). The experimental data reported for both the V2 and the V3 cowls had this correction for the sting applied to the measured drag to provide a corrected nacelle drag coefficient (C_{Dc}) (Equation 3.23). This method was applied in the CFD validation activity in the same way as in the wind tunnel analysis.

4.1.1 Axisymmetric nacelle aerodynamic validation

In this section, the validation of the 2D axisymmetric simulations is summarised. This consisted of test cases for the first two nacelles (V1, V2). The V1 nacelle was simulated across 62 aerodynamic simulations whilst the V2 was simulated across 90. These geometries differed greatly in design which allowed for a test of the CFD method for different aerodynamic profiles and at greatly different operating points.

Experimental data for the V1 were reported as adjusted nacelle $C_{D'}$ values (Equation 3.18), which include the viscous drag of $0.12L_t$ of cylindrical sting section downstream of the nacelle trailing edge. CFD analyses were carried out with the medium domain and coarse mesh for the V1 with the sting (NS, Figure 4.2) for a range of Mach numbers from 0.65 to 0.95. The adjusted drag coefficient $C_{D'}$ predicted by the CFD was typically within 4% of the experimental data prior to drag rise (Figure 4.3a, 4.3b, 4.3c). The CFD accuracy for M_{DR} , using equation 2.1, was between $\Delta M_{DR} = -0.001$ for $MFCR = 0.6$ and $\Delta M_{DR} = -0.014$ for

MFCR	M_{DR}	
	CFD	Measurement
0.6	0.825	0.826
0.7	0.845	0.848
0.76	0.852	0.866

TABLE 4.1: Measured and CFD predicted drag rise Mach number for V1

$MFCR = 0.76$ (Table 4.1). The CFD simulations slightly under-predicted the influence of the spillage drag measured at lower MFCRs (Figure 4.3d). For example, the measured $C_{D'}$ increased from 0.036 to 0.064 when the MFCR was reduced from 0.78 to 0.51 at $M_\infty = 0.85$ to give $C_{D'} = 0.028$. Across the same range of MFCR the CFD calculated $\Delta C_{D'} = 0.02$. The M_{DR} for both the experiment and the CFD data show that with a lower MFCR, the drag divergence occurs at a lower Mach number (Table 4.1). This is caused by the local incidence of the flow increasing at the highlight for a lower MFCR. A lower MFCR causes the stagnation point on the nacelle to move closer to the throat and therefore the flow has a greater distance to travel around the lip. Since this region is highly curved a large acceleration occurs which gives a higher peak Mach number and results in a stronger shock. The presence of the higher Mach numbers then expedites the drag divergence due to increased Mach number.



FIGURE 4.2: Nacelle/sting configuration (NS)

The V2 nacelle was also tested in a sting mounted configuration in the wind tunnel experiments.⁶⁹ Due to the placement of the rake downstream of the nacelle trailing edge, some of the measured momentum loss arises from the skin friction which acts on the sting. This should not be included in the nacelle drag and it was corrected for in the wind tunnel experiments by the assumption of a flat plate skin friction correction factor which has been applied to the reported measurements¹⁰¹. To reproduce this correction in the current computational work a method based on both the flat plate Reynolds number and Mach number was used to estimate the skin friction caused by the sting as was the case in the experimental procedure¹⁰¹. The V2 nacelle therefore was modelled computationally with the sting attached (NS) to give an accurate representation of the experimental set up (Figure 4.2).

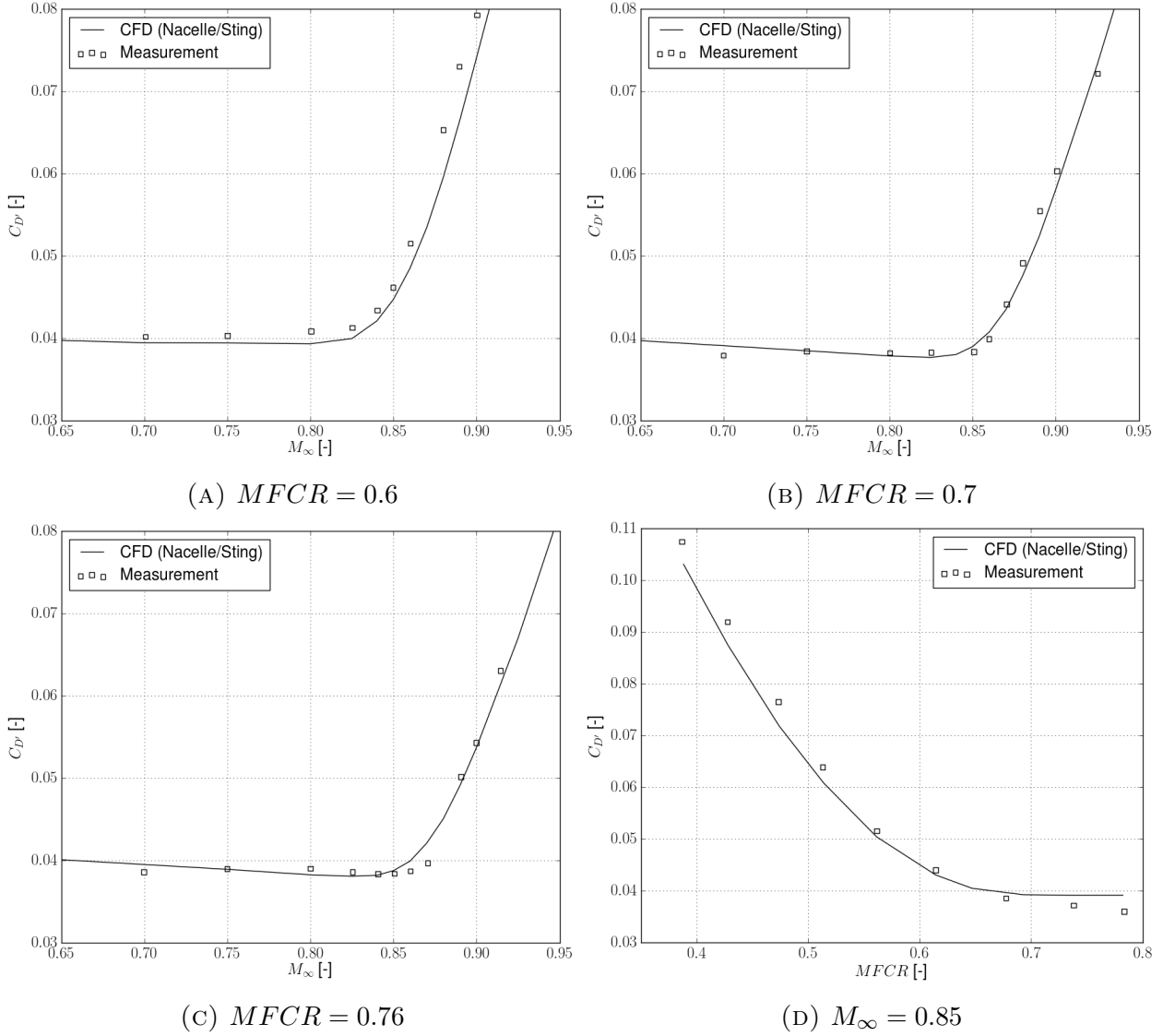


FIGURE 4.3: Drag coefficient of nacelle and sting section ($C_{D'}$) as a function of freestream Mach number and MFCR for the V1 nacelle in the nacelle/sting configuration (NS) from CFD analysis and experimental data

The CFD data was also post-processed in the same way as the experimental data (equation 3.23) to determine the corrected drag coefficient (C_{Dc}). A comparison of this corrected drag to the actual drag is made in Appendix C.

The V2 cowl is a configuration which was generated using an inverse design approach and is a more modern design compared to the V1 nacelle. One of the benefits of this design is that, relative to the V1 cowl (Figure 4.3b), there is a notable increase in the measured M_{DR} from 0.845 to 0.880 at a reference MFCR of 0.7 (Figure 4.4a). Another benefit of this detailed design is the improved performance at lower MFCR (Figures 4.3d, 4.4d). This lower gradient of drag coefficient with MFCR makes the

nacelle less penalising in the portions of a flight where a lower power setting is used. The improved control over the curvatures in the V2 nacelle resulted in a modest 20% increase in drag between $MFCR=0.7$ and $MFCR=0.5$ compared to an 85% increase for the V1 nacelle. At a $MFCR$ of 0.7, and below the M_{DR} of 0.880, the CFD for the V2 nacelle predicts the corrected drag coefficient C_{Dc} with an RMS ΔC_{Dc} of 0.0001 (Figure 4.4a). Above the M_{DR} the difference increases and at the highest M_∞ of 0.89 with $MFCR$ of 0.7, this ΔC_{Dc} is -0.0015 although there is also notable uncertainty in the experimental data at these higher Mach numbers. At this same $MFCR$ of 0.7, the measured M_{DR} , was 0.879 and the CFD predicted 0.880. For the effect of spillage across the range of $MFCR$ from 0.7 to 0.45 at $M_\infty = 0.8$, the difference between the CFD and the measurements is an RMS ΔC_{Dc} of 0.0013. The calculated CFD ΔC_{Dc} between $MFCR = 0.7$ and $MFCR = 0.5$ is 0.0036 in comparison with the measured value of 0.0023.

Whilst the agreement between the CFD simulations and the experimental data worsened as the $MFCR$ was reduced (Figure 4.4), the overall trends in drag rise were similar. The drag rise metric can be defined by different gradients as discussed (Section 2.1). Due to the limited number of experimental data points, more drag rise Mach numbers can be calculated by using a gradient of 0.05, however for completeness both criteria have been evaluated (Figure 4.5). The closest agreement based on the criterion of 0.05 occurs at the lowest $MFCR$ of 0.47 where the experiment predicted $M_{DR} = 0.867$ and the CFD predicted $M_{DR} = 0.868$ and the highest $MFCR$ of 0.7 where the calculated drag rise was $M_{DR} = 0.876$ and $M_{DR} = 0.875$ based on the experiment and CFD respectively. However, in between these two $MFCRs$ the agreement worsens with the measured data experiencing a precipitous drop in the drag rise below $MFCR = 0.55$ and such a change not present in the CFD simulations. It is important to note however that these changes all occur within a relatively small band of drag rise Mach numbers ($\Delta M_{DR} \leq 0.01$). For the current project the drag rise Mach number is only sought to be modelled at $MFCR = 0.7$ and above so this result was considered adequate to validate the capability of doing so. It was decided to use the higher drag rise criterion of 0.1 despite the slightly worse agreement between the measurement and CFD since this should be a more robust criterion in general. In addition, the trend in the isentropic Mach number distributions with an increase in M_∞ was predicted well by the CFD (Figure 4.6) which generally estimated the experimental pre-shock Mach number within $M = 0.02$.

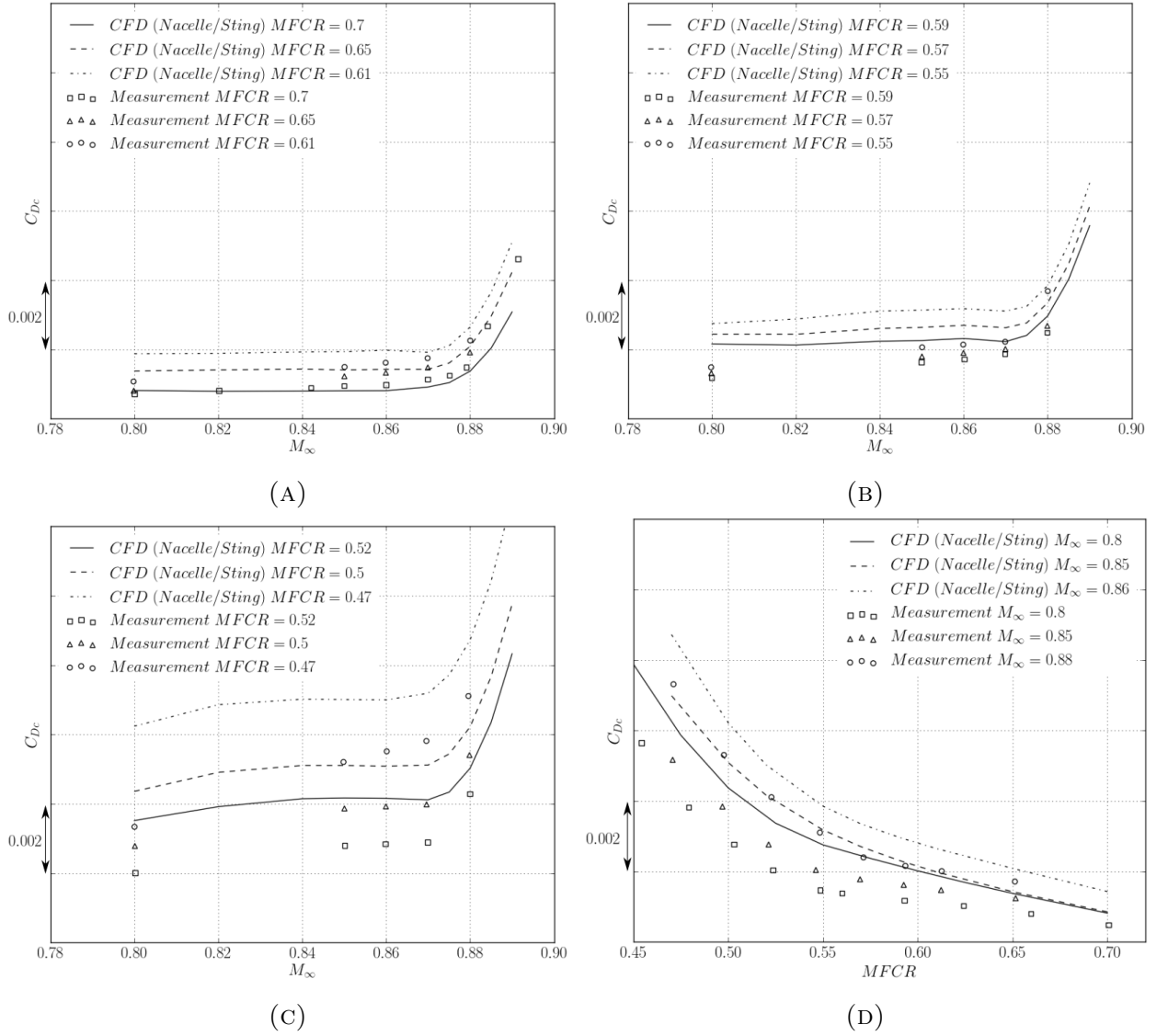


FIGURE 4.4: Drag coefficient (C_{Dc}) as a function of freestream Mach number and MFCR for the V2 cowl in the nacelle/sting configuration (NS) from CFD analysis and experimental data

4.1.2 3D nacelle aerodynamic validation

Across the flight envelope, the nacelles will operate at different angles of attack and the V2 nacelle and the V3 nacelle were investigated at a range of incidence up to 6° . Since it has been demonstrated that the absolute values from the experimental data can be largely influenced by the spatial discretisation of the rake, the analysis centred on the characteristics of drag rise and nacelle Mach number distributions at incidence. Nacelle incidence causes the stagnation of the streamtube to move into the intake near the top surface of the nacelle and further onto the outside of the

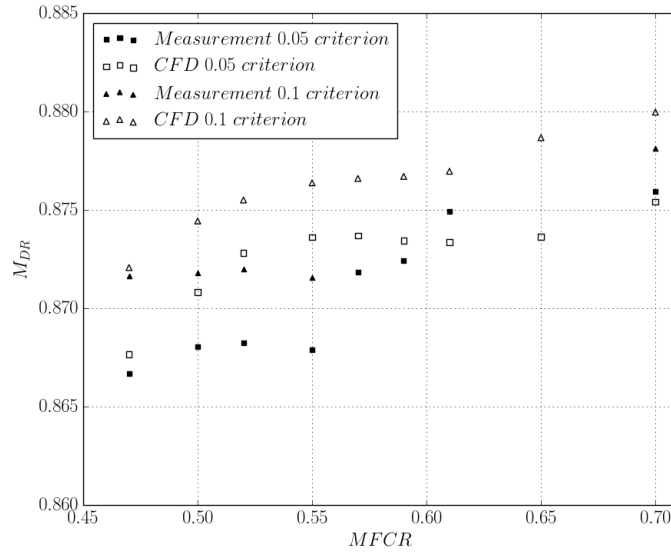


FIGURE 4.5: M_{DR} as a function of M_{FCR} for the V2 nacelle from both CFD and experimental data evaluated by two criteria for drag rise

nacelle lower surface. The influence of nacelle drag is then more pronounced on the topline, partly due to the increased length and therefore skin friction associated with the drag domain. The topline streamline must also curve more due to the relocation of the stagnation point on the nacelle which causes greater accelerations and may result in stronger shocks in supercritical designs. This then causes a change in the drag rise Mach number.

The V2 Nacelle was simulated at incidence between 0 and 4 degrees. Since the discretisation and solver differed slightly between 2D and 3D the same cases were computed in both to assess the compatibility between the two for the V2 nacelle. At a freestream Mach number of $M_\infty = 0.8$ all three lines of the 3D CFD solution are in good agreement with each other and predict the same peak Mach number as the 2D CFD calculation (Figure 4.8a). This agreement worsens due to an increase in the freestream Mach number and by $M_\infty = 0.88$ there is a notable offset between the 2D and 3D computations (Figure 4.8b). In terms of the integral metrics, the absolute levels of drag are similar between the 2D and 3D calculations (Figure 4.7) with offset from 2D to 3D varying from $C_{Dc} = -0.0003$ at $M_\infty = 0.8$ to $C_{Dc} = 0.0008$ at $M_\infty = 0.89$. This means that the 3D predicted drag rise is $M_{DR} = 0.878$ whilst the 2D drag rise was $M_{DR} = 0.880$ and the test measured $M_{DR} = 0.878$. Whilst this suggests that the 3D simulations were more accurate, the pre-drag-rise gradient of drag in the test data was between those predicted in 2D and 3D CFD. Both CFD strategies were considered validated, but due to the differences between 2D and 3D direct comparison of metrics between the two was not deemed reliable.

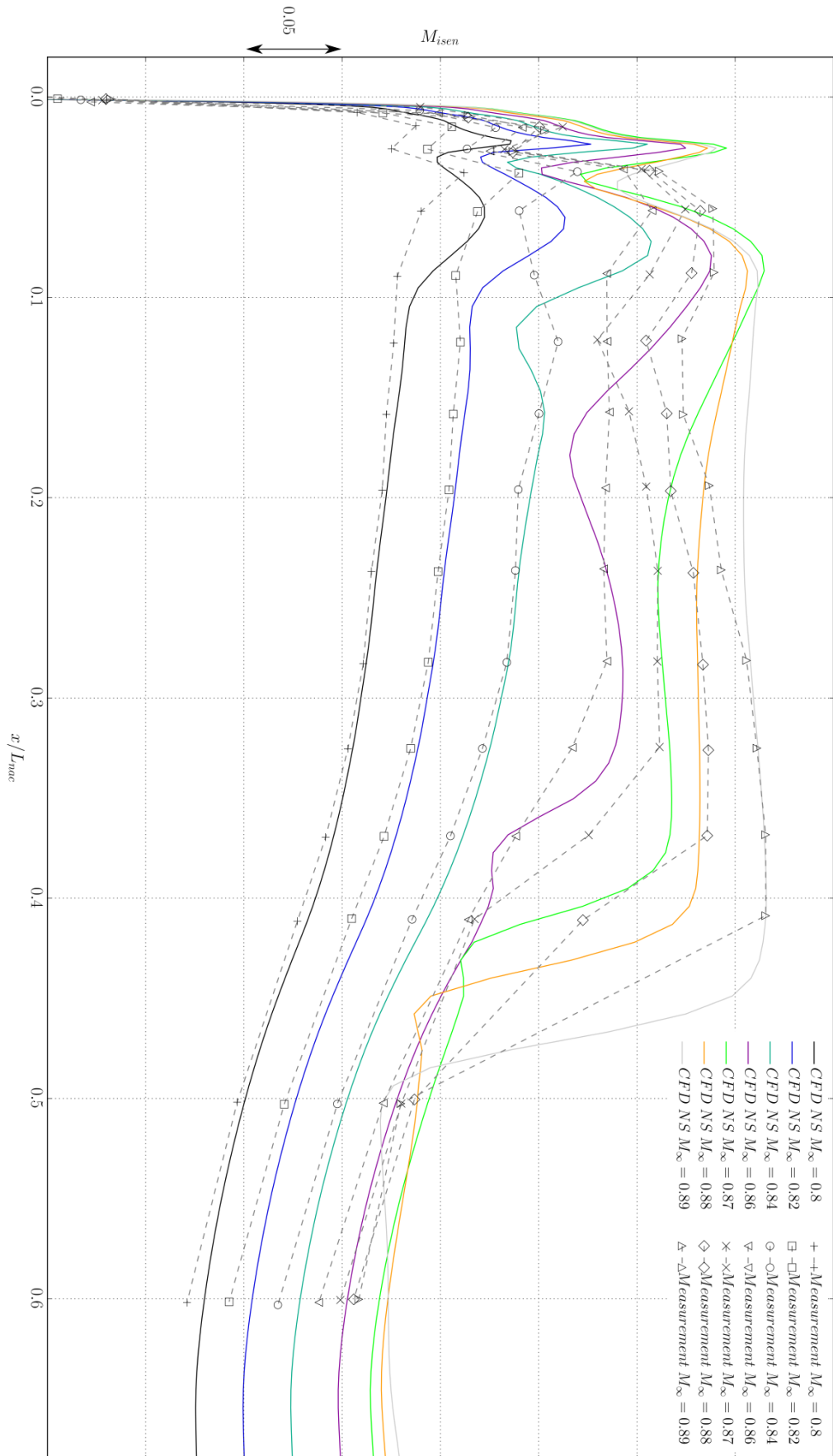


FIGURE 4.6: M_{isen} as a function of x/L_{nac} for the V2 nacelle from both CFD and experimental data across a range of freestream Mach numbers (M_∞)

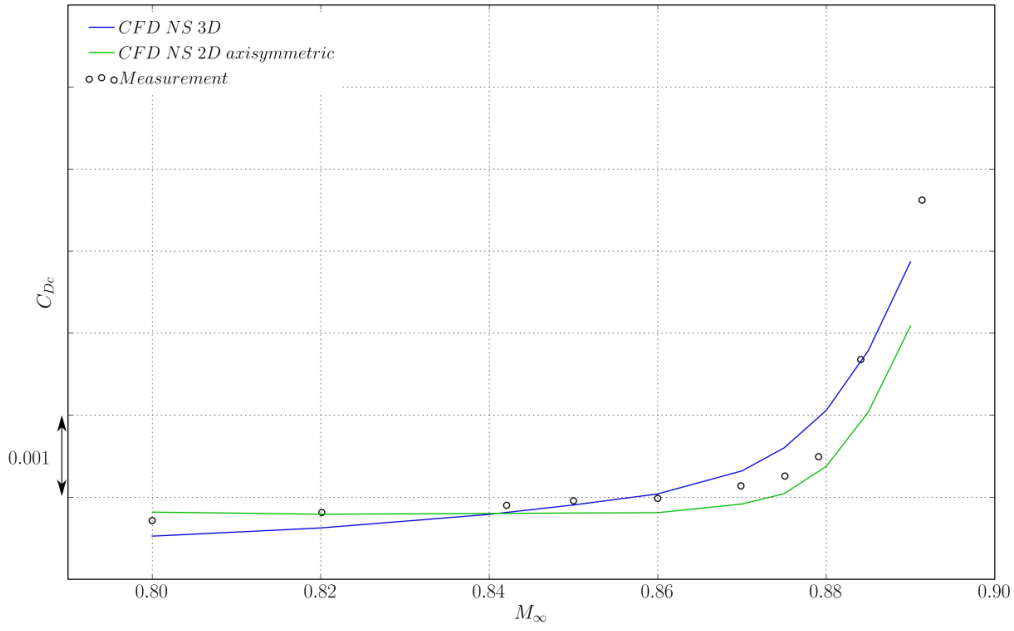


FIGURE 4.7: Drag coefficient (C_{Dc}) as a function of freestream Mach number V2 cowl in the nacelle/sting configuration (NS) from CFD analyses, in 2D and 3D, and experimental data

α_{CL}	V2			V3	
	2D CFD	CFD	Measurement	CFD	Measurement
0	0.880	0.878	0.878	-	-
2	-	0.877	0.876	-	-
4	-	0.874	0.873	0.866	0.859
6	-	-	0.863	0.860	

TABLE 4.2: Measured and CFD predicted drag rise Mach numbers for V2 and V3 cowls at incidences between $\alpha_{CL} = 0^\circ$ and $\alpha_{CL} = 6^\circ$

In comparison with the measured data (Table 4.2), the predicted drag rise Mach number from the CFD was in good agreement with differences in M_{DR} of $\Delta M = 0.002$ at 0° to $\Delta M = 0.001$ at 4° for the V2 nacelle. The local aerodynamics demonstrated the same flow features for the incidence cases between the CFD and the experiment (Figure 4.10). At $\alpha_{CL} = 2^\circ$ the CFD slightly overpredicted the peak Mach numbers on the sideline, with a value of $M_{peak} = 1.03$ relative to $M_{peak} = 1.02$ in the experiment. The topline at the same incidence was underpredicted by the CFD however: $M_{peak} = 1.17$ relative to $M_{peak} = 1.19$ in the experiment. The prediction of the topline peak Mach number from the CFD improved at $\alpha_{CL} = 4^\circ$ with both CFD and experiment predicting a value of $M_{peak} = 1.31$. The sideline was again overpredicted by CFD having a peak of $M_{peak} = 1.03$ and the experiment $M_{peak} = 1.00$. However, these offsets are relatively small and since the general flow

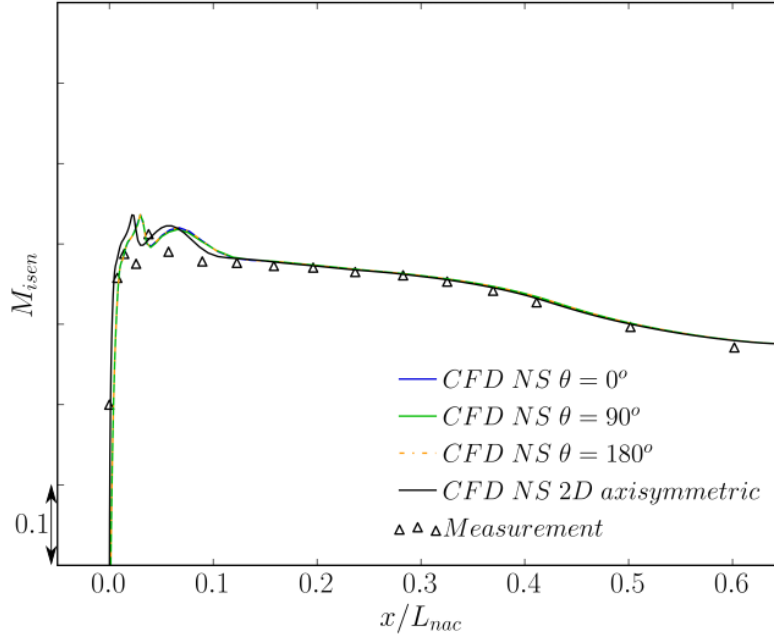
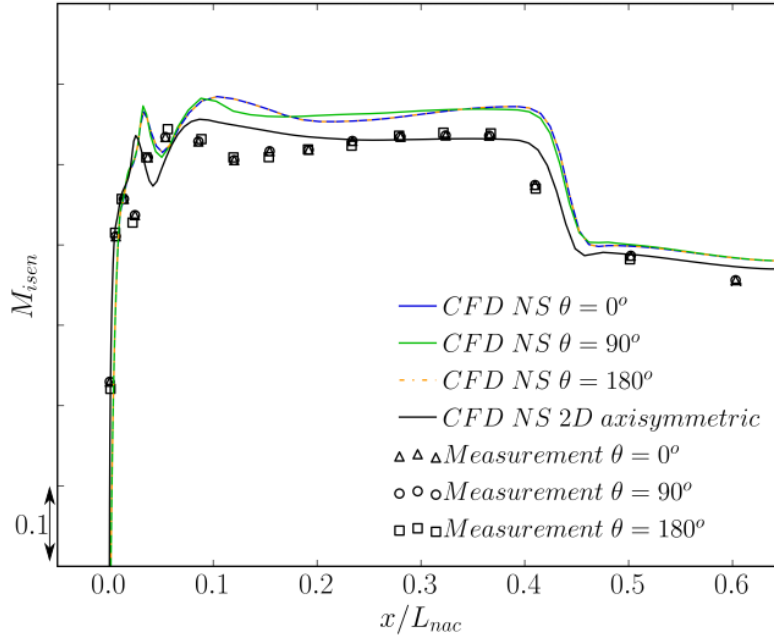
(A) $M_\infty = 0.8$ (B) $M_\infty = 0.88$

FIGURE 4.8: Isentropic Mach number distributions as a function of axial length for the V2 nacelle in the nacelle/sting configuration (NS) from CFD analysis, computed in 2D CFD and 3D CFD, and experimental data

topology is followed by the CFD, it was considered adequate for predicting trends from incidence.

The measurement data for the V2 nacelle predicts very little change in the drag gradients (Figure 4.9a) between 0 and 2 degrees; for an axisymmetric nacelle a more pronounced change would be expected and is observed in the CFD data (Figure 4.9a). However, at 4 degrees, the measured impact of incidence is more prominent in the un-drooped configuration, the V2 nacelle (Figure 4.9a) with the measured M_{DR} decreased by 0.005 to $M_{DR} = 0.874$ due to a 2° increase in incidence compared with 0.001 for V3 nacelle (Table 4.2). There is less impact on the drag rise from incidence for the drooped nacelle (V3), since the drag can be reduced with correct application of droop, scarf and detailed nacelle design. The use of droop and scarf is to allow a nacelle to operate in a range of low incidence common in aircraft due to cruise incidence, engine installation and wing upwash. For the V3 nacelle, the calculated drag rise Mach number from the CFD was greater by between $\Delta M_{DR} = 0.007$ at 4° to $\Delta M_{DR} = 0.003$ at 6° relative to the measured values (Table 4.2). This modest difference is because the CFD tends to slightly under-predict the severity of the post drag rise drag values with a consistent offset after the drag rise Mach number (Figure 4.9b).

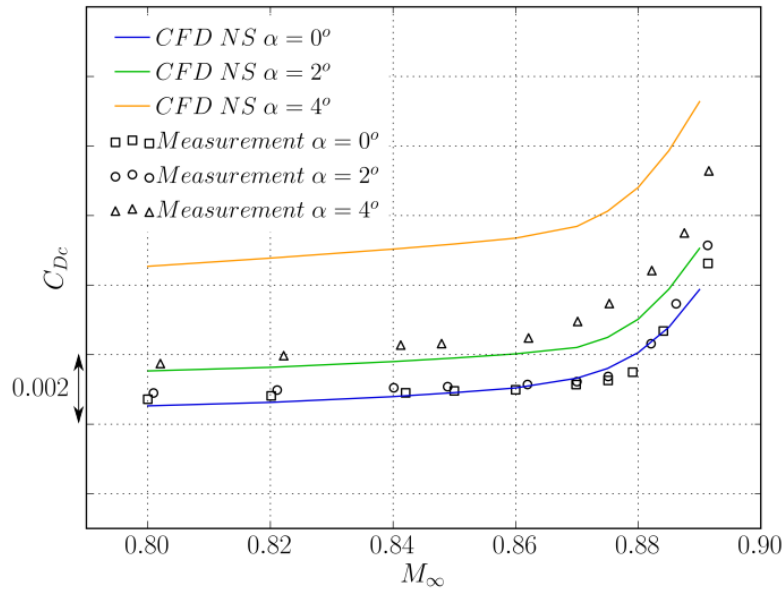
4.1.3 Intake aerodynamic validation

4.1.3.1 Geometry

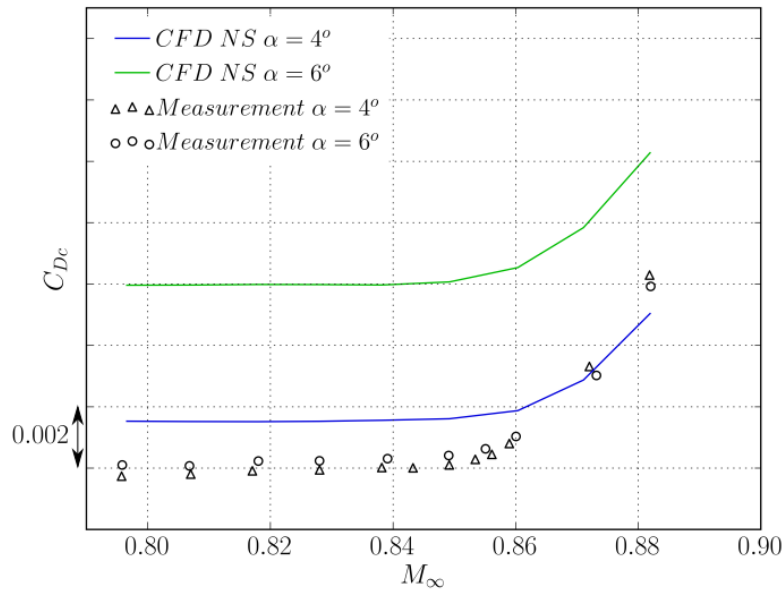
The geometry used for the analysis (I1) was provided by the industrial sponsor¹⁰⁹ and consisted of a half model of a drooped intake and a cylindrical centrebody. The geometry included an extended spinner model and had an exit plane downstream of the fan face.

4.1.3.2 Validation

The initial study for the intake validation was a high incidence, high mass flow, low freestream Mach number case, i.e. climb out under engine failure. This case, in which $M_\infty = 0.25$, WRR=2142 lbs^{-1} and $\alpha = 33^\circ$ will from here on be referred to as the datum case as it was the first case to be analysed and used for further studies. The lip profiles for this case are summarised in Figure 4.11 at three azimuthal



(A) V2 nacelle



(B) V3 nacelle

FIGURE 4.9: Drag coefficient (C_{Dc}) as a function of freestream Mach number in the nacelle/sting configuration (NS) from CFD analyses and experimental data.

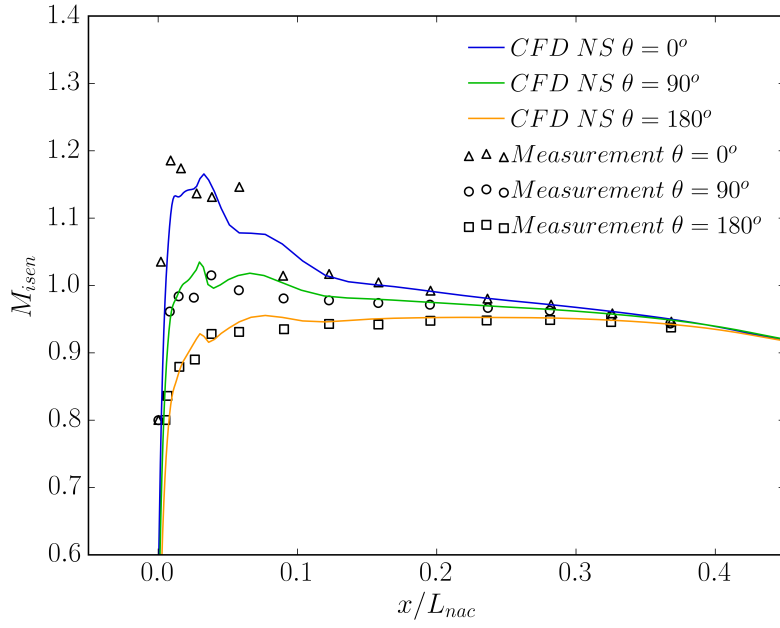
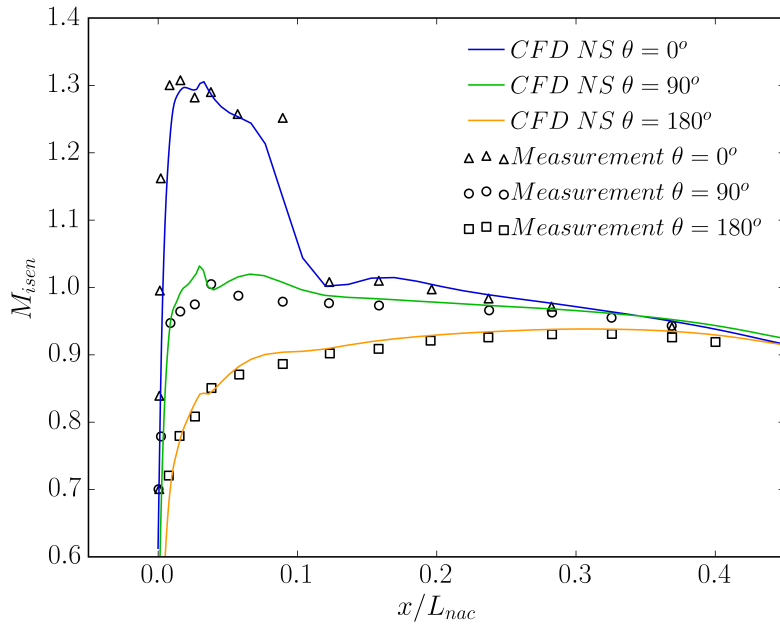
(A) $\alpha_{CL} = 2^\circ$ (B) $\alpha_{CL} = 4^\circ$

FIGURE 4.10: Isentropic Mach number as a function of axial length for the V2 cowl in the nacelle/sting configuration (NS) from CFD analysis and experimental data.

positions. The experimental data showed that the flow was symmetrical across the bottom 60 degree sector. Since the CFD had a symmetry plane enforced in the XY plane, all CFD data is symmetrical. The peak Mach numbers predicted by the CFD for the datum case were lower by a maximum of $\Delta M = 0.05$ compared to the measured values. The location of the shock was predicted within 0.1 X/L though the CFD fails to predict the small recirculation bubble at the bottom dead centre (Figure 4.11a) after the shock.

The agreement with the experimental data is largely dependent on the extent to which the assumption of symmetry holds. As the mass flow is increased to 2142 lbs^{-1} (Figure 4.12) the flow topology observed in the experiment became less symmetrical and the supersonic region became more prominent on one particular side of the bottom line (Figure 4.12b, 4.12c). Since the simulations were run using an enforced symmetry plane and with RANS calculations they become less accurate as the flow becomes more unsteady. In spite of this, the peak Mach numbers from the CFD are in good agreement with the measurements on the side where the suction peak has not collapsed, i.e. $\theta = 150^\circ$ and $\theta = 165^\circ$ in the experimental data. For the $\theta = 165^\circ$ position the peak Mach number is within agreement of $\Delta M = 0.03$ of the $\theta = 165^\circ$ position from the wind tunnel. Similarly for the higher incidence case of $\alpha = 33.25^\circ$ (Figure 4.12), once the separation of the flow in the intake becomes more unsteady the location of the shock in the CFD is less representative with the experimental data showing a shock which spans a greater axial distance. The peak Mach numbers for this case have a maximum deviation of $M=0.01$ from the experimental data.

4.1.3.3 Influence of turbulence model and discretisation order

Further studies were performed to ascertain the influence of the turbulence model used for closure of the RANS equations. For this purpose, the datum case was re-evaluated using the two-equation Spalart-Allmaras model and the Fluent transition $k - \omega$ SST model based on the $\gamma Re - \theta$ model from Menter and Langtry.¹¹⁰ In addition, the datum case was evaluated with a third order discretisation scheme to determine its influence. Full details of these studies are provided in Appendix B.

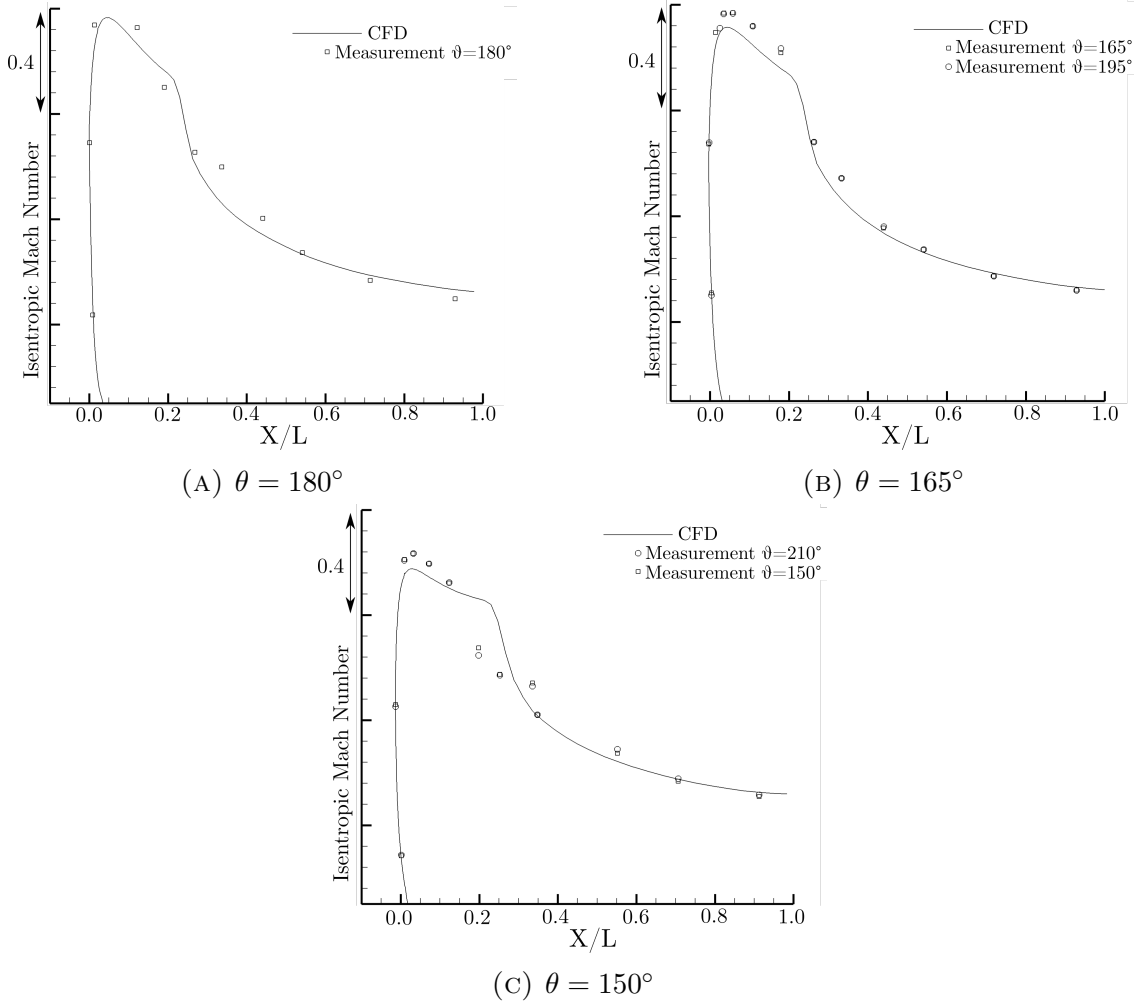


FIGURE 4.11: Isentropic Mach distributions from CFD method and measurement at $M_\infty = 0.25$, $WRR=2133 \text{ lbs}^{-1}$ at $\alpha = 33^\circ$ at three azimuthal positions

4.2 Optimisation Verification

This section contains a summary of the work which was carried out to evaluate the implementation and calibration of the optimisation routine.

4.2.1 Convergence to optima

Three analytical optimisation problems will be discussed in this section each of which tested different aspects of the performance of the NSGA-II implementation. Two of these problems are multi-objective and require simultaneously minimising multiple functions whilst the third is a single-objective problem. A single-objective

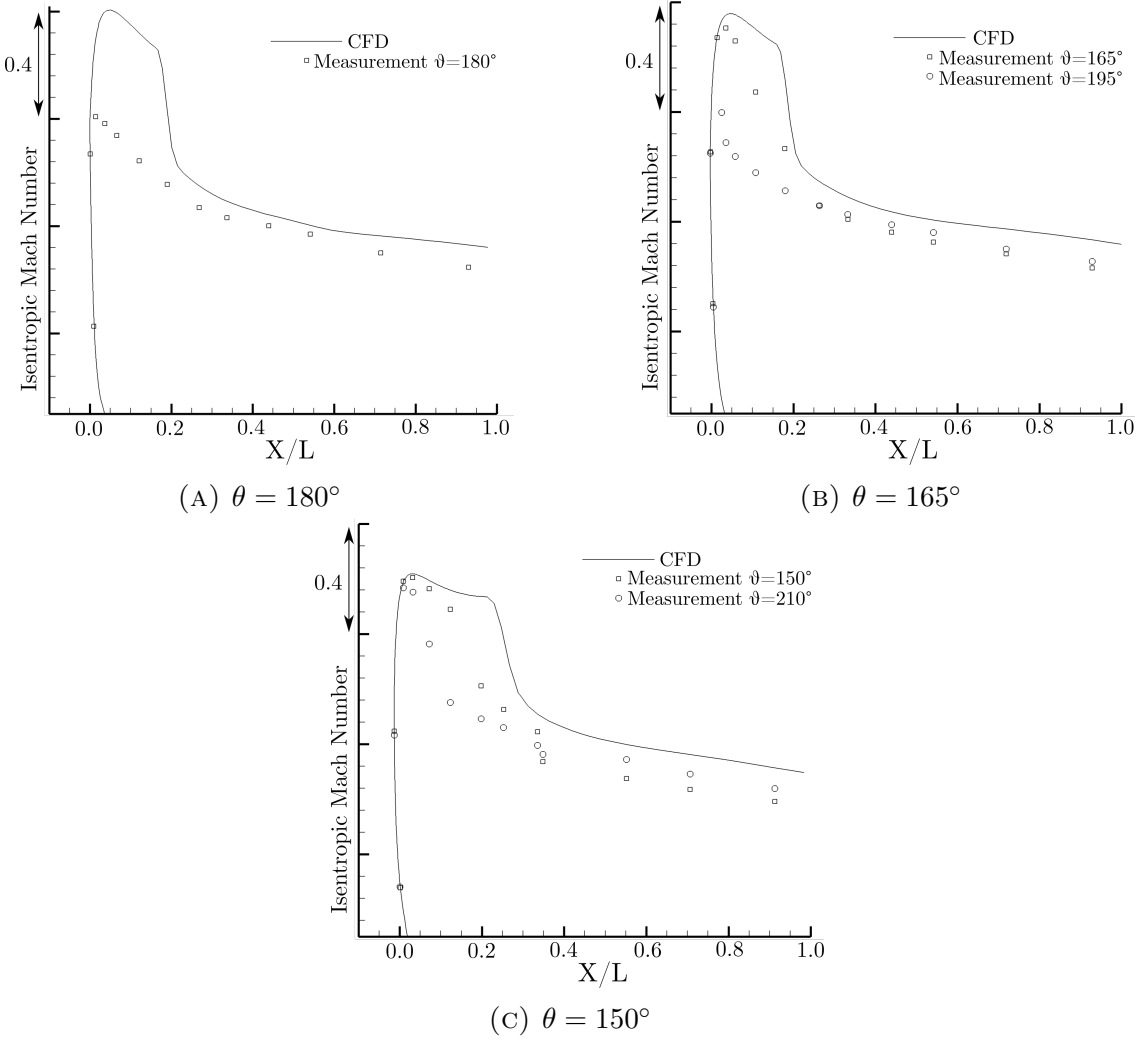


FIGURE 4.12: Isentropic Mach distributions from CFD method and measurement at $M_\infty = 0.25$, $WRR=2142 \text{ lbs}^{-1}$ at $\alpha = 33^\circ$ at three azimuthal positions

optimisation problem is one in which a single mathematical function must be minimised. This typically presents a simpler problem than a multi-objective optimisation since there will only be one optimal objective value. The Booth function¹¹¹ requires minimising equation 4.1 within the bounds 4.2. There is a single global optimum at $\mathbf{x}^* = (1, 3)$ where $f(x) = 0$. The Kursawe function¹¹² is a two-objective problem with a disconnected Pareto front which adds to the complexity in acquiring a distribution along the entire front. It involves minimisation of two functions (equations 4.3 and 4.4) based on the bounds given in equation 4.5. The Kursawe function does not have an analytical solution, however estimations of the optimal set have been extensively published. Finally, the three objective version of the DTLZ2 problem¹¹³ was tested on. This problem required the minimisation of three functions (equations 4.6, 4.7, 4.8) which had a common function $g(\mathbf{X}_M)$ (equation 4.8).

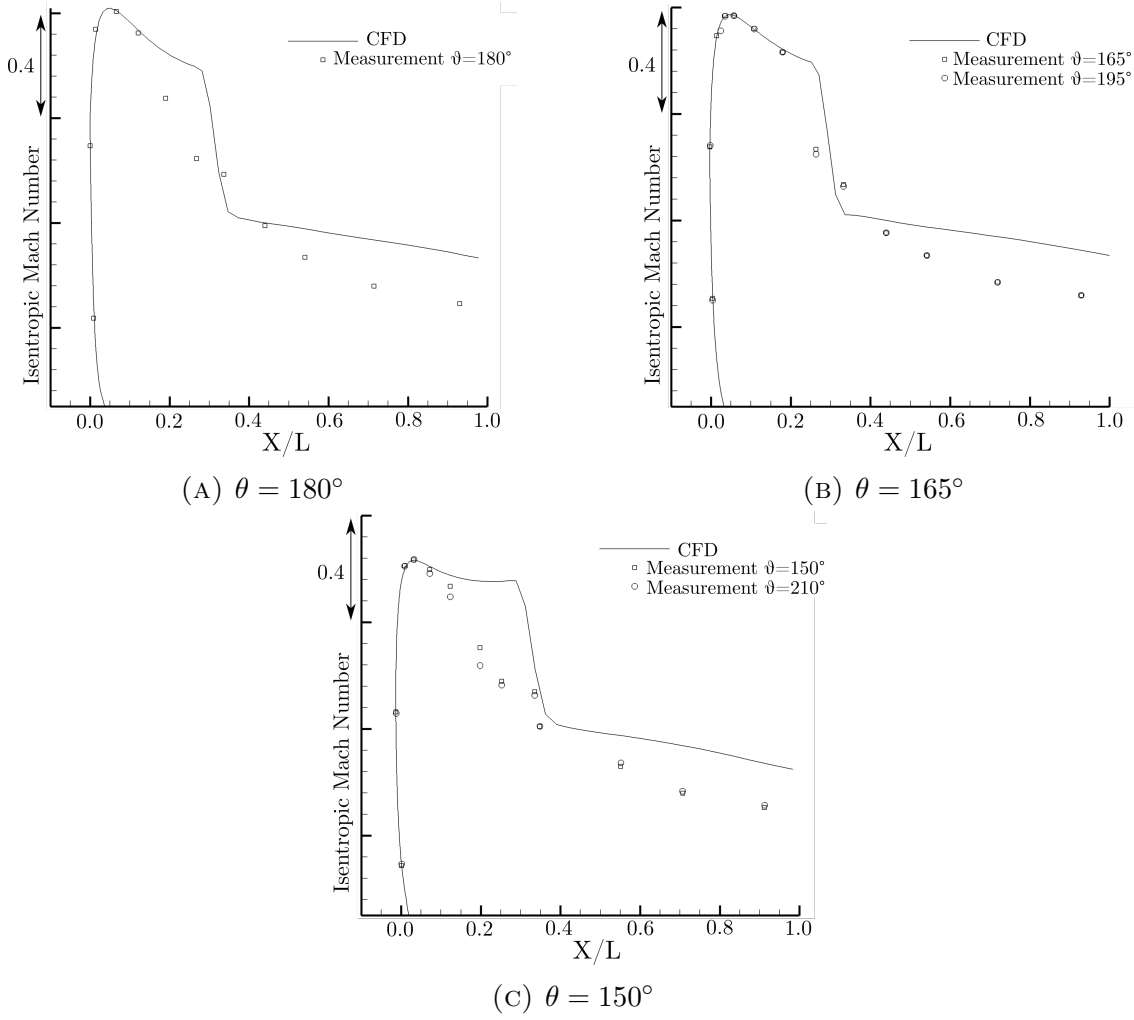


FIGURE 4.13: Isentropic Mach distributions from CFD method and measurement at $M_\infty = 0.25$, $WRR=2129 \text{ lbs}^{-1}$ at $\alpha = 33.25^\circ$ at three azimuthal positions

This problem is evaluated within the bounds given in equation 4.10. The analytical Pareto set to this problem includes any point which lies on the orthant of a sphere of radius 1 with its centre at the origin and satisfies equation 4.10. Alternatively stated the optimal solutions satisfy both equation 4.10 and $\sum_{i=1}^3 (f_i^*)^2 = 1$.

$$f(\mathbf{x}) = (x_1 + 2x_2 - 7)^2 + (2x_1 + x_2 - 5)^2 \quad (4.1)$$

$$x_i \in [-10, 10], i = 1, 2 \quad (4.2)$$

$$f_1(\mathbf{x}) = \sum_{i=1}^{i=2} \left[-10 \exp \left(-0.2 \sqrt{x_i^2 + x_{i+1}^2} \right) \right] \quad (4.3)$$

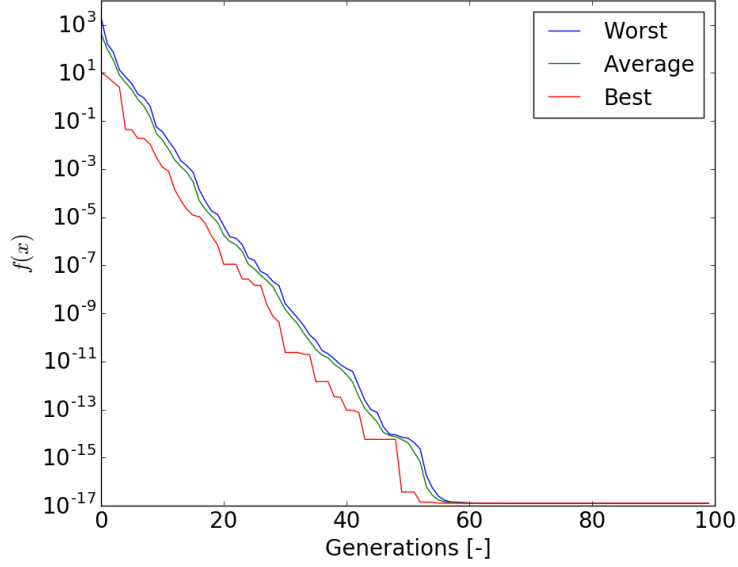


FIGURE 4.14: Highest, mean, and lowest objective values in the mating pool at given generations for the Booth function as calculated by the NSGA-II algorithm

$$f_2(\mathbf{x}) = \sum_{i=1}^{i=3} \left[|x_i|^{0.8} + 5 \sin(x_i^3) \right] \quad (4.4)$$

$$x_i \in [-5, 5], i = 1, 2, 3 \quad (4.5)$$

$$f_1(\mathbf{x}) = (1 + g(\mathbf{X}_M)) \cos\left(\frac{x_1\pi}{2}\right) \cos\left(\frac{x_2\pi}{2}\right) \quad (4.6)$$

$$f_2(\mathbf{x}) = (1 + g(\mathbf{X}_M)) \cos\left(\frac{x_1\pi}{2}\right) \sin\left(\frac{x_2\pi}{2}\right) \quad (4.7)$$

$$f_3(\mathbf{x}) = (1 + g(\mathbf{X}_M)) \sin\left(\frac{x_1\pi}{2}\right) \quad (4.8)$$

$$g(\mathbf{X}_M) = \sum_{x_i \in \mathbf{X}_M} (x_i - 0.5)^2, x_3, x_4, x_5 \in \mathbf{X}_M \quad (4.9)$$

$$x_i \in [0, 1], i = 1, 2, 3, 4, 5 \quad (4.10)$$

The implementation of NSGA-II used in this project converged to a value of $f(\mathbf{x}) = 1.3 \times 10^{-17}$ for equation 4.1 (Figure 4.14). Since the true optimal value is zero and the maximum value of $f(\mathbf{x})$ within the bounds is $f(\mathbf{x}) = 2.6 \times 10^3$, this suggests that the optimisation has successfully approximated the optimal value within excellent accuracy.

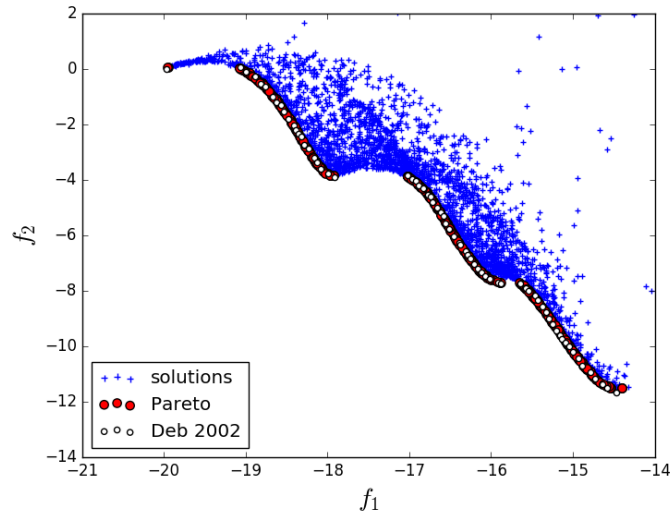
The optimisation tool was applied on the Kursawe problem using a generation size of 40 for 100 generations. The convergence of the optimisation process was assessed based on the evolution of the hypervolume throughout the generations. This analysis demonstrated a monotonic progression and that beyond generation 61 there is less than 1% increase in the hypervolume relative to the initial hypervolume indicating that the solution is converged. The optimisation tool successfully converged to a Pareto set which covered all disconnected sections of the Kursawe function (Figure 4.15a) as have been reported in previous literature.

In addition, the optimisation of the DTLZ2 problem resulted in a Pareto set which closely approximated the true spherical orthant Pareto front (Figure 4.15b). The convergence of this problem can be measured as a volume relative to the true volume of the Pareto set. In this case, a volume was measured from the coordinate (2,2,2) to the set determined by the optimisation and divided by the analytical volume of the spherical orthant from (2,2,2). The final Pareto set which was converged upon for the DTLZ2 problem was 99.6% of the true volume.

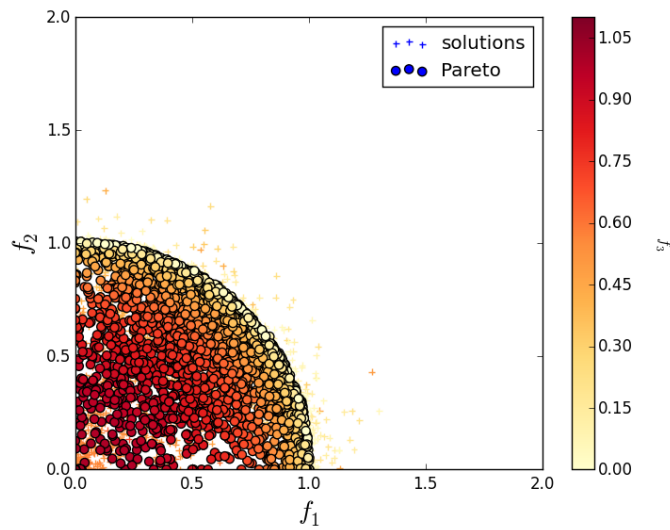
4.2.2 Mutation operator

The Booth function was used to assess the correct implementation and also the use of two different mutation operators: Gaussian and non-uniform mutation, see section 3.5. Gaussian mutation consists of a mutation rate, a mean value and a standard deviation which together determine the frequency and size of mutations applied to the mating pool of the optimisation. The frequency is given as a fraction of 1 and occurs at that rate for all individual values within the input vector. The mean and standard deviation prescribe a normalised Gaussian distribution which is used to determine a mutation value. This mutation value is then added to the normalised input variable.

Since the optimisation method incorporates stochastic elements in the initial population creation, mutation operator and crossover operator, the optimisation was performed 50 times to give generic results. Through the completion of 50 independent optimisations it was possible to extract average values for the performance of the routine. Since the Booth function is a single objective function the optimal individual in any given generation is simply the one with the lowest value of $f(\mathbf{x})$ (equation 4.1). This minimum value was recorded as a function of the generation for the 50 optimisations and the minimum, average and maximum taken across those



(A) Final Pareto set as determined by the NSGA-II implementation with all previous solutions calculated and example Pareto from literature for the Kursawe functions



(B) Final Pareto set as determined by the NSGA-II implementation with all previous solutions calculated for the DTLZ2 functions

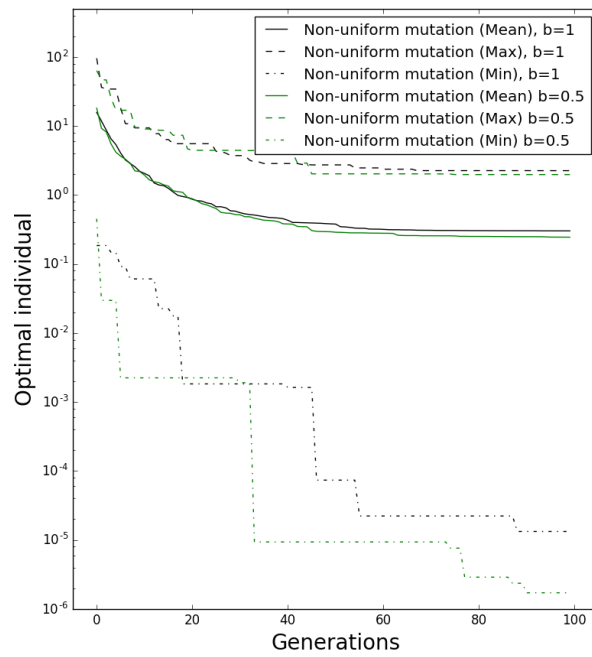
FIGURE 4.15: Optimal Pareto sets for analytical optimisation problems

50 optimisations. This analysis was performed for optimisation which incorporated Gaussian mutation and optimisation which incorporated non-uniform mutation.

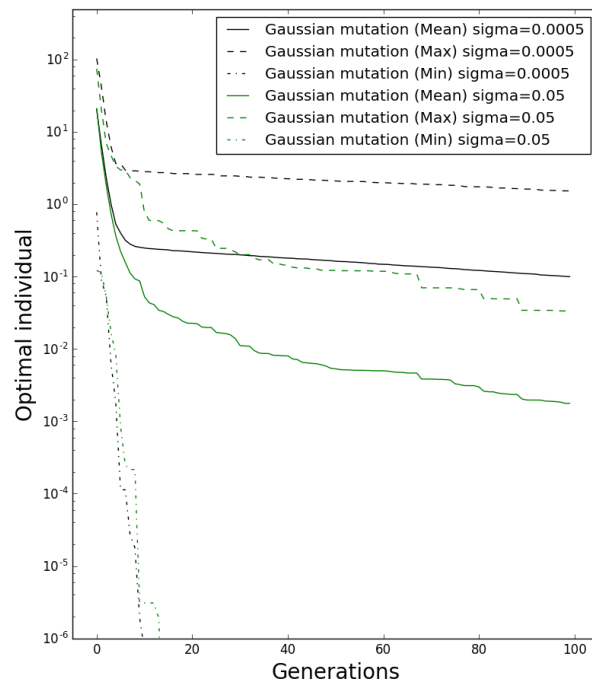
The outcome of this analysis (Figure 4.16) demonstrated that the two different mutation operators could give different optima. In addition, the calibration of these two mutation operators had an influence on the final optimal result as well. Specifically, it was observed that Gaussian mutation, with $\sigma_{mutation} = 0.05$ gave a mean (across the 50 optimisations) minimum $f(\mathbf{x})$ which was two orders of magnitude lower than the mean optimal value from the non-uniform mutation. Since the Booth function has a monotonic shape it is not possible for a genetic algorithm to converge to a false, local minimum. The tendency to not decrease below a particular value of convergence must therefore reflect the mutation and/or crossover operators from producing small enough variance to approximate $f(\mathbf{x}) = 0$. Too small a variance however would greatly increase the number of generations required to converge the optimisation. The use of the Gaussian mutation operator gave on average the better performance with a lower value after 100 generations for both values of $\sigma_{mutation}$. This suggests that the Gaussian distribution used to centralise the mutation is more effective at producing appropriately sized mutations and can give greater control through the $\sigma_{mutation}$ value than does non-uniform mutation. Therefore, for further optimisation work the Gaussian mutation operator was used exclusively.

4.2.3 Initial generation size

In addition to a verification of the ability of the routine to fully resolve a discontinuous front, the Kursawe function was also used to perform preliminary tests of the potential benefits of increasing the size of the initial generation. Two configurations were tested which each consisted of 4000 evaluations of the Kursawe functions. For configuration A, the optimisation was performed with a constant generation size of 40 individuals over 100 generations. Configuration B had an enlarged initial generation size of $i_{gen0} = 400$ and the total number of generations reduced to 91 to retain the same computational expense of 4000 evaluations. Each configuration was used for 40 separate optimisations to enable averaged values to be considered. The hypervolume was calculated for all 80 optimisations across both configurations and compared at the initial and final generations. When i_{gen0} was increased from 40 to 400, there was an initial set with a higher median hypervolume in the 40 instances of the optimisation from which the optimisation process was initiated (Figure 4.17). It also had less scatter with a range of 53.6 compared to 89.4 from the standard



(A) Non-uniform mutation



(B) Gaussian mutation

FIGURE 4.16: Comparison of non-uniform mutation and Gaussian mutation for the Booth function

$i_{gen0} = 40$. This ultimately gave a similar median and interquartile range (IQR) for the final generation $n_{gen} = N$ for both configurations. However, there were fewer outliers in the configuration with an enlarged zeroth generation suggesting a more robust implementation for the same number of evaluations. This therefore indicates that an enlarged random sample may be of use to improve the likelihood of convergence to the Pareto front without any increase in computational cost.

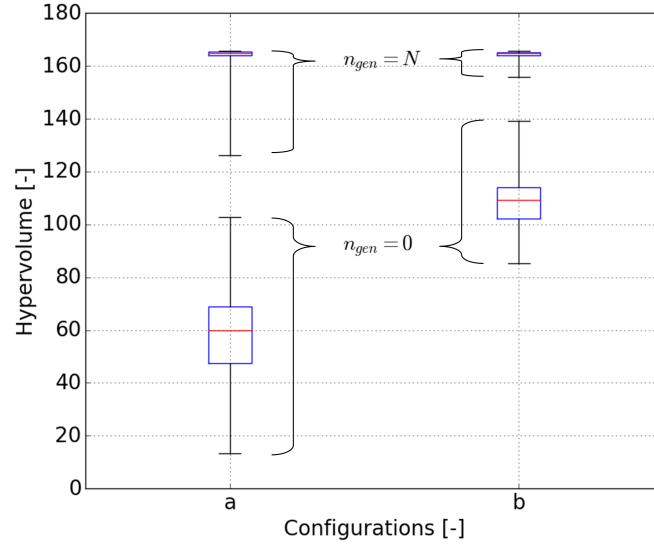


FIGURE 4.17: Median hypervolume of zeroth and final generations with IQR and range for 40 optimisation analyses of the Kursawe function with (a) $i_{gen0} = 40$, $N=100$ and (b) $i_{gen0} = 400$, $N=91$

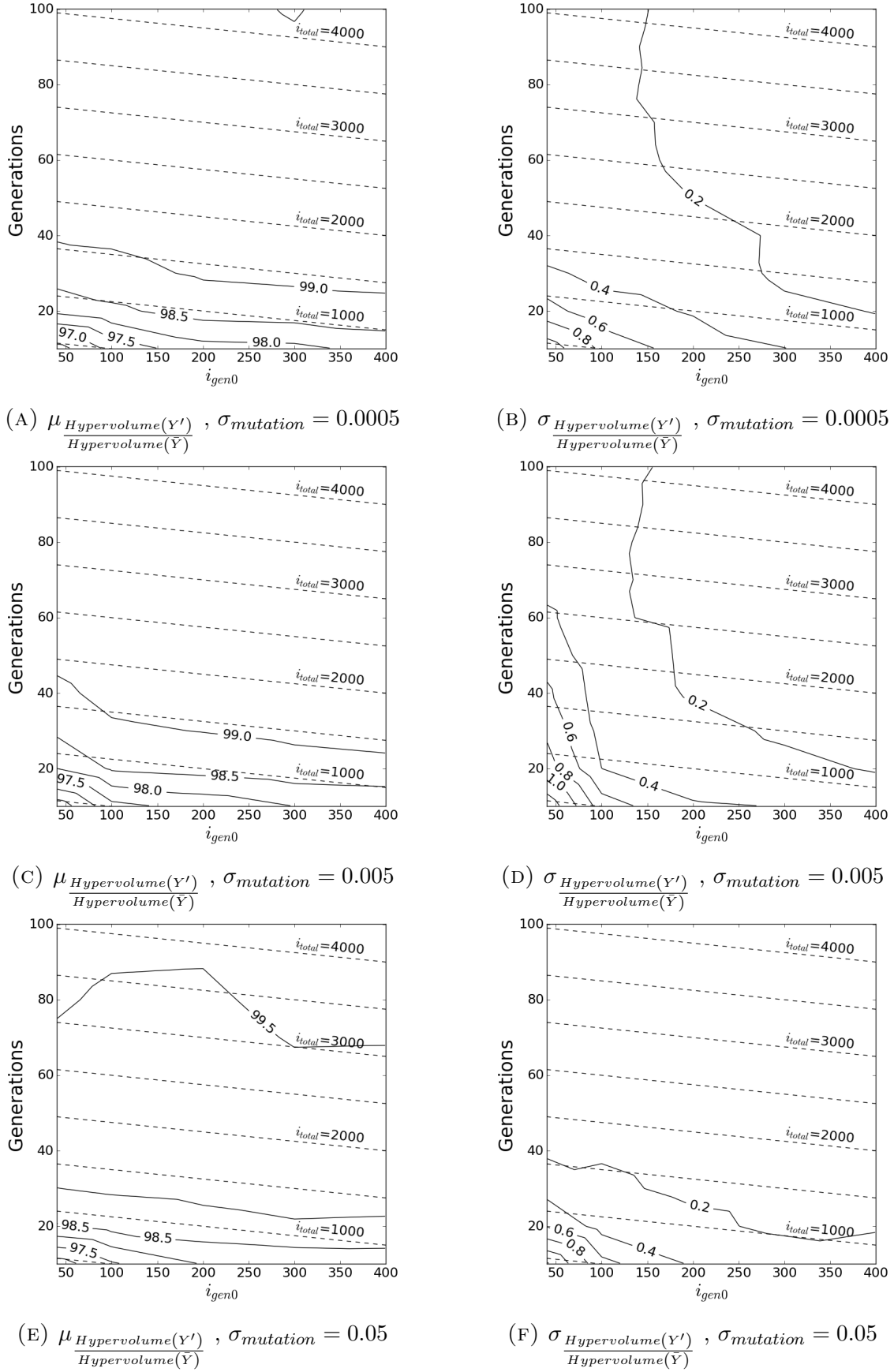


FIGURE 4.18: Influence of initial generation size, generation number and Gaussian mutation size on the ratio of the true hypervolume, relative to (2,2,2), determined across 50 independent optimisations of the DTLZ2 problem

Further to the test on the Kursawe problem, the influence of an enlarged initial generation was tested on the DTLZ2 problem. A series of optimisations were carried out for a range of initial generation sizes, number of generations and mutation standard deviations. In order to make an assessment of the performance without the stochastic element of the algorithm skewing the results, each analysis was repeated 50 times. The mean hypervolume and standard deviation of the hypervolume was then calculated from all 50 of these optimisations (Figure 4.18). The lowest value of mean hypervolume for each $\sigma_{mutation}$ was for 10 generations with $i_{gen0} = 40$ which produced a minimum of 96% of the true hypervolume. The highest mean hypervolumes were mostly obtained with the highest value of $\sigma_{mutation}$ of 0.05 which reached over 99.5%. The influence of mutation standard deviation is otherwise limited largely to the cases with a small initial generation size. An increase from $\sigma_{mutation} = 0.0005$ to $\sigma_{mutation} = 0.05$ resulted in the rate of change of hypervolume increasing over the first 40-50 generations. The increase in mutation standard deviation also decreased the standard deviation of the hypervolumes found for the cases with $i_{gen0} = 40$ and $i_{gen0} = 100$ from under 0.4 to under 0.2. Only at the highest mutation standard deviation tested ($\sigma_{mutation} = 0.05$) was the computational efficiency in terms of the hypervolume achieved for a given number of evaluations, relatively unaffected by the size of the initial generation. For lower mutation standard deviations the use of an enlarged generation size results in both a higher average hypervolume and a lower standard deviation of hypervolume. Overall, these results suggest that there is an optimal value of $\sigma_{mutation}$ for a given optimisation problem which will reduce the computational expenditure. However without a priori knowledge of the design space, this value may be difficult to correctly estimate. In such cases the use of an enlarged i_{gen0} can provide an improved hypervolume. In addition, regardless of the value of $\sigma_{mutation}$ the use of an enlarged i_{gen0} reduces the standard deviation in the optimisation results for low generation numbers giving a better probability of any one optimisation acquiring a Pareto set close to the true Pareto front.

4.2.4 Initial generation sampling

A genetic algorithm must have an initial generation of solutions before it can begin the process of evolution through mutation, crossover and tournaments which will ultimately approximate the Pareto front. Typically this is a sample of points randomly placed within the bounds of the degrees of freedom. As was demonstrated in subsection 4.2.3, this random sample, if not suitably large, can result

in different final Pareto sets if algorithm must start from poor solutions and the mutation is not well calibrated. This is particularly a problem if an optimisation can only be run once and a priori knowledge of the solution space is not available. It was therefore proposed to remove some of the randomness from the optimisation through a structured initial generation. Subsequent mutation would still provide the random variation required to find additional minima, but a structured initial generation would ensure that the full design space is relatively evenly sampled. Two approaches to sampling were tested: a full factorial sample and a Latin Hypercube sampling. These changes were again tested on the 3-objective, 5-variable DTLZ2 optimisation problem. A full factorial sample is a regular n -level sample across the 5 variables. For this test, a 2-level, 3-level, and 4-level full factorial were used. This corresponded to initial generation sizes of 32, 243, and 1024, i.e. 2^5 , 3^5 , and 4^5 . The Latin Hypercube initial generation sizes were the same as used for the random sample, i.e. 40, 100, 200, 300, and 400. The Latin Hypercube sampling was completed using the *diversipy* Python module which implements the algorithm proposed by Wessing¹¹⁴. All optimisations were completed with a Gaussian mutation operator and $\sigma_{mutation} = 0.05$.

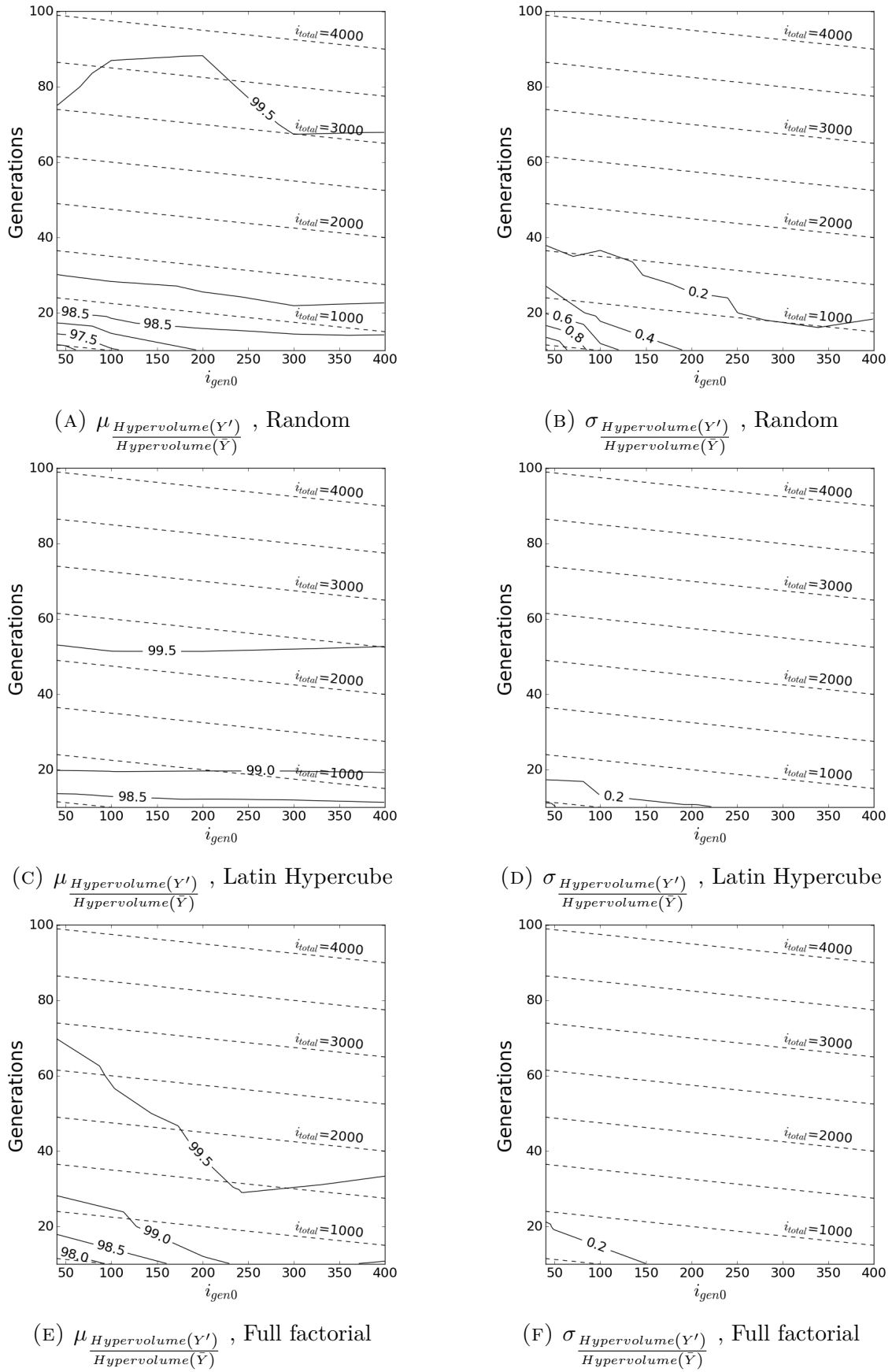


FIGURE 4.19: Influence of initial generation size, generation number and sampling method on the ratio of the true hypervolume, relative to (2,2,2), determined across 50 independent optimisations of the DTLZ2 problem

Both structured initial generation sampling methods predictably decreased the standard deviation of all of the hypervolumes across the 50 optimisations (Figure 4.19) relative to the random case. Both Latin Hypercube sampling and full factorial sampling for the initial generation resulted in less than $\sigma_{\frac{\text{Hypervolume}(Y')}{\text{Hypervolume}(Y)}} = 0.2\%$ for anything over 1000 evaluations (Figures 4.19d, 4.19f). To achieve this level of $\sigma_{\frac{\text{Hypervolume}(Y')}{\text{Hypervolume}(Y)}}$ for the random sampling in some cases took twice as many evaluations as for the Latin Hypercube and full factorial samples. In addition, the value of $\mu_{\frac{\text{Hypervolume}(Y')}{\text{Hypervolume}(Y)}}$ was generally improved for the structured sample methods with over 98% of the true hypervolume found after 10 generations for all initial sample sizes with Latin Hypercube sampling (Figure 4.19c). The highest $\mu_{\frac{\text{Hypervolume}(Y')}{\text{Hypervolume}(Y)}}$ achieved after 1500 evaluations was from the full factorial approach with an initial population size of $i_{gen0} = 243$ which converged to over 99.5% of the true hypervolume (Figure 4.19e). However, this appears to worsen with an increase in the i_{gen0} size. This restriction on this sampling method to $i_{gen0} = n^5$ means that it is more difficult to achieve a detailed contour of the response relative to the Latin Hypercube sampling. In addition this restriction makes it more difficult to implement for an arbitrary number of variables.

4.2.5 Additional tests

Further to the work demonstrated on analytical optimisation problems, the routine was applied to a response surface model (RSM) of nacelle aerodynamic drag. This allowed the quantification of the ability of the algorithm to approximate Pareto fronts of far more representative shapes of the ones considered in this project. For additional details on this, see Appendix D.

4.3 Conclusions

Throughout this chapter validation and verification activities have been detailed. The following conclusions can be made:

- The CFD analysis of an aero-engine nacelle was demonstrated to be within 4% of the nacelle drag at pre-drag rise Mach numbers relative to test data.

- The drag rise Mach number (M_{DR}) was predicted within $\Delta M_{DR} = 0.003$ at a typical mass flow capture ratio of 0.70 for the nacelle most representative of those considered in the project.
- The optimisation method was able to find Pareto sets for 1,2 and 3 objective mathematical problems.
- Without a priori knowledge of a suitable mutation value for the optimisation problem, a large initial population can provide a much improved allocation of evaluations than a fixed population size.
- For a constant number of evaluations, a genetic algorithm should be initialised with a Latin hypercube to improve the final Pareto set which is determined.

CHAPTER 5

Axisymmetric Engine Design Studies

This chapter describes a range of studies carried out on axisymmetric engine nacelles. In this simplified model it is assumed that a single aeroline rotated around the x -axis is a suitable representation of an engine cowl. Throughout this chapter the nacelles which are discussed will only be axisymmetric and as such are modelled as a 2D plane in the CFD model which assumes axis symmetry. The exception to this is when the flowfield is not axisymmetric, i.e. when a nacelle is simulated at incidence to the freestream flow.

5.1 Configuration Testing

The aerodynamic performance of a nacelle is often measured in a wind tunnel with a simplified model of the geometry which excludes the efflux.^{46,115} These simplified models generally assume that there is no significant interaction between the exhaust flow and the nacelle external flow. The two parts of the engine are therefore modelled separately with the nacelle mounted on a sting in a configuration which allows the engine mass flow to be controlled. This removes the upstream influence of the exhaust jet on the fan cowl. To determine the differences in aerodynamics between a sting-mounted wind tunnel sized nacelle model and a full sized nacelle with exhaust jet, several intermediary configurations were considered in this study to determine the level of fidelity required. These configurations were a nacelle mounted on a sting (configuration NS, Figure 5.1a), a nacelle mounted on a cylindrical sting (configuration NCS, Figure 5.1b), a nacelle with a simplified cylindrical exhaust (configuration N, Figure 5.1d), and a nacelle with a representative exhaust geometry (configuration NE, Figure 5.1e).

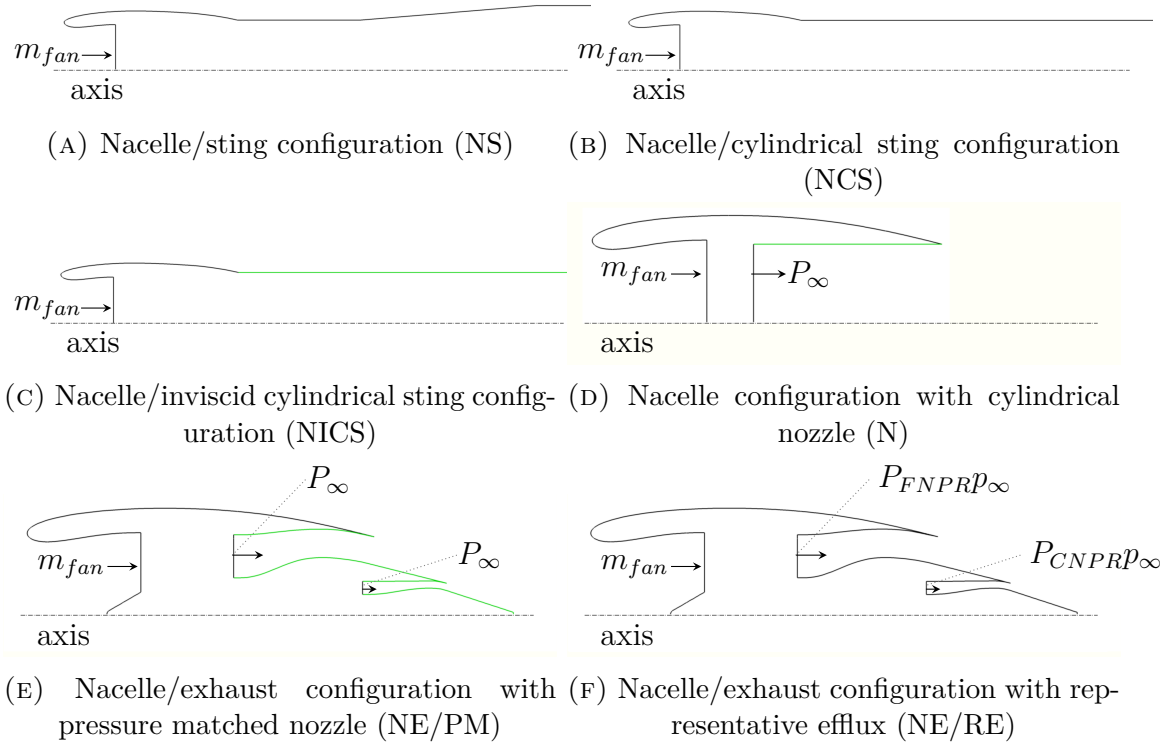


FIGURE 5.1: Configurations tested for nacelle models. Green lines indicate inviscid walls.

As an initial test, the influence of the sting at wind tunnel model scale was considered for the second validation (V2) nacelle (section 4.1). In this study the V2 nacelle was modelled with a sting (configuration NS) and without a sting (configuration N) at model scale. To represent the simplest reference condition of a nacelle without a propulsive jet, and in order to remove any post exit drag terms, the freestream total pressure and temperature were imposed in the cylindrical nozzle duct. A direct comparison can then be made to the V2 nacelle as it was modelled in the validation activity, i.e. sting mounted. As in the validation studies (section 4.1), a series of calculations were performed in which the freestream Mach number was increased, whilst the $MFCR$ was maintained at $MFCR = 0.7$. The results of these test cases indicate that the calculated modified nacelle drag ($c_{D*_{nac}}$, Equation 3.8) for a nacelle without a sting is around 40% higher than a simulation with a sting (Figure 5.4a). In the case of the V2 nacelle the presence of the sting also caused an artificially high M_{DR} value where the nacelle and sting configuration (NS) has $\Delta M_{DR} = 0.004$ higher than the nacelle (N) case at $\alpha_{CL} = 0^\circ$, $MFCR = 0.7$. The CFD calculated $\Delta c_{D*_{nac}}$ between $MFCR = 0.7$ and $MFCR = 0.5$ is $\Delta c_{D*_{nac}} = 0.0037$ for the computations with V2 with a sting (NS) in comparison with a value of $\Delta c_{D*_{nac}} = 0.0031$

for the cases with no sting (N). This suggests that the spillage characteristics are not highly influenced by the presence of the sting (Figure 5.4a).

Whereas the drag (D_{nac} , Equation 3.9) and modified drag (D_{nac}^* , Equation 3.7) for the NS configuration are equivalent since there are no post exit terms (ϕ_{post}) to account for, the same is not true for the N configuration. In the N configuration the post exit terms (ϕ_{post}) should be minimal, because the exhaust conditions are matched to the freestream total conditions, however there is still a notable impact by their inclusion (Figure 5.4a). For the N configuration, the post exit terms are of the order of between 10% and 12% of the modified drag. This is significant since in these simulations the post exit streamtube is close to a cylinder and only varies due to the radial momentum imparted from the nacelle flow. In a case where the jet has a higher pressure ratio and must expand to freestream conditions these terms would be expected to be larger.

The influence of the sting in conjunction with angle of attack was assessed with the V2 nacelle. Simulations were carried out at $\alpha_{CL} = 2^\circ$ and $\alpha_{CL} = 4^\circ$. A similar influence of the sting was observed at incidence as was observed at $\alpha_{CL} = 0^\circ$. The discrepancy between the modified drag (D_{nac}^*) of the configurations with sting (NS) and without (N) remained roughly constant, in the region of $\Delta C_{D^*_{nac}} = 0.0070$ and $\Delta C_{D^*_{nac}} = 0.0090$, as incidence was increased from $\alpha_{CL} = 0^\circ$ to $\alpha_{CL} = 4^\circ$. The influence of the sting had a similar impact on the drag rise Mach number. At $\alpha_{CL} = 0^\circ$ the predicted drag rise Mach number changed from $M_{DR} = 0.874$ to $\Delta M_{DR} = 0.880$ by the inclusion of the sting. For the case at $\alpha_{CL} = 2^\circ$ this change was from $M_{DR} = 0.871$ to $\Delta M_{DR} = 0.877$ and at $\alpha_{CL} = 4^\circ$ from $M_{DR} = 0.868$ to $\Delta M_{DR} = 0.874$.

The presence of the sting had a notable influence on the pressure field around the nacelle (Figure 5.2) which resulted in these different drag estimates. The main impact of the sting is the bulk displacement of the flow downstream of the nacelle trailing edge which creates a forward horizontal buoyancy force on the nacelle through locally increased pressures at the trailing edge position. The turning of the streamlines at the aft of the nacelle comes partially from the presence of the solid body and partially from the additional boundary layer growth on the sting. In addition to this there is a conical section with an increase in diameter on the sting, at which the measurement rake is located, which creates an additional pressure rise (Figure 5.2 position B). These changes in the pressure field caused by having a solid sting in place of a wake propagate forwards and influence the nacelle aerodynamics. At

higher freestream Mach numbers, where a shock wave is present on the nacelle, the pressure increase from the sting also acts to move the shock more upstream in the presence of the sting (Figure 5.2 position A). This will result in a variable impact from the sting for designs with different pressure distributions. The distribution of isentropic Mach numbers also shows that the effect of the sting mounting is to move the shock further upstream and to increase the pressure on the aft 80% of the nacelle (Figure 5.2). In the cases with the nacelle and idealised efflux (N) there is a lower static pressure at the trailing edge of the nacelle as the streamlines curve slightly towards the axis whereas for the sting cases (NS) the boundary layer of the sting pushes them slightly outwards producing a higher pressure (Figure 5.2).

A large static pressure increase was observed at the conical section of the sting (Figure 5.2 position B). To determine if this is the dominant effect of the sting, an alternative geometry was produced which incorporated an entirely cylindrical sting from the nacelle trailing edge (NCS). Finally, an analysis was carried out for a nacelle mounted on an inviscid cylindrical sting (NICS) to demonstrate how much influence the boundary layer growth has on the measured nacelle drag since this also leads to an increased buoyancy force. Relative to the nacelle cases with the wind tunnel sting (NS), with a viscous cylindrical sting (NCS) there is an increase in modified drag (c_{D*}) of between 0.0018 at $M_\infty = 0.8$ and 0.0035 at $M_\infty = 0.89$ (Figure 5.3). The only cause of streamline turning aft of the trailing edge in these cases is from boundary layer growth over the sting. For the inviscid cylindrical sting (NICS) there is an increase in drag (c_{D*}) of between 0.0024 at $M_\infty = 0.8$ and 0.0042 at $M_\infty = 0.89$ relative to the baseline nacelle and sting cases (NS). This increase is due to the streamlines from the trailing edge being parallel to the axis and creating no additional buoyancy force. Therefore the modified nacelle drag (c_{D*}) measured with the inviscid sting (NICS) contains no additional drag terms (F_{interf}). As a result, the additional drag measured from the cases with no sting (N) relative to the inviscid cylindrical sting (NICS) is due to the streamlines which curve towards the axis, and thus cause a lower pressure on the nacelle, due to the shear layer between the idealised efflux and the flow from the nacelle. This demonstrates that the idealised efflux model creates an additional negative interference term and has a non-cylindrical post exit streamtube. The increase in drag rise Mach number observed in the nacelle with sting cases (NS) is not present in either cylindrical sting configurations (NCS/NCIS) suggesting that this is due to the conical section of the sting.

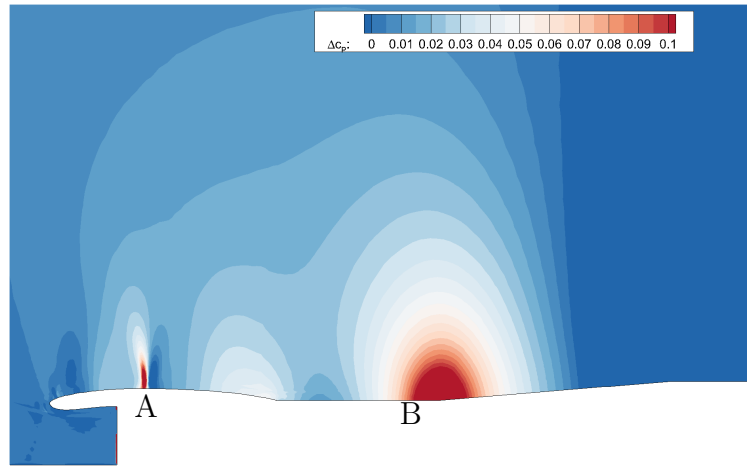


FIGURE 5.2: Change in pressure coefficient caused by the presence of the sting at $M_\infty = 0.87$, $MFCR = 0.7$ relative to a nacelle with no sting (N)

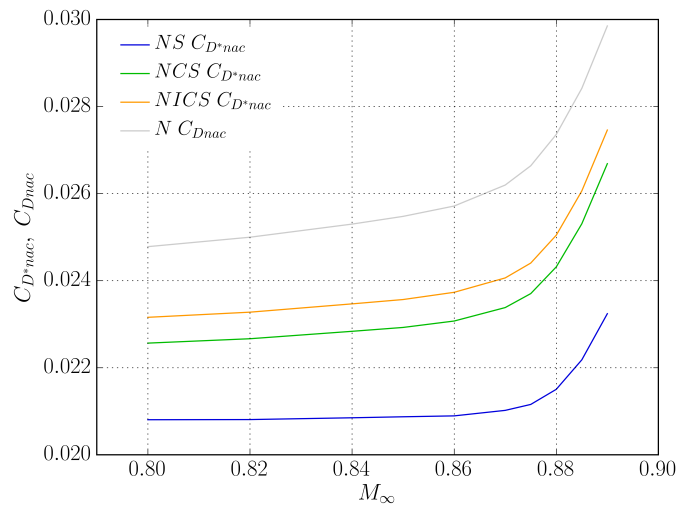


FIGURE 5.3: Drag coefficient as a function of M_∞ for V2 at $MFCR = 0.7$ for CFD analysis with (NS) and without sting (N), with a cylindrical sting (NCS) and with an inviscid cylindrical sting (NICS)

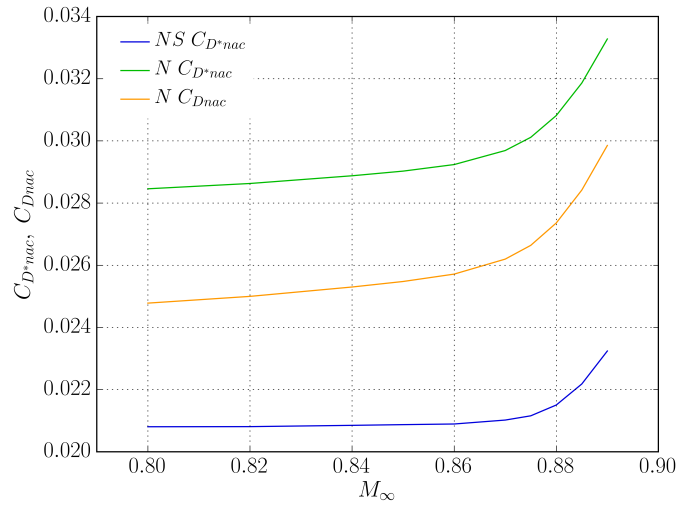
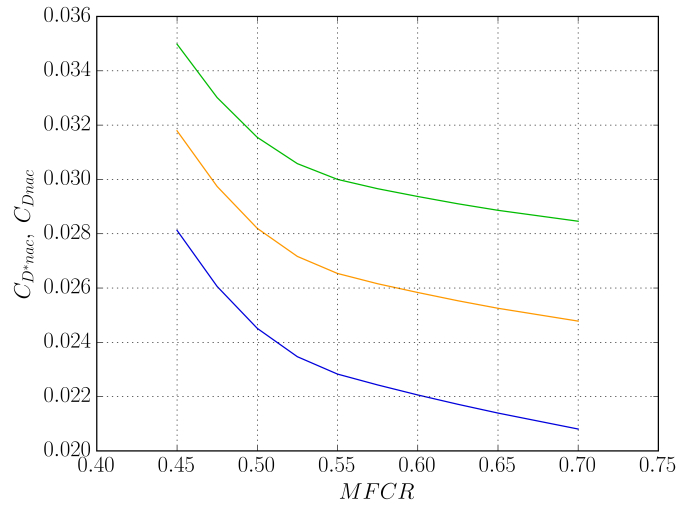
(A) $MFCR = 0.7$ (B) $M_\infty = 0.8$

FIGURE 5.4: The influence of (A) freestream Mach number and (B) MFCR on drag for the V1 nacelle modelled with the sting (NS) and without (N)

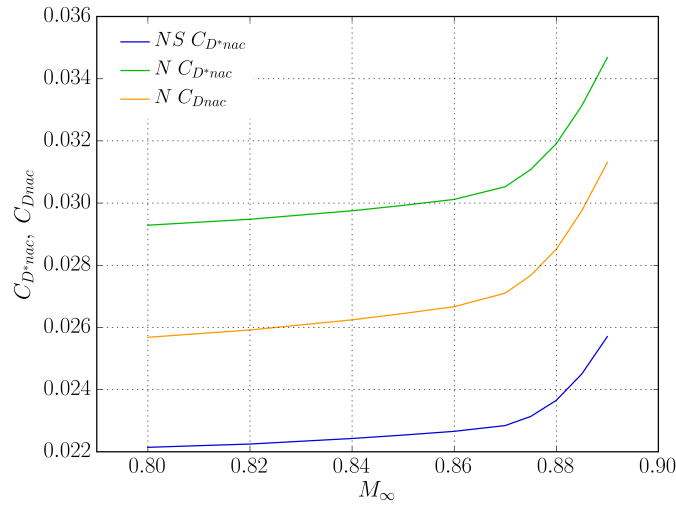
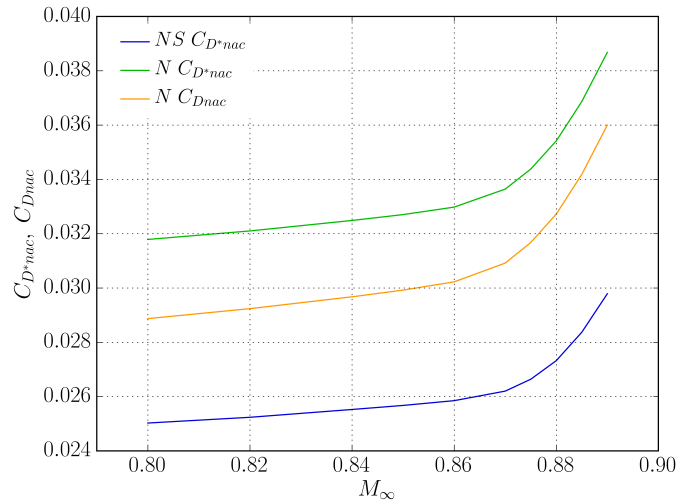
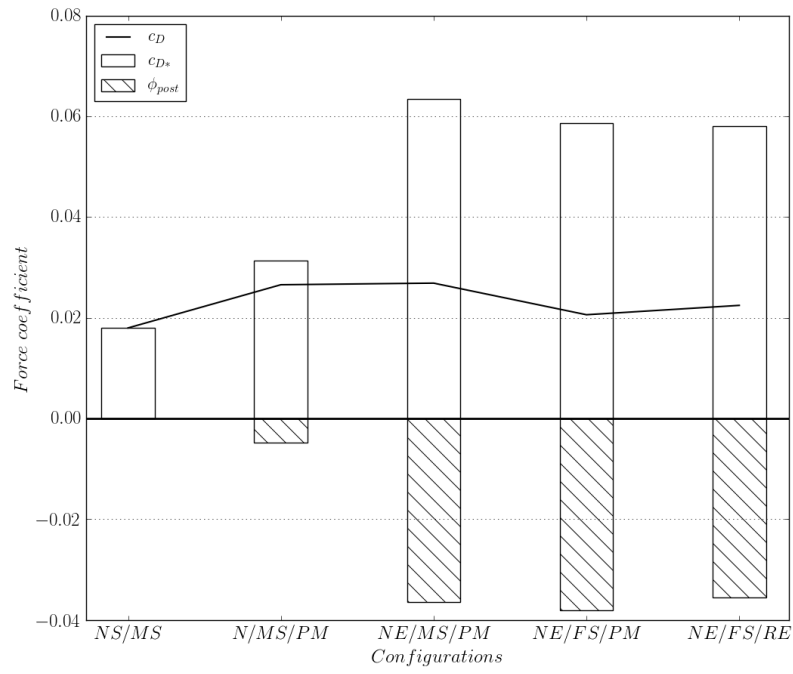
(A) $\alpha_{CL} = 2^\circ$ (B) $\alpha_{CL} = 4^\circ$

FIGURE 5.5: The influence of freestream Mach number on drag for the V1 nacelle modelled with the sting (NS) and without (N)

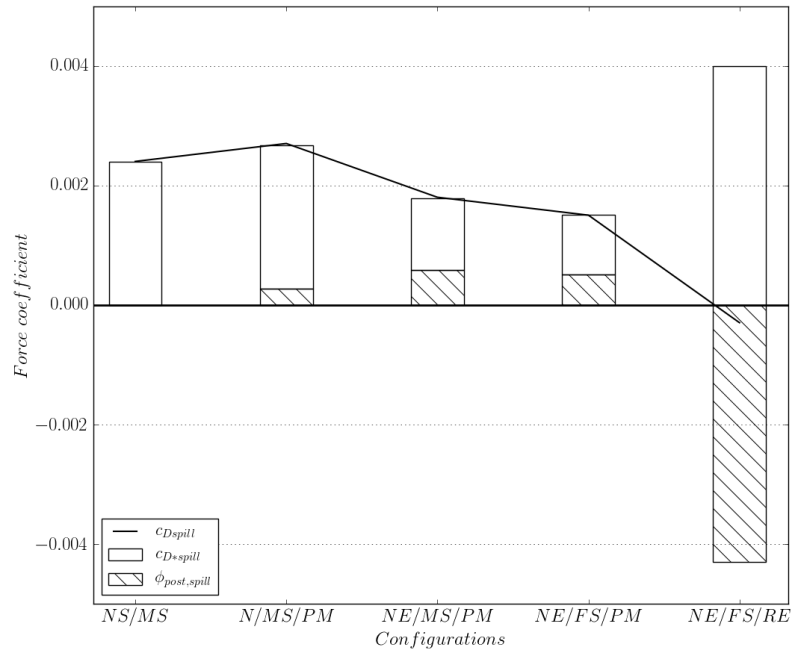
Evidently the presence of a sting as in a wind tunnel test can have a major influence on the integral metrics, however the simplified exhaust geometry and efflux boundary conditions used for the V2 study are not truly representative of an engine in flight either. To assess how this simplification influences the nacelle aerodynamics a subsequent study was carried out with the E2 nacelle. With a CFD model it is relatively simple to add the exhaust flow compared to a wind tunnel configuration and as such several configurations were tested. This was to determine the required complexity of the engine model as well as to quantify the interference of the sting used in a wind tunnel model. The first set of computations were performed at wind tunnel model scale (MS) and included nacelle/sting (NS) (Figure 5.1a), nacelle with

a cylindrical nozzle (N) (Figure 5.1d) and a nacelle with a freestream total pressure matched dual stream exhaust (NE)(Figure 5.1e). The cylindrical nozzle model was used again to represent the simplest reference condition of a nacelle without a jet, and in order to minimise post exit drag terms, the freestream total pressure and temperature were imposed in the cylindrical nozzle duct.

The second set of computations sought to model the engine at altitude and included two different configurations of a nacelle with dual stream exhaust (NE). One configuration had efflux matched to the freestream total conditions (PM) and one had the boundary conditions representative of an UHBPR at altitude (RE). The wind tunnel model scale (MS) data was computed with a constant free stream total pressure and total temperature which gave a Reynolds number range based on nacelle maximum diameter of 4.5×10^6 to 5.0×10^6 . Based on the International Standard Atmosphere conditions for 10,668m altitude the full scale Reynolds number range was 22.8×10^6 to 29.6×10^6 . For the configuration with an exhaust geometry and representative efflux (RE) the nozzle pressure ratios were varied with the engine cycle across the Mach number range to ensure a constant MFCR. The fan nozzle pressure ratio ranged between 1.77 and 2.36 whilst the core nozzle pressure ratio varied between 1.37 and 2.00 between $M_\infty = 0.7$ and $M_\infty = 0.9$. The cruise condition for this engine is at $M_\infty = 0.82$ and $MFCR = 0.75$. The effect of the engine power setting is also of interest therefore the influence of MFCR was also assessed for these configurations.



(A) Cruise drag



(B) Spillage drag

FIGURE 5.6: Drag decomposition for CFD results from five configurations of E2 engine at $M_\infty = 0.82$, $MFCR = 0.75$, $\alpha_{CL} = 0^\circ$

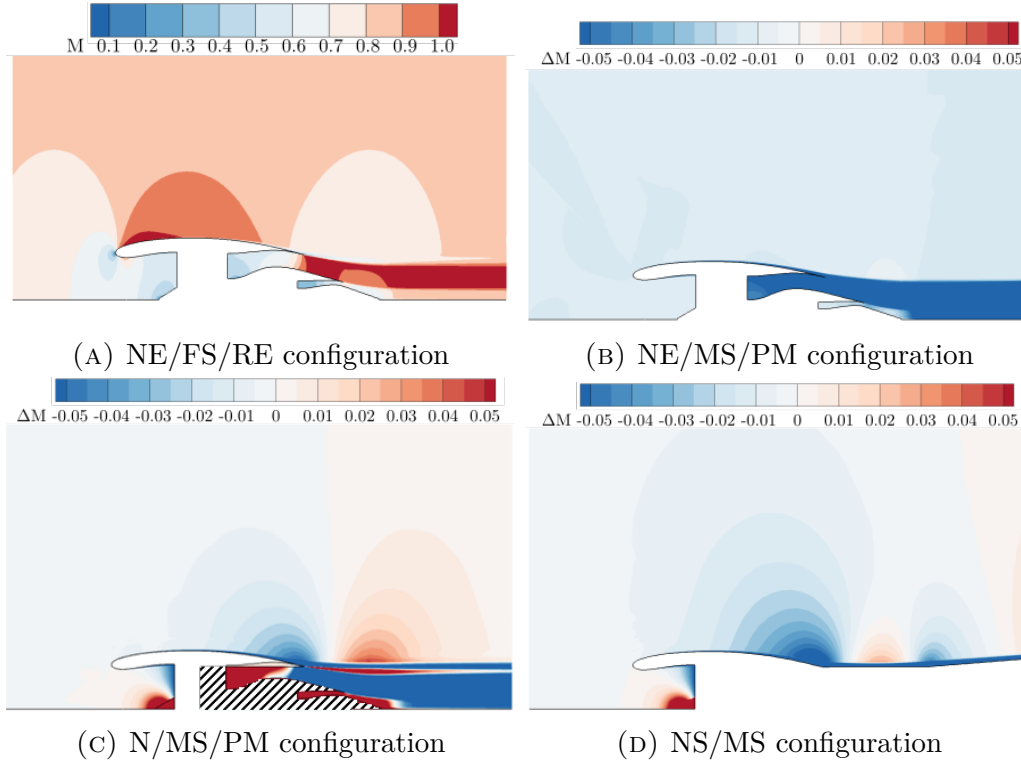


FIGURE 5.7: (A) Mach contours for E2 datum nacelle and (B)-(D) relative Mach number increase from the NE/FS/RE configuration, at $M_\infty = 0.82$, $MFCR = 0.75$, $\alpha_{CL} = 0^\circ$

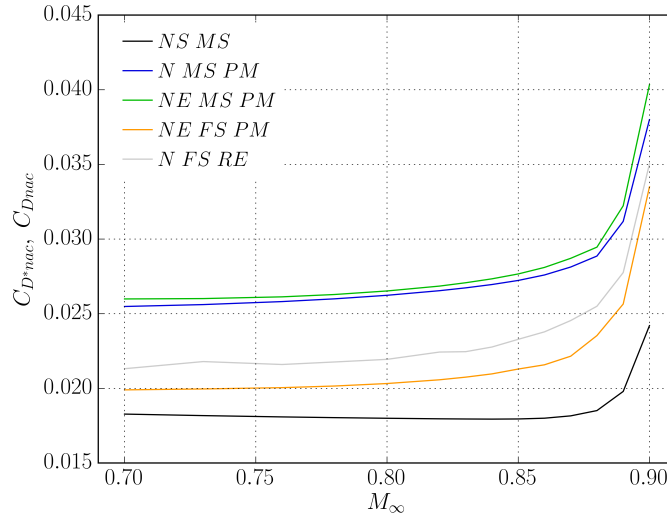


FIGURE 5.8: CFD results for E2 nacelle in different configurations evaluated in terms of nacelle drag coefficient ($c_{D_{nac}}$) as a function of M_∞ at constant $MFCR = 0.75$, $\alpha_{CL} = 0^\circ$

	$M _{\frac{\partial C_D^*}{\partial M_\infty}=0.1}$	$M _{\frac{\partial C_D}{\partial M_\infty}=0.1}$
Nacelle/sting model scale (NS/MS)	0.882	0.882
Nacelle model scale (N/MS/PM)	0.877	0.877
Nacelle/exhaust model scale (NE/MS/PM)	0.850	0.876
Nacelle/exhaust full scale (NE/FS/PM)	0.855	0.870
Nacelle/exhaust full scale with efflux (NE/FS/RE)	0.849	0.875

TABLE 5.1: Drag rise Mach number for different configurations of E2 nacelle

The cruise drag, assessed at $M_\infty = 0.82$, $MFCR = 0.75$, demonstrates the large discrepancies between the different configurations. The post exit force is an important contributor to the overall drag which is around 61% of the magnitude of the modified drag (Figure 5.6a) for the NE/FS/RE case. In the configuration with a cylindrical exhaust (N/MS/PM), the post exit terms were predictably much smaller than for the NE/FS/RE case however still reduced the drag by 15% relative to the modified drag (Figure 5.6a). The sting mounted configuration (NS/MS) gave the lowest overall drag (Figure 5.6a). The predicted value was around 20% lower than the full configuration (NE/FS/RE) (Figure 5.6a). The source of this lower drag can be identified within the flowfield (Figure 5.7) as an increase in pressure caused by the sting blockage. Locally, the streamlines are turned by both the sting and the boundary layer growth over it which results in a higher relative pressure exerted on the afterbody of the nacelle compared to the configurations without the sting. Relative to the case with the sting (NS/MS), the case with a simple exhaust (N/MS/PM) predicts a 48% higher cruise drag in terms of C_{Dnac} (Figure 5.6a). This gives a direct quantification of the additional force generated by this streamline turning. The inclusion of the cycle efflux (NE/FS/RE) relative to a pressure matched equivalent (NE/FS/PM) results in a 9% increase in the measured drag which is partly caused by a change in the post exit force but also in the modified drag as a result of the trailing edge pressure (Figure 5.8). These comparisons demonstrate that the post exit force has a large influence on the predicted drag and must be accounted for even when an idealised jet is simulated. In addition, the sting mounted nacelle should be treated with caution unless the additional pressure force acting on the afterbody is accounted for and the drag increased accordingly.

The different configurations were also modelled across a range of Mach numbers and the results indicated that the configuration influenced the predicted trends in drag relative to the Mach number (Figure 5.8). This resulted in different values of drag rise Mach number for the different configurations (Table 5.1). The model with the sting (NS/MS) once again gave the most optimistic prediction, with a

$\Delta M_{DR} = 0.007$ higher drag rise Mach number than the full (NE/FS/RE) configuration. Drag rise predicted from a model with a sting will again be influenced by the buoyancy provided by the sting presence and should be treated cautiously. Table 5.1 however demonstrates that the inclusion of the post exit force is only influential on the configurations which include a non-cylindrical exit streamtube. For the full configuration (NE/FS/RE) the omission of this term (ϕ_{post}) results in an under-predicted drag rise by $\Delta M_{DR} = 0.026$. Consequently, it is important to include this value as it can have a large influence on the predicted M_{DR} for designs with an exhaust system.

A spillage increment was calculated between $M=0.82$, $MFCR=0.55$ and $M=0.82$, $MFCR=0.75$ which was defined for both the modified nacelle drag ($c_{D*_{nac}}$) (Equation 3.8) and the nacelle drag (c_D) (Equation 3.10). The difference between the two metrics is therefore the change in the force on the post exit streamtube due to the change in the mass flow ($\phi_{post,spill}$). There was a large difference in the spillage drag predicted by the simulation of different configurations (Figure 5.6b). The change in exhaust configuration changed both the size of the post exit force ($\phi_{post,spill}$) and the modified spillage drag ($c_{D*_{spill}}$). The influence of the trailing edge condition therefore influences both the exit streamtube and the forebody spillage. This resulted in the sting mounted (NS/MS) and the nacelle with cylindrical exhaust (N/MS/PM) simulations to predict the highest spillage drags. Conversely, the full configuration (NE/FS/RE) was seemingly affected by a greater modified spillage drag ($c_{D*_{spill}}$) but once the post exit terms were included a negative overall spillage drag was predicted. This is a result of the different shaped streamtube for the lower MFCR with the representative efflux. As the mass flow is reduced the nozzle pressure ratio is also reduced which results in more jet expansion. For this case, the beneficial forward force acting from this offsets any forebody spillage drag and gives a net improvement. To assess the system performance at a lower mass flow the full drag should be considered.

This study concluded that the performance metrics were all influenced by changing the configuration which was being simulated. It was therefore determined that to accurately model the aerodynamic performance of different nacelle geometries the full model (configuration NE/FS/RE) had to be simulated and subsequent results, except where stated, all use this configuration.

	E1	E2	E3
BPR	10	17.8	17.8
OPR	50	64	64
$FNPR$	2.7	2.1	2.1

TABLE 5.2: Cruise design point for engine cycles

	E1/E2	E3
l_{nac}/r_{hi}	4.3	3.1
r_{TE}/r_{hi}	0.955	0.911
f_{if}	1.07	1.29
r_{max}/r_{hi}	1.3	1.15
l_{fb}/l_{nac}	0.4	0.4
β_{nac}	14.0°	13.0°

TABLE 5.3: Summary of baseline engine nacelle design parameters

5.2 Baseline Engine Performance

Two cycles were created which represented two approaches to ultra-high bypass ratio (UHBPR) engine design and a technology level increment between them. The first of these cycles was a three-spool architecture intended to represent a current engine technology scaled to an ultra-high bypass ratio size. The second of the cycles was two-spool and incorporated a gearbox, which is expected to be a feature of some future ultra-high bypass designs.¹⁹ One pre-defined cycle is also included in some studies for this work. This engine, defined by Stańkowski¹¹, is representative of a 3-spool turbofan engine with $BPR=10$ and $OPR=50$. The cycle parameters are summarised in Table 5.2. Each of the three cycles have the same cruise standard net thrust (F_{Nref}) which is therefore constant for all engines in this thesis.

Throughout this thesis a large range of nacelle designs will be considered and it will be important to have a baseline design to compare these to. As such a baseline nacelle has been designed for each of the engine models. E1 had a nacelle designed¹¹ based on traditional design rules using the geometry tool GEMINI¹¹⁶. The baseline nacelle for engine E2 was a scaled up version of the E1 nacelle with a 23% larger fan diameter but the other nacelle geometric parameters were non-dimensionally the same (Table 5.3). Finally, the E3 engine had an aggressive baseline to represent a technology level change and an example of a novel design space nacelle.

The flowfields around the three baseline nacelles (Figure 5.9) demonstrate the increased difficulty of the shorter, slimmer nacelle. As was expected the scaled E2

engine (Figure 5.9c) had the same nacelle flow features as the smaller E1 engine in the cruise condition (Figure 5.9a). Due to its larger size however the nacelle drag (D_{nac}) was 42% larger for the baseline E2 than the baseline E1. This drag accounted for 7.4% of the reference standard net thrust (F_{Nref}) for the E2 engine nacelle whilst it accounted for 5.2% for the E1 engine nacelle. The reference standard net thrust used is the design intent at a 1D level and is constant between all engines in the thesis. For comparison, the E3 engine simulated at the same aerodynamic condition (Figure 5.9e) demonstrated a double flow acceleration on the nacelle surface and a higher peak Mach number ($M_{peak} = 1.18$) than E1 and E2 ($M_{peak} = 1.06$). However, because the E3 nacelle is 23% shorter and has a 4.5% lower r_{max} the drag at this condition was reduced by 17% relative to the E2 nacelle. The E3 baseline nacelle has a drag of 6.1% of F_{Nref} at $M_\infty = 0.82$ and $MFCR = 0.75$. This reduction is as a result of less viscous drag from the cowl surface. The viscous forces on the cowl of the E2 nacelle are 4.6% of F_{Nref} whilst for the E3 nacelle this contribution is reduced to 3.5% of F_{Nref} .

However, when taken to the more aggressive operating point of $M_\infty = 0.85$, $MFCR = 0.7$ the E3 engine exhibits even worse flow features (Figure 5.9f). This results in an increase in the drag to 7.5% of F_N whilst the viscous force contribution only increased to 3.6% of F_{Nref} . The increase in drag due to the change in operating point for the E2 nacelle is lower, up to 8.1% of F_N . Whilst this is still higher than the E3 nacelle drag it is an indication of the increased sensitivity of the shorter nacelle to Mach number and MFCR changes. The lower influence of the operational point change on the longer nacelle is evident in the more benign aerodynamics (Figure 5.9d) which do not exhibit the double shock pattern of the shorter nacelle. These different aerodynamics are a result of the shape of the baseline designs. Each of the designs must have a high curvature at the leading edge to transition to a blunt leading edge (Figure 5.10). This high curvature however must also rapidly reduce to ensure smooth transition to a change in the gradient polarity at the maximum radius (Figure 5.10). This smooth behaviour is a key benefit of the underlying Bernstein polynomials however results in a large $\frac{d\kappa}{dx}$ at the leading edge. In the extreme case in which the fineness ratio is 1, then $l_{nac} = 2r_{max}$ and the changes in curvature would be minimised. However, to reduce drag and create a more streamlined shape, fineness ratio is generally increased and $\frac{d\kappa}{dx}$ must increase to ensure a smooth shape. This then relates directly to the choice of lower r_{max}/r_{hi} for the E3 engine nacelle. The higher curvature and the greater change in curvature at the leading edge of the E3 nacelle result in a higher peak Mach number as the flow is accelerated more.

This more supercritical design then produces more wave drag and may lead to worse off design or installed performance.

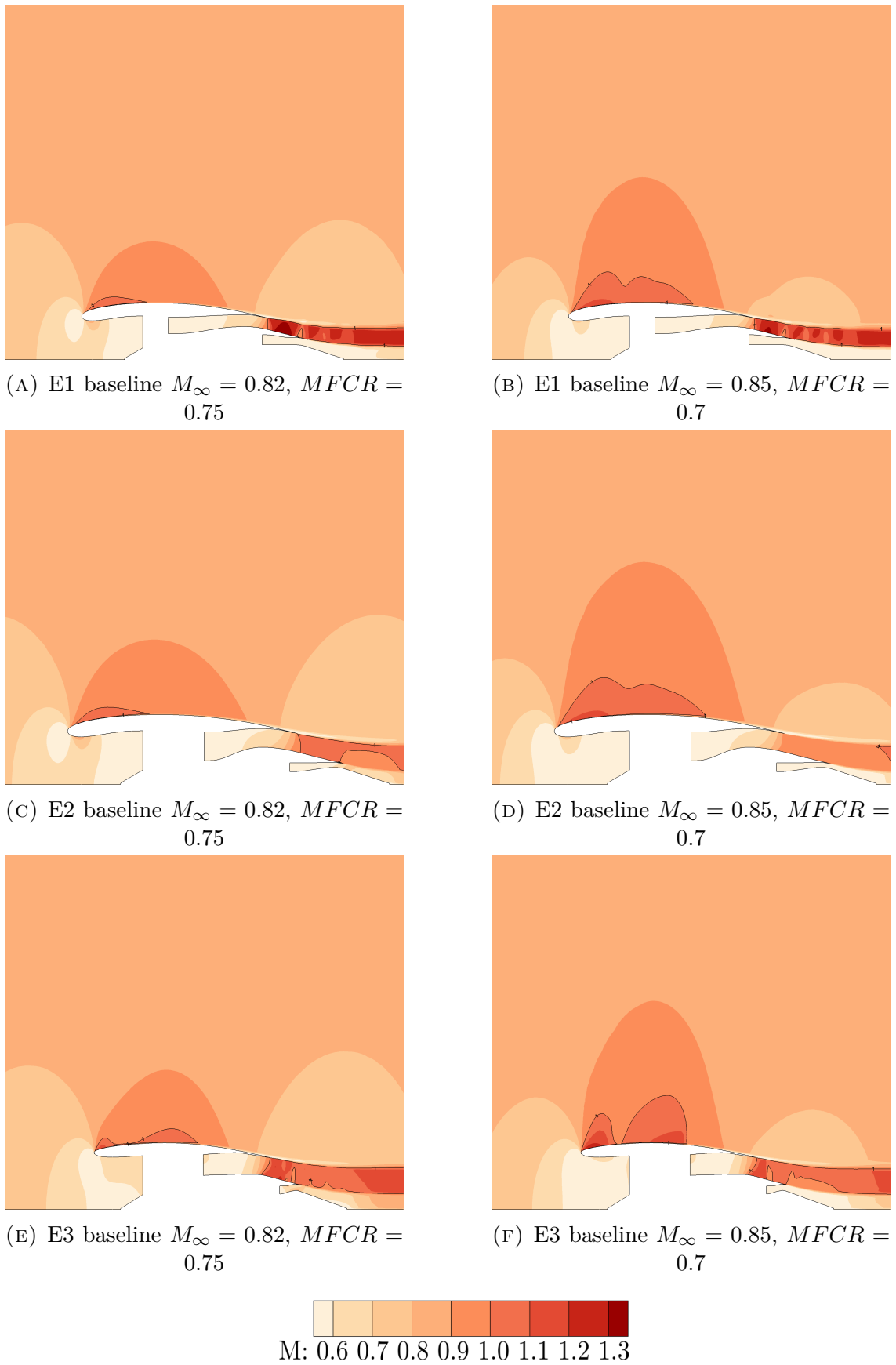


FIGURE 5.9: Three engine nacelle designs at two different aerodynamic conditions

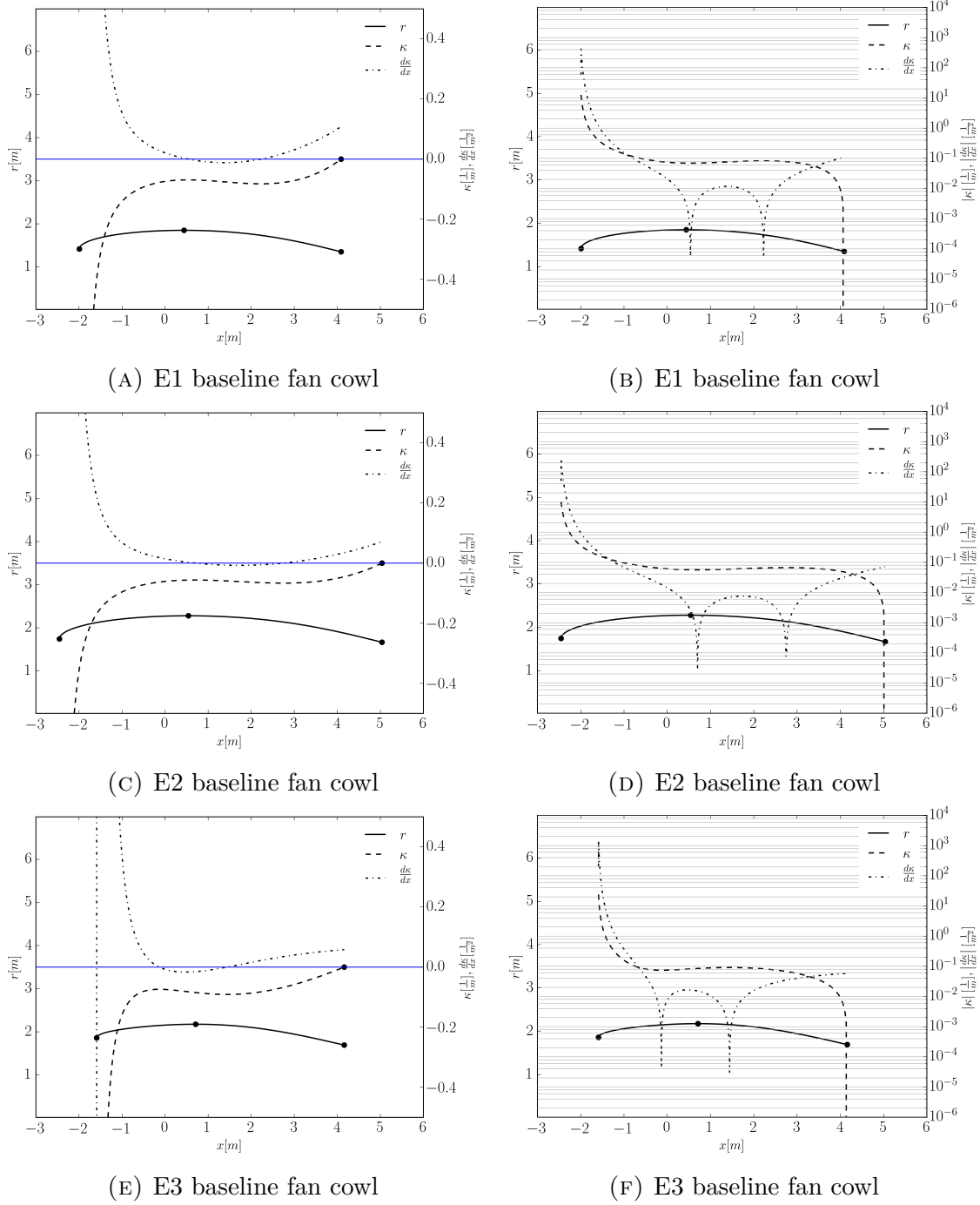


FIGURE 5.10: Three baseline engine nacelle designs

5.2.1 Influence of freestream Mach number

The drag rise Mach number is an important performance indicator which indicates the onset of large amounts of wave drag due to the compressibility of the airflow. At certain points in the flight envelope it may be necessary for the aircraft to

operate at higher than cruise Mach numbers. It is therefore important to ensure that the aerodynamic components are not close to the transonic boundary at which the drag will diverge. It is also important to consider that interference effects between components which were designed in isolation may result in higher local accelerations once combined than were present in isolation. A margin of safety is therefore introduced between the design point and the drag rise Mach number (also known as the drag divergence Mach number) of each component. Knowledge of the drag rise Mach number is therefore important in preliminary design. The three baseline engines were simulated as a function of freestream Mach number to determine the baseline drag rise Mach numbers.

The E1 and E2 engines have a similar aerodynamic response to a change in freestream Mach number (Figure 5.11). As the freestream Mach number is increased the acceleration over the forebody increases and around $M_\infty = 0.8$ exceeds the speed of sound. As the sonic region grows, shock waves result in an increase in the drag (Figure 5.12). This results in the drag rise performance being very similar (Figure 5.12) however the larger E2 engine is offset by roughly +1.4% of F_{Nref} at $M_\infty = 0.7$. This offset between E1 and E2 increases to 4.4% of F_{Nref} at $M_\infty = 0.9$. The drag rise of E1 is $M_{DR} = 0.878$ at $MFCR = 0.75$ and the drag rise of E2 is $M_{DR} = 0.875$ at the same MFCR. Since the nacelle of E2 is a scaled version of the E1 nacelle then this small change in the gradient is a result of the Reynolds effect of a larger engine. The E3 nacelle is much more aggressive with greater values of κ and $\frac{d\kappa}{dx}$ (Figure 5.10). This results in higher peak Mach numbers on the forebody (Figure 5.11c) compared to the E1 or E2 nacelle. As a result, the critical Mach number of the E3 nacelle is approximately $M_\infty = 0.73$. Strong shock waves therefore develop at lower freestream Mach numbers than for the E1 or E2 nacelles. In terms of nacelle drag this means that above $M_\infty = 0.87$ the E3 nacelle produces more drag than the longer E2 nacelle (Figure 5.12). The drag rise of the E3 nacelle is at the much lower value of $M_{DR} = 0.848$.

The E3 engine was expected to operate at the more aggressive $MFCR = 0.7$. A sweep of M_∞ was therefore carried out at this lower MFCR for the E3 nacelle. Since this reduced MFCR gives a locally higher flow incidence at the lip, this results in higher peak Mach numbers on the forebody (Figure 5.11d) compared to the higher MFCR (Figure 5.11c). Between $M_\infty = 0.7$ and $M_\infty = 0.87$ this results in a higher D_{nac} at the lower MFCR. Above $M_\infty = 0.87$ however the double shock pattern experienced for the lower MFCR simulations becomes a single shock as the sonic regions merge (Figure 5.11d). This results in a reduction in the drag before the

increase continues as the single shock strengthens (Figure 5.12). This is above drag rise and the reduction in $MFCR$ from $MFCR = 0.75$ to $MFCR = 0.7$ results in a decrease in the M_{DR} from $M_{DR} = 0.848$ to $M_{DR} = 0.845$

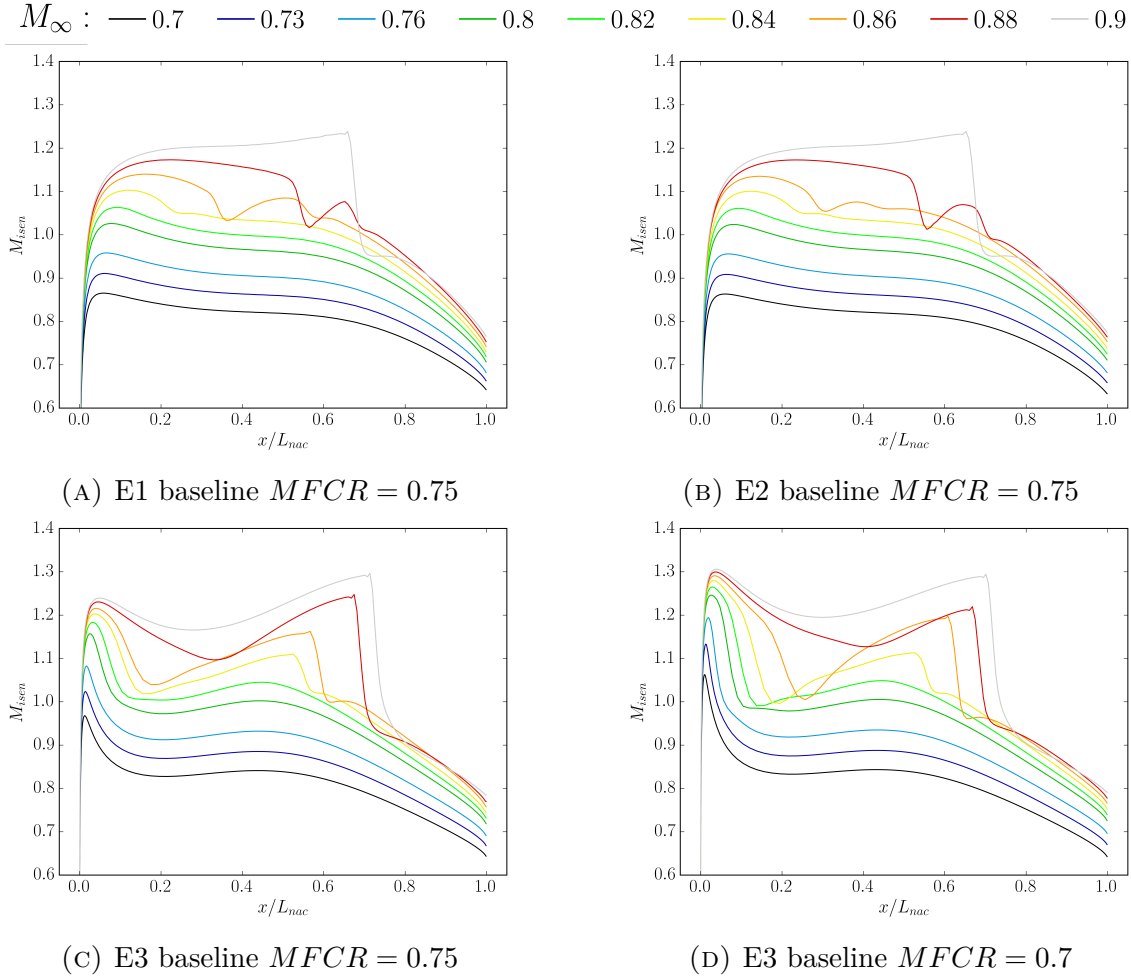


FIGURE 5.11: Three engine nacelle designs at $MFCR = 0.75$ as a function of M_∞

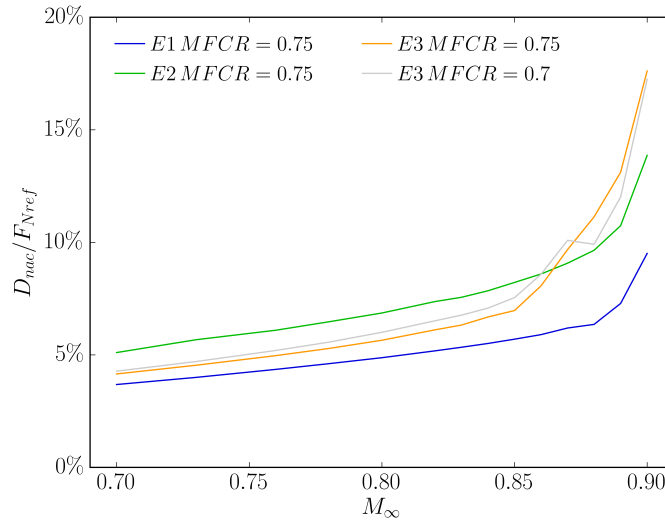


FIGURE 5.12: D_{nac}/F_{Nref} as a function of M_∞ for the three baseline nacelles

5.2.2 Influence of engine mass flow and power setting

Across the flight envelope an engine will operate at a range of power settings which will cover a range of altitudes, Mach numbers and mass flows. Throughout the cruise portion of the flight for instance the mass flow through the engine will be decreased to reduce the thrust of the engine. This is to compensate for the lower mass of the aircraft at the end of cruise compared to the start of cruise. As the mass flow is decreased, the streamtube which enters the engine has a smaller upstream cross-sectional area, which results in a higher angle of the flow at the highlight. This increased angle will cause a greater acceleration around the nacelle lip which will lead to higher velocities.

Simulations were performed which included representative nozzle boundary conditions which reflected the reduced power setting on the engine as were computed from the cycle analysis. This then gives an indication of the overall performance of the engine drag under different power settings. The cycle model for the E2 engine could not be converged at $MFCR = 0.5$ so the CFD simulation was not computed. In general the aerodynamic response from the E1 nacelle and the E2 nacelle were the same (Figures 5.13a, 5.13b) with small changes at the trailing edge present due to the different efflux conditions of the two cycles. Specifically since the E1 engine was a higher specific thrust engine the FNPR range between $MFCR = 0.55$ and $MFCR = 0.75$ was 1.938 to 2.713 whilst for the E2 engine across the same MFCR range the FNPR only varied from 1.599 to 2.147. The larger change in FNPR

means that the post exit force for the E1 nacelle varies more than it does for the E2 nacelle (Figure 5.14). The drag of the E1 and E2 nacelles do not have a large response to a lower MFCR (Figure 5.15a). In this case the change in the post exit force largely compensates for the nacelle force which increases due to the increased wave drag (Figure 5.15b). The E3 nacelle however was affected more detrimentally by the reduced MFCR. The peak Mach number was increased to $M_{peak} = 2.0$ by $MFCR = 0.5$ (Figure 5.13c) and this resulted in a worse drag performance than the E1 or E2 nacelles (Figure 5.15a). At the higher Mach number ($M_\infty = 0.85$) which the E3 nacelle is expected to operate at, there is an increase in the drag of around 1% of F_{Nref} relative to the lower Mach number of $M_\infty = 0.82$ (Figure 5.15a). This offset decreases as the MFCR is reduced however is fairly constant in terms of modified nacelle drag (Figure 5.15b). This indicates that the change in the offset present in the drag values (Figure 5.15a) is influenced by the post exit forces. At $M_\infty = 0.82$, the change in FNPR between $MFCR = 0.5$ and $MFCR = 0.75$ is from $FNPR = 1.555$ to $FNPR = 2.328$, whilst at $M_\infty = 0.85$ this change is from $FNPR = 1.602$ to $FNPR = 2.430$. This larger change results in a larger change in the post exit force for the cases at $M_\infty = 0.85$ and thus the offset reduces. Most significantly however, at the desired operating Mach number $M_\infty = 0.85$, significant spillage occurs and between $MFCR = 0.75$ and $MFCR = 0.5$ the drag increases by 90%.

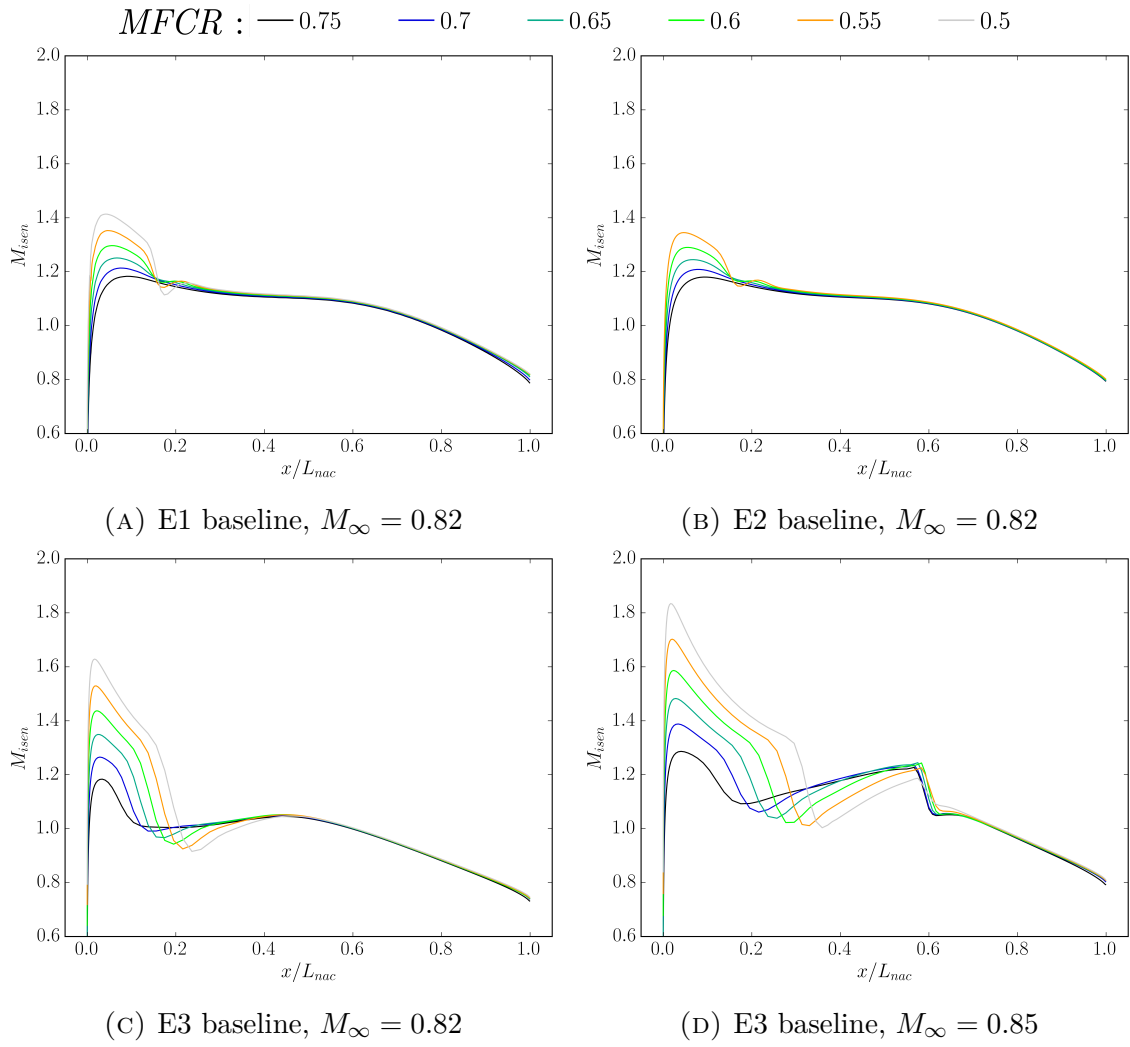
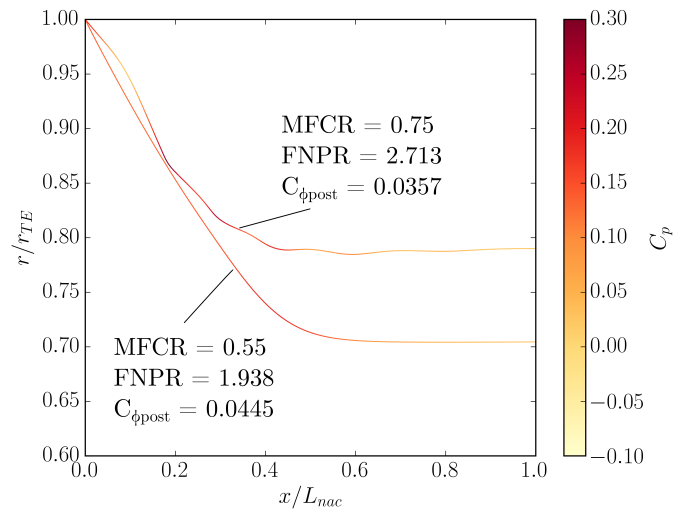
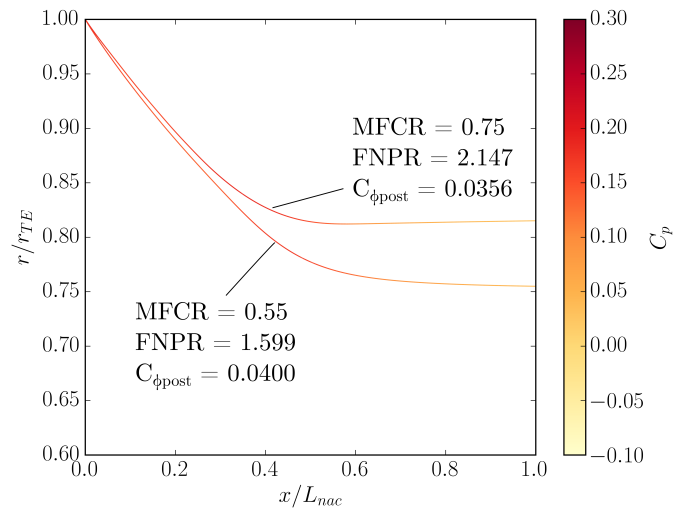


FIGURE 5.13: The isentropic Mach number distributions of three engine nacelle designs as a function of $MFCR$

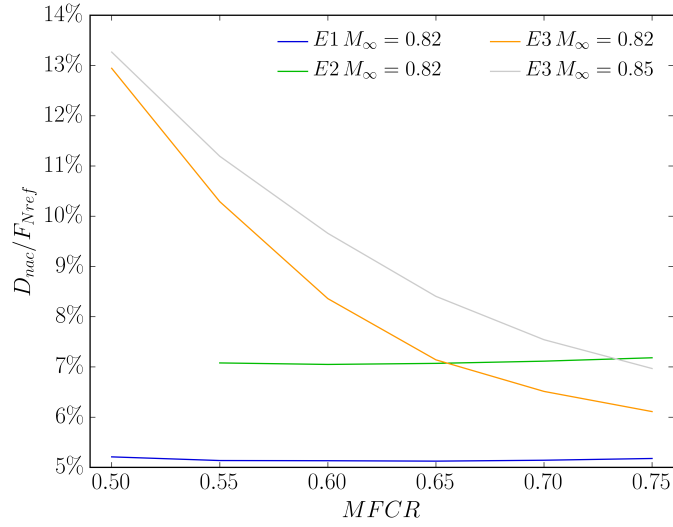
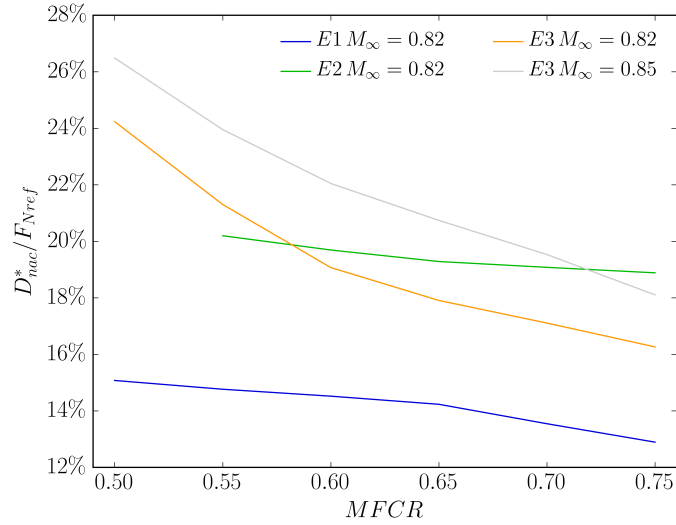


(A) E1



(B) E2

FIGURE 5.14: Streamlines from the trailing edge of the nacelle

(A) D_{nac}/F_{Nref} (B) D_{nac}^*/F_{Nref} FIGURE 5.15: Drag performance as a function of $MFCR$ for the three baseline nacelles

5.3 Axisymmetric nacelle design by full-factorial approach

A two parameter design space exploration was carried out based on the baseline E3 engine by varying the nacelle length, l_{nac} , and maximum radius, r_{max} (Figure 5.16). This was to determine the viability of using a novel (slim and short) nacelle design for future UHBPR engines. The CFD analyses were computed on an engine

with the two spool architecture. The parameterisation used for this full factorial approach did not include a hybrid-iCST nacelle since the curvatures were set to zero at the trailing edge. All other parameters were fixed to the values of the E3 baseline engine (Table 5.3). The range of the full factorial of analyses was based on previous work which assessed a smaller region of the design space⁸. The range of cases was established to incorporate slimmer and shorter nacelles (Figure 5.16). The dual stream engine configurations include the core cowl and plug, however the exhaust design was unchanged throughout the design space exploration.

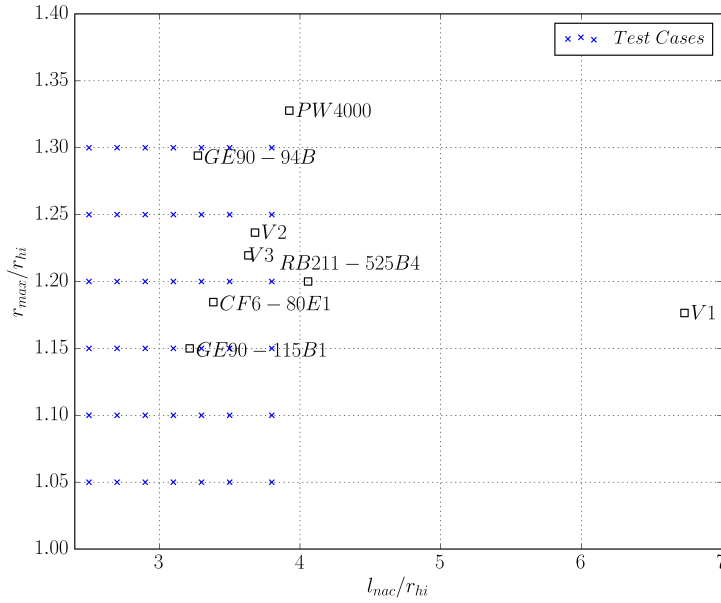


FIGURE 5.16: 2 dimensional full factorial design space with example produced engines and validation engines

The results from the CFD analyses (Figure 5.18) demonstrated that there are some areas of this particular design space which had very high levels of cruise drag. The highest drag ($c_{Dcruise} = 0.3625$) was encountered at $l_{nac}/r_{hi} = 2.5$ and $r_{max}/r_{hi} = 1.3$, i.e. the shortest nacelle with the highest forward facing area. This can largely be attributed to the particular parameterisation employed and other constraints on the geometry which resulted in a heavily inflected design (5.17). The nacelle inflects if the curvature (κ) is positive for any region as this curves the aeroline away from the axis. The resultant aerodynamics for this inflected design were poor and the high drag was a result of the large separation which started around 45% of the nacelle length. This corresponds with heavy local diffusion which triggered the separation. In addition to the separation, the flow around the nacelle has two regions of sonic flow which incur wave drag. Due to the parameterisation and the constraints used,

many of nacelle geometries included inflections (Figure 5.17, 5.18a). These inflected geometries performed badly in general with values of $c_{Dcruise} > 0.05$ for 75% of the cases. For the shorter and thicker nacelles, significant regions of separation were present (Figure 5.19b) as a result of the aggressive diffusion on the afterbody of the nacelle. In this corner of the design space the separation is the main reason for the high drag values. As the length was increased and the thickness reduced these inflections were removed (5.17). This resulted in a sizable reduction in drag (5.18b). At the thinnest end of the design space as r_{max}/r_{hi} approaches 1.05 there is another increase in drag (Figure 5.18a) which is largely caused by high accelerations (Figures 5.19a 5.20) triggered by the large changes in curvatures associated with the slim profiles (5.17). In addition at this low level of r_{max}/r_{hi} there are inflections on the forebody (5.17) for all but the shortest nacelle. A discontinuity in the peak Mach number exists for the case at $l_{nac}/r_{hi} = 3.3$, $r_{max}/r_{hi} = 1.05$ due to a shock driven separation which prevents a higher acceleration. The removal of the inflected designs and the very high drag design demonstrates an obvious trend in the cruise drag which increases as the nacelle is either shortened or made slimmer (Figure 5.18c). The lowest drag corresponds to the longest nacelles at which the lowest peak Mach number is also present (Figure 5.19a). These benign designs, typified by $l_{nac}/r_{hi} = 3.8$, $r_{max}/r_{hi} = 1.2$ and $r_{max}/r_{hi} = 1.25$ in Figure 5.20, accelerated steadily over the forebody and decelerated without any strong shock.

It was established from this full factorial study that the initial geometry parameterisation was inappropriate to produce well designed novel nacelles. It had been used previously to successfully design conventional nacelles. For the novel configurations a greater control of the nacelle curvature was required to lessen the forebody accelerations and to prevent inflections. A conclusion therefore from this study was that the geometry parameterisation should be changed to provide greater flexibility and, given the associated substantial increase in dimensionality, that an optimisation design method would be required instead of the full factorial exploration approach. The three most important aerodynamic aspects of the design found by this study were the removal of geometric inflections, absence of separation and limitation of the peak Mach number.

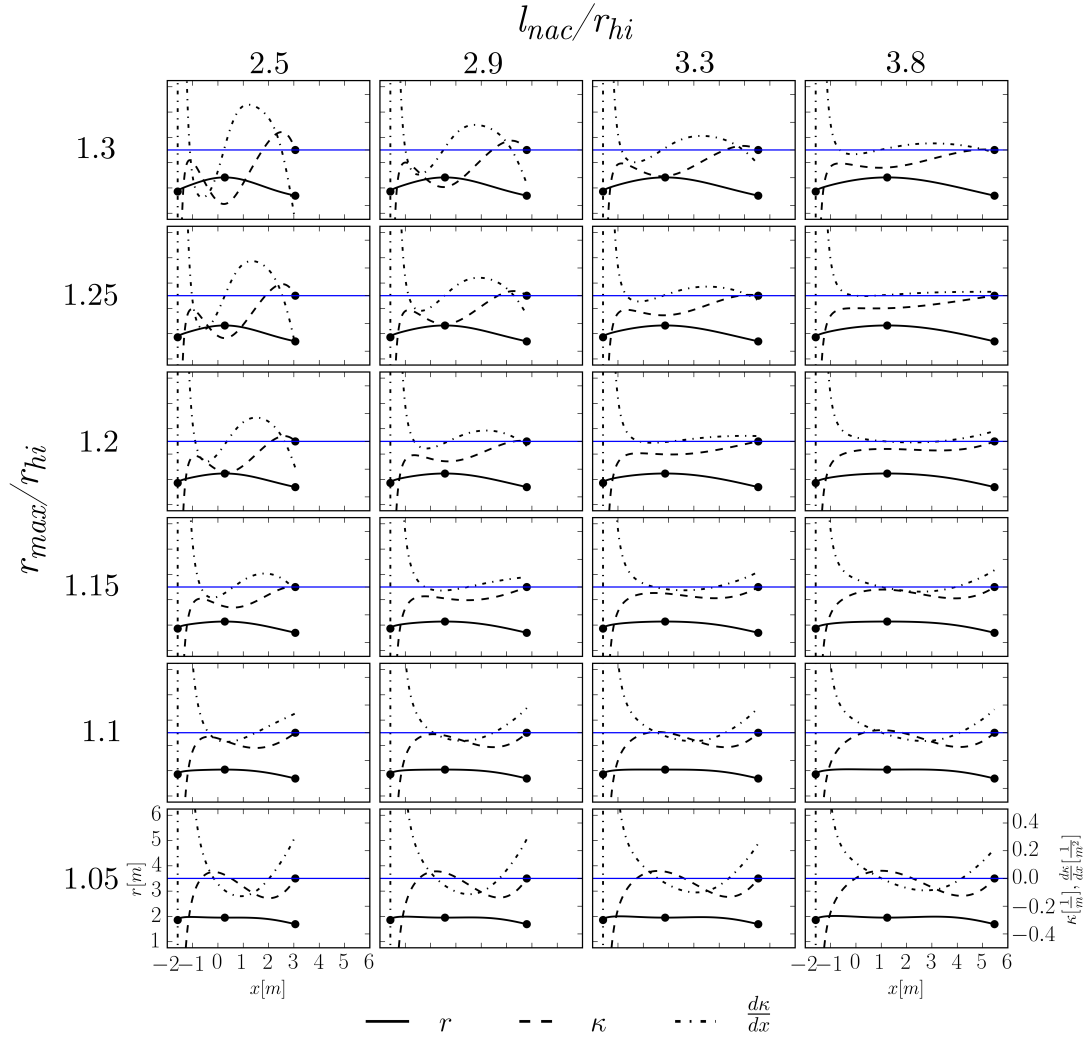


FIGURE 5.17: Subset of designs from full factorial of engine nacelles

5.4 An improved parameterisation

One problem with the parameterisation method established from section 5.3 was that the slimmer and shorter nacelles are more likely to inflect. As an example, the design of $l_{nac}/r_{hi} = 3.3$, $r_{max}/r_{hi} = 1.3$ inflects with the given set of parameters on the afterbody at around $x = 2.6m$ (Figure 5.17). One way to remove this inflection would be to increase the boattail angle (Figure 5.21). This increase in boattail angle however will result in a more rapid diffusion on the afterbody of the nacelle which may be less preferable from a design perspective. A rapidly diffusing nacelle for instance is more likely to separate at high incidence. The boattail angle therefore is usually limited to within a relatively narrow range which will be compatible

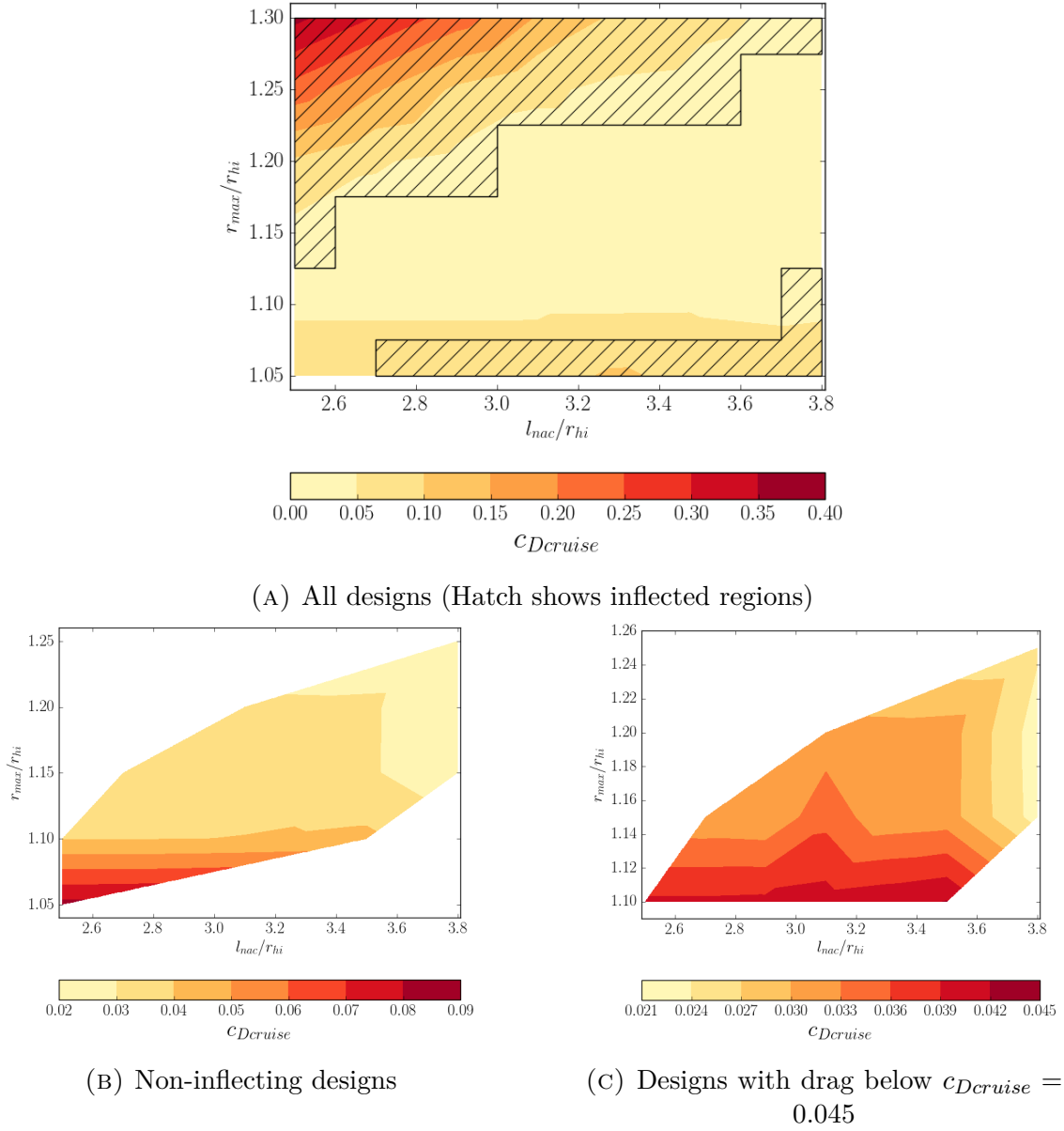
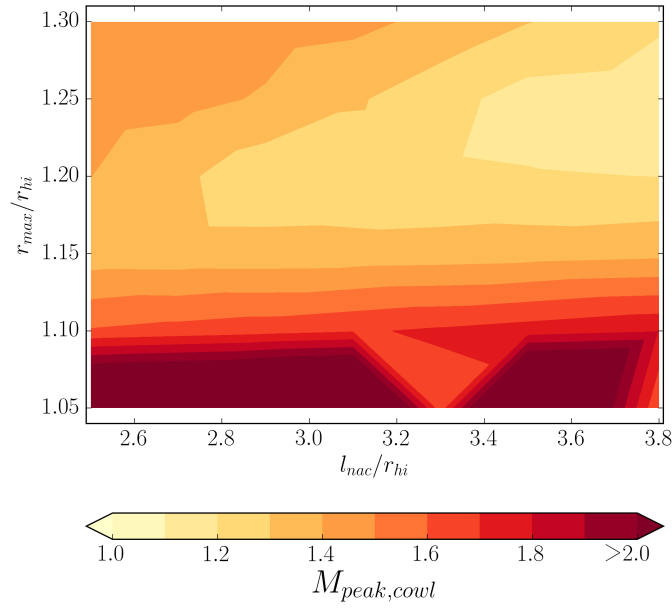


FIGURE 5.18: Cruise drag ($c_{Dcruise}$) as a function of non dimensional length (l_{nac}/r_{hi}) and thickness ratio (r_{max}/r_{hi}) for a full factorial of engine designs

with the range of operations and the integration with exhaust system and wing aerodynamics.

Another variable which can have a strong influence on the geometry produced is the imposed value of trailing edge curvature (κ_{TE}). In the previous study (section 5.3) this was set to a value of $\kappa_{TE} = 0$ but varying this value can result in more favourable designs. For instance the design of $l_{nac}/r_{hi} = 3.3$, $r_{max}/r_{hi} = 1.1$ had a fairly strong shock with trailing edge curvature of zero (Figure 5.20). The curvature distribution can be controlled more through the addition of non-zero curvature at



(A) Non-inflecting designs

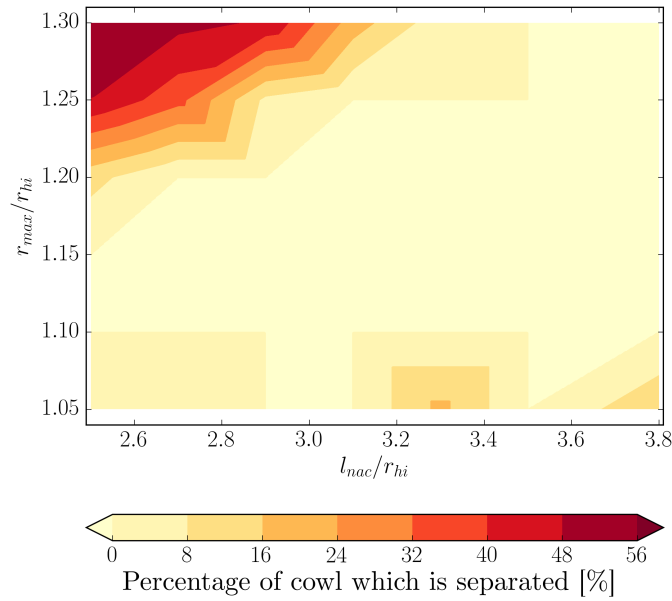
(B) Designs with drag below $c_{Dcruise} = 0.045$

FIGURE 5.19: Cruise drag ($c_{Dcruise}$) as a function of non dimensional length (l_{nac}/r_{hi}) and thickness ratio (r_{max}/r_{hi}) for a full factorial of engine designs

the trailing edge (Figure 5.24). This addition allows for the curvature distribution to adopt different shapes and the rate of change of curvature ($\frac{d\kappa}{dx}$) to be reduced (Figure 5.22b) or even increased (Figure 5.22d) according to the designers requirements. It is expected that the preferred design would have low values of $\frac{d\kappa}{dx}$ to avoid rapid accelerations which are associated with large changes in curvature.

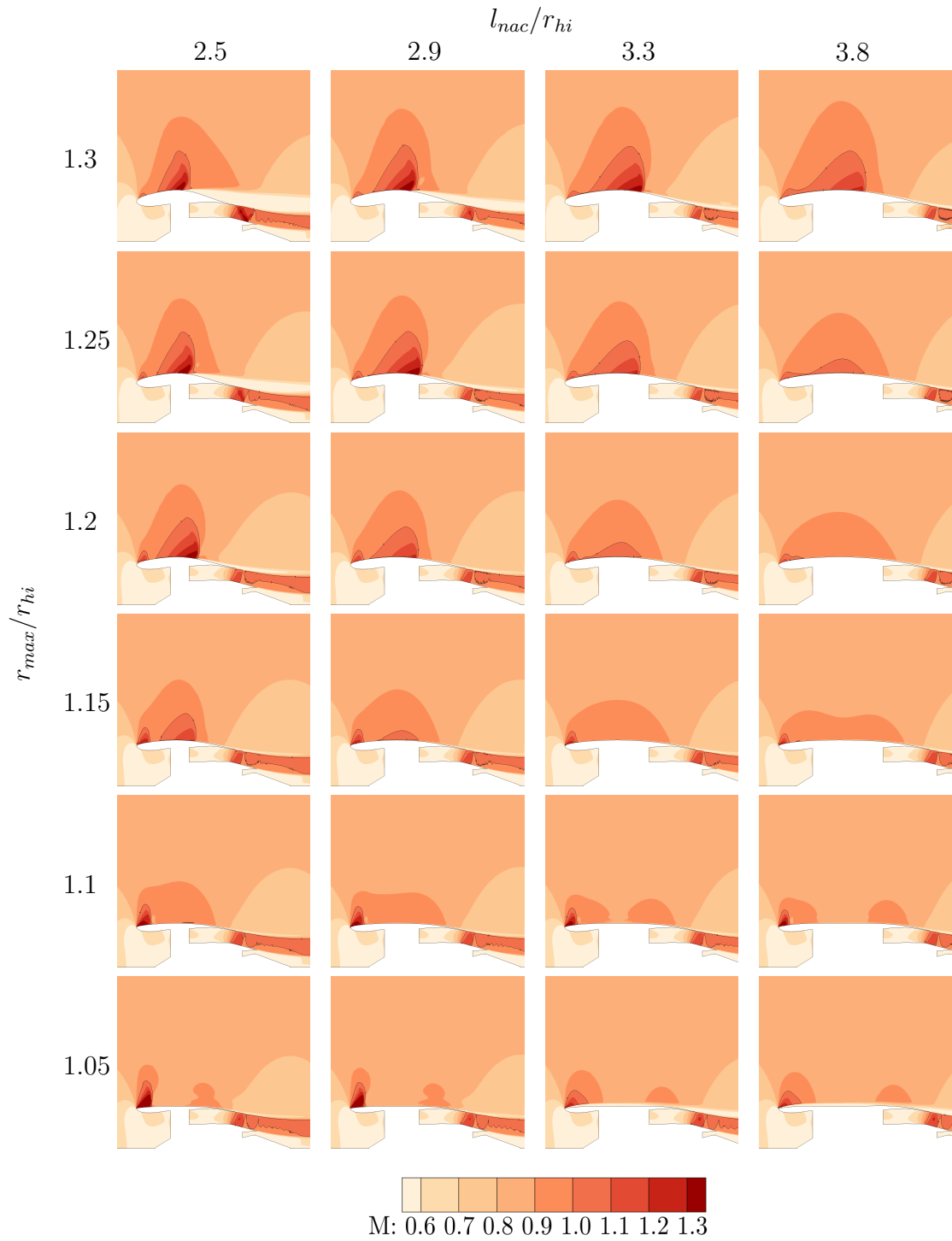
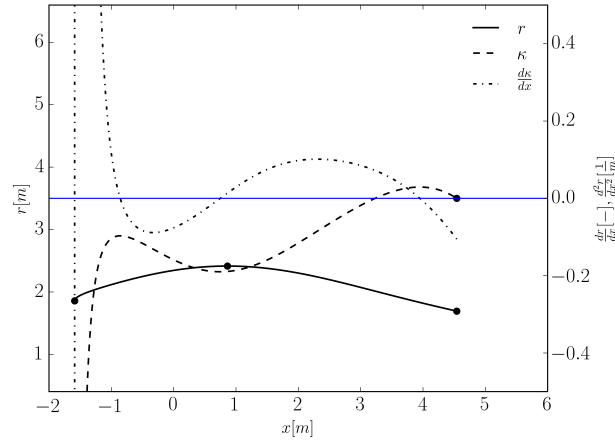
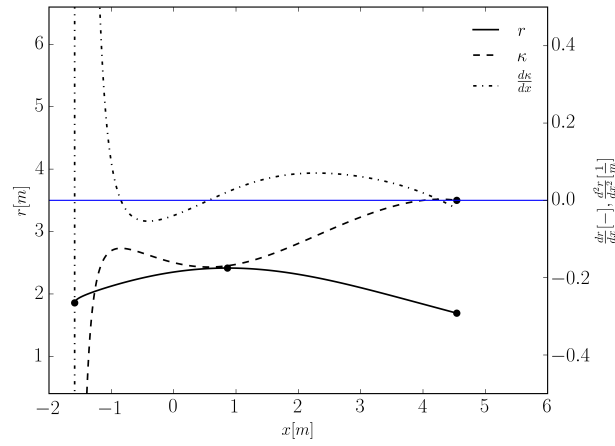
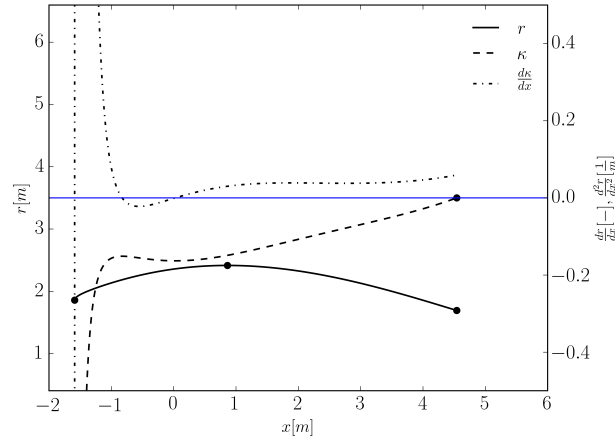


FIGURE 5.20: Subset of designs from full factorial of engine nacelles

The problem of inflections on the forebody was generally caused by too high a value of r_{if} . An example of this was the nacelle designed for $l_{nac}/r_{hi} = 3.3$ and with $r_{max}/r_{hi} = 1.05$ (Figure 5.17). This design curved away from the engine axis for approximately $\frac{1}{3}$ rd of the fan cowl (Figure 5.23a). A reduction in r_{if} from

(A) $\beta = 13.0^\circ$ (B) $\beta = 15.0^\circ$ (C) $\beta = 17.0^\circ$ FIGURE 5.21: Influence of β on geometrical inflections

$r_{if} = 0.041m$ to $r_{if} = 0.011m$ reduced the extent of the convex section and moved it aft such that the fan cowl started to curve away from the axis on the forebody and then stopped on the afterbody (Figure 5.23b). It was required to also reduce

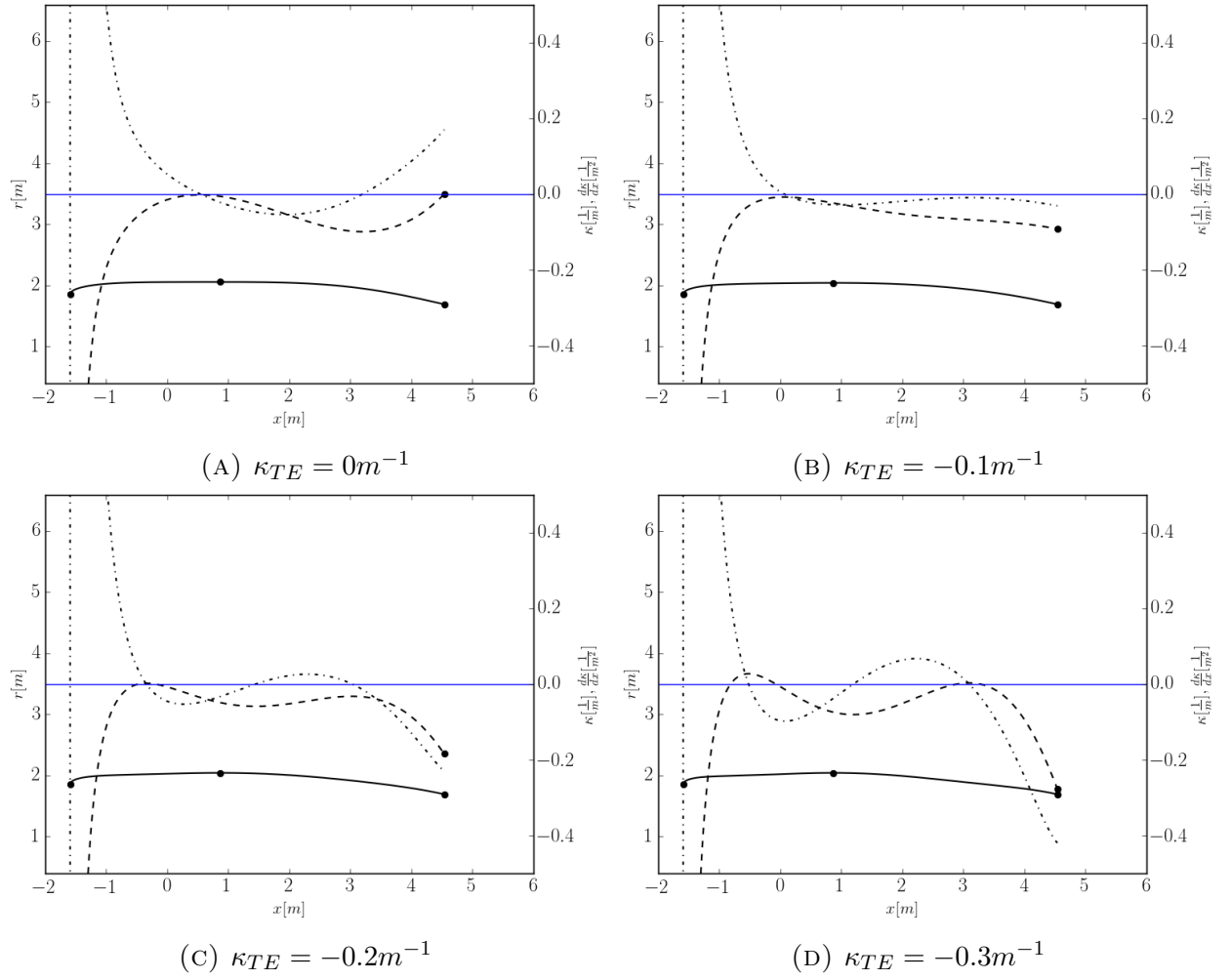


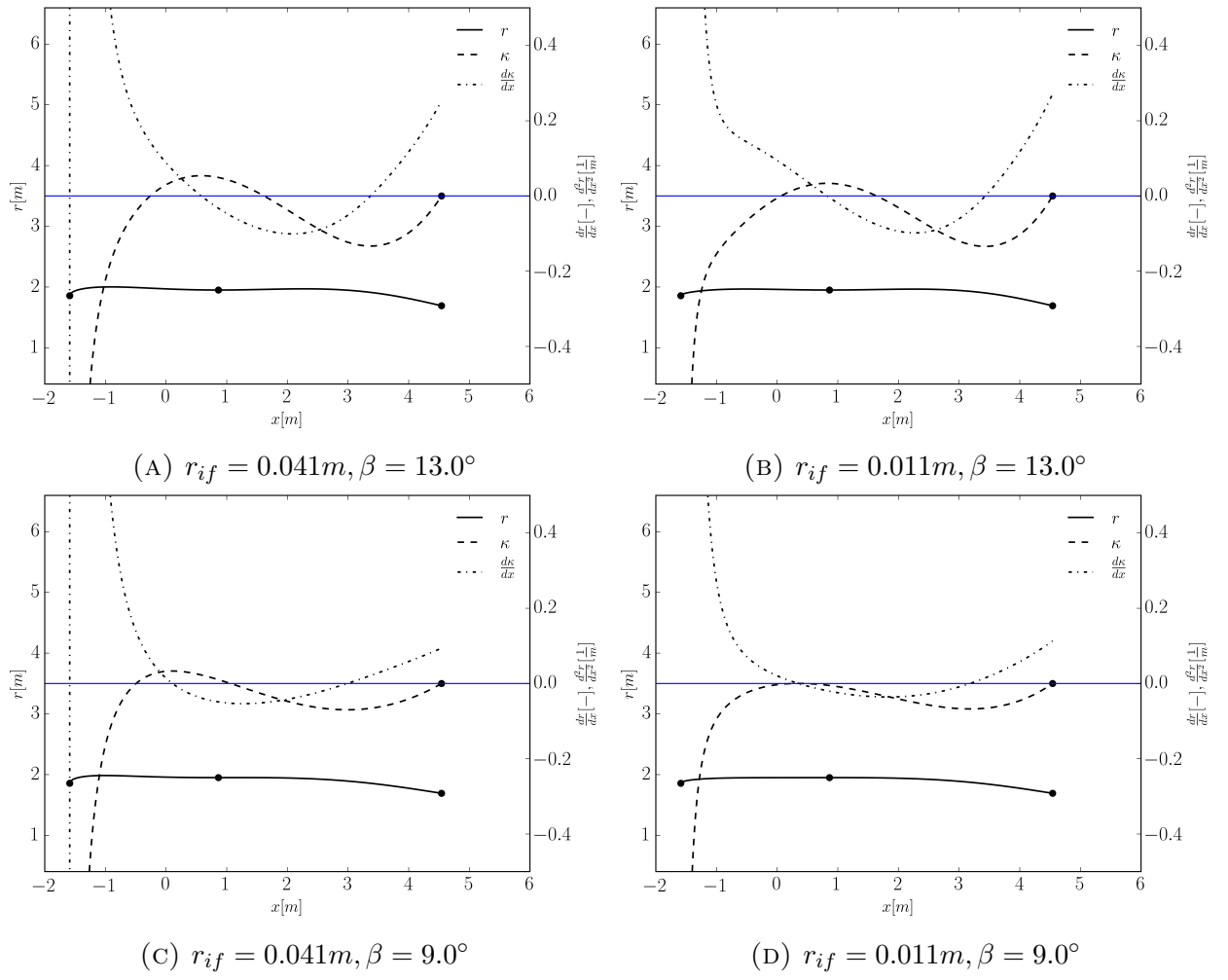
FIGURE 5.22: Influence of κ_{TE} on geometrical inflections and curvature distributions

the boattail angle (Figure 5.23d) from $\beta = 13.0^\circ$ to $\beta = 9.0^\circ$ to completely remove this inflection. This demonstrates a problem with the parameterisation used until now as it required both r_{if} and β to be changed to produce a non-inflected forebody design. Ideally the forebody design would be decoupled to allow more flexibility and less reliance on a good afterbody design.

An alternative approach was considered which decoupled the forebody and the afterbody somewhat whilst retaining the same number of degrees of freedom. In this method the same constraints are applied but the forebody and afterbody are split into two separate iCST curves at the position of maximum radius. Both of these curves are created from the highlight to the trailing edge but they are then trimmed to just forebody or afterbody. Additional constraints are added to the afterbody curve which ensures continuity at the maximum radius between forebody

and afterbody. Specifically, the second and third derivatives at this axial location are extracted from the forebody curve and imposed at the same position on the afterbody curve. The zeroth and first derivatives are already constrained at this position to ensure it is the maximum radius. This constraint ensures that the curvature and the rate of change of curvature are consistent between the two curves however higher derivatives will have a discontinuity at the maximum radius. With this level of decoupling between the forebody and afterbody it is possible to achieve a non-inflecting design with either of the two boat-tail angles desired (Figure 5.24) where previously the only viable design had a low boat-tail angle (Figure 5.23). This beneficial decoupling allows additional viable designs in the extreme, previously inflecting, regions of the design space.

This process of varying independently the design variables to produce a non-inflecting design is clearly a cumbersome method of exploring the design space. In addition, these permutations of the variables do not necessarily result in well designed, representative nacelles. Since an objective of this project was to determine the impact of a future UHBPR engine nacelle on the drag performance of the engine it was established that a more exhaustive method was required to produce the designs. One approach could be an extension of the full factorial approach design of experiments conducted above. If the length were fixed for a given problem and the other variables identified (r_{max}/r_{hi} , f_{max} , f_{if} , β , κ_{TE}) were not then there would be 5 degrees of freedom. If six levels of each variable were maintained then this would result in 7776 separate designs. Since it was also desired to look at off design aerodynamics, this would be prohibitive. A 3 level full factorial could be performed with 243 designs however whilst this would provide information on the influence of each variable it would not necessarily provide representative designs. The desired outcome of any design space exploration in this project would rather be a trade-off surface in which all the designs can be said to be well designed for a given set of aerodynamic requirements, i.e. performance at different aircraft operating points. Such a trade-off surface is defined by a Pareto set which can be determined through the process of optimisation. The following sections will therefore outline the results of optimisation activities performed on this problem.

FIGURE 5.23: Influence of r_{if} and β on geometrical inflections

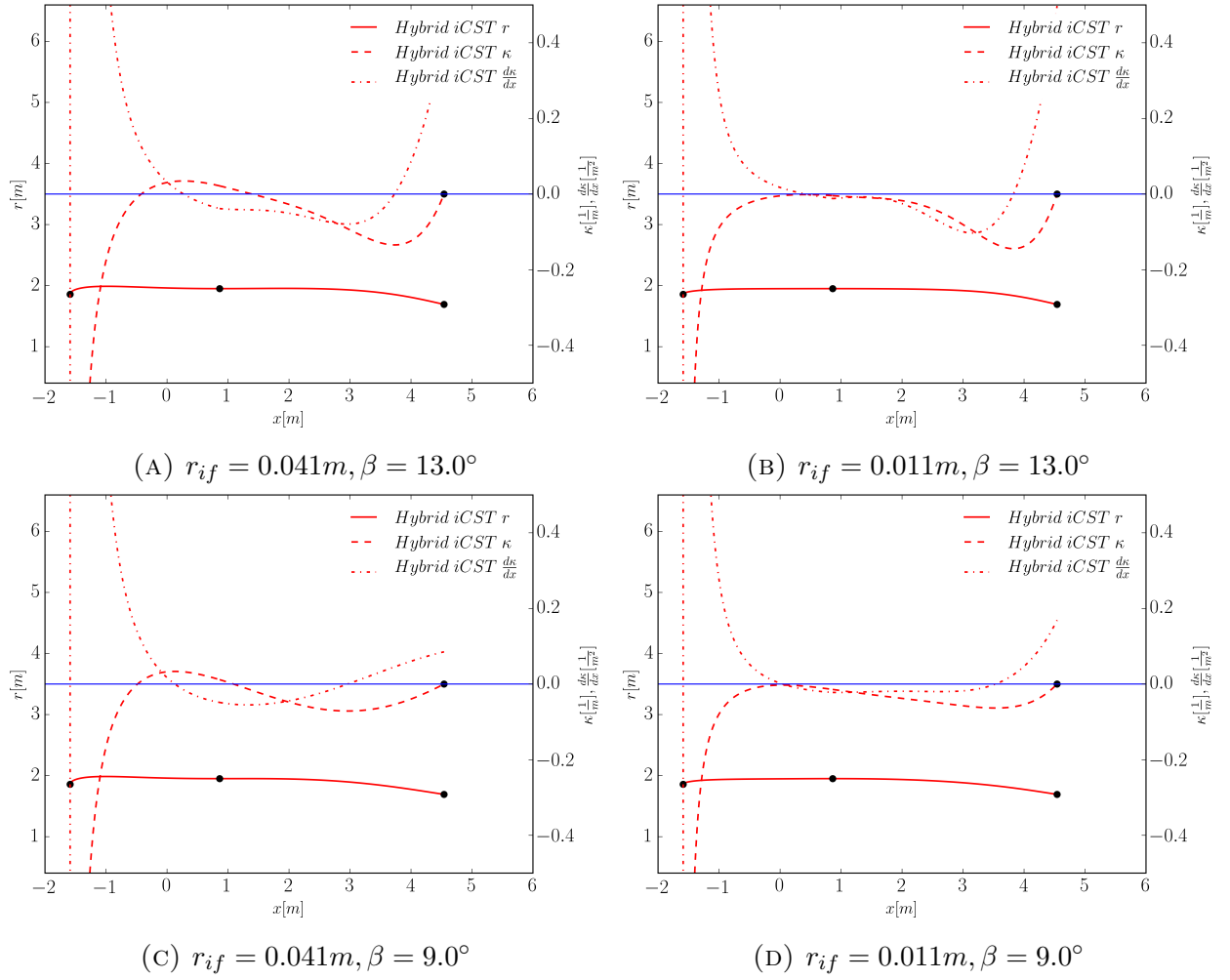


FIGURE 5.24: Influence of r_{if} and β on geometrical inflections with a hybrid iCST fan cowl

5.5 Axisymmetric nacelle optimisation

5.5.1 Optimisation on existing design

To demonstrate the optimisation method which has been outlined (Section 3.5) it was applied to the V2 nacelle used for validation (Section 4.1). This allowed for determination of whether the optimisation algorithm had been correctly calibrated and if the CFD method could determine the response from the design space accurately enough for optimisation. The validation nacelle was chosen as a test case since it had been designed in detail for good aerodynamic performance so should provide a high benchmark.

	Lower bound	Upper bound
f_{if}	0.6	1.4
r_{max}/r_{hi}	1.08	1.4
l_{fb}/l_{nac}	0.3	0.5
y''_{TE}	-0.3	0.0
β_{nac}	11.0°	14.0°

TABLE 5.4: Parametric geometry bounds used for the optimisation of the V2 nacelle

CFD simulations were run as the means of assessing the nacelle performance metrics. The exhaust condition was matched to the total conditions of the freestream flow and as such the configuration was N (Figure 5.1d) and drag was evaluated with equation 3.8. Three objective functions were used as described in section 3.4.4 ($c_{D,cruise}, M_{DR}, \Delta c_{Dspill}$). In this case the cruise condition was considered to be $M_\infty = 0.85$, $MFCR = 0.7$ whilst the spillage was evaluated as a change between the cruise and $M_\infty = 0.85$, $MFCR = 0.65$ condition. The optimisation was run for 50 generations with an initial generation size of 400 and subsequent generation size of 40. The bounds used were chosen to expand wide enough beyond the conventional design space but also include the original design (Table 5.4). This optimisation was initialised from 400 randomly generated samples within the bounds of the search space (Figure 5.25a). This random sample was sorted into non-dominated sets in which each design was mutually non-dominated. A mating pool of 40 was then filled from the lowest non-dominated sets. This mating pool was updated over the subsequent generations as mutation and crossover produced new designs in the search space (Figures 5.25b, 5.25c, 5.26a, 5.26b, 5.26c). Over the course of the generations, the objective functions were normalised and the hypervolume calculated. These values can be used to determine the convergence of the optimisation. The hypervolume rapidly increased in the first 11 generations and then reached an asymptote as the optimisation converged (Figure 5.27). Very little improvement was noted beyond 20-30 generations and as such the maximum number of generations was reduced for subsequent optimisations.

A final Pareto front was resolved by combining all the results from each generation and evaluating a non-dominated set from these (Figure 5.28). By doing this, the resolution of the Pareto set was not limited to the 40 individuals of the mating pool. The results from this optimisation demonstrated that a Pareto front could be reached through direct CFD simulations which could improve on a well-designed baseline nacelle (Figure 5.28). A trade-off was evident between the three objective

functions, i.e. neither $C_{Dcruise}$, M_{DR} nor C_{Dspill} were collinear. This confirms that in the design of a nacelle, the cruise drag can be traded for improved drag rise or spillage, or vice versa.

Compared to the baseline design, improvements in the individual objective functions were available which gave an increase of $\Delta M_{DR} = 0.014$ in drag rise Mach number, $\Delta C_{Dnac} = -0.0057$ cruise drag and an equivalent C_{Dspill} . Since the drag rise Mach number is typically dictated by the required design Mach number, this was maintained in selecting a design to compare to the original V2 nacelle. As all Pareto optimal designs had lower drag than the baseline, the design with the lowest spillage drag was chosen for comparison (Figure 5.29). This design had an improved drag rise of $\Delta M_{DR} = 0.005$ higher, and a reduced drag of $\Delta C_{Dnac} = -0.0040$, though the spillage increased by $\Delta C_{Dspill} = 0.0003$. This increase is negligible in the context of a reduction of $\Delta C_{Dnac} = -0.0040$ in cruise drag however. In addition, the aerodynamics were greatly improved by the optimisation process relative to the baseline design (Figure 5.29). The optimal design exhibited a single acceleration followed by a deceleration with no strong shock wave, whereas the baseline had multiple re-accelerations over the forebody of the cowl.

The Pareto set can also be examined in the design space (Figure 5.30, 5.31, 5.32) as well as the objective space. In this context, the optimal regions of the design space can be determined and the influence of each design parameter is related to the objective functions. For this optimisation, the optimal set converged to a region which was not coincident with any of the bounds of the design space. This indicates that the search space was probably well suited to the design problem. There were a few outliers, but the vast majority of the designs were in a single group. These outlying designs generally had high cruise drag, low M_{DR} but compensated with the lowest spillage (Figure 5.32). The outlying designs were located in the higher values of r_{max}/r_{hi} and f_{if} relative to the larger group of optimal designs. This signifies a more rounded, i.e. greater radius of curvature, nacelle highlight with a larger forward facing area. As shown in section 5.4, this will give a greater drag and worse drag rise but lower accelerations across the highlight, which turn provides the improved spillage performance as evidenced (Figure 5.32). Designs with high r_{max}/r_{hi} or f_{if} have the worst M_{DR} performance (Figure 5.31) and this is a key driver which has resulted in the optimal set located here. In addition, there is a narrow band of designs within the r_{max}/r_{hi} design space which have acceptable levels of cruise drag (Figure 5.30). This further demonstrates the trade-off evinced in section 5.4 whereby a very low r_{max}/r_{hi} design resulted in a strong shock due to

high forebody curvature whilst a high r_{max}/r_{hi} would result in high curvatures at the maximum radius and an afterbody shock which greatly thickened the boundary layer in a region of high diffusion. These trade-offs between the objective variables limit the design space and whilst good designs in any one objective appear throughout much of the search space it is effectively bounded by the requirement to balance all three objectives. The other three variables (f_{max} , β_{nac} and y''_{TE}) whilst important to avoid inflected or rapidly diffusing nacelles do not seem to limit the viable search space as much as r_{max}/r_{hi} or f_{if} . This can be demonstrated by the fact that good values for each objective function can independently be obtained in the entire ranges, whereas this is not true for r_{max}/r_{hi} or f_{if} .

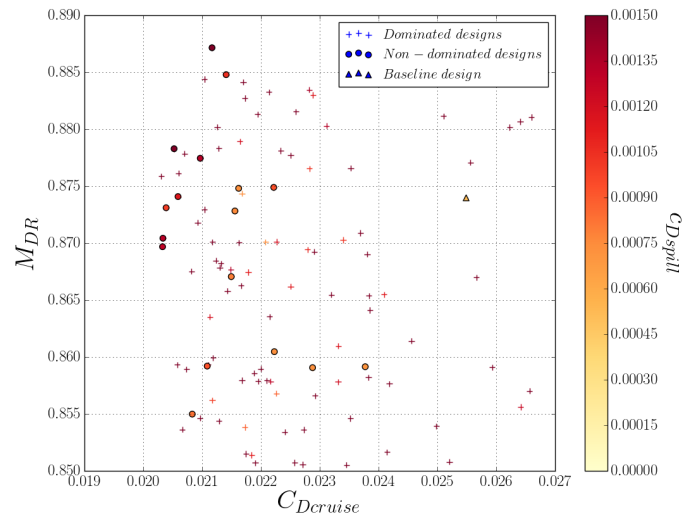
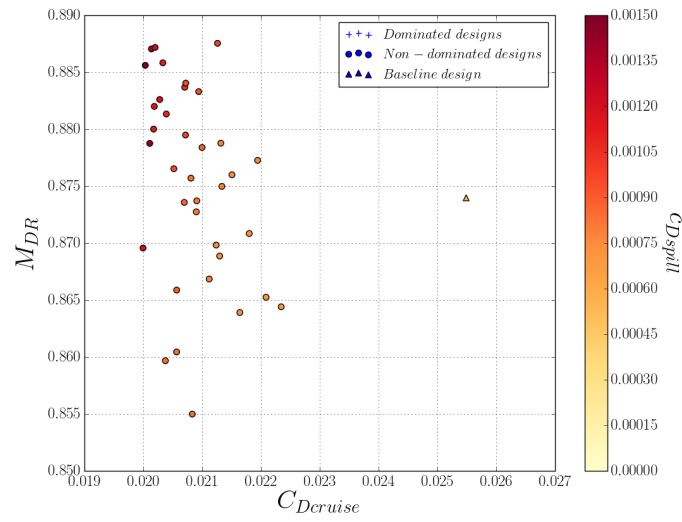
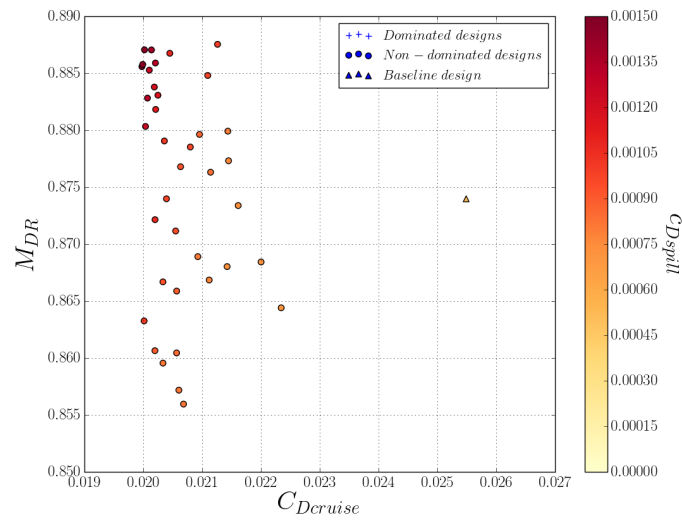
(A) $n_{gen} = 0$ (B) $n_{gen} = 5$ (C) $n_{gen} = 10$

FIGURE 5.25: Evolution of the mating pool for the V2 nacelle optimisation

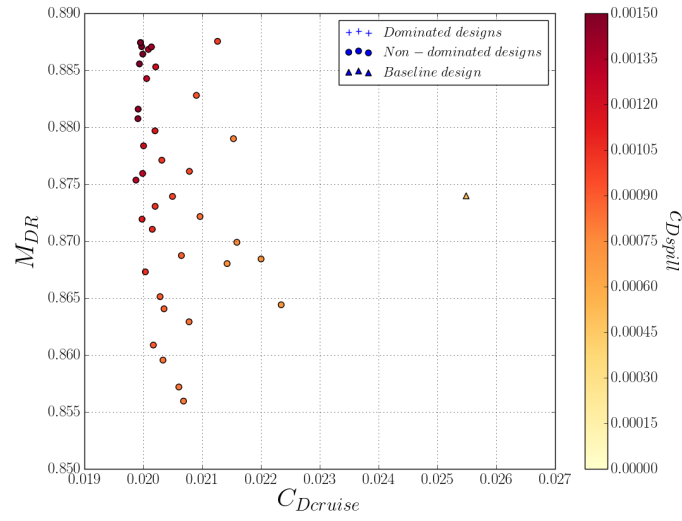
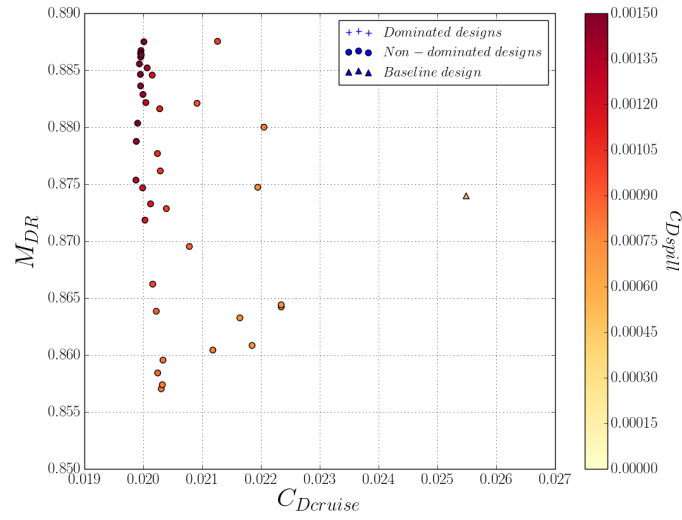
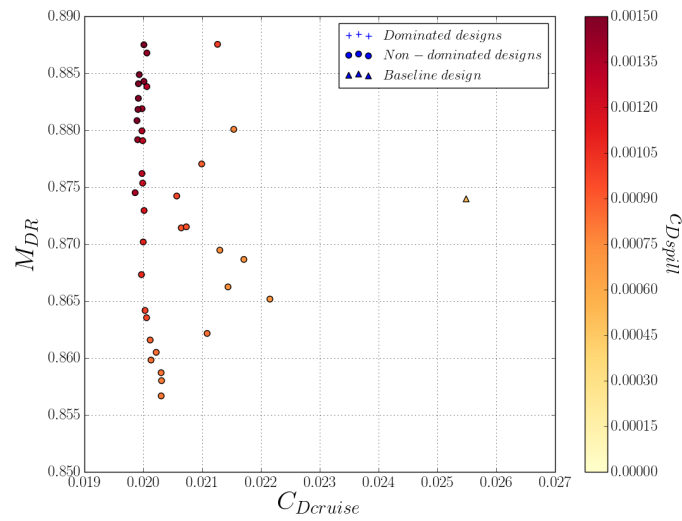
(A) $n_{gen} = 15$ (B) $n_{gen} = 30$ (C) $n_{gen} = 50$

FIGURE 5.26: Evolution of the mating pool for the V2 nacelle optimisation

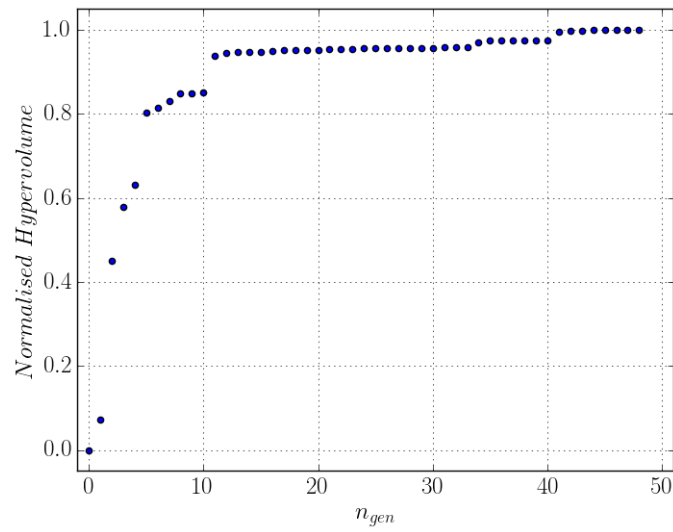


FIGURE 5.27: Hypervolume convergence for the V2 nacelle optimisation case

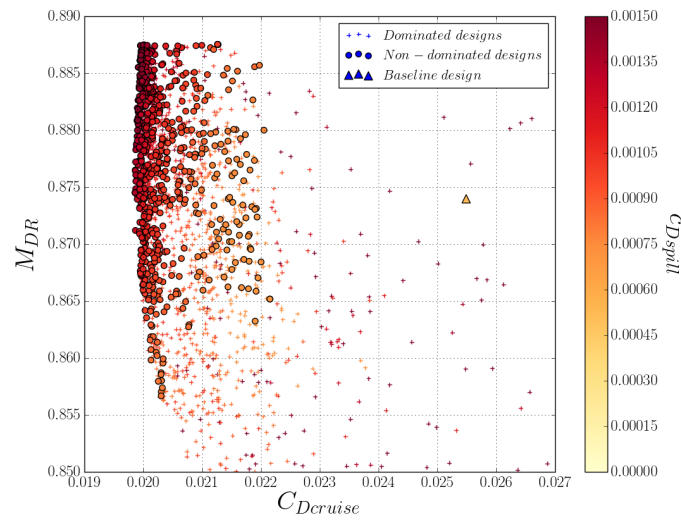


FIGURE 5.28: Pareto optimal set for optimisation of the validation nacelle 2 in objective space

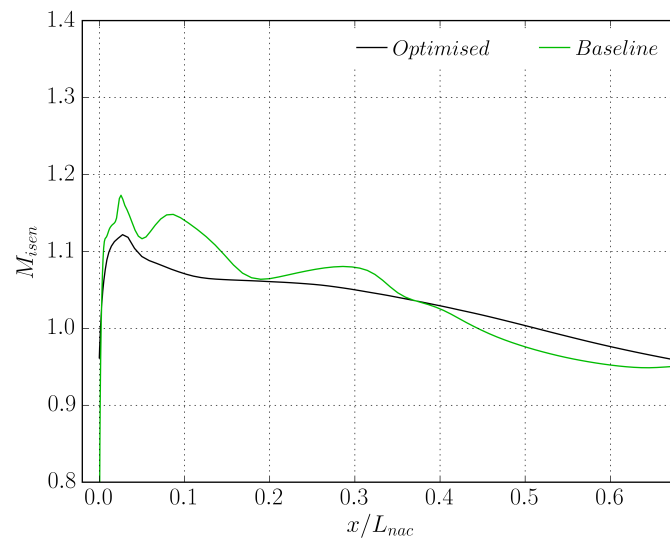
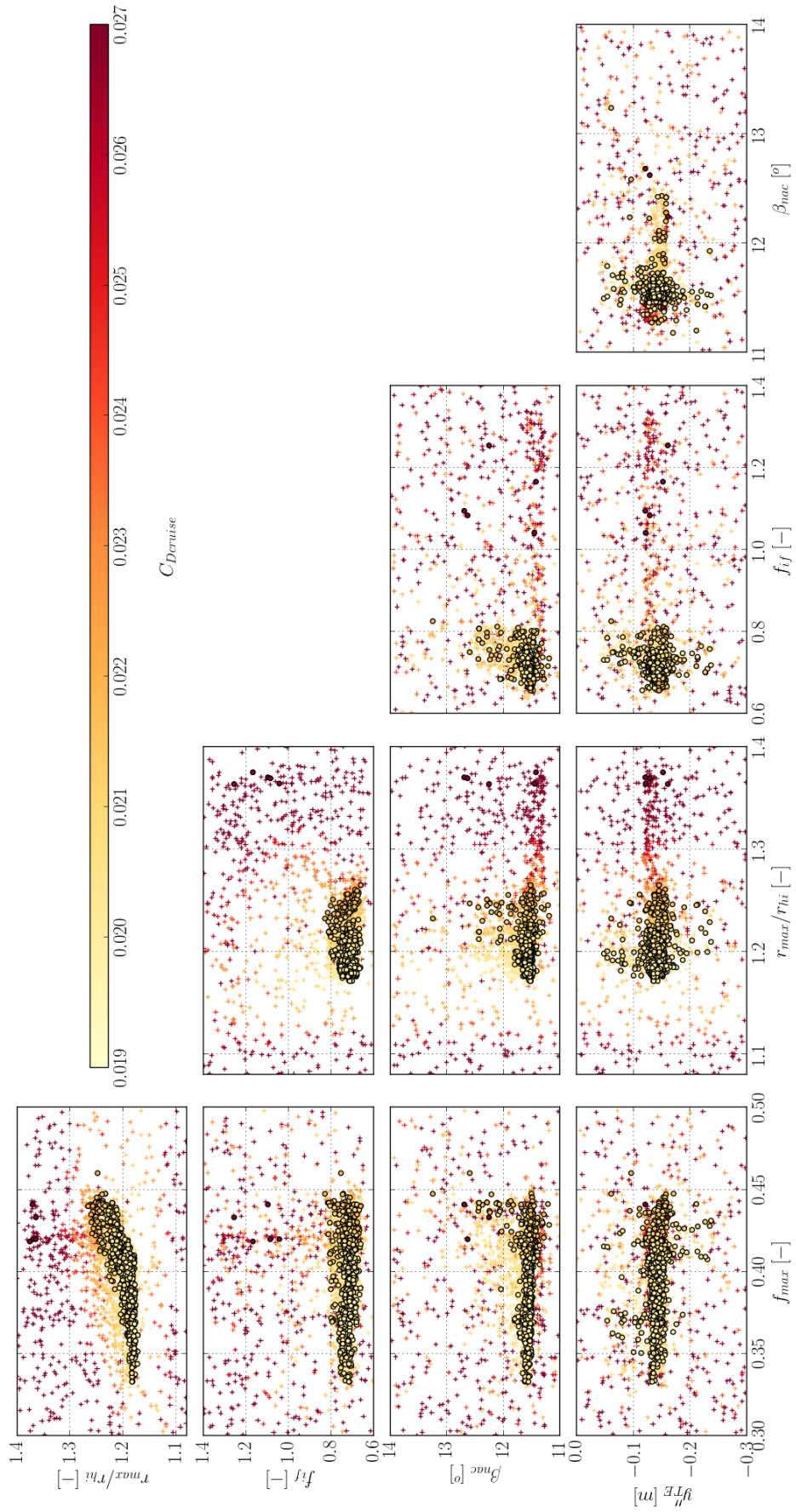


FIGURE 5.29: Example optimal design cowl Mach number distribution

FIGURE 5.30: Pareto set in the design space coloured by $C_{D_{ruise}}$

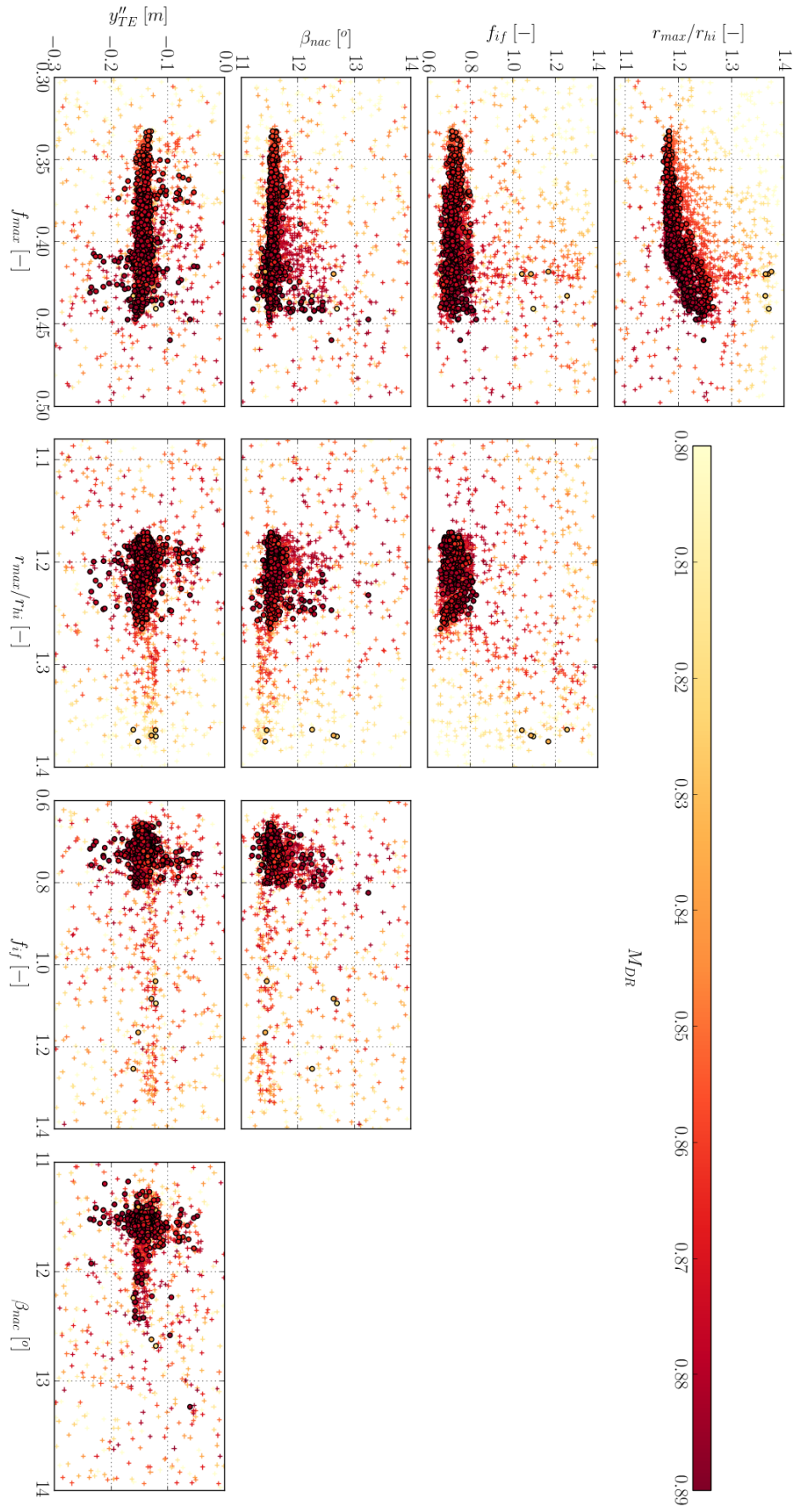
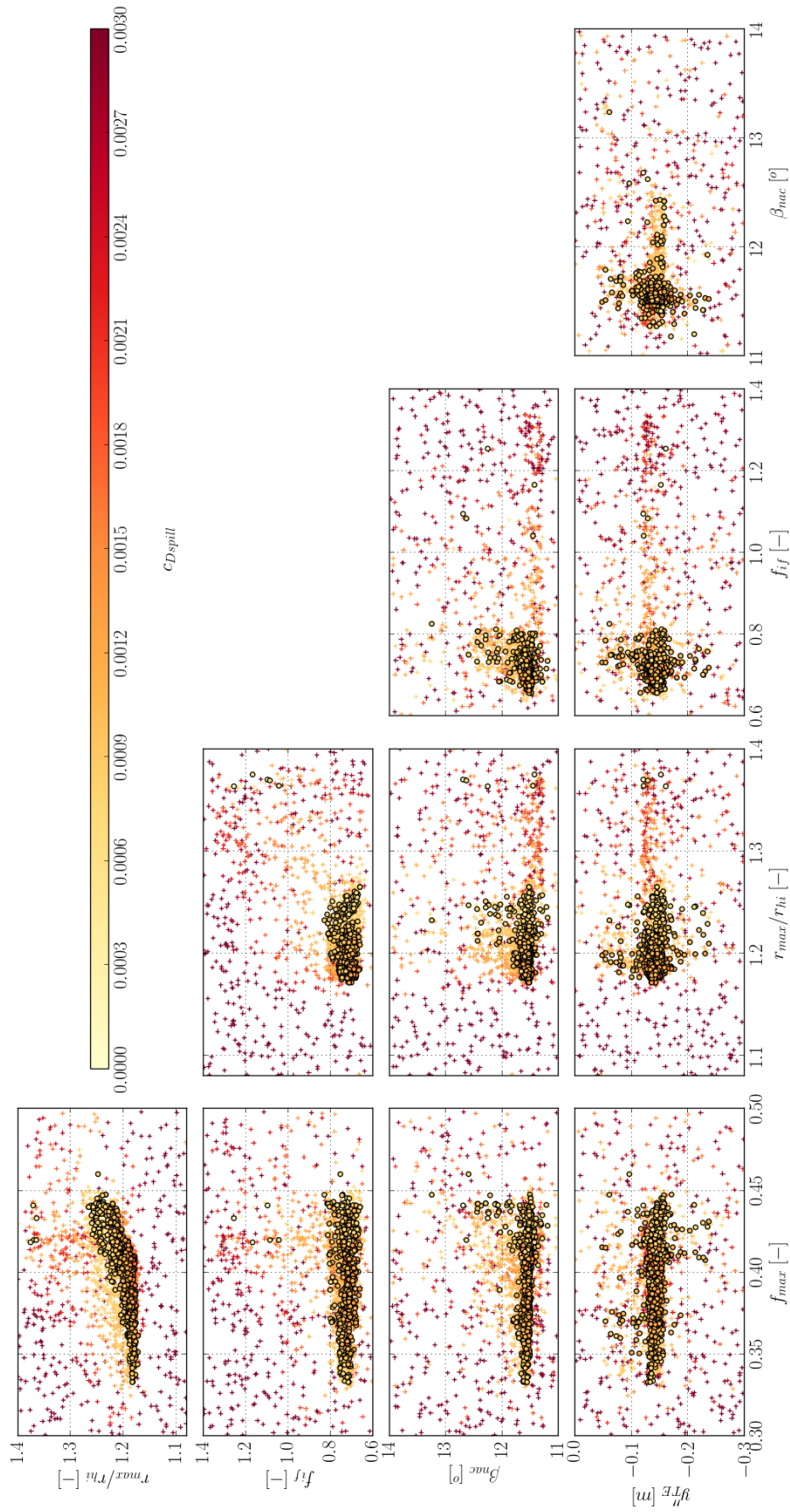


FIGURE 5.31: Pareto set in the design space coloured by MDR

FIGURE 5.32: Pareto set in the design space coloured by C_{Dspill}

5.5.2 Optimisation on Novel Nacelles

An optimisation was carried out for a non-dimensional nacelle length of $l_{nac}/r_{hi} = 3.8$. This represents a relatively long nacelle. For comparison the study by Fang et al⁷⁷ considered a length of roughly $l_{nac}/r_{hi} = 3.28$ ¹¹⁷ to operate at $M_\infty = 0.8$. The GE90-115B1 has a nacelle length of $l_{nac}/r_{hi} = 3.22$ and the GE90-94B has a nacelle length of $l_{nac}/r_{hi} = 3.27$ ¹¹⁷; both of these engines operate at $M_\infty = 0.84$. The PW4000 and CF6-80E1 were both designed to fly at a cruise Mach number of $M_\infty = 0.82$ and had lengths of $l_{nac}/r_{hi} = 3.92$ and $l_{nac}/r_{hi} = 3.38$ respectively.¹¹⁸ In this context, the length of $l_{nac}/r_{hi} = 3.8$ is relatively conservative however it is aimed to operate at a higher Mach number of $M_\infty = 0.85$. The bounds used for this optimisation, summarised in Table 5.5, were chosen to cover a wide range of the available design space and expand beyond conventional design parameters. The engine was designed to cruise at a flight $M_\infty = 0.85$, which once a safety factor³⁶ is considered, the drag rise Mach number should be above $M_{DR} = 0.87$. A constraint was therefore applied of $M_{DR} > 0.85$ to resolve this difficult region of the design space. The hypervolume of the Pareto set was calculated at each generation to determine the level of convergence of the optimisation process. 30 generations were computed and over the final 5 generations there was less than a 1% increase in hypervolume which indicated that the optimisation was converged close to the optimal set.

Variable	Lower Bound	Upper Bound
β	11.0°	14.0°
r_{max}/r_{hi}	1.08	1.4
$f_{max} = l_{fb}/l_{nac}$	0.2	0.5
y''_{TE}	-0.5	0.5
$f_{if} = r_{if}l_{fb}/(r_{max} - r_{hi})^2$	0.6	1.35

TABLE 5.5: Optimisation bounds for nacelle geometric parameters

The improvement from the initial generation, a random sample, resulted in an improvement in M_{DR} from $M_{DR} = 0.8672$ to $M_{DR} = 0.8716$. The mean value of drag ($c_{Dcruise}$) reduced by $\Delta c_{Dcruise} = 0.0009$ from $c_{Dcruise} = 0.313$ to $c_{Dcruise} = 0.304$, however the mean spillage (c_{Dspill}) increased from $c_{Dspill} = 0.0027$ to $c_{Dspill} = 0.0040$ as the optimisation populated the objective space with high drag rise designs. This

indicated that the population increased disproportionately in the higher M_{DR} region or that the initial population was skewed to the lower spillage regions. NSGA-II has been observed to exhibit uneven distributions across a Pareto set in comparison to some similar algorithms¹¹³, however good convergence is still achieved. The minimum spillage drag which met the criterion of $M_{DR} = 0.87$ decreased by $\Delta C_{Dspill} = 0.0011$ from $C_{Dspill} = 0.0016$ to $C_{Dspill} = 0.0005$. The results of this optimisation demonstrate that the higher drag rise Mach numbers are achieved at the expense of the spillage drag performance. Above the required M_{DR} of 0.87, the spillage drag is consistently above 5% of the cruise drag. The lower values are achieved at the expense of higher mid cruise drag (Figure 5.34b).

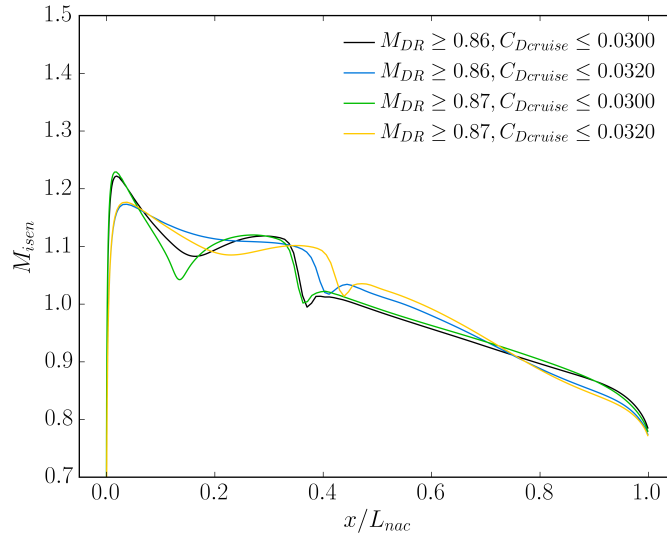


FIGURE 5.33: Example optimal fan cowl Isentropic Mach number profiles at a length of $l_{nac}/r_{hi} = 3.8$ for different required M_{DR} and $C_{Dcruise}$

For example, if a cruise Mach number of $M_\infty = 0.84$ is required then a drag rise Mach number of $M_{DR} = 0.86$ would be needed. The Boeing 777 with GE90 engines operates at this flight Mach number so this may be a comparable scenario. The optimisation process has resulted in designs which meet this criterion (Figure 5.34b) and have generally favourable aerodynamics, i.e. no strong shocks or separation at cruise (Figure 5.33). With the requirement of $M_{DR} \geq 0.86$ there is still the decision of minimising the drag or the spillage. A lower cruise drag can be achieved at the expense of both the spillage (Figure 5.34b) and the Mach number distribution (Figure 5.33). An increase in the acceptable $C_{Dcruise}$ from $C_{Dcruise} \leq 0.0300$ to $C_{Dcruise} \leq 0.0320$ for example, gives optimal designs which resulted in a reduced peak Mach number from $M_{peak} = 1.22$ to $M_{peak} = 1.17$. This gave a concomitant

reduction in spillage drag from $C_{Dspill} = 0.0008$ to $C_{Dspill} = -0.0002$. As the required M_{DR} is increased the minimum levels of spillage which are experienced by the cowls are also increased (Figure 5.34b). An increase in the desired operational Mach number to $M_\infty = 0.85$ and the requisite increase to $M_{DR} = 0.87$ also begins to make the aerodynamics of the possible designs less favourable (Figure 5.33). The aerodynamic behaviour becomes more characterised by multiple accelerations and wave drag during the cruise operation. As at the lower required M_{DR} the decision of an acceptable $C_{Dcruise}$ has a strong influence on the spillage and the pressure distributions. An increase in the acceptable $C_{Dcruise}$ from $C_{Dcruise} \leq 0.0300$ to $C_{Dcruise} \leq 0.0320$ for example resulted in optimal designs with a reduced peak Mach number from $M_{peak} = 1.23$ to $M_{peak} = 1.18$ and a reduction in spillage drag from $C_{Dspill} = 0.0016$ to $C_{Dspill} = 0.0010$.

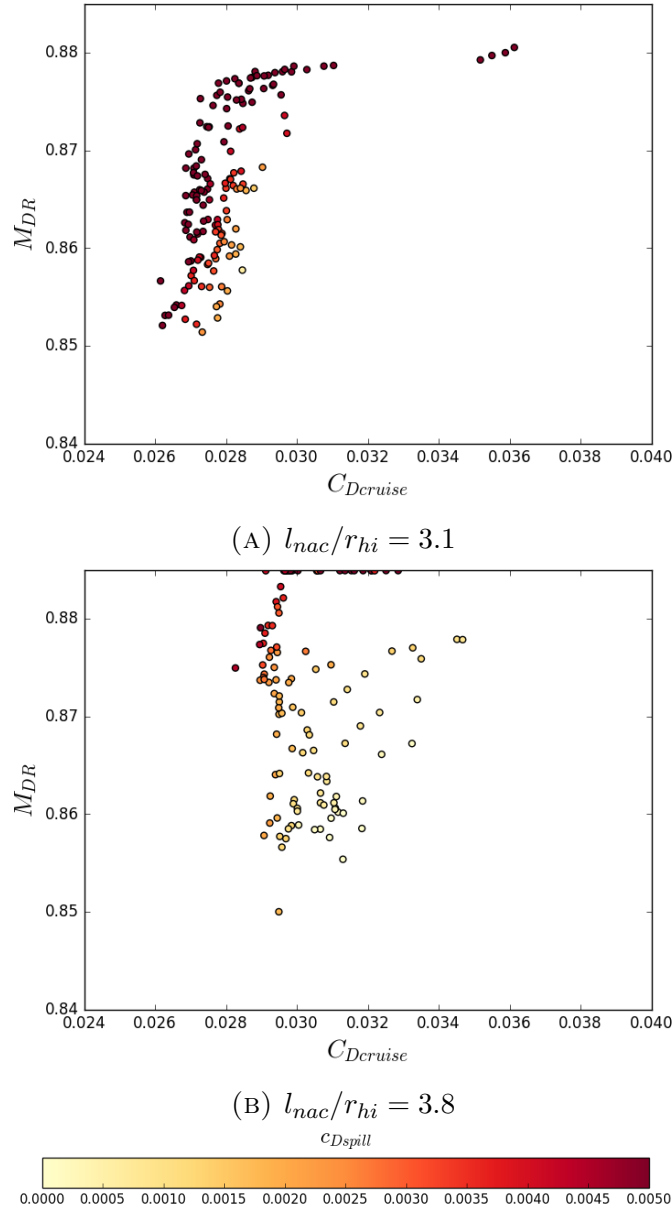


FIGURE 5.34: Pareto optimal sets for optimisations performed at two nacelle lengths, plotted as a function of the performance metrics

The second nacelle length assessed was $l_{nac}/r_{hi} = 3.1$ which is in a more aggressive region of the design space than the previous length $l_{nac}/r_{hi} = 3.8$ and those in the open source. The same bounds were used for all five variables $[\beta, r_{max}, l_{fb}, r_{if}, y''_{TE}]$ as were used for the optimisation of the longer nacelle ($l_{nac}/r_{hi} = 3.8$) (Table 5.5). Again the convergence of the optimisation procedure was monitored by the hypervolume across all generations. Over the final 5 generations there was less than a 0.1% increase in the hypervolume, which again indicated a good level

of convergence from the algorithm. The improvement from the random sample resulted in a $\Delta M_{DR} = 0.004$ higher mean value of M_{DR} with an increase from $M_{DR} = 0.867$ to $M_{DR} = 0.881$. The mean value of cruise drag ($C_{Dcruise}$) reduced by $\Delta C_{Dcruise} = 0.0028$ from $C_{Dcruise} = 0.0311$ to $C_{Dcruise} = 0.0283$, whilst the mean spillage (C_{Dspill}) decreased by $\Delta C_{Dspill} = 0.0048$ from $C_{Dspill} = 0.0101$ to $C_{Dspill} = 0.0053$. No individuals in the initial random design space exceeded a drag rise Mach number of $M_{DR} = 0.87$. However after the optimisation 34% of the Pareto set had the desired drag rise Mach number of $M_{DR} \geq 0.87$ with the minimum spillage at $C_{Dspill} = 0.0040$ for a design which meets the drag rise criterion of $M_{DR} \geq 0.87$. It is clear from the Pareto set (Figure 5.34a) that the optimal designs at this shorter nacelle length ($l_{nac}/r_{hi} = 3.1$) allow for around 10% lower nacelle drag than at the longer length ($l_{nac}/r_{hi} = 3.8$). This is to be expected since there is less wetted area for a shorter nacelle and the skin friction drag of the nacelle typically accounts for 50-60% of the nacelle drag. However, these optimal designs also demonstrate that reaching the higher values of drag rise Mach numbers is more penalising to the spillage drag further emphasising the need for multi-objective optimisation. This also resulted in less desirable pressure distributions on the nacelle surface which produced this additional wave drag at off design conditions. This highlights the increased difficulty in designing a nacelle of shorter length as the spillage drag (C_{Dspill}) is around twice as large at $M_{DR} = 0.87$ for the shorter length ($l_{nac}/r_{hi} = 3.1$) in comparison to the longer nacelle ($l_{nac}/r_{hi} = 3.8$).

5.5.3 Optimisation with directly optimised pressure distributions

The optimisations for the two nacelle lengths were repeated with the limitations on the aerodynamic pressure distributions outlined in (section 3.4.4.3). This provided an additional constraint, limiting the factor $C_{shock} < 0.8$ which prevented the sum of pre-shock c_p from exceeding -0.8, and an additional objective function to minimise C_{infect} . These two additional controls were expected to limit the designs within the final Pareto set to those with weak or no shocks and smoother pressure distributions compared to the uncontrolled data (section 5.5.2). Due to the added objective (C_{infect}) and the additional constraint (C_{shock}), the generation size was increased from 40 to 50. The bounds of the optimisation were kept the same (Table 5.5).

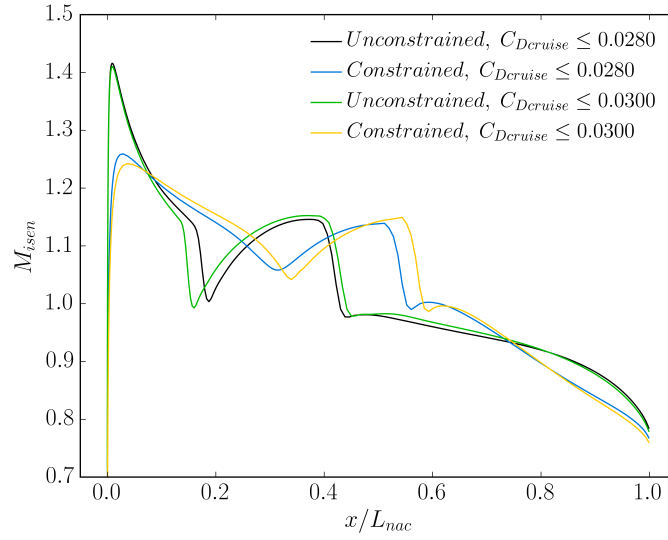
The addition of the controls (C_{shock} , C_{infect}) resulted in higher values of M_{DR} found for the short nacelle (Figure 5.37a) relative to the unconstrained optimisation shown

previously (Figure 5.34a). However, above $M_{DR} = 0.87$, the minimum cruise drag is around 4% higher than the optimisation without the two additional metrics (C_{shock} , $C_{inflect}$). This indicates that the lowest drag designs have undesirable aerodynamic distributions which are mitigated in the more constrained optimisation. There is also an increase in the average drag for the longer nacelle once the constraints are applied (Figure 5.37b) relative to the unconstrained optimisation (Figure 5.34b).

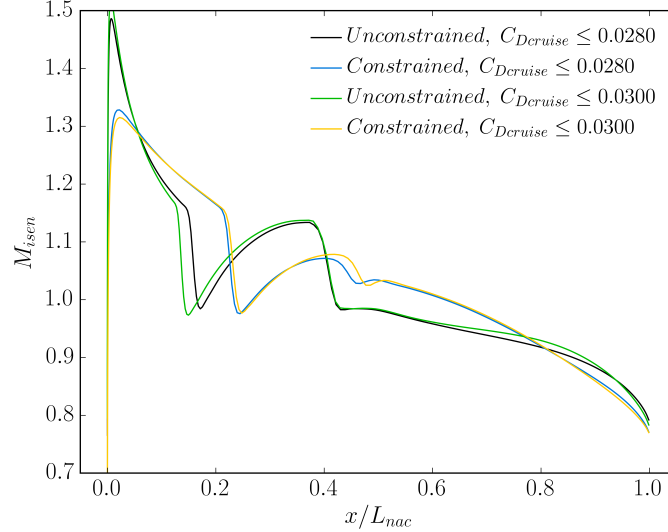
To understand these changes incurred by the addition of the pressure limitations (C_{shock} , $C_{inflect}$) comparable designs were chosen from the unconstrained Pareto sets and the constrained sets for the short nacelle length of $l_{nac}/r_{hi} = 3.1$. These were chosen to meet certain varied requirements, specifically two target drag rise Mach numbers ($M_{DR} \geq 0.86$ and $M_{DR} \geq 0.87$) and two target cruise drag values ($C_{Dcruise} \leq 0.0300$ and $C_{Dcruise} \leq 0.0320$). The nacelle design with the lowest spillage drag which also met these requirements was picked from the two Pareto sets. The controls on the pressure distributions (C_{shock} , $C_{inflect}$) produced, in general, designs with isentropic Mach number distributions that had less severe pressure gradients (Figure 5.35). The introduction of the pressure distribution controls reduced the peak Mach numbers on all four nacelle designs significantly (Figures 5.35a, 5.35b). As a result the sum of the pre-shock Mach numbers was reduced relative to the unconstrained.

At the lower required M_{DR} of $M_{DR} \geq 0.86$ there was either equivalent or better spillage performance. At a required $M_{DR} \geq 0.86$ and $C_{Dcruise} \leq 0.0280$ the spillage decreased from $c_{Dspill} = 0.0025$ to $c_{Dspill} = 0.0017$ through the addition of pressure controls. At a required $M_{DR} \geq 0.86$ and $C_{Dcruise} \leq 0.0300$ the spillage remained unchanged at $c_{Dspill} = 0.0014$. However at the higher required M_{DR} of $M_{DR} \geq 0.87$, whilst the aerodynamics looked improved the spillage was worsened through addition of the pressure controls. Specifically, at a required $M_{DR} \geq 0.87$ and $C_{Dcruise} \leq 0.0280$ the spillage increased from $c_{Dspill} = 0.0050$ to $c_{Dspill} = 0.0070$ through the addition of pressure controls and at $C_{Dcruise} \leq 0.0300$ it increased from $c_{Dspill} = 0.0040$ to $c_{Dspill} = 0.0064$. This is due to the unconstrained optimal designs exhibiting relatively unstable pressure distributions which when subjected to lower MFCRs change shock topology from a double shock pattern to a single shock pattern. This response, whilst providing improved metrics, is not desirable as it likely to result in sudden changes in drag across the operating envelope. This is evidenced in the nacelle drag as a function of freestream Mach number (Figure 5.36). The designs which came from the unconstrained optimisations were far more sensitive to changes in the freestream Mach number whereas the designs from

the constrained optimisations had a monotonic relation between C_{Dnac} and M_∞ . A danger of optimisation, even multi-objective optimisation, is that the resultant designs will be over-designed for the metrics which are given as objectives. The introduction of the pressure controls evidently reduces this somewhat by penalising pressure distributions which aren't smooth.



(A)



(B)

FIGURE 5.35: Example optimal Isentropic Mach number profiles at a length of $l_{nac}/r_{hi} = 3.1$ with (a) a required $M_{DR} \geq 0.86$ and (b) a required $M_{DR} \geq 0.87$

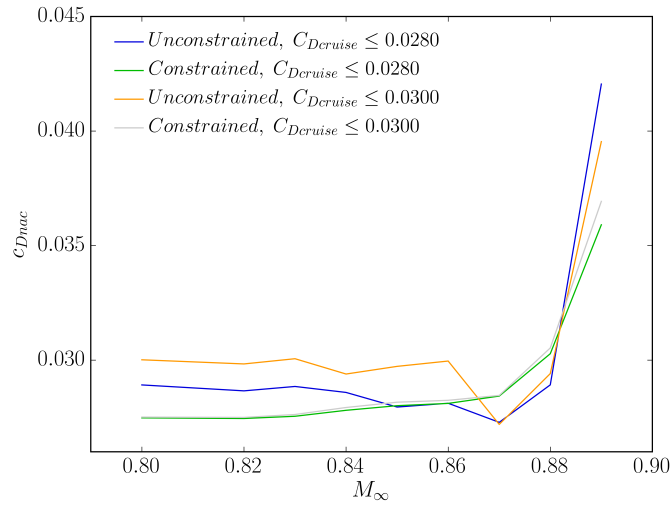


FIGURE 5.36: C_{Dnac} as a function of M_∞ for designs with different $C_{Dcruise}$ criteria for a nacelle length of $l_{nac}/r_{hi} = 3.1$ with and without pressure controls at a required $M_{DR} \geq 0.87$

5.5.4 Influence of length and trailing edge offset on optimal designs

Since the previous analyses failed to produce a nacelle which had good aerodynamics and performance metrics the end points were relaxed somewhat. Since the optimisation starts from a given x and r coordinate for the highlight and trailing edge, the length and radial offset ($r_{TE} - r_{hi}$) are prescribed. To determine the influence of the trailing edge radius on the aerodynamic performance of a nacelle a matrix of three trailing edge radii ($r_{TE}/r_{hi} = 0.911, r_{TE}/r_{hi} = 0.95, r_{TE}/r_{hi} = 1.0$) was optimised for across both lengths. An increase in the trailing edge radius reduces the radial offset which must be achieved by the optimal fan cowl curve.

Each of these optimisations incorporated the c_p controls (C_{shock}, C_{infect}) described in section 3.4.4.3. The outcome of these six optimisations were six Pareto sets (Figure 5.37) which enabled a comparison of the optimal designs available for each length and trailing edge radius. The highest drag rise Mach number was $M_{DR} = 0.885$ which was limited by the range of Mach numbers computed for each case. As before, the designs with the highest drag rise Mach number are also those with the most adverse spillage as the requirement to minimise the frontal facing area of the nacelle to improve the M_{DR} also makes the accelerations around the lip more severe and the spillage worse. However, for both lengths ($l_{nac}/r_{hi} = 3.1$ and $l_{nac}/r_{hi} = 3.8$) the spillage at higher M_{DR} is reduced by an increase in the trailing edge radius.

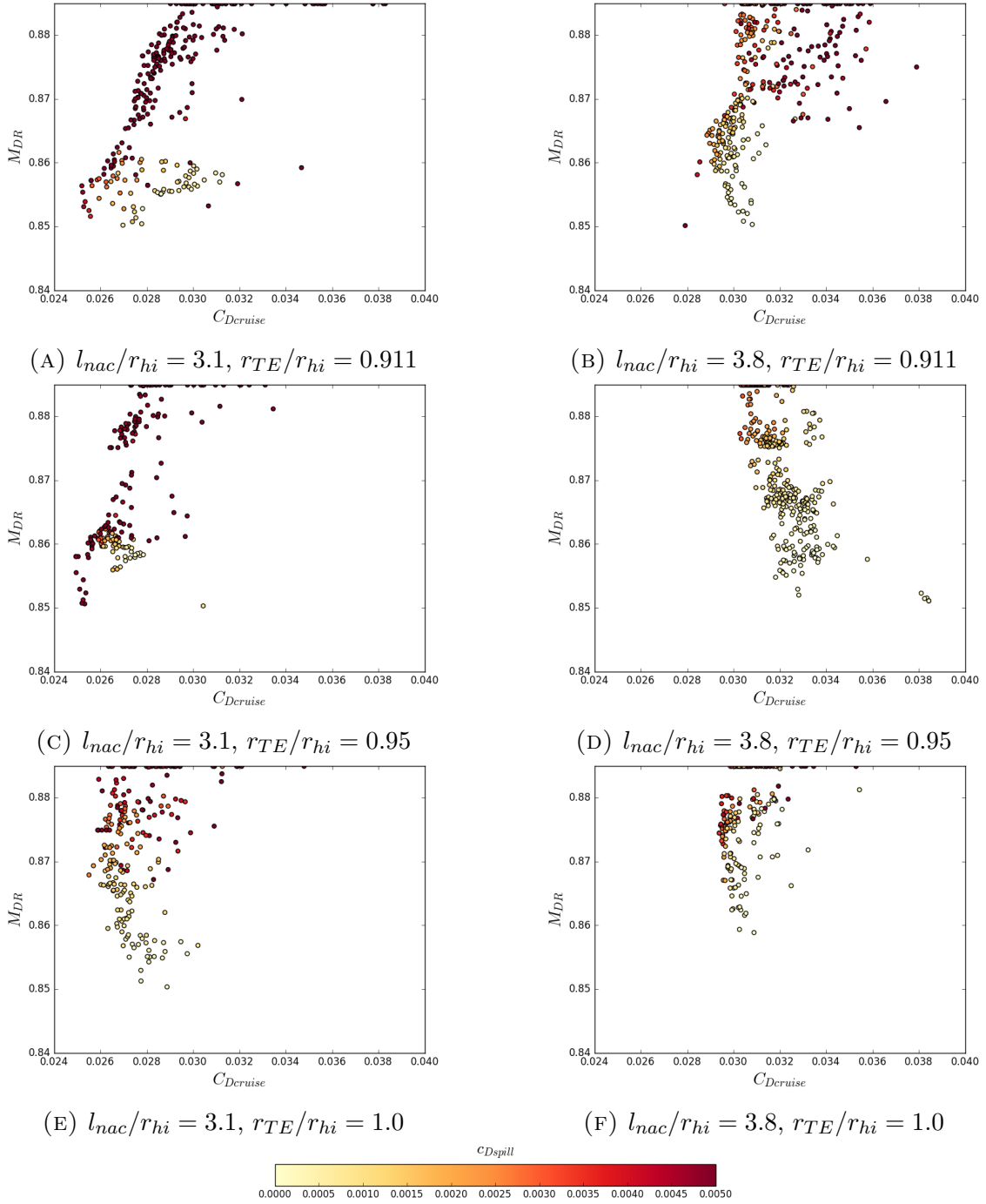


FIGURE 5.37: Pareto optimal sets for optimisations performed at different lengths and trailing edge radii, plotted as a function of the performance metrics

To compare these data it is useful to consider the possible designs available given a particular design requirement, in this case M_{DR} (Figure 5.38). The drag and spillage of the shorter ($l_{nac}/r_{hi} = 3.1$) nacelle rapidly worsens as a higher M_{DR} is targeted

(Figure 5.38). For instance, whilst at a required $M_{DR} = 0.855$ the achievable level of drag for the shortest nacelle ($l_{nac}/r_{hi} = 3.1$), with a low trailing edge radius ($r_{TE}/r_{hi} = 0.911$), is $c_{Dcruise} = 0.0252$ (Figure 5.38a). Conversely, the longer nacelle ($l_{nac}/r_{hi} = 3.8$) with the same trailing edge radius ($r_{TE}/r_{hi} = 0.911$) had a minimum achievable drag of $c_{Dcruise} = 0.0289$. This represents an improvement from a shorter nacelle of $\Delta c_{Dcruise} = -0.0037$. However, if the required drag rise was increased to $M_{DR} = 0.88$ then the improvement in drag available from a shorter nacelle would decrease to $\Delta c_{Dcruise} = -0.0011$ which reflects the increased difficulty in producing a good design for a short nacelle at high Mach numbers. A similar problem is evident in the spillage drag present at different drag rise Mach numbers (Figure 5.38b). For a required $M_{DR} = 0.855$ the minimum spillage which can be attained for both nacelle lengths is below $c_{Dspill} = 0.0001$. However the performance of the short nacelles worsens greatly at the higher drag rise Mach numbers and by $M_{DR} = 0.88$ the minimum spillage from the short nacelle is $c_{Dspill} = 0.0070$ which would represent around a quarter of the cruise drag. The longer nacelle on the other hand has a less detrimental response to an increase in the required M_{DR} . As a result the minimum spillage at $M_{DR} = 0.88$ is $c_{Dspill} = 0.0016$ which indicates the relative ease of design. The influence of the trailing edge radius is also evident in these trends, most notably for the short nacelle. For these short optimisations the lowest trailing edge radius ($r_{TE}/r_{hi} = 0.911$) resulted in the highest drag and spillage at $M_{DR} = 0.88$. This represented a substantial rise from the minimum value at $M_{DR} = 0.855$. With an increase in the trailing edge radius to $r_{TE}/r_{hi} = 1.0$ the gradient of minimum drag against M_{DR} decreases which results in a much improved drag at $M_{DR} = 0.88$ relative to the lower trailing edge $r_{TE}/r_{hi} = 0.911$. Specifically this increase in trailing edge radius allows a minimum drag of $c_{Dcruise} = 0.0259$ at M_{DR} compared to $c_{Dcruise} = 0.0285$ for the lower trailing edge radius.

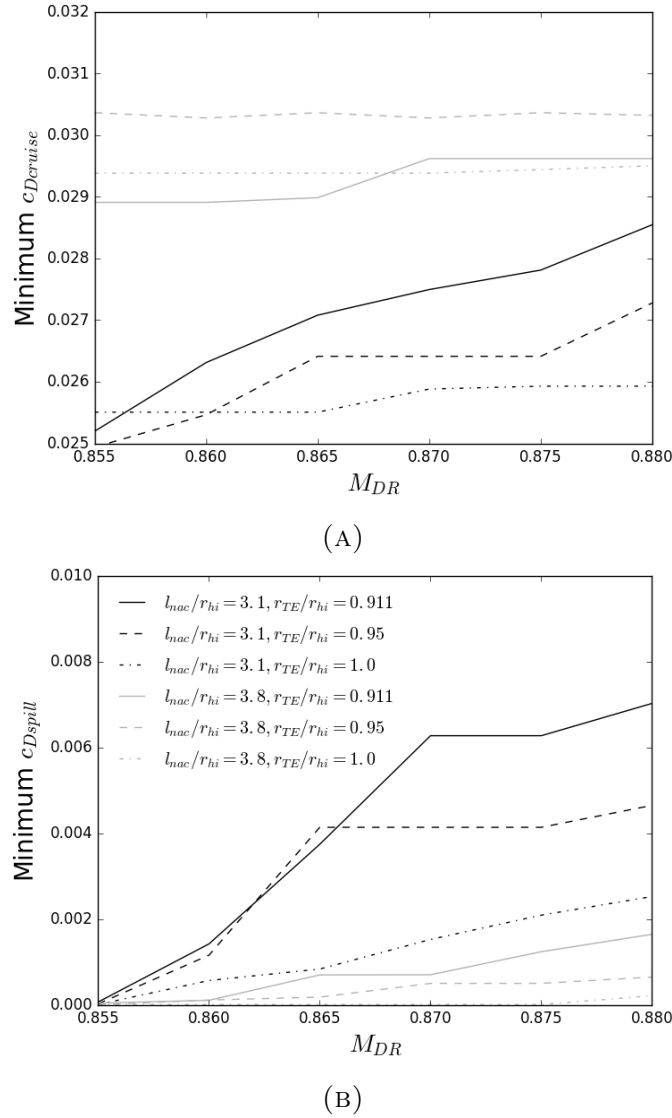


FIGURE 5.38: Minimum achievable (a) $c_{Dcruise}$ and (b) c_{Dspill} for a given M_{DR}

The reason for the different performances across the range of lengths and trailing edge radii can be explained through the aerodynamic loadings on the cowl (Figure 5.39). Six designs were chosen from the Pareto sets which all met the criterion of $M_{DR} > 0.87$. The shortest nacelle with the lowest trailing edge had the highest peak Mach number of $M_{peak} = 1.33$. This high peak Mach number results in a stronger shock and additional wave drag. As the MFCR is decreased from MFCR=0.7 and MFCR=0.65 this peak Mach number increases from $M = 1.33$ to $M = 1.46$ and this results in the high spillage value of $c_{Dspill} = 0.0069$. An increase in the length improves the isentropic Mach distribution which can be achieved (Figure 5.39b). A change in the length from $l_{nac}/r_{hi} = 3.1$ to $l_{nac}/r_{hi} = 3.8$ reduced the peak Mach number of the optimal design by $\Delta M = 0.09$ from $M = 1.33$ to $M = 1.24$. This

resulted in less wave drag during cruise and less spillage, $c_{Dspill} = 0.0015$ compared to $c_{Dspill} = 0.0069$.

An alternative to lengthening the nacelle would be to increase the trailing edge radius which also improves the peak Mach number. For the shortest nacelle length investigated ($l_{nac}/r_{hi} = 3.1$), the improvement in the Mach number distribution from an increase in the trailing edge radius is evident (Figure 5.39a). In addition to the removal of the double shock pattern by increasing from $r_{TE}/r_{hi} = 0.911$ to $r_{TE}/r_{hi} = 1.0$, the peak Mach number was reduced by $\Delta M = 0.12$ from $M = 1.33$ to $M = 1.21$. In addition, the nacelle design with $r_{TE}/r_{hi} = 1.0$ had $\Delta c_{Dspill} = -0.0053$ (-76%) less spillage and $\Delta c_{Dcruise} = -0.0016$ (-6%) less cruise drag, compared to the nacelle design with $r_{TE}/r_{hi} = 0.911$, whilst still meeting the drag rise criterion of $M_{DR} > 0.87$. A similar influence of trailing edge radius was observed for the longer nacelle (Figure 5.39b) where the peak Mach number was reduced from $M = 1.24$ to 1.13 by an increase in trailing edge radius from $r_{TE}/r_{hi} = 0.911$ to $r_{TE}/r_{hi} = 1.0$.

5.6 Analysis of design parameters

The trends in the design space to generate optimal nacelles for different lengths and trailing edge radii were assessed for each of the optimal sets in terms of each degree of freedom (Figure 5.40). Several key aspects can be determined from these trends which demonstrate the differences in the design approach which could be taken for UHBPR engine nacelles. The longer nacelle designs consistently have larger r_{max}/r_{hi} in the optimal sets, with higher median values for all three trailing edge radii (Figure 5.40a). In addition an increase in the trailing edge radius resulted in an increase of the mean value for r_{max}/r_{hi} for the short nacelle, which indicates that as the trailing edge radius is reduced the profile is reduced to avoid thick, short designs. Such a trend is less evident for the longer nacelle. In addition to a higher r_{max}/r_{hi} , there was a trend for higher f_{max} as the trailing edge radius was increased (Figure 5.40c). The different average values for the design metrics result in different iCST shapes produced. These iCST curves each have a curvature distribution which has been through the solution of the linear set of equations which define the iCST curve (section 3.2.2). The curvature (κ) distributions which define the fan cowl geometry have a first order influence on the aerodynamics since they determine the amount of local diffusion and compression around the nacelle. The curvature

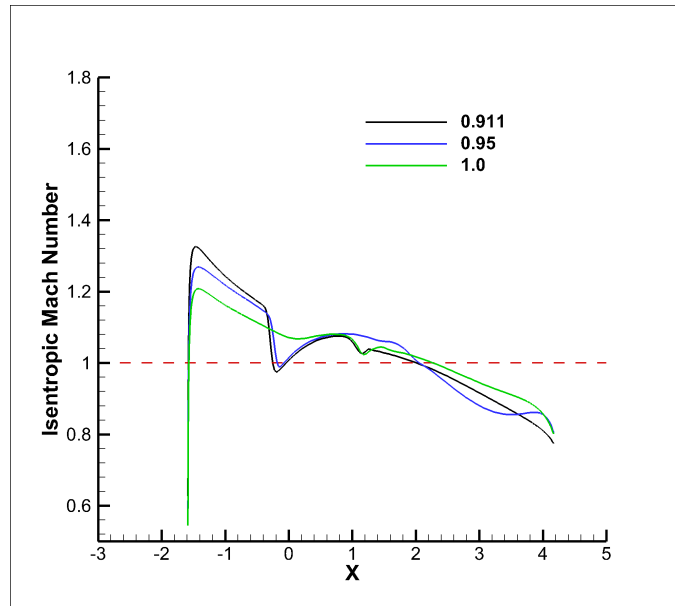
distributions of the optimal nacelles which were chosen from the Pareto fronts all exhibit some similar features (Figure 5.41). For instance the forebody curvatures of all six designs are monotonic (Figure 5.41a). The peak acceleration will always occur around the lip and all six of the chosen designs exhibit a peak Mach number just after this. The monotonic curvature distribution which follows this allows a smooth deceleration. This is important in cruise and also in an off design conditions where it is also preferable to avoid a shock too close to the position of maximum Mach number. To achieve a similar profile of curvature for three different trailing edge radii whilst avoiding an inflecting afterbody curve the maximum radius is increased and moved axially rearwards. Five of the six Pareto sets have an average f_{if} of around 0.85 except for the optimal set for the shortest nacelle with the lowest trailing edge radius where on average $f_{if} = 0.95$. Non-dimensionally this makes a blunter nacelle, however since the radius of curvature is non-dimensionalised by $l_{fb}/(r_{max} - r_{hi})^2$ and this value is generally higher for the shorter nacelles the radius of curvature is similar to the five less aggressive designs.

The highest peaks in isentropic Mach number (Figure 5.39) correlate with the highest rates of change in curvature (Figure 5.41b). Subsequent to the peak in $\Delta\kappa/\Delta x$ there is a steady decrease which for all six chosen designs reduced the curvature to less than $\kappa = 0.1m^{-1}$ (Figure 5.41a). The aft 80% of the nacelle is characterised by small changes in the curvature which mostly remain below $\kappa = 0.1m^{-1}$. In addition the magnitude of the rate of change of curvature remains below $0.1m^{-2}$ for most of the afterbody until around the last 10% of the cowl.

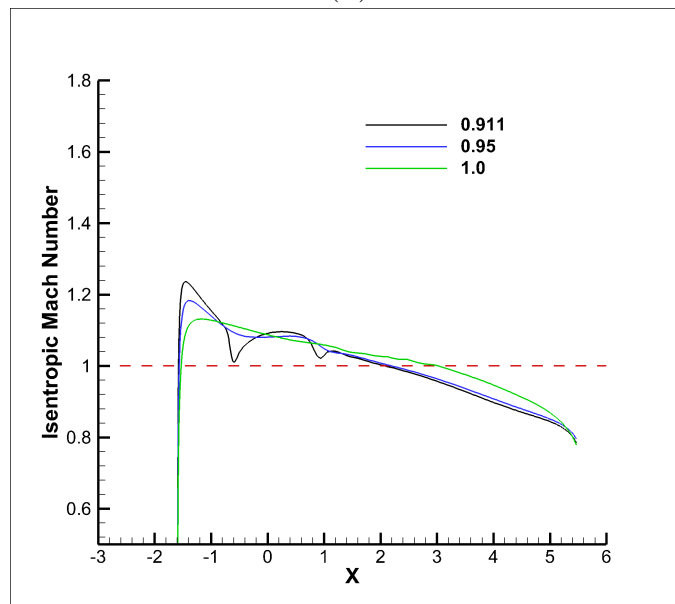
The boattail angle tended to increase with a greater trailing edge radius for the shorter nacelle whilst the opposite trend was observed for the longer nacelle (Figure 5.40d). There were no clear trends in the trailing edge second derivative as a function of the length or trailing edge radius.

5.7 Influence of Incidence on Aerodynamic Performance

An aero-engine is unlikely to operate at zero degrees incidence at the cruise condition. The aircraft will cruise at an incidence to generate the requisite lift for level flight. In addition to this the engine may be installed on the wing at an angle of incidence to aid the installation. Finally, the circulation around the wing will provide



(A)



(B)

FIGURE 5.39: Example optimal Isentropic Mach number profiles for all trailing edge radii at a length of (a) $l_{nac}/r_{hi} = 3.1$ and (b) $l_{nac}/r_{hi} = 3.8$

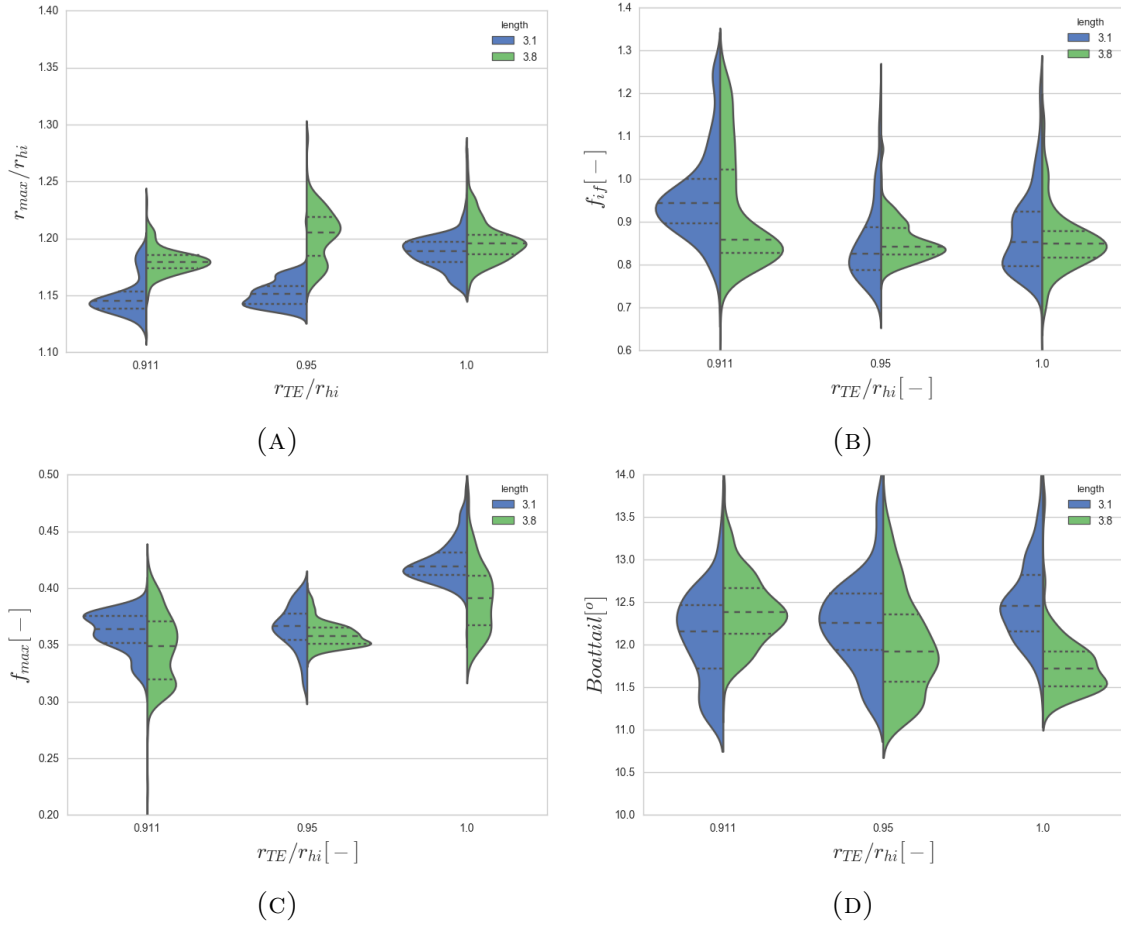


FIGURE 5.40: Pareto optimal sets for optimisations performed at different lengths and trailing edge radii, plotted as a function of the design metrics. Dashed lines indicate median value and dotted lines indicate the quartiles. Number of individuals are indicated by the relative thickness of the distributions

an upstream upwash which the engine nacelle will probably be influenced by. Due to these reasons it is important to assess how well an engine nacelle performs at incidence. In this study the baseline E2 engine was simulated at the cruise altitude ($h = 10668m$) and operating condition ($M_\infty = 0.82$, $MFCR = 0.75$) between and incidence of $\alpha = 0^\circ$ and $\alpha = 8^\circ$.

A baseline performance of a nacelle as a function of incidence is demonstrated in the E2 nacelle (Figure 5.43). As the incidence of the nacelle is increased the streamtube of air entering it shifts down from the highlight plane. As a result the local stagnation point on the topline ($\theta = 0^\circ$) moves further into the intake and the stagnation point on the bottomline ($\theta = 180^\circ$) moves onto the outside of the fan cowl. Due to this movement of the streamtube around the leading edge of the nacelle, the acceleration on the topline will be greater at high incidence as a result of

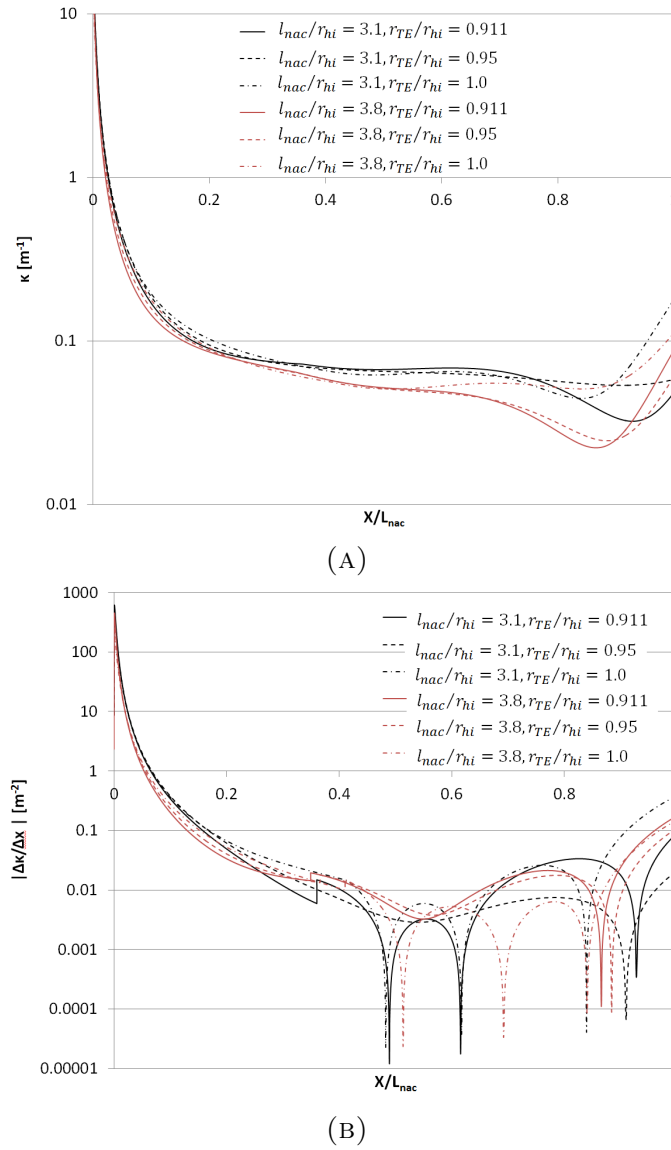


FIGURE 5.41: (a) Curvature distribution and (b) Rate of change of curvature distribution for six chosen optimal designs for different lengths and trailing edge radii

the air curving further around the leading edge. The distributions of isentropic Mach number as a function of incidence (Figure 5.43) therefore shows that from $\alpha_{CL} = 0^\circ$, where the three azimuthal lines all have the same distribution of isentropic Mach number, to $\alpha_{CL} = 8^\circ$ there is a divergence of the three azimuthal lines in terms of isentropic Mach number. For the E2 nacelle at $M_\infty = 0.82$ the topline operates at higher incidence as the centreline incidence increases and this results in the peak Mach number increasing from $M_{peak} = 1.10$ to $M_{peak} = 1.57$ between $\alpha_{CL} = 0^\circ$ and $\alpha_{CL} = 10^\circ$. The bottomline however experiences the opposite trend as the local incidence for the fan cowl decreases, which decreases the peak Mach number.

Across the three axisymmetric designs simulated in this incidence range the worst performing was the shortest nacelle ($E3 \ l_{nac}/r_{hi} = 3.1$) (Figure 5.43). The influence of incidence on this shortest nacelle rapidly strengthened the shock on the topline of the nacelle significantly by $\alpha_{CL} = 8^\circ$. At $\alpha_{CL} = 8^\circ$ the peak Mach number is $M_{peak} = 1.65$ which results in a strong shock. The increase in topline peak Mach number has a positive linear relationship with the incidence for all three engines (Figure 5.44). The gradient of these linear correlations vary however between the 3 engines with the shortest nacelle ($E3 \ l_{nac}/r_{hi} = 3.1$) having the highest gradient and the longest nacelle ($E2$) having the lowest. The gradient of these linear correlations also increase with the peak Mach number of the nacelle at $\alpha_{CL} = 0^\circ$, i.e. a design with higher baseline peak Mach number has a more severe response to incidence in terms of peak Mach number. The peak Mach number of the sideline is relatively independent of the incidence (Figure 5.44).

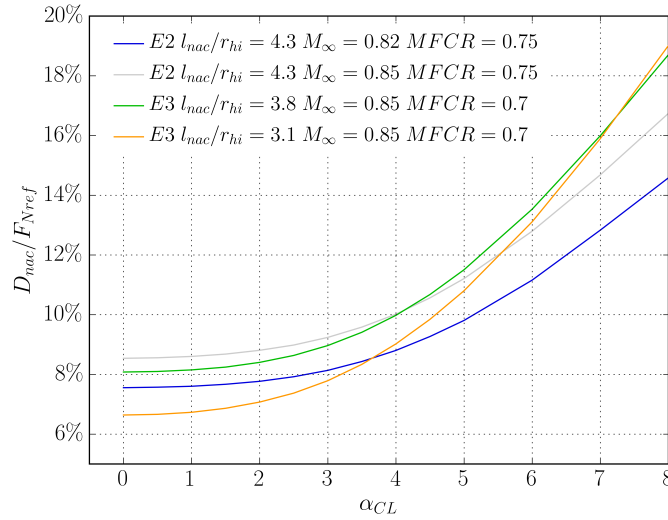


FIGURE 5.42: C_{Dnac}/F_{Nref} as a function of α_{CL} for three different nacelle designs

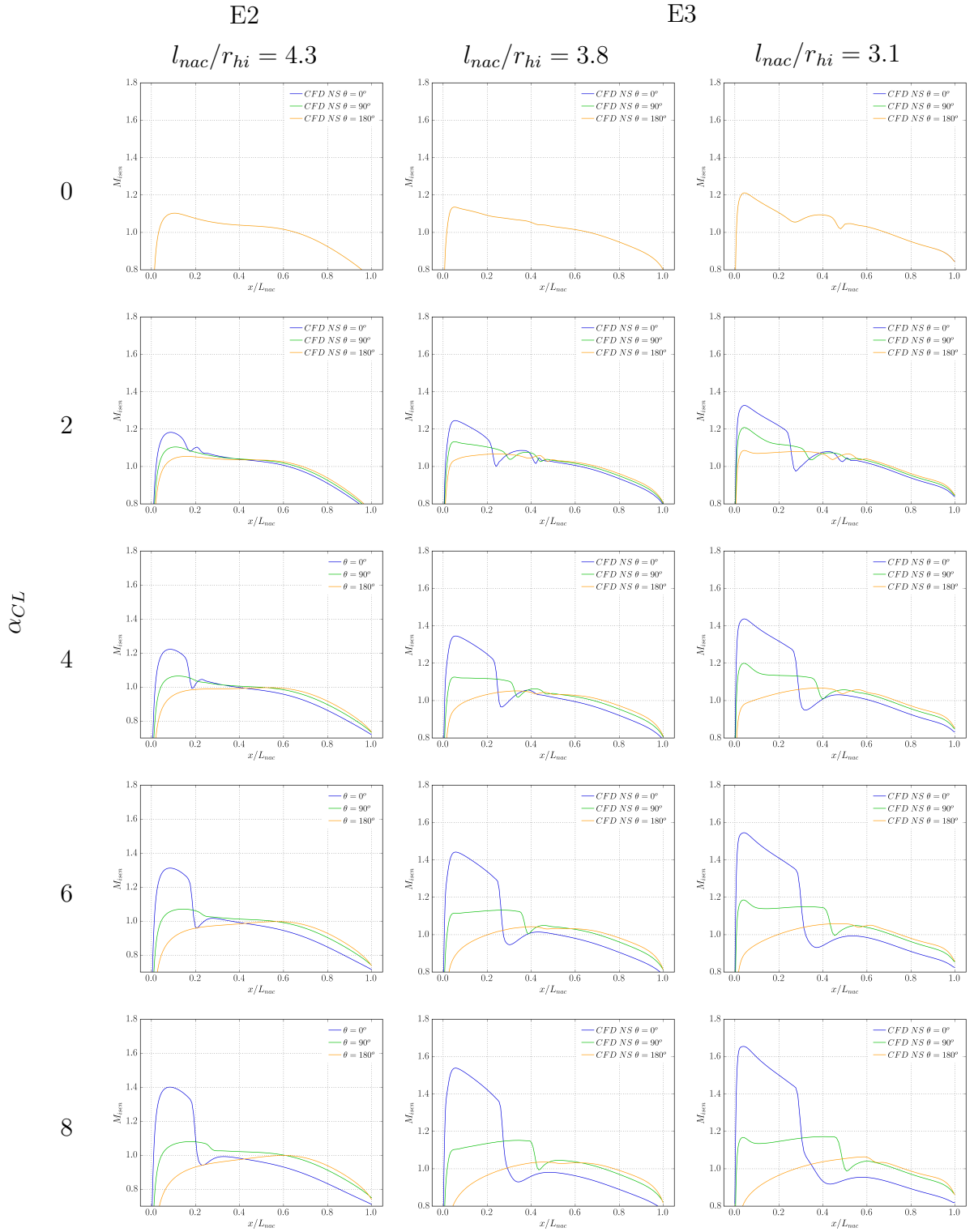


FIGURE 5.43: Isentropic Mach number distributions for three different engine designs as a function of incidence (α_{CL}) in the cruise condition, E2: $M_{\infty} = 0.82$, MFCR = 0.75; E3: $M_{\infty} = 0.852$, MFCR = 0.7

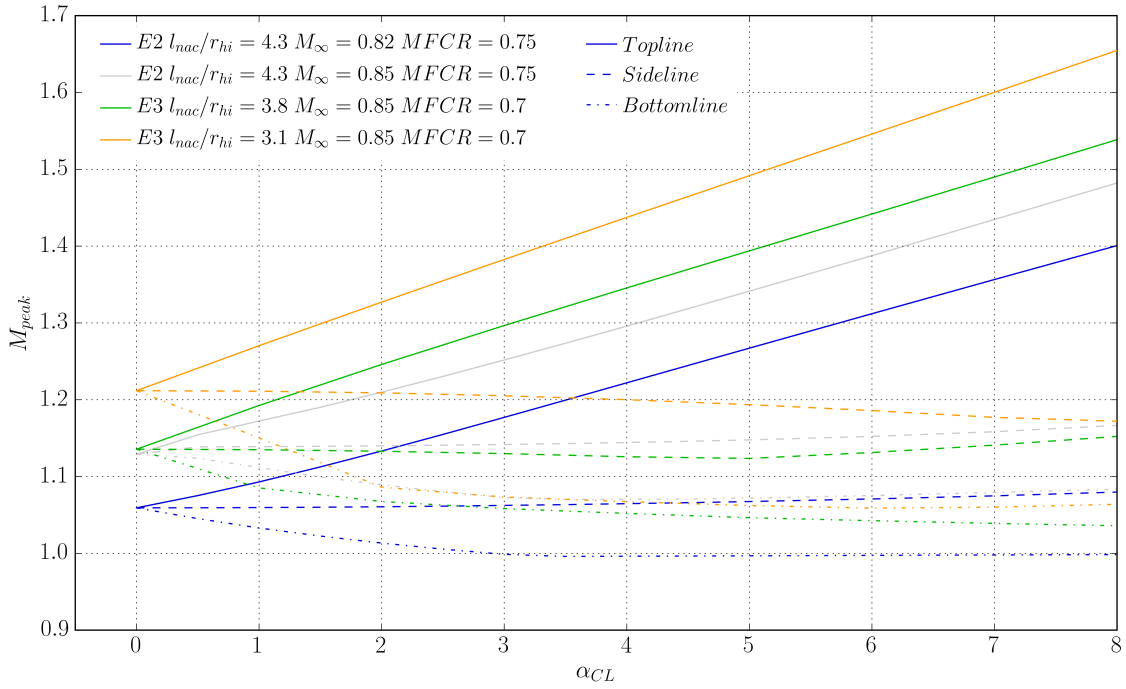


FIGURE 5.44: M_{peak} as a function of α_{CL} for three different nacelle designs

5.8 Influence of Windmill on Aerodynamic Performance

Another concern for aggressive nacelle designs is how they will perform under extreme off design conditions. It has already been demonstrated that the influence of MFCR can be greater for a nacelle with a lower value of l_{nac}/r_{hi} . Another condition which relates to low MFCR is that of a windmilling engine. Under this condition an engine has failed but the fan is free to rotate. As such, the passage of air through the engine results in drag from friction losses and the energy extracted to rotate the engine. It is important to minimise the drag in this condition since in the critical engine out scenario the second engine will likely be at a higher than normal power setting and a large yaw moment will be produced. To maintain static stability, the vertical stabiliser will be sized under these conditions. For instance, a previous study identified that an increase in fan diameter from 2.84m to 3.78m on the B777-200ER at $M_\infty = 0.2$, similar to a take-off condition, resulted in an increase of windmill drag of over 150% and an increase in the tail size of 15% was required to balance this yaw moment.⁴⁷

Since the influence of MFCR is greater on a shorter nacelle it is likely that meeting a given criterion with a windmilling engine will be more difficult. An engine failure at altitude with a diversion mission profile was considered a 2D axisymmetric problem with zero incidence across a range of Mach numbers ($M_\infty = [0.2, 0.85]$) and altitudes ($h = [0m, 10668m]$). In a diversion due to engine failure an aircraft will need to maximise its range by flying at the maximum L/D ratio. It is important to determine what influence a novel nacelle design might have on the optimal L/D. Conversely, engine failure at the end of runway was considered with incidence at a constant Mach number and altitude. Four engines, the baseline E1 and E2 engines and optimal designs for the E3 engine at $l_{nac}/r_{hi} = 3.1$, $r_{TE}/r_{hi} = 1.0$ and $l_{nac}/r_{hi} = 3.8$, $r_{TE}/r_{hi} = 1.0$, were simulated across all of the diversion conditions.

Representative mass flows for the windmill condition were calculated from a method outlined by ESDU¹³ (Equation 5.1, 5.2) which is based on a curve fit (Equation 5.3) as a function of freestream Mach number (M_∞) and maximum sea level thrust divided by the weight of the air flow ($w = mg$) (Equation 5.4) at maximum sea level thrust. The mass flow was a function of the freestream conditions and thus the altitude however since the MFCR is non-dimensional this is constant for a given engine cycle and freestream Mach number (Figure 5.45). The engine with the highest specific thrust (E1) therefore had the lowest MFCR in the windmill conditions. However, since E1 is a smaller engine overall with a very conservative nacelle design it is expected to be less susceptible to spillage than the E3 engines.

Across the entire range of conditions the E1 engine experienced the lowest nacelle drag (Figure 5.46). For instance at $M_\infty = 0.85$ and $h = 10668m$ the E1 drag is 8.4% of F_{Nref} whilst the larger E2 engine had drag of 10.6% of F_{Nref} . This increase in drag from E1 to E2 would be greater if the engine had been simply scaled since the reduced specific thrust of E2 increased the windmill MFCR and thereby reduced the spillage. The E3 engines have the worst external windmill at $M_\infty = 0.85$, $h = 10668m$. The longer E3 nacelle ($l_{nac}/r_{hi} = 3.8$) drag was 14.2% of F_{Nref} whilst the shorter E3 nacelle ($l_{nac}/r_{hi} = 3.1$) had drag of 22.2% of F_{Nref} . This significant increase in windmill drag at the cruise Mach number is caused by the presence of shock waves on the E3 nacelles. In cruise, where $MFCR = 0.7$, the shorter E3 nacelle has a peak Mach number of $M_{peak} = 1.21$ but once the MFCR is reduced to the windmill value of $MFCR = 0.347$ the peak Mach number increases to $M_{peak} = 1.91$. This is a trait of its aggressive design which features high accelerations due to the high curvature changes. Conversely for the E1 engine the increase due to windmill is from $M_{peak} = 1.12$ to $M_{peak} = 1.47$. At freestream Mach numbers below $M_\infty \leq 0.5$

the shorter E3 nacelle has marginally lower drag than the E2 nacelle at all altitudes (Figure 5.46). This has implications for the diversion planning since this will usually be carried out at lower than cruise Mach numbers. However, the shortest nacelle also encountered wave drag at the lowest freestream Mach number, slightly above $M_\infty = 0.5$. As a result of this increase in wave drag above $M_\infty = 0.5$ the M_∞ at which the minimum drag is encountered is at $M_\infty = 0.45$ (Figure 5.47d). This is significantly lower than for the other designs. For instance the longer nacelle of $l_{nac}/r_{hi} = 3.8$ encountered minimum drag at $M_\infty = 0.55$ (Figure 5.47c) whilst the conventional nacelle had a minimum drag at $M_\infty = 0.65$ (Figure 5.47b). This will have an influence on the overall aircraft L/D as a function of M_∞ and may ultimately result in a lower speed being optimal for the diversion. As an example, the Airbus A350-XWB airliner was certified for an ETOPS (extended operations) time of 370 minutes by the EASA with a diversion range of 2500 nautical miles under standard atmospheric conditions.¹¹⁹ This corresponds to a diversion velocity of $U = 209 \text{ ms}^{-1}$ and dependent on the diversion altitude the aircraft can be assumed to fly between $M_\infty = 0.6$ and $M_\infty = 0.7$. Assuming a diversion altitude of $h = 6706\text{m}$, the flight Mach number would be $M_\infty = 0.66$. This would be plausible with the E1 or E2 nacelles which would be close to the Mach number of minimum drag (Figures 5.47a, 5.47b). However, at this altitude of $h = 6706\text{m}$, to operate the longer E3 nacelle ($l_{nac}/r_{hi} = 3.8$) at the optimal Mach number for drag would reduce the diversion range from 4630km (2500nm) to 3828km (2067nm). This worsens for the shorter E3 nacelle ($l_{nac}/r_{hi} = 3.1$) where the range is reduced to 3132km (1691nm) based on the optimal flight Mach number of $M_\infty = 0.45$. These reductions in range would ultimately dictate the acceptable routes for long-haul flights. At $M_\infty = 0.6$, the external drag of the shortest nacelle ($l_{nac}/r_{hi} = 3.1$) was estimated to be around 50% of the internal drag predicted by an existing method.⁴² This indicates that the external drag is a considerable component of the overall windmill drag and should be considered for compact nacelles.

$$m = \frac{\mathcal{A}A_{fan}P_\infty f(M_\infty)}{\sqrt{RT_\infty}} \quad (5.1)$$

$$f(M) = \frac{\gamma M_\infty}{\left[1 + \frac{\gamma-1}{2} M_\infty^2\right]^{\frac{\gamma+1}{2(\gamma-1)}}} \quad (5.2)$$

$$\begin{aligned}
\mathcal{A} = & 0.4825866 - 1.553662 \times 10^{-3}\zeta - 0.1794865 \times 10^{-3}\zeta^2 + 1.450817 \times 10^{-6}\zeta^3 \\
& + 0.4699035M_\infty - 0.03533626M_\infty\zeta + 0.9184379 \times 10^{-3}M_\infty\zeta^2 - 6.034876 \times 10^{-6}M_\infty\zeta^3 \\
& - 0.9263036M_\infty^2 + 0.0565954M_\infty^2\zeta - 1.364252 \times 10^{-3}M_\infty^2\zeta^2 + 9.174115 \times 10^{-6}M_\infty^2\zeta^3 \\
& + 0.5980928M_\infty^3 - 0.0276658M_\infty^3\zeta + 0.60149 \times 10^{-3}M_\infty^3\zeta^2 - 4.05015 \times 10^{-6}M_\infty^3\zeta^3
\end{aligned} \tag{5.3}$$

$$\zeta = [F_N/w]_{s,0} \tag{5.4}$$

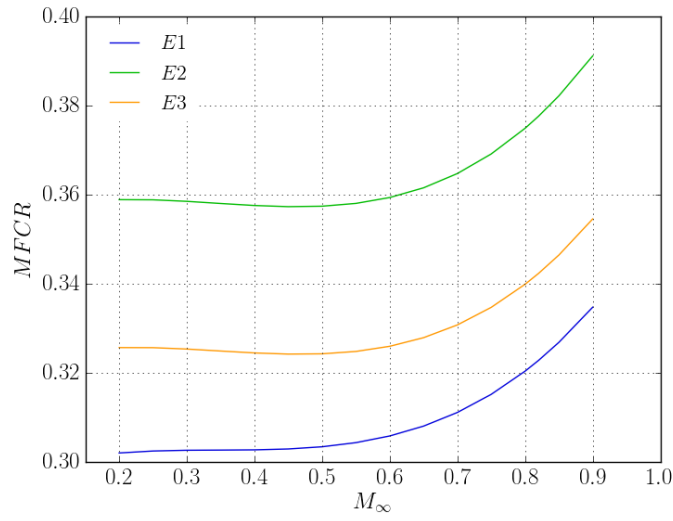


FIGURE 5.45: MFCR as a function of M_∞ for three engine cycles in the windmill condition based on ESDU data¹³

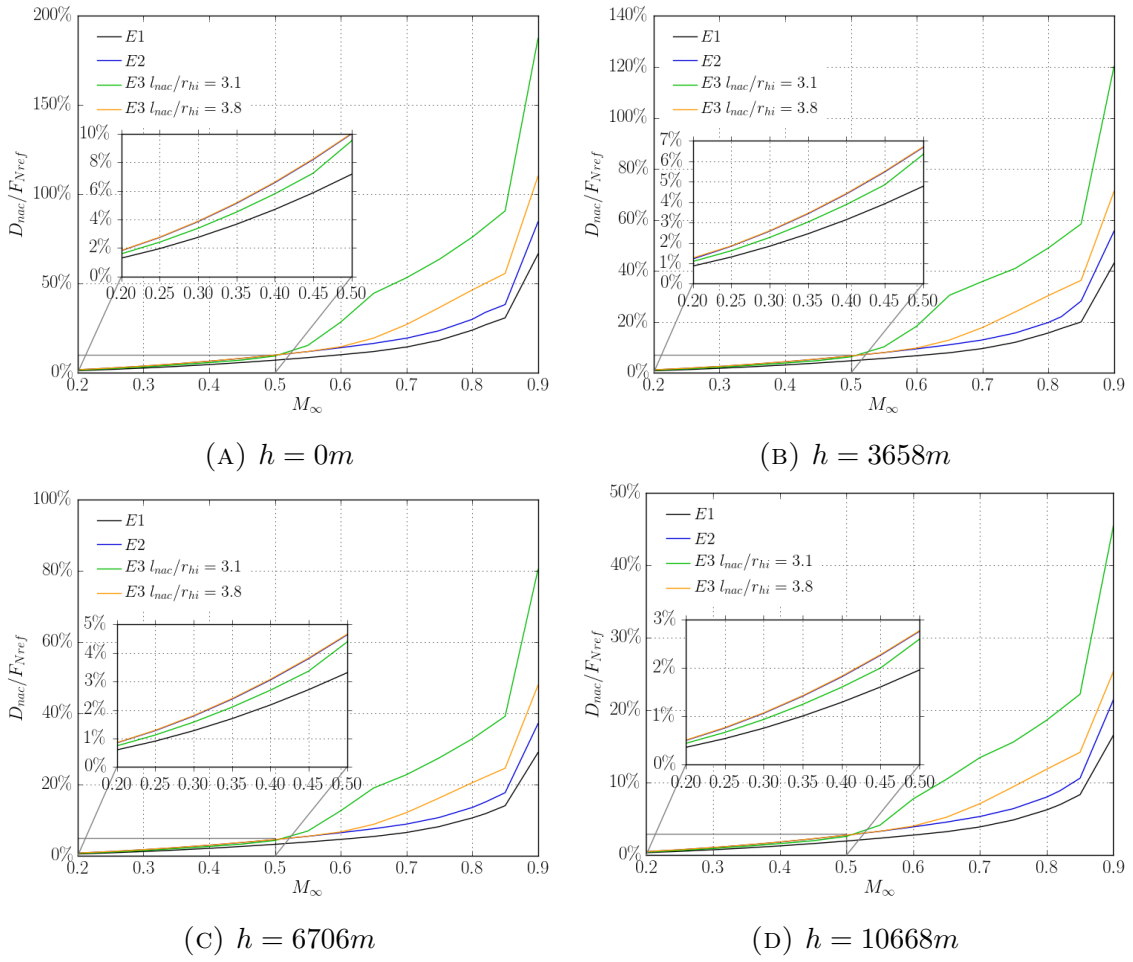


FIGURE 5.46: Influence of M_∞ and h on semi-dimensional windmill drag (c_{DA}) for four different engine designs

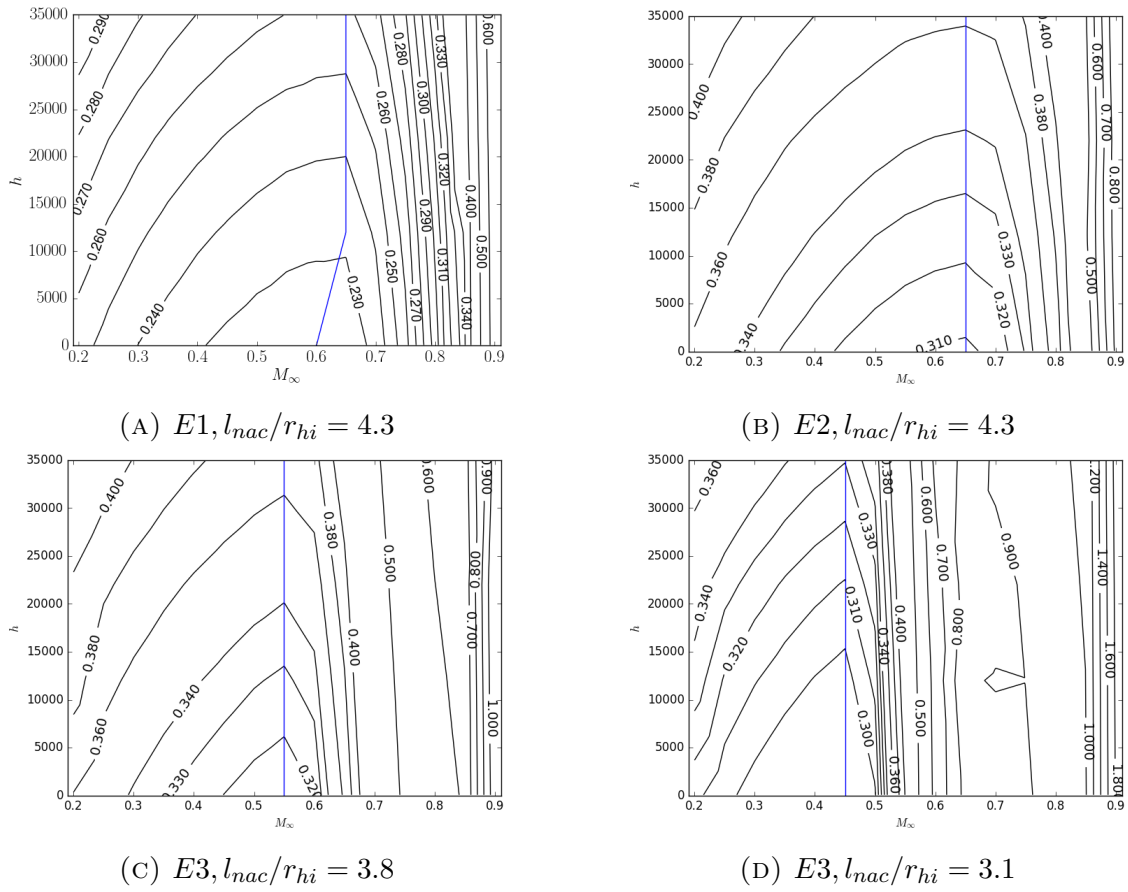


FIGURE 5.47: Influence of M_∞ and h on semi-dimensional windmill drag ($c_D A$) for four different engine models

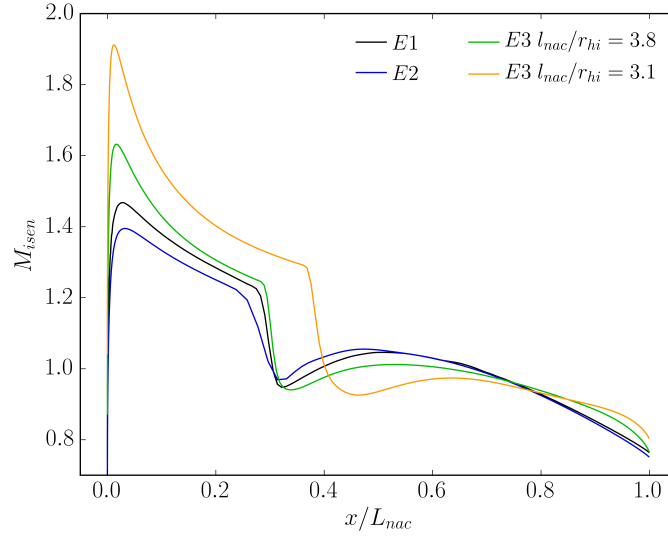
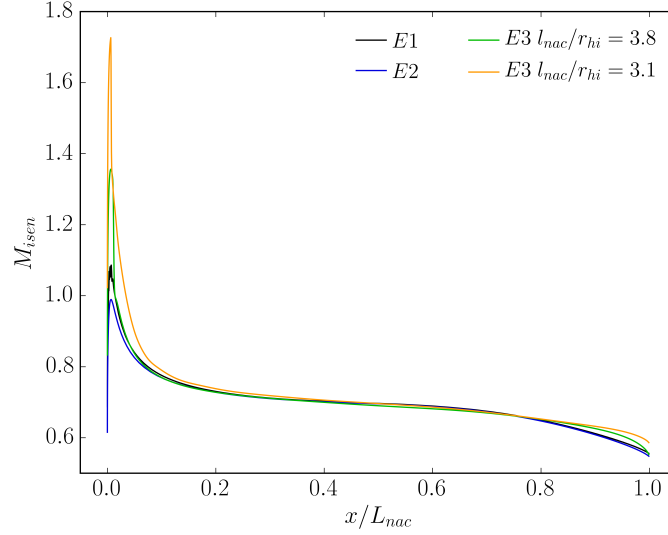
(A) $M_\infty = 0.85$, $h = 10668m$ (B) $M_\infty = 0.6$, $h = 6706m$

FIGURE 5.48: Isentropic Mach distributions for four engine designs under the windmill condition

An incidence study was performed for an end of runway condition for the E2 nacelle and the two optimised E3 nacelles discussed in the previous paragraph ($l_{nac}/r_{hi} = 3.8$, $r_{TE}/r_{hi} = 1.0$ and $l_{nac}/r_{hi} = 3.1$, $r_{TE}/r_{hi} = 1.0$). This analysis was performed at $h = 0m$, $M_\infty = 0.25$ and between $\alpha = 0^\circ$ and $\alpha = 30^\circ$. At this Mach number and altitude the drag performance will not be dominated by wave drag as in the cruise comparisons but due to the high incidences separation may become a problem. As the incidence increases the pressure gradient will become more adverse as the flow must diffuse more on the upper surface of the nacelle. At a high enough incidence

the adverse pressure gradient the flow may separate from the surface and recirculate before re-attaching. At higher incidence the separation region will grow until it no longer re-attaches. This flow regime is associated with very high drag.

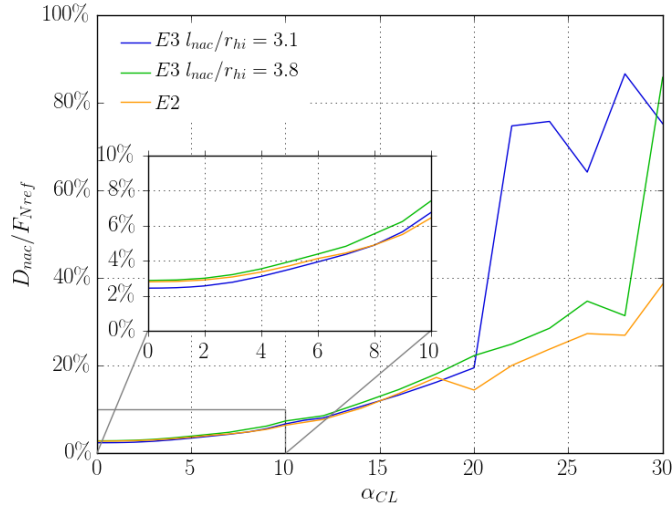


FIGURE 5.49: Influence of α_{CL} on windmill drag (D_{nac}) for four different engine designs in an end of runway condition ($M_\infty = 0.25$, $h = 0m$)

The precipitous rise in drag which is encountered at higher incidence (Figure 5.49) is due to the separation of the flow along the upper surface of the fan cowl. This occurs at a different value of α_{CL} depending on the length of the nacelle. Whilst the conventionally designed E2 nacelle does not have a fan cowl separation even at the high incidence of $\alpha_{CL} = 30^\circ$, the shorter E3 nacelles have both separated by this point (Figure 5.50). For both E3 nacelles the separation is triggered over the leading edge of the nacelle which results in a large region of reversed flow (Figure 5.50). This leading edge separation is the cause of a sudden change in drag above a given incidence (Figure 5.49). This indicates that the shortest nacelle cannot operate above $\alpha_{CL} = 20^\circ$ under the windmill condition.

5.9 Influence of Installation on Aerodynamic Performance

This section aims to determine the influence of the engine installation on the Common Research Model (CRM) aircraft. The section demonstrates the performance of the E2 engine, with the $l_{nac}/r_{hi} = 4.3$ nacelle, initially as a baseline at the two

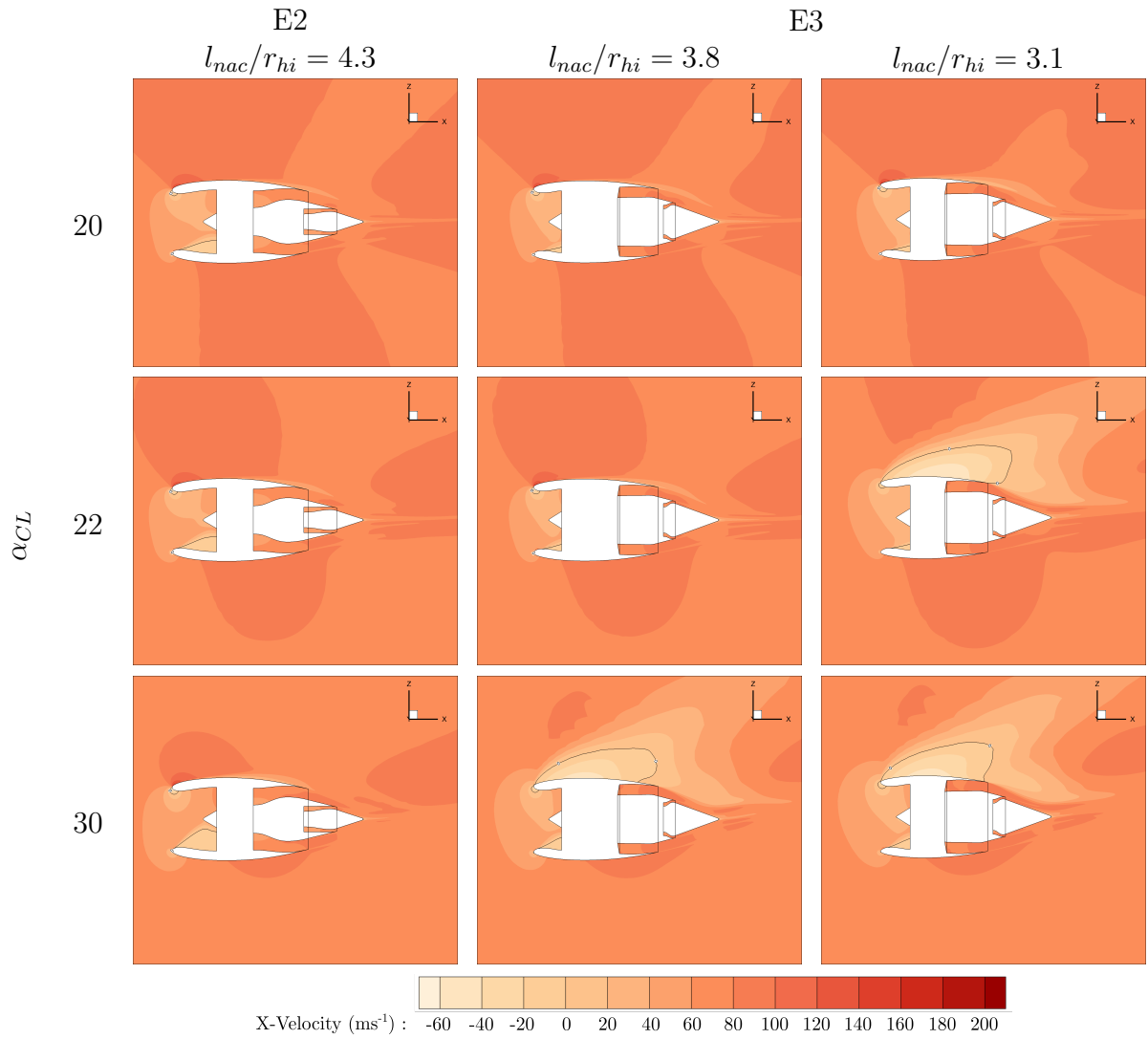


FIGURE 5.50: Contours of X-Velocity (U_x) for three different engine designs as a function of incidence (α_{CL})

different operating Mach numbers ($M_\infty = 0.85$ and $M_\infty = 0.82$). Then the relative performance of the E3 engine with the short ($l_{nac}/r_{hi} = 3.1$) nacelle is quantified. Finally, two installation angles are assessed for the E3 engine to establish whether this value can be reduced to improve aerodynamic interaction with the aircraft as the short nacelle is more sensitive to incidence (Section 5.7).

5.9.1 Clean Wing

The clean wing configuration (WBT0) consists of the airframe (fuselage, wing, tailplane) without any engine installed. It was computed¹¹ to establish the performance of the airframe without any interference from an underwing engine. From this starting point, the installation and interference terms can be deduced by installing the engine and comparing the results. The CRM aircraft was designed to operate at $M_\infty = 0.85$ and $C_L = 0.5$.⁶⁴ The CRM clean wing configuration (WBT0) has been calculated at $M_\infty = 0.85$ ¹²⁰ and $M_\infty = 0.82$.¹¹ At the design lift coefficient of $C_L = 0.5$ the L/D ratio at both $M_\infty = 0.85$ and $M_\infty = 0.82$ is around $L/D = 22$ for these clean wing calculations (Figure 5.51). The operating incidence at $M_\infty = 0.85$ is lower at around $\alpha_{a/c} = 2.1^\circ$ in comparison to $\alpha_{a/c} = 2.3^\circ$ at $M_\infty = 0.82$ (Figure 5.52). However above $\alpha_{a/c} = 3.5^\circ$ the lift at $M_\infty = 0.85$ reduces below the lift at $M_\infty = 0.82$ as transonic effects begin to worsen the performance. This worsened relationship with incidence is as a direct result of the increased loading on the wing which is present at the higher Mach number (Figure 5.53).

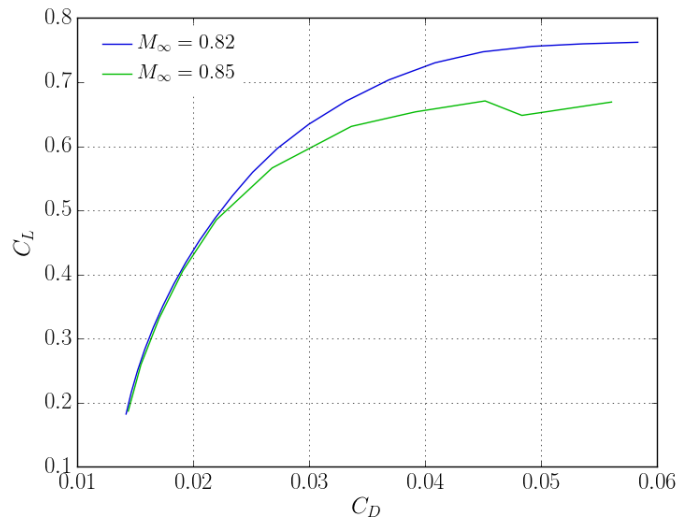


FIGURE 5.51: Drag polar for the CRM clean wing aircraft at $M_\infty = 0.82$ and $M_\infty = 0.85$

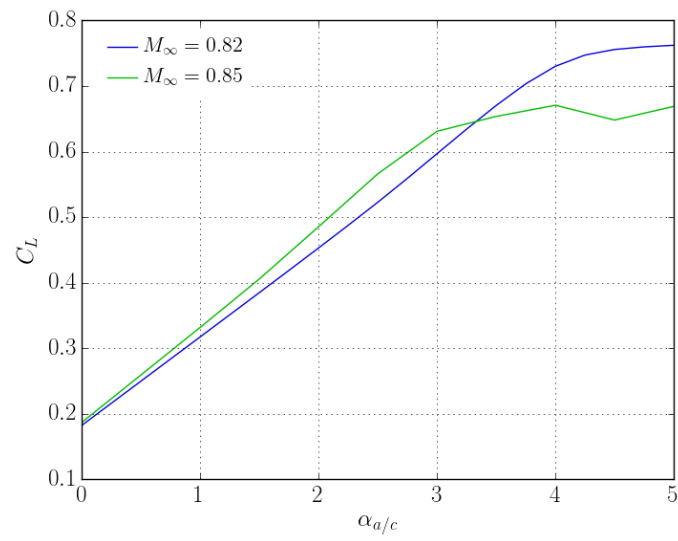


FIGURE 5.52: $C_L - \alpha$ curve for the CRM clean wing aircraft at $M_\infty = 0.82$ and $M_\infty = 0.85$

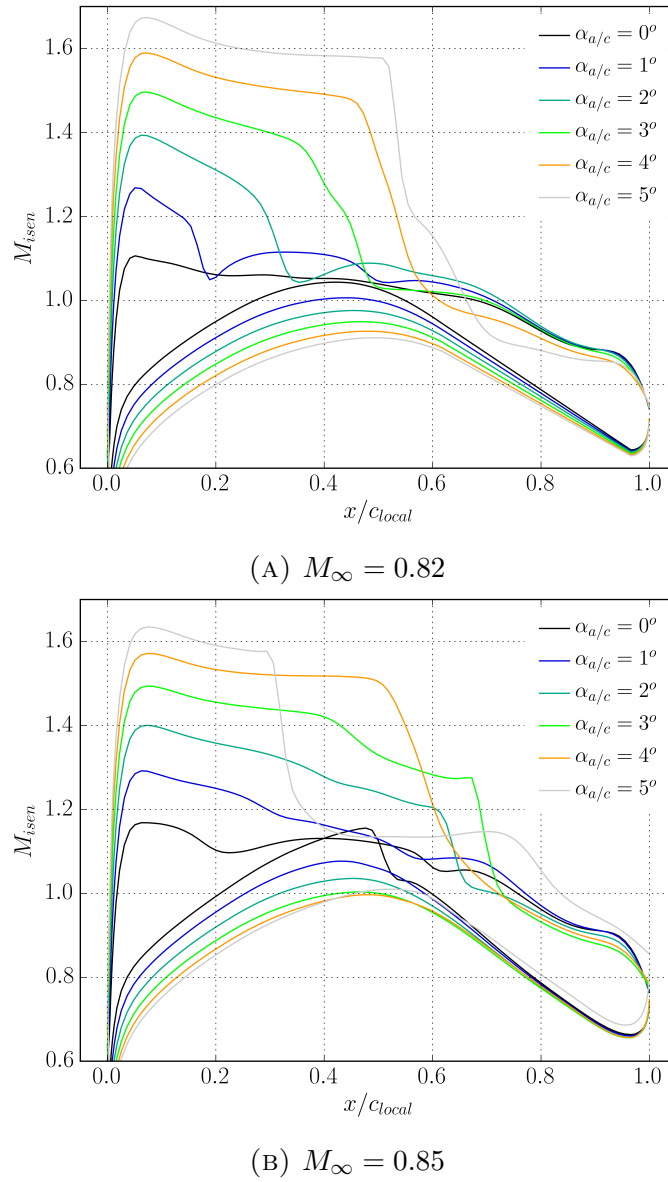


FIGURE 5.53: Isentropic Mach distributions on the CRM wing in the engine installation plane at two freestream Mach numbers across a range of incidences

5.9.2 E2 Engine

The E2 engine with datum nacelle was simulated in the datum installation position (Figure 3.6) at $M_{\infty} = 0.85$ for a range of incidences to establish a drag polar. Relative to the clean wing configuration WBT0, the installed configuration WBT0N had lower lift for a given incidence due to the influence the engine had on the airframe. As a result of this the required incidence to achieve the cruise lift coefficient

of $C_L = 0.5$ increased from $\alpha_{a/c} = 2.1^\circ$ to $\alpha_{a/c} = 2.3^\circ$ due to the addition of the engine. The increase in the aircraft drag between the clean wing configuration and the installed configuration at this lift coefficient is the installation drag. The installation drag for the E2 engine installed in the C3 position was $C_{D,inst} = 0.0022$ (Figure 5.54). To establish how much this increase in drag is due to the interference it is important to determine the difference between the installation drag ($C_{D,inst}$) and the isolated nacelle drag ($C_{D,nac}$). The drag of the superposition model (SM) was therefore calculated where the isolated nacelle drag is added to the clean aircraft drag. The superposition model consists of the clean wing configuration (WBT0) and an isolated nacelle summed in a condition at which they produce the same lift coefficient as the installed case. The installation angle of the E2 engine was $\alpha_{inst} = 1.75^\circ$ relative to the aircraft axis. The upwash was extracted at the position of the engine highlight centre from the clean wing configuration (WBT0) analyses. These values were calculated for the required cruise lift coefficient of $C_L = 0.5$. The upwash was approximately $\alpha_{upwash} = 0.3^\circ$. The sum of the aircraft incidence ($\alpha_{a/c}$), the installation incidence (α_{inst}) and the upwash incidence (α_{upwash}) give the effective engine incidence which the isolated engine must be considered at to obtain the superposition model drag. For the E2 engine installed in the datum position this effective incidence was $\alpha_{CL} = 4.4^\circ$ at $C_L = 0.5$. The interference term is then the difference between the installed configuration and the superposition model.

An interference benefit is present in the D_{*nac} force for the installed case relative to the superposition model (Figure 5.55). However, the forward force acting on the post exit streamtube is reduced once installed in comparison to the superposition model (Figure 5.55). Due to this and the relatively unchanged airframe drag, there is a net interference drag term of 0.3% of F_{Nref} (Figure 5.55). The source of these interference terms can be identified in the local aerodynamics. A direct comparison of the nacelle aerodynamics from isolated to installed is difficult since CFD was not computed at the exact effective incidences which the engine operates at when installed on the wing. However there is a close approximation and the trends in the peak Mach numbers have been demonstrated to respond quite linearly to incidence (section 5.7). The main effects of the wing pressure field on the nacelle was an increase in the pressure on the aft 60% of the cowl, and a relocation of the shock. The beneficial reduction in drag on the nacelle is caused by the increase in the pressure on the forward facing aft 60% of the cowl (Figure 5.56). This provides a greater benefit than the increase in drag due to the stronger shock.

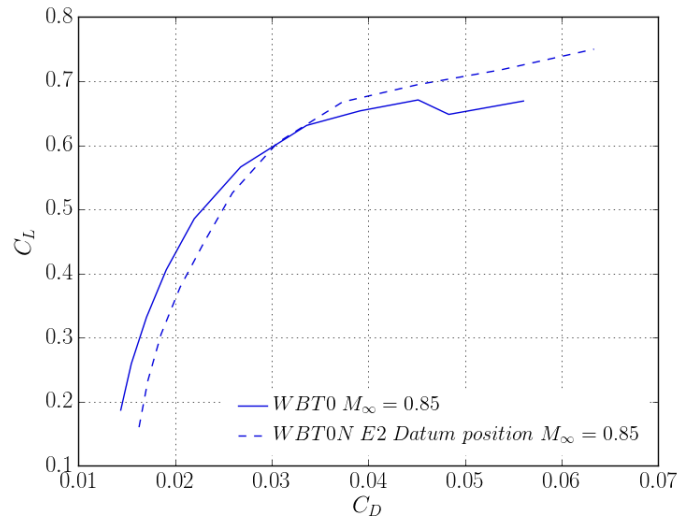


FIGURE 5.54: Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E2 engine installed in the datum position

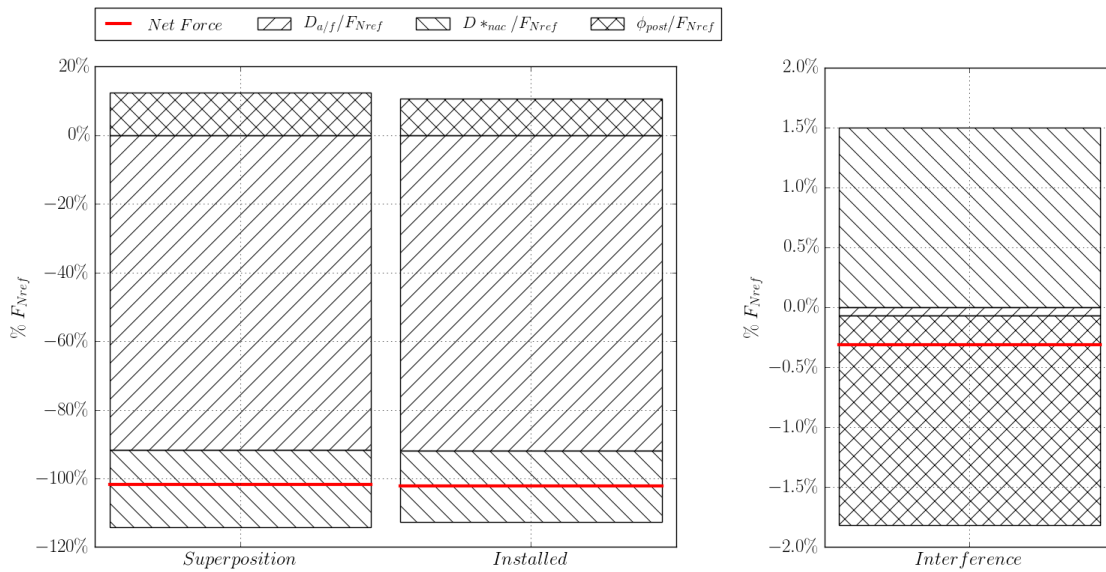


FIGURE 5.55: Stackup of drag forces acting on the isolated and installed E2 engine and airframe

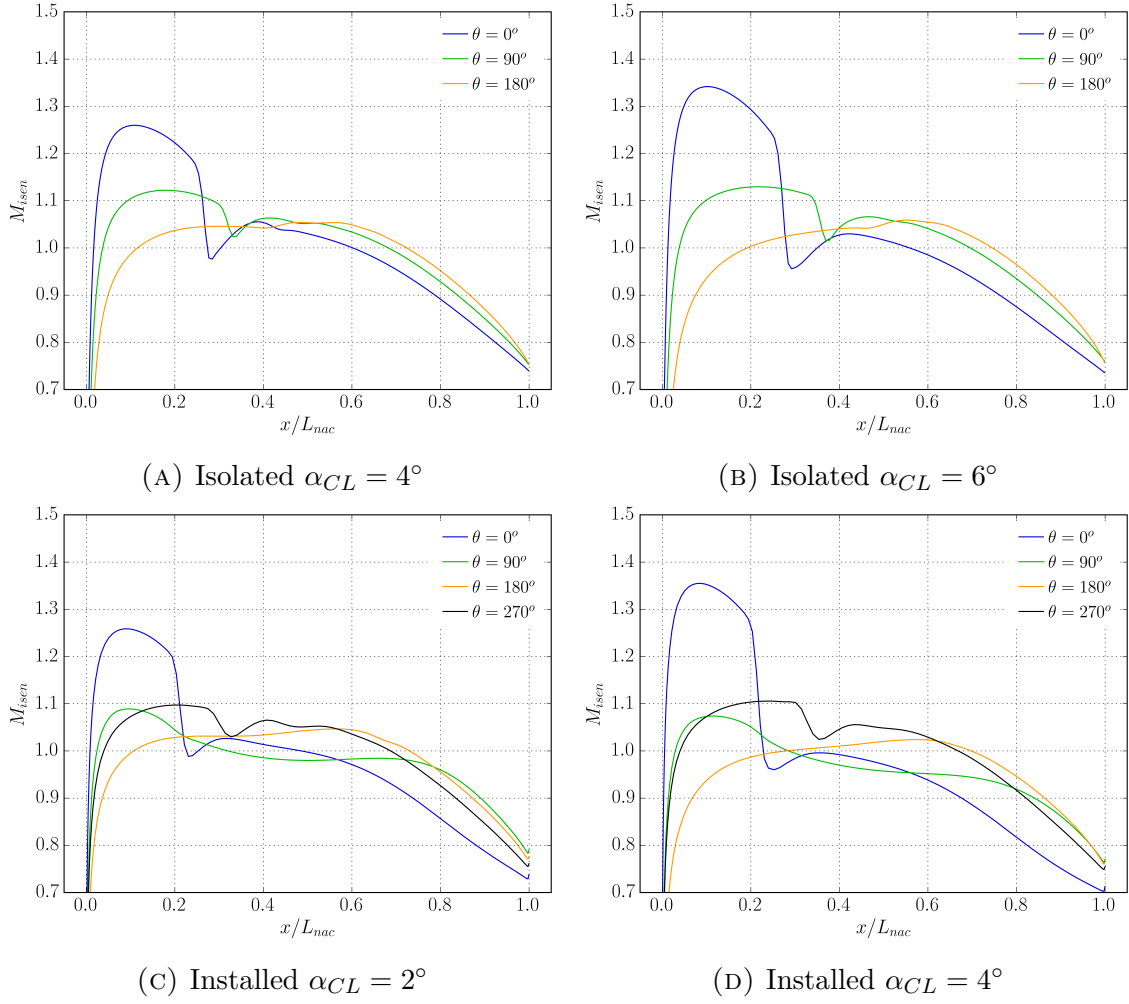


FIGURE 5.56: Isentropic Mach distributions on the E2 nacelle at two incidences, in an isolated and installed (C3) configuration at $M_\infty = 0.85$

5.9.2.1 Influence of Freestream Mach Number

To determine the sensitivity of the installation to freestream Mach number, the data at $M_\infty = 0.85$ was compared to data at $M_\infty = 0.82$.¹¹ For the clean wing case the increase in Mach number increased the lift at a given incidence. This trend remains true for the installed configuration. However, the higher freestream Mach number also resulted in an increase in the drag of both the airframe and the engine. As a result of this the required incidence to achieve the cruise lift coefficient of C_L at $M_\infty = 0.82$ increased from $\alpha_{a/c} = 2.3^\circ$ to $\alpha_{a/c} = 2.5^\circ$ due to the addition of the engine, whilst at the higher Mach number the required incidence with the engine was $\alpha_{a/c} = 2.3^\circ$. The installation drag for the E2 engine installed in the datum position was $C_{D,inst} = 0.0015$ at $M_\infty = 0.82$, which is lower than at $M_\infty = 0.85$

where it was $C_{D,inst} = 0.0022$ (Figure 5.57). However, as demonstrated previously (Section 5.7) the increase in Mach number does also cause an increase in the drag for the isolated engine configuration. Therefore, again the installed case was compared to the superposition model. Whilst for the case at $M_\infty = 0.85$, the interference force resulted in an $0.3\%F_{Nref}$ increase in the drag, for the case at $M_\infty = 0.82$ there was a net benefit of $1.3\%F_{Nref}$ due to the interference. At the lower Mach number the benefit is due to a much larger beneficial interaction on the nacelle (Figure 5.58), whilst there is also a detrimental interaction on the wing for this case (Figure 5.58).

The sources of these interference terms can again be identified in the local aerodynamics. At $M_\infty = 0.82$ the addition of the engine causes a large reduction in the peak suction on the upper surface of the wing relative to the clean wing configuration (WBT0) at every incidence tested (Figure 5.59a). This is a consequence of the local region of the wing being washed by the lower momentum wake of the nacelle. At $\alpha_{a/c} = 2^\circ$ for instance, the peak Mach number is reduced from $M = 1.40$ to $M = 1.12$. This reduction in suction of course reduces the lift, however as this suction is on the forward facing section of the wing any reduction also increases the drag. In contrast, the configuration at the higher Mach number of $M_\infty = 0.85$ experiences less reduction in peak suction as a result of engine installation, reducing from $M = 1.40$ to $M = 1.22$ at $\alpha_{a/c} = 2.0^\circ$. This is largely a result of the different flow topology between the two Mach numbers, whereby the clean wing configurations have the same peak Mach number at the two different freestream Mach numbers as the shock moves from forward of the crest to aft as the freestream Mach number is increased from $M_\infty = 0.82$ to $M_\infty = 0.85$ at $\alpha_{a/c} = 2.0^\circ$ (Figure 5.59). With an engine installed, the aft shock remains in a similar position and a second shock wave is present on the forebody for the case at $M_\infty = 0.85$ and $\alpha_{a/c} = 2.0^\circ$. This two shock formation retains more suction on the forward facing region of the wing than the shock free configuration at the lower Mach number relative to the respective clean wing configurations. In the regions of the wing away from the engine, the flow is also greatly affected by the installation of the engine. At both freestream Mach numbers a high Mach region is present in the installed configuration inboard of the engine location on the upper surface of the wing (Figure 5.60). This relative decrease in pressure compared to the clean wing configuration provides a reduction in the drag as it is present on a region of high forward projected area. However for the case at $M_\infty = 0.82$, this also results in the addition of a shock wave which is not present in the clean wing configuration (Figure 5.60), which increases the drag and causes an overall increase in the airframe drag due to installation (Figure 5.58). For

the case at $M_\infty = 0.85$ however, a shock is present in the clean wing configuration at $\alpha_{a/c} = 2.0^\circ$ which extends for most of the wingspan (Figure 5.60). As such, the acceleration caused by the addition of the engine is less penalising in terms of wave drag and overall the installation has almost no detrimental impact on the airframe drag relative to the clean wing configuration.

The nacelle aerodynamics are important for these installed comparisons as the beneficial contributions to the interference force for both Mach numbers were acting on the nacelle (Figure 5.58). The interference force resulted in four times as much drag reduction in the $M_\infty = 0.82$ than it did in the $M_\infty = 0.85$ case (Figure 5.58). For instance at $\alpha_{a/c} = 2.0^\circ$ the effective nacelle incidence is calculated to be $\alpha_{CL} = 4.1^\circ$ at $M_\infty = 0.82$ once the installation angle and the wing upwash is accounted for. The $\alpha_{a/c} = 2.0^\circ$, $M_\infty = 0.82$ installed case is therefore comparable to the $\alpha_{CL} = 4.0^\circ$ at $M_\infty = 0.82$ isolated case. The effective incidences at the other conditions are: $\alpha_{CL} = 6.4^\circ$ at $\alpha_{a/c} = 4.0^\circ$ at $M_\infty = 0.82$, $\alpha_{CL} = 4.0^\circ$ at $\alpha_{a/c} = 2.0^\circ$ at $M_\infty = 0.85$, and $\alpha_{CL} = 6.3^\circ$ at $\alpha_{a/c} = 4.0^\circ$ at $M_\infty = 0.85$. The re-trimmed aircraft incidence is closer to $\alpha_{a/c} = 2.0^\circ$ than $\alpha_{a/c} = 4.0^\circ$ in both freestream Mach number cases. The benefit of installation on the nacelle drag is due to the increase in pressure which acts on the afterbody of the nacelle relative to the isolated case (Figure 5.61). In addition to this the position of the shock is moved upstream by around $0.02L_{nac}$ however this does not appreciably increase the strength. At $M_\infty = 0.85$ a similar trend is seen where the afterbody experiences a beneficial increase in pressure (Figure 5.56). However, as it is at a higher freestream Mach number the accelerations on the cowl are generally higher in both installed and isolated. This results in a greater penalty from installation as the shock is moved upstream and the shock occurs at a higher Mach number. The same interference is therefore acting at both freestream Mach numbers however the balance between shock strength increase and beneficial pressure force is what results in much less interference benefit for the $M_\infty = 0.85$ case.

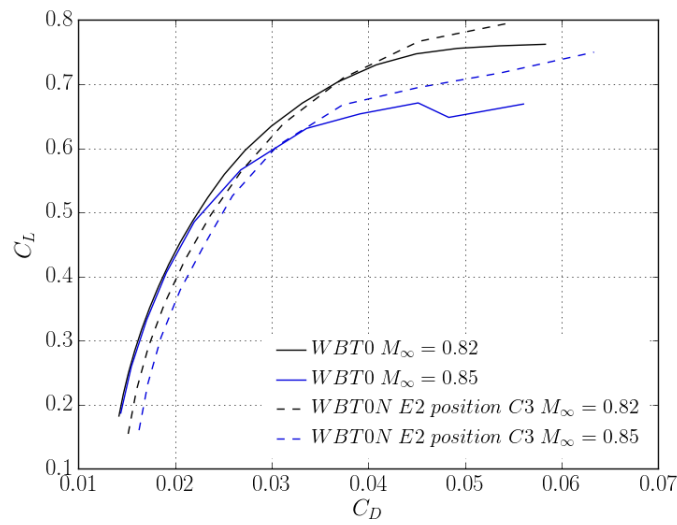


FIGURE 5.57: Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.82$ and $M_\infty = 0.85$ and for the CRM with the E2 engine installed in position C3

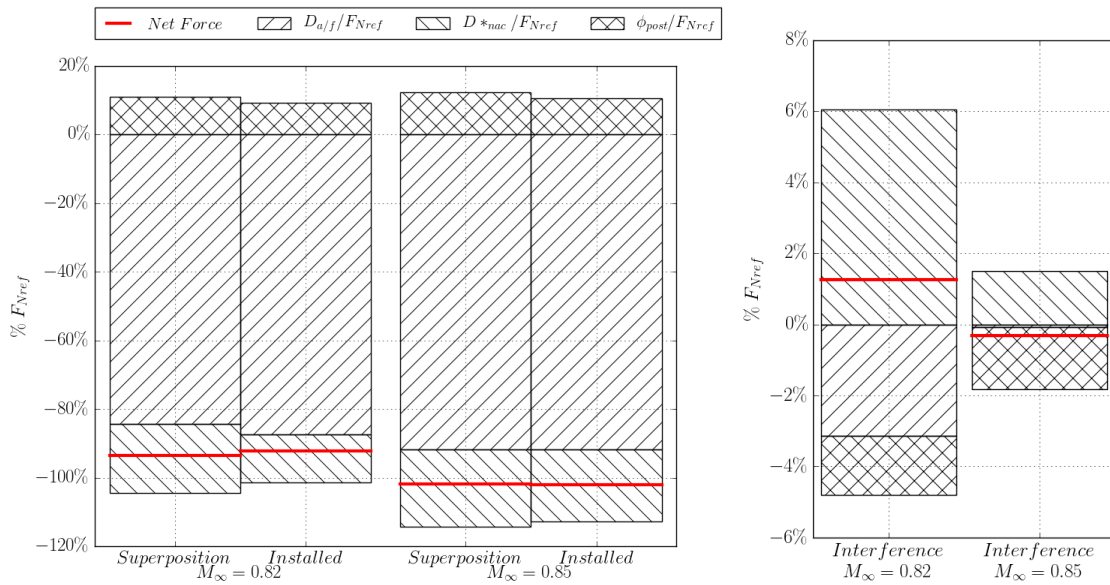


FIGURE 5.58: Stackup of drag forces acting on the isolated and installed E2 engine and airframe

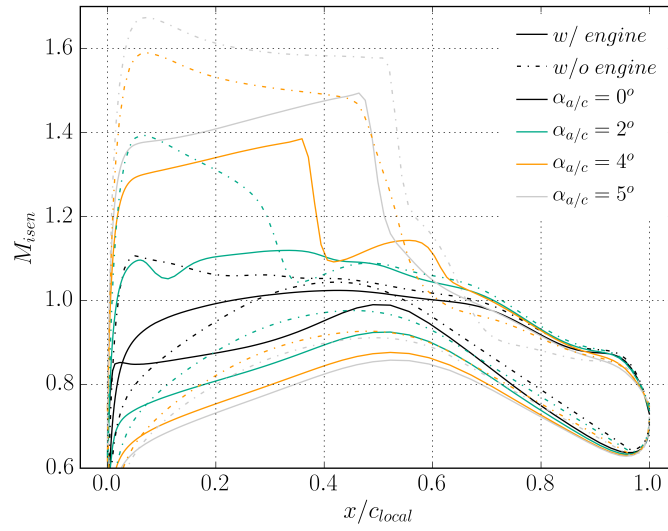
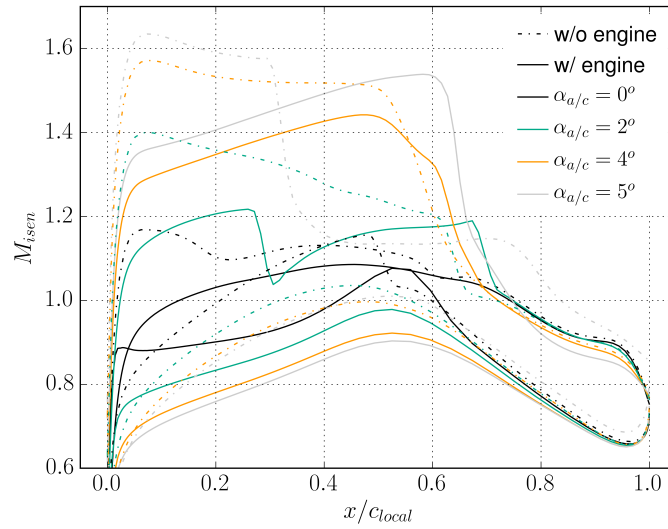
(A) $M_\infty = 0.82$ (B) $M_\infty = 0.85$

FIGURE 5.59: Isentropic Mach distributions on the CRM wing in the engine installation plane at two freestream Mach numbers across a range of incidences with and without engine E2 installed in the baseline position (C3)

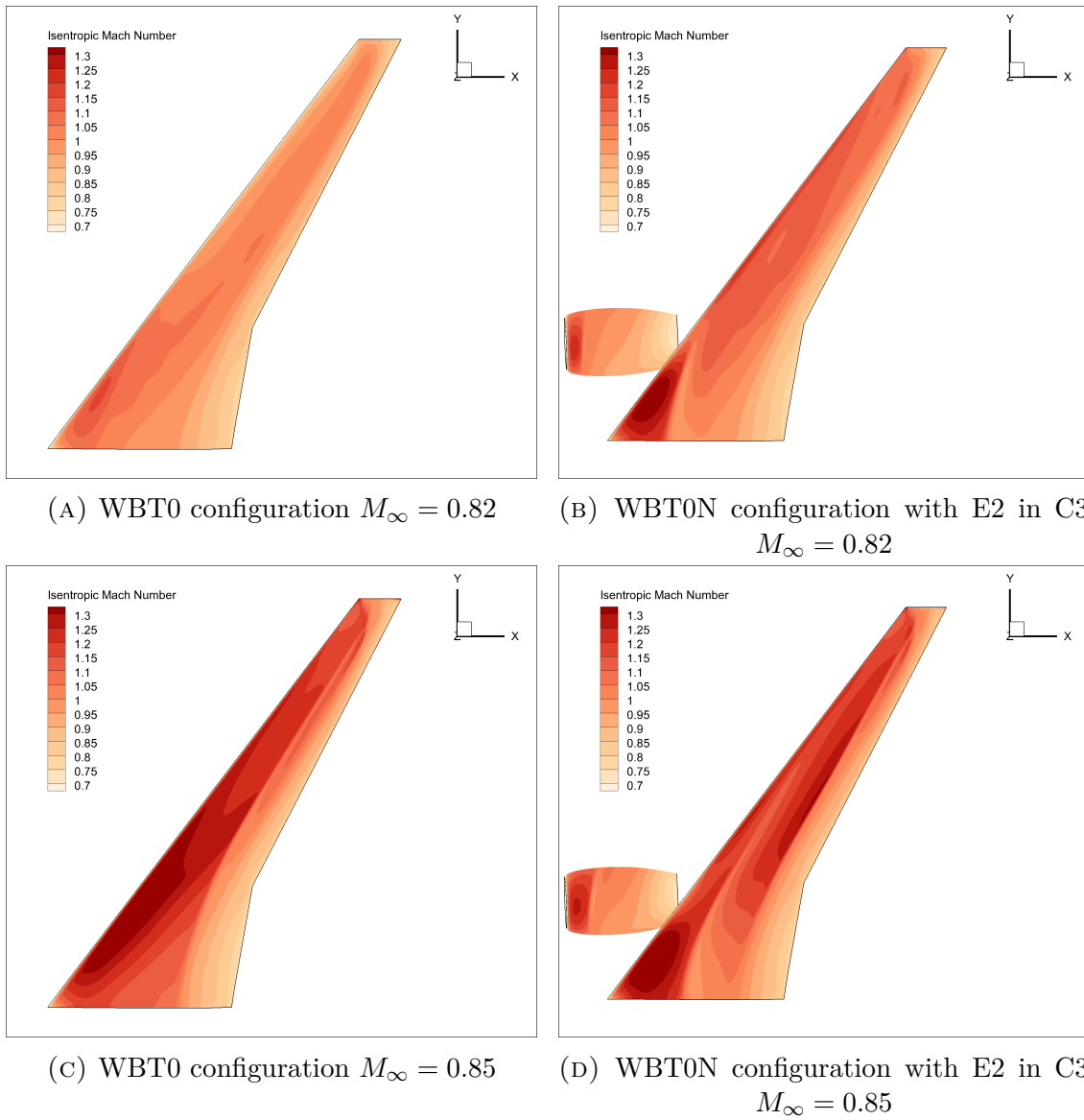


FIGURE 5.60: Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the clean wing (WBT0) and with E2 installed in C3 configuration

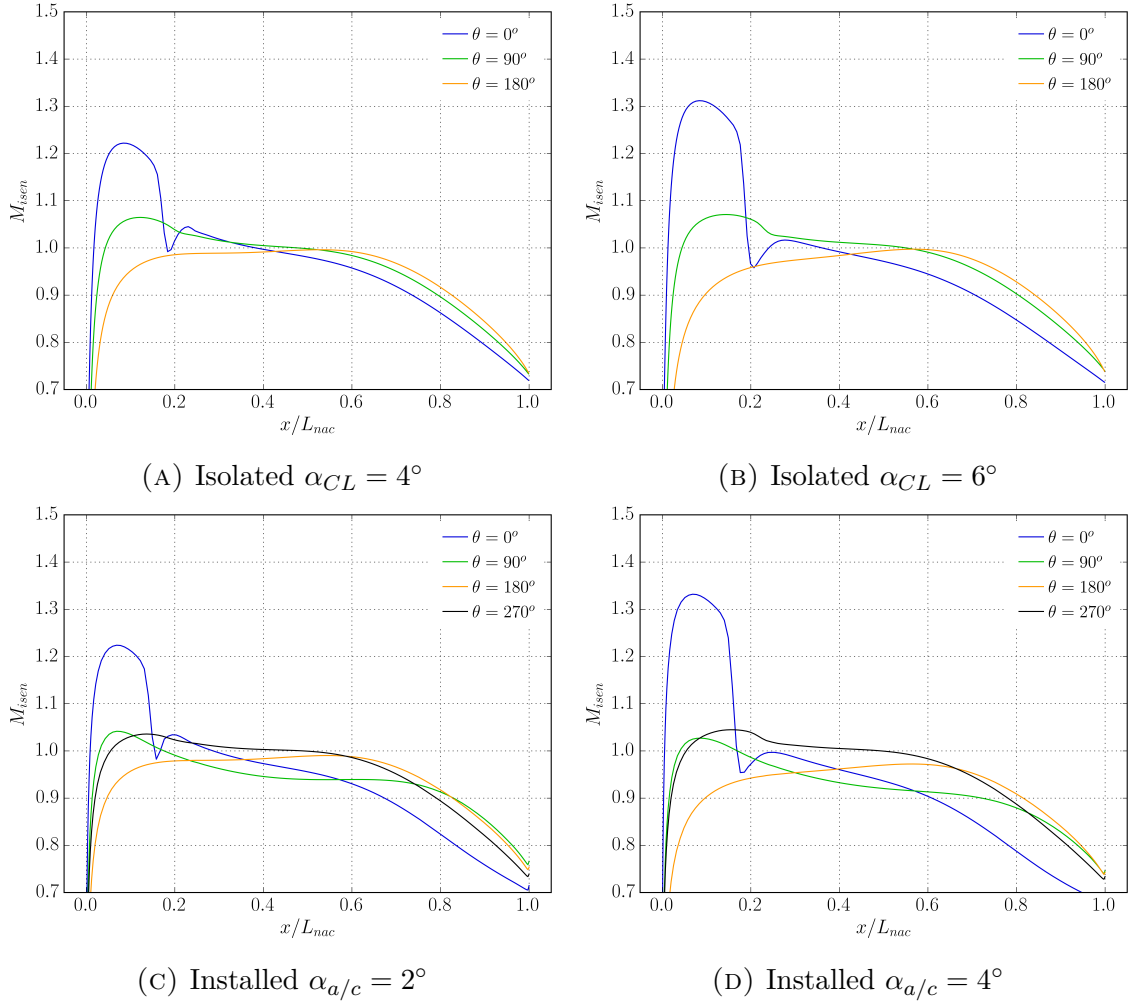


FIGURE 5.61: Isentropic Mach distributions on the E2 nacelle at two incidences, in an isolated and installed (C3) configuration at $M_\infty = 0.82$

5.9.3 E3 engine

The E2 $l_{nac}/r_{hi} = 4.3$ engine nacelle was compared to the E3 $l_{nac}/r_{hi} = 3.1$ nacelle in the same datum installation position. The drag polars indicated that below $C_L = 0.6$ the configuration with the E3 engine installed had less drag than the longer E2 engine installed in the same position of C3 and operating at the same Mach number of $M_\infty = 0.85$ (Figure 5.62). At the design lift coefficient of $C_L = 0.5$ the configuration with the E3 engine had an overall drag of $C_{D,a/c} = 0.0241$ which compares favourably to the E2 engine configuration which had an overall drag coefficient of $C_{D,a/c} = 0.0248$ at the same re-trimmed condition. The influence of the installation in comparison to the superposition model however is much different between the E2 and the E3 engines, which indicates how this lower drag is achieved.

The superposition model was again calculated at the incidence of the airframe and nacelle which would give a combined lift of $C_L = 0.5$ whilst accounting for the installation angle of the engine and the upwash incidence seen by the nacelle. This resulted in a similar incidence and isolated nacelle drag between the E2, $D_{*nac} = 22.5\%F_{Nref}$, and E3 engines, $D_{*nac} = 21.6\%F_{Nref}$ (Figure 5.63). The influence of installation however changes these numbers much more for the E3 engine (Figure 5.63). Specifically, the airframe experiences a large reduction in drag of $11.2\% F_{Nref}$, whilst the modified nacelle drag is increased by 4.0% of F_{Nref} and the beneficial post exit force is reduced by 6.4% of F_{Nref} . This indicates a far greater level of interaction for the E3 engine than was experienced with the E2 engine (Figure 5.63). Ultimately, this results in a $0.9\% F_{Nref}$ reduction in the drag relative to the superposition model for the E3 engine whilst a detriment of $0.3\% F_{Nref}$ was observed for the E2 engine.

The reduction in airframe drag is a result of a change in the pressure field which acts on the wing. At the lower incidences, there is a higher acceleration on the lower surface of the wing relative to the clean wing case (Figure 5.64) which provides a suction on the forward facing section of the wing. This suction does not appear in the higher incidences and the flowfield is similar to the configuration with the E2 engine installed (Figures 5.59b, 5.64). This increase in suction is largely confined to the inboard region of the wing as this area will be most influenced by the gully formed by the wing, fuselage and engine (Figure 5.65). This increase in suction also would reduce the lift, though the Mach number is reduced further aft on the lower surface of the wing, which will help to regain the lift (Figure 5.64). The upper surface of the wing is influenced in a similar way to with the E3 engine installed where the single shock becomes two separate shocks. Relative to the E3 configuration, the upstream shock is stronger on the E3 installed wing with a relatively weak aft shock. This is also caused by an increase in the acceleration at the leading edge of the wing.

In contrast to the E2 engine, the drag of the E3 engine was worse in the installed case relative to the superposition model (Figure 5.63). In the case of the E3, which had a larger amount of wave drag in the isolated case, the interaction in the wing resulted in an increase in the wave drag (Figure 5.66) as had been observed for the E2 engine at $M_\infty = 0.85$. This displacement of the shock also reduces the sonic region by around 50% relative to the isolated case and this provides less forward force due to suction loss. As with the E2 engine the loss in forebody suction is accompanied by an increase in aftbody pressure, however for this more supercritical

design the reduction in suction and increased wave drag is far more penalising than the aftbody improvement.

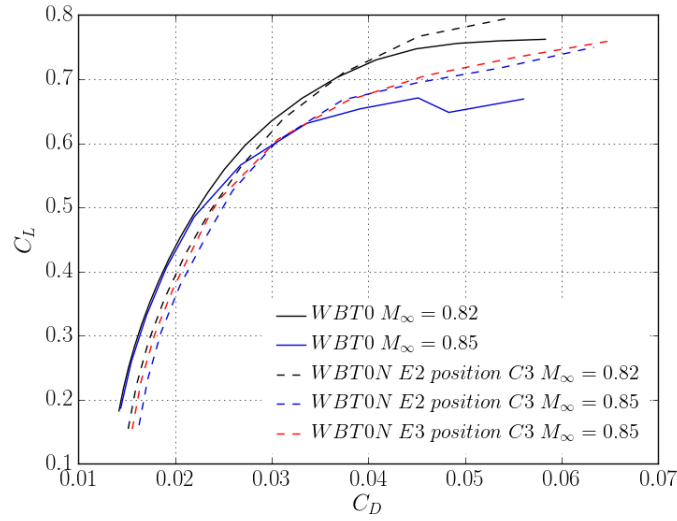


FIGURE 5.62: Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.82$ and $M_\infty = 0.85$ and for the CRM with the E2 and E3 engines installed in position C3

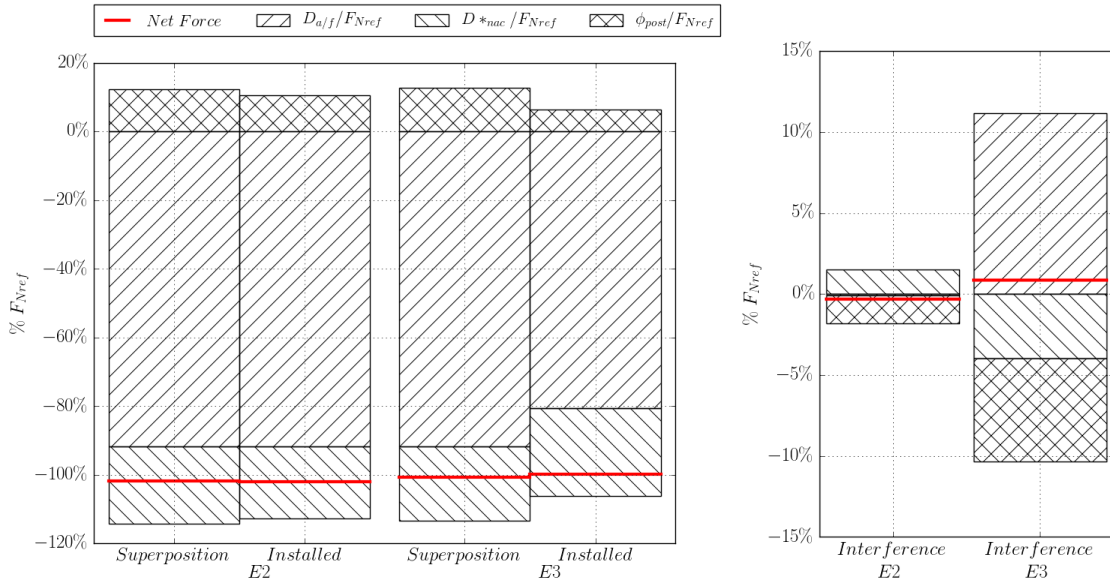


FIGURE 5.63: Stackup of drag forces acting on the isolated and installed E2 and E3 engine and airframe in position C3

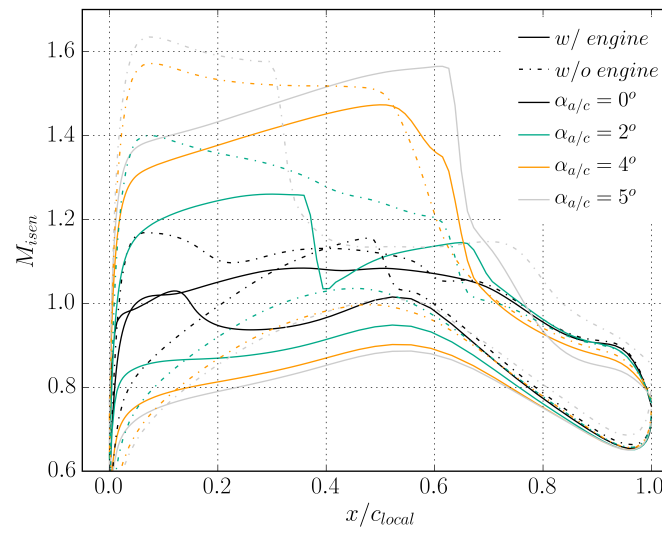
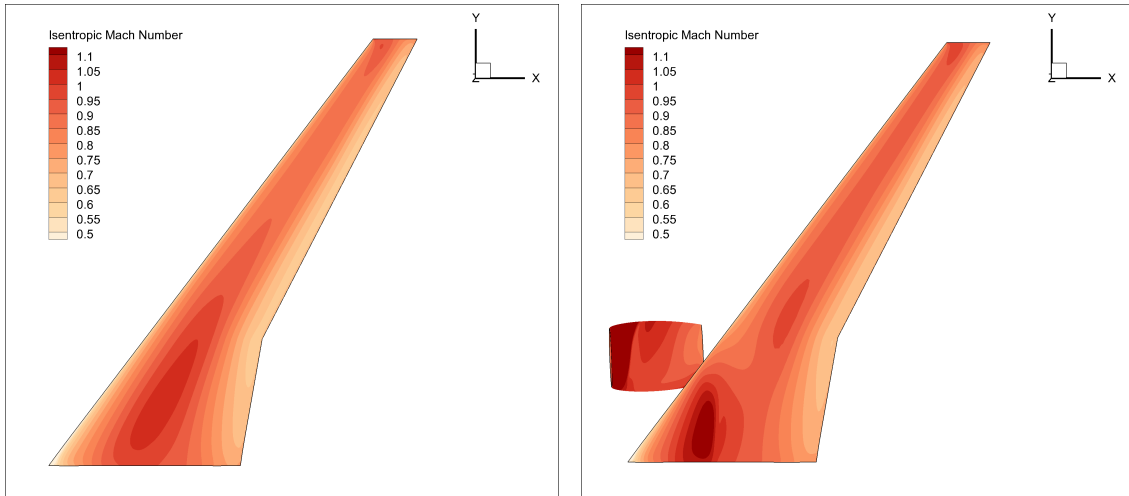
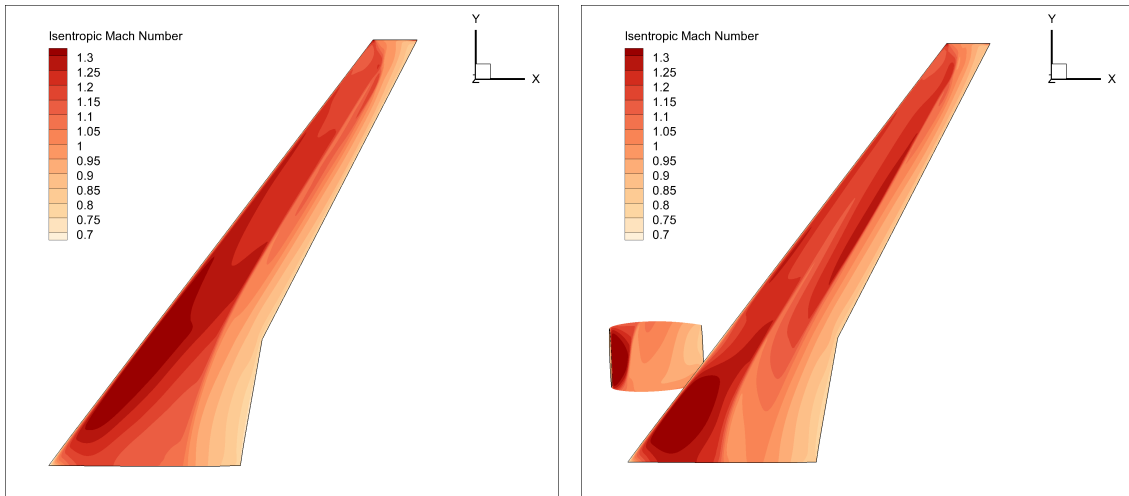


FIGURE 5.64: Isentropic Mach distributions on the CRM wing in the engine installation plane at two freestream Mach numbers across a range of incidences with and without engine E3 installed in the baseline position (C3)



(A) WBT0 configuration wing lower surface $M_\infty = 0.85$ (B) WBT0N configuration wing lower surface with E3 in C3 $M_\infty = 0.85$



(C) WBT0 configuration wing upper surface $M_\infty = 0.85$ (D) WBT0N configuration wing upper surface with E3 in C3 $M_\infty = 0.85$

FIGURE 5.65: Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the clean wing (WBT0) and with E3 installed in C3 configuration

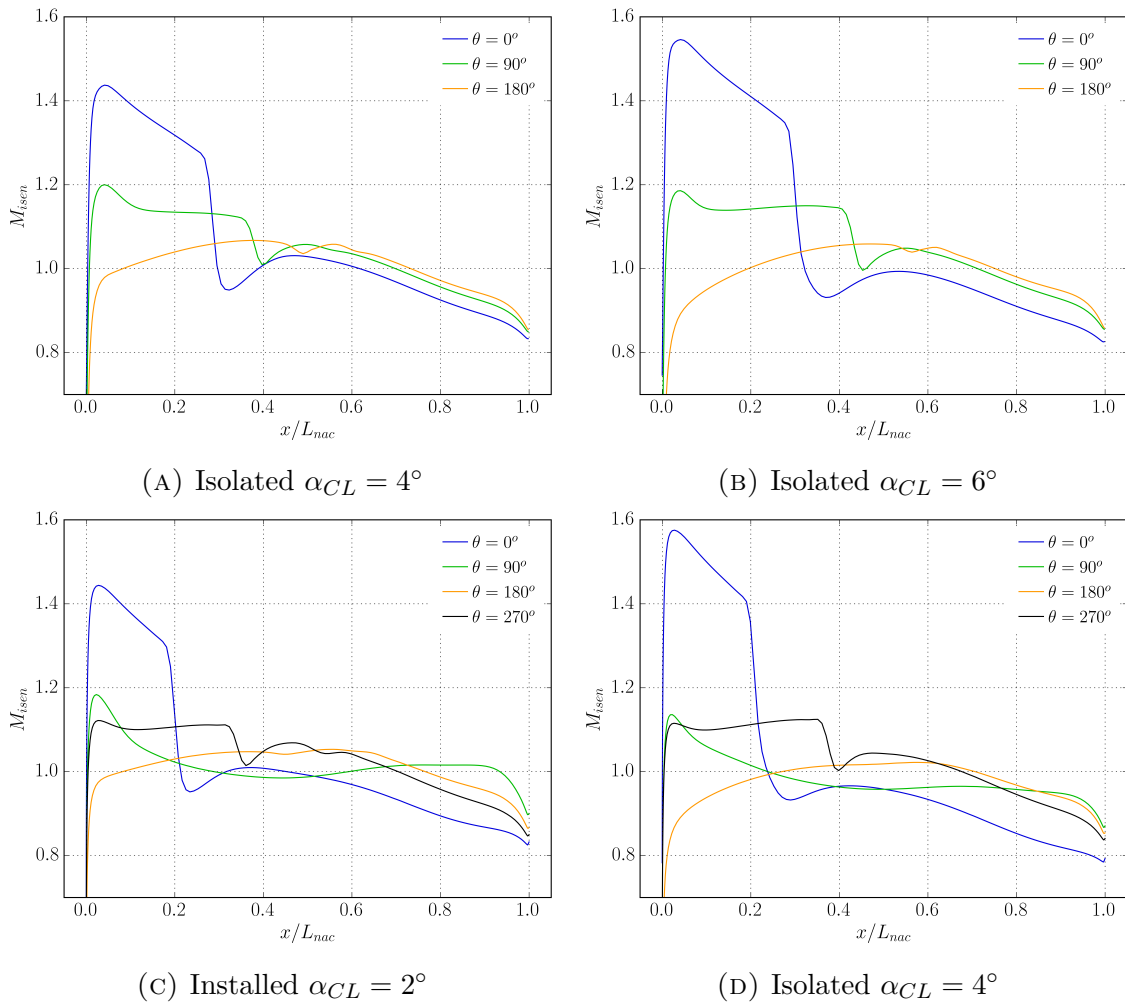


FIGURE 5.66: Isentropic Mach distributions on the E3 nacelle at two incidences, in an isolated and installed in the datum position

5.9.4 Influence of Installation Angle

In previous work¹¹ and in the previous section, the engine was installed at a constant angle ($\alpha_{Inst} = 1.75^\circ$) relative to the wing chord. This angle is determined to ensure that a reasonable trade off is achieved between a minimisation of the nacelle incidence and the minimisation of the interaction between the wing and the jet. Whilst the nacelle aerodynamics could conceivably be optimised by orienting the engine axis to coincide with the flow direction, this would likely result in the jet impinging on the wing and the thrust vector resulting in negative lift. The impact of the installation angle was very small on the overall drag polar (Figure 5.67), which suggests a very similar influence between the wing and engine between the

two installation angles. The influence of the engine on the wing appears to be minimal at all incidences tested (Figure 5.68) and restricted to the upper surface of the wing. This is not localised to the engine plane and the spanwise distributions are generally unchanged by the change in installation angle (Figure 5.69). However, since the nacelle is operating at a slightly lower incidence, there is a reduction in the peak Mach number on the nacelle at a given aircraft incidence (Figure 5.70). Since the two configurations retrimmed to an equivalent incidence, this improvement can be achieved without an appreciable loss of lift from the thrust vector. This reduction in peak Mach number will improve the off design performance and make the nacelle easier to design. Ultimately, this seemed to justify the reduction in the installation angle which enables a more favourable nacelle aerodynamics which will give improved characteristics for the nacelle off design.

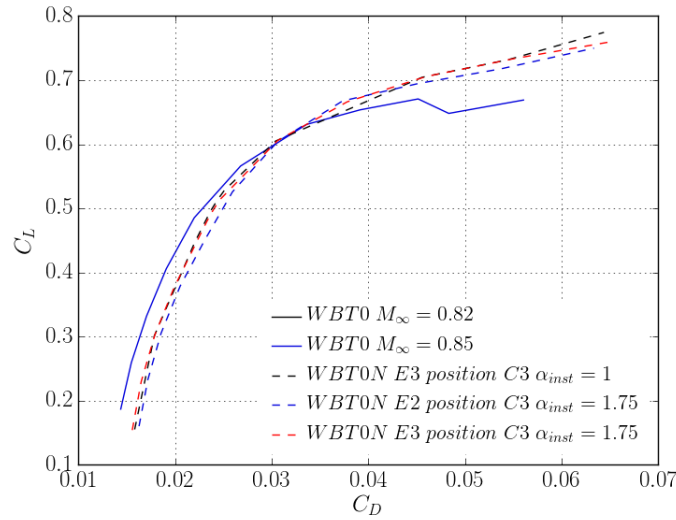


FIGURE 5.67: Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E2 and E3 engines installed in position C3 with two installation angles (α_{inst})

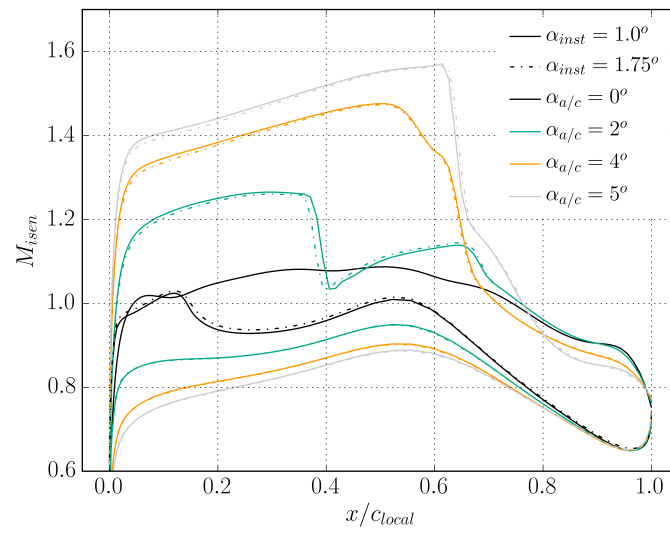
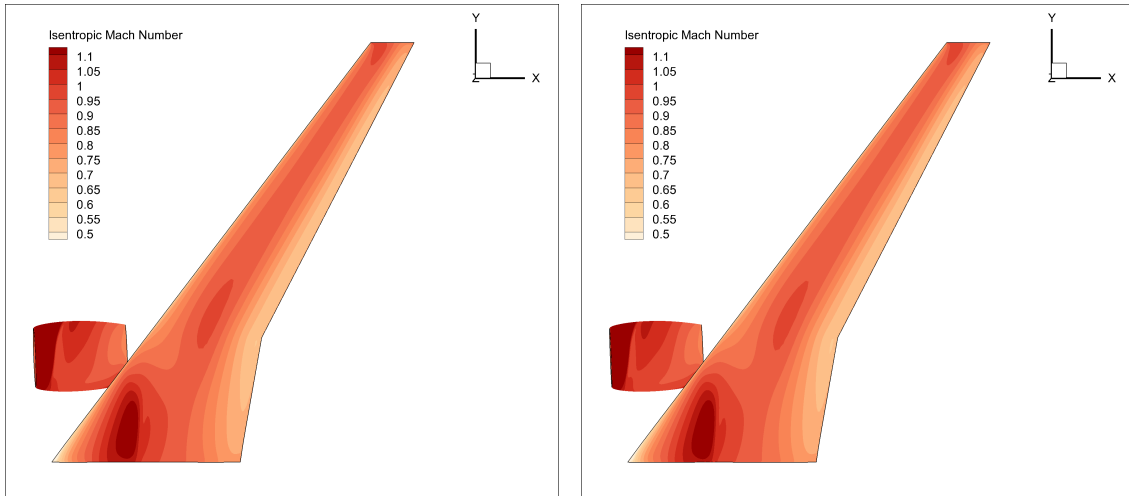
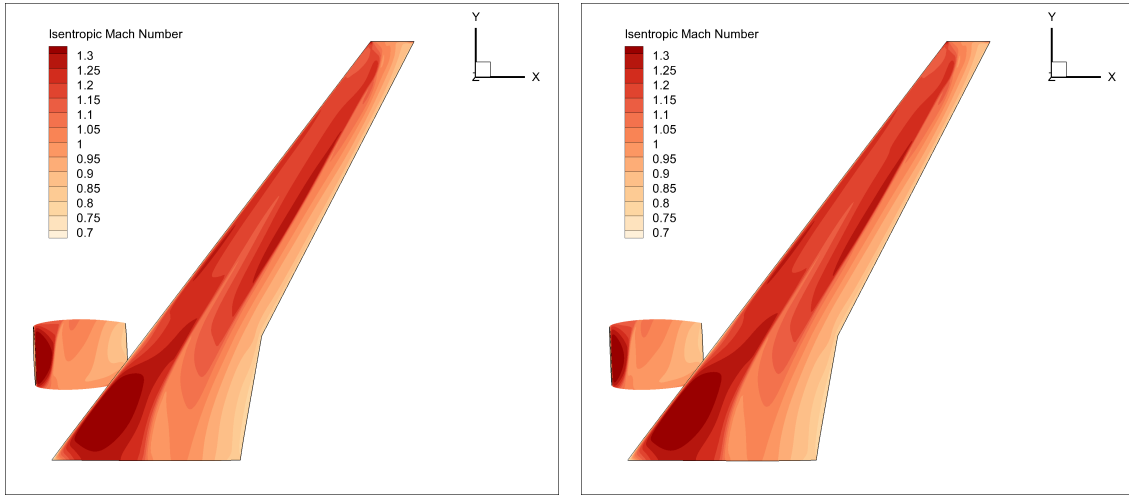


FIGURE 5.68: Isentropic Mach distributions on the CRM wing in the engine installation plane for two installation angles with engine E3 installed in the baseline position (C3)



(A) WBT0N configuration wing lower surface with E3 in C3 with $\alpha_{inst} = 1.75^\circ$

(B) WBT0N configuration wing lower surface with E3 in C3 with $\alpha_{inst} = 1.0^\circ$



(C) WBT0N configuration wing upper surface with E3 in C3 with $\alpha_{inst} = 1.75^\circ$

(D) WBT0N configuration wing upper surface with E3 in C3 with $\alpha_{inst} = 1.0^\circ$

FIGURE 5.69: Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the clean wing (WBT0) and with E3 installed in C3 configuration

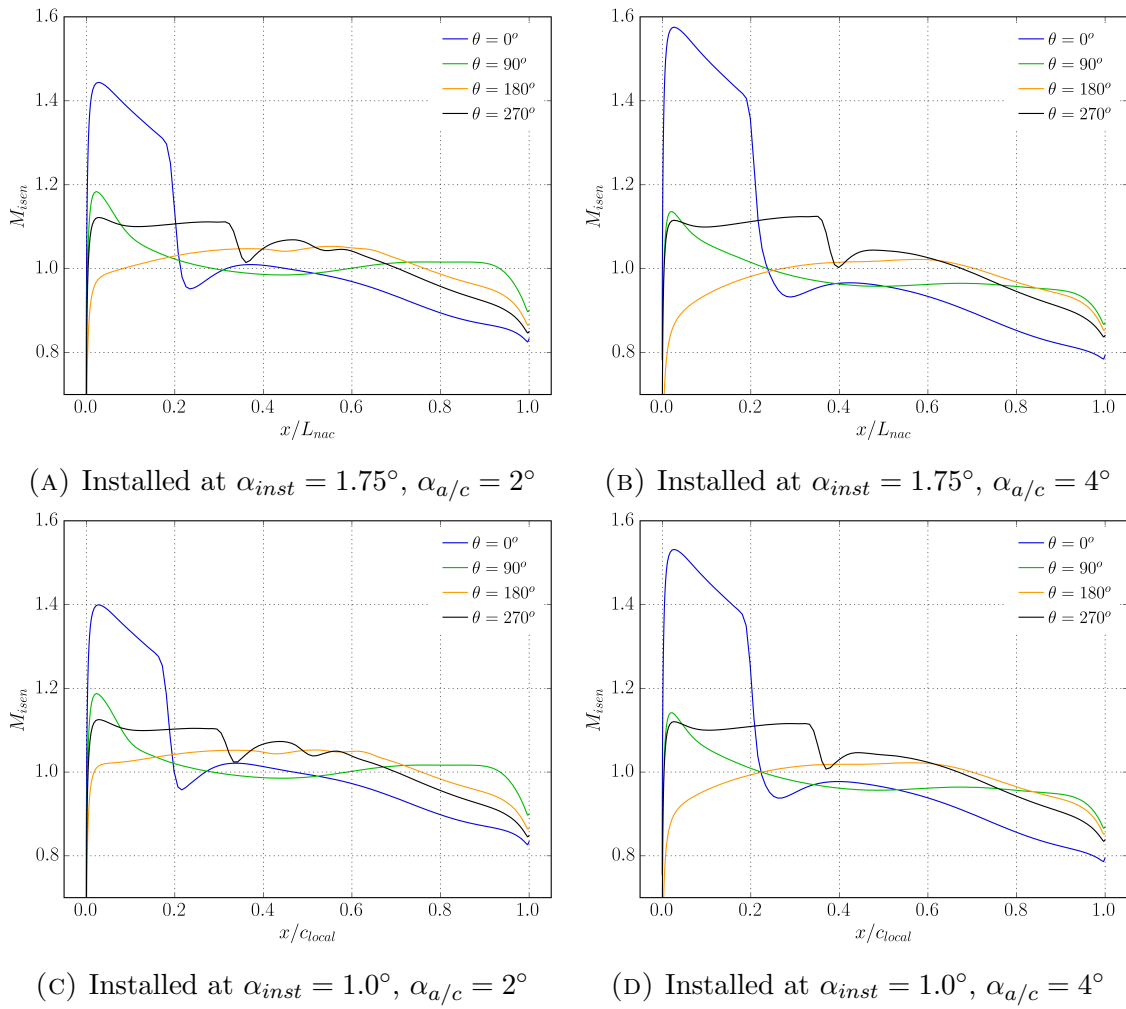


FIGURE 5.70: Isentropic Mach distributions on the E3 nacelle for two installation angles

5.9.5 Net Propulsive Force

As well as considering the impact of installation on drag, it is important to recognise that within the thrust domain there are interference terms which will impact on the overall net propulsive force of the system. For a trimmed aircraft, the NPF should be zero at the design lift coefficient of $C_L = 0.5$. However, the E2 engine had a design Mach number of $M_\infty = 0.82$, so in operation at $M_\infty = 0.85$ the NPF invariably changes as it operates off-design. To consider the influence of the installation the forces which act on the full configuration (WBT0N) are compared to a summation of the forces which act on the isolated engine (NE) and isolated airframe (WBT0) at an equivalent lift. At the design Mach number of $M_\infty = 0.82$, the modified standard net thrust (F_{N*}) was around 18% higher than the reference standard net

thrust (F_{Nref}) due to the forces acting on the core cowl and plug which give a beneficial forward force. The F_{N*} increases with an increase in Mach number in isolation (Figure 5.71) due to an increase in the engine mass flow as the MFCR was held constant. The airframe and modified nacelle drags are also increased as a result of an increase in the wave drag acting on both (Figure 5.71). As a result the NPF for the case at $M_\infty = 0.82$ is 11.5% of F_{Nref} whilst at $M_\infty = 0.85$ it was 11.3%. This excess thrust would typically prompt a reduction in engine size to reduce the NPF to zero, which would achieve weight and drag benefits. The installation of the engine on the aircraft results in the NPF for both freestream Mach number cases increasing. At $M_\infty = 0.82$ the increase is 1.9% of F_{Nref} whilst at $M_\infty = 0.85$ the increase is 1.7% (Figure 5.71).

For the E3 engine there was a greater level of interference from the wing on the F_{N*} (Figure 5.72) relative to the E2 engine. This resulted in an overall reduction in the net interference force and a reduction in the NPF for the full configuration relative to the E2 engine. The influence of the wing on the modified standard net thrust is due to the influence on the $\theta_{exhaust}$ terms, i.e. the forces which act on the core cowl and plug. The interaction between the exhaust system and the wing results in a displacement of the core cowl shock in the installed case relative to the isolated case (Figure 5.73).

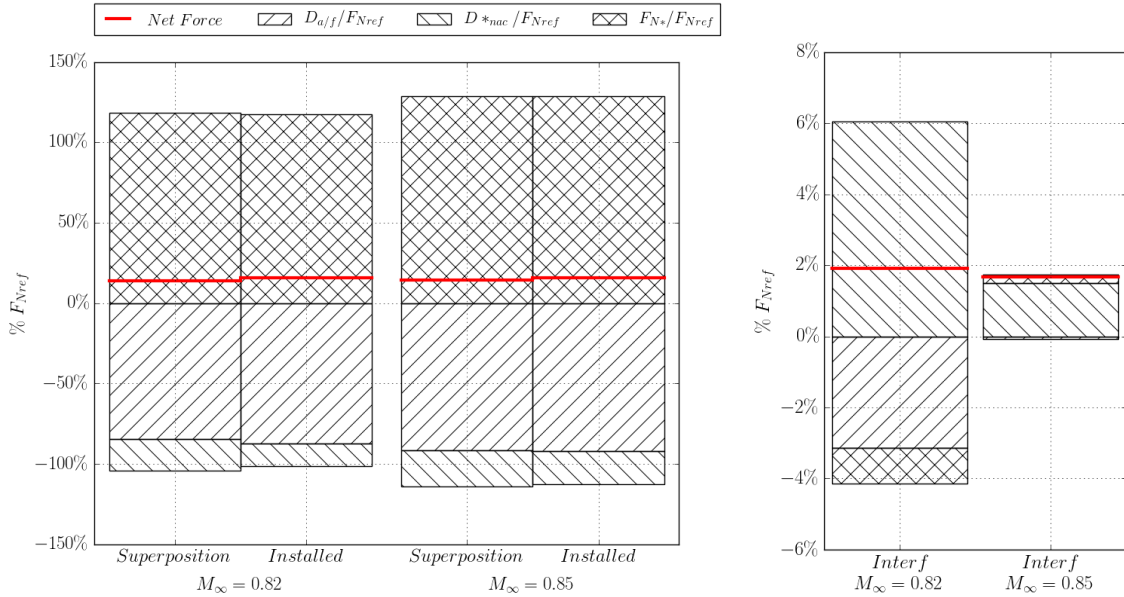


FIGURE 5.71: Stackup of forces acting on the isolated and installed E2 engine and airframe in datum position

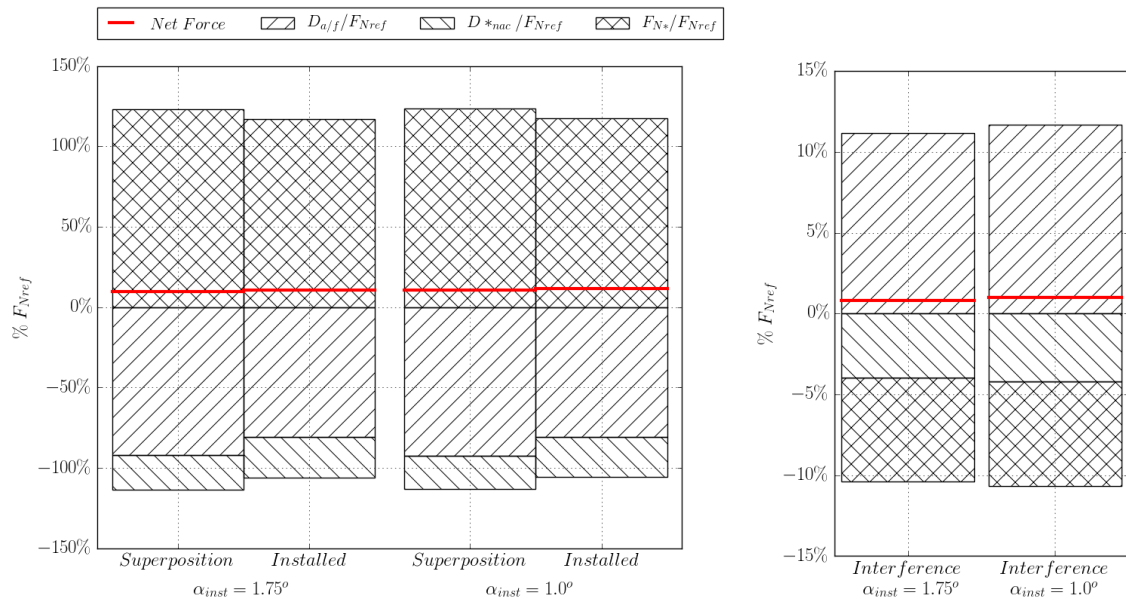


FIGURE 5.72: Stackup of forces acting on the isolated and installed E3 engine and airframe in datum position

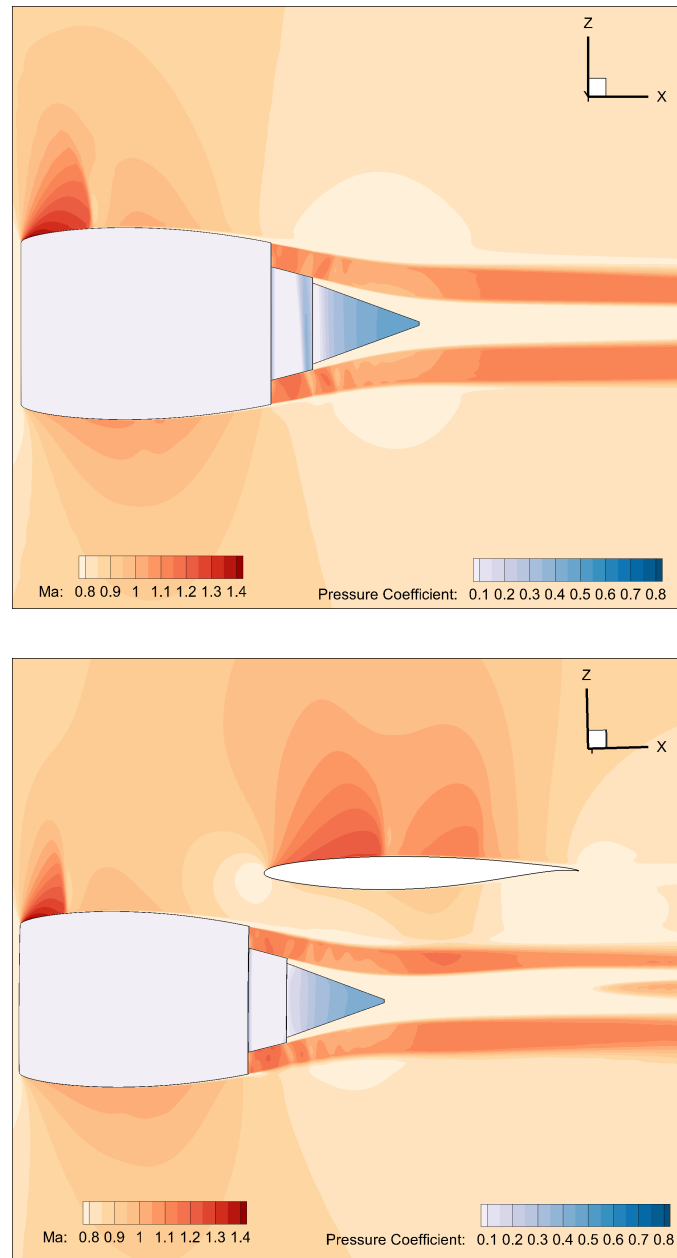


FIGURE 5.73: Isolated, $\alpha_{CL} = 4^\circ$ and installed, $\alpha_{a/c} = 2^\circ$, flowfields for the E3 engine installed in position C3

5.10 Conclusions

Throughout this chapter a number of conclusions have been arrived at and the consequences of these have a direct impact on the design of future UHBPR engines.

- The specific configuration used to model an engine in CFD has a large impact on the predicted drag. Notably, the inclusion of an experimental sting resulted in around a 20% reduction in the drag relative to a simple pressure matched exhaust.
- The inclusion of an exhaust streamtube can double the modified drag and as such the post exit terms must be included for a representative drag prediction.
- Using the parameterisation outlined in section 3.2.2 a design space of novel nacelles can be explored using relatively few variables. However, simply adjusting length and thickness can result in nacelles which have convex sections. These nacelles with gradient inflections have very poor drag characteristics and should be avoided.
- In order to allow an exploration of the novel short and slim design space, an optimisation approach can provide well performing nacelles in terms of cruise drag, drag rise Mach number and spillage drag for conventional nacelles. An improvement has been demonstrated by the application of constraints on pressure distribution which results in nacelles with more conventional pressure distributions in a novel design space.
- A nacelle with a 23% reduction in length resulted in a 22% reduction in nacelle drag. However, these compact designs are more sensitive to off design condition.
- A reduction in nacelle length for a given required drag rise Mach number will result in worse spillage performance, which may be 9 times higher for the reduced length nacelle of $l_{nac}/r_{hi} = 3.1$ compared to a $l_{nac}/r_{hi} = 4.3$ nacelle. This can be alleviated with an increase in the trailing edge radius.
- A short and slim nacelle with typically exhibit a higher peak Mach number than a conventional nacelle as the curvatures are increased at the leading edge. In cruise, this can be expected to be around $\Delta M_{peak} = 0.1$ to $\Delta M_{peak} = 0.15$ higher for a $l_{nac}/r_{hi} = 3.1$ compared to a $l_{nac}/r_{hi} = 4.3$ nacelle at $M_\infty = 0.85$
- Whilst in cruise the short nacelle gave benefits in drag, the windmill condition has been demonstrated to be critical. In an end of runway incidence sweep, the shortest nacelle examined separated on the external fan cowl at $\alpha_{CL} = 22^\circ$ whilst the longest nacelle had not separated by $\alpha_{CL} = 30^\circ$.

- Additionally, a short nacelle in the windmill condition may result in a significant reduction in the diversion range due to slower optimal Mach number in diversion.
- Once installed on an aircraft the impact of reducing the nacelle length to $l_{nac}/r_{hi} = 3.1$ was a decrease in overall cruise aircraft drag of 3% relative to a conventional length nacelle ($l_{nac}/r_{hi} = 4.3$). Installation of an engine with a short nacelle can result in a greater interaction as a forebody shock may be brought closer to the wing pressure field which indicates to exploit the reduced drag of a short nacelle it should be designed in conjunction with the wing.

CHAPTER 6

Non-axisymmetric Engine Design Studies

Until this point, the design studies carried out have focused on an idealised engine model which incorporated no three dimensional geometry changes. This assumed the sideline of the nacelle was an accurate representation of the full geometry once revolved around the x-axis. However, in practice nacelles are designed in greater detail than this and include droop, scarf, and azimuthal variation of design parameters. This chapter focusses on the influence that these 3D elements have to the aerodynamics, performance and design approach.

6.1 Baseline non-axisymmetric design

To allow a reference point for subsequent design studies a naive design approach was taken to create a non-axisymmetric nacelle geometry. In this approach, droop and scarf were simply applied to a 2D nacelle aeroline. An optimal axisymmetric was chosen from the optimisation at $l_{nac}/r_{hi} = 3.1$ and $r_{TE}/r_{hi} = 1.0$ which was the same design used for incidence (section 5.7) and installation (section 5.9) studies in the previous chapter. A droop of $\theta_{droop} = 3.0^\circ$ and scarf of $\theta_{scarf} = 4.0^\circ$ was applied to the axisymmetric design to obtain new highlight coordinates for top and bottom aerolines. A 3D design was then created by applying the same non-dimensional parameters and boattail angles used for the sideline design. This approach is expected to be sub-optimal since the parameters applied were optimised for the sideline length and highlight position, whereas the topline is slightly longer and the bottomline is slightly shorter due to the addition of scarf. This resulted in a higher acceleration on the bottomline at $\alpha_{CL} = 0^\circ$ and a stronger shock relative to the bottomline

of the axisymmetric design at $\alpha_{CL} = 0^\circ$ (Figure 6.1). However, as the centreline incidence is increased to $\alpha_{CL} = 3^\circ$ the topline incidence increases whilst the bottomline incidence decreases. The impact of this acts to reduce acceleration on the bottomline and increase it on the topline as shown for the axisymmetric nacelle. For the non-axisymmetric nacelle, this means that the highest surface Mach number moves from the bottomline to the topline between $\alpha_{CL} = 0^\circ$ and $\alpha_{CL} = 3^\circ$ (Figure 6.1). If correctly designed this would result in the minimum peak Mach number occurring at the incidence at which the engine operates once installed on an aircraft. Since the non-axisymmetric nacelle was not designed in detail however the drag was $8.9\%F_{Nref}$ at $M_\infty = 0.85$, $MFCR = 0.7$, $\alpha_{CL} = 3^\circ$, whilst the axisymmetric nacelle had a drag of $7.8\%F_{Nref}$.

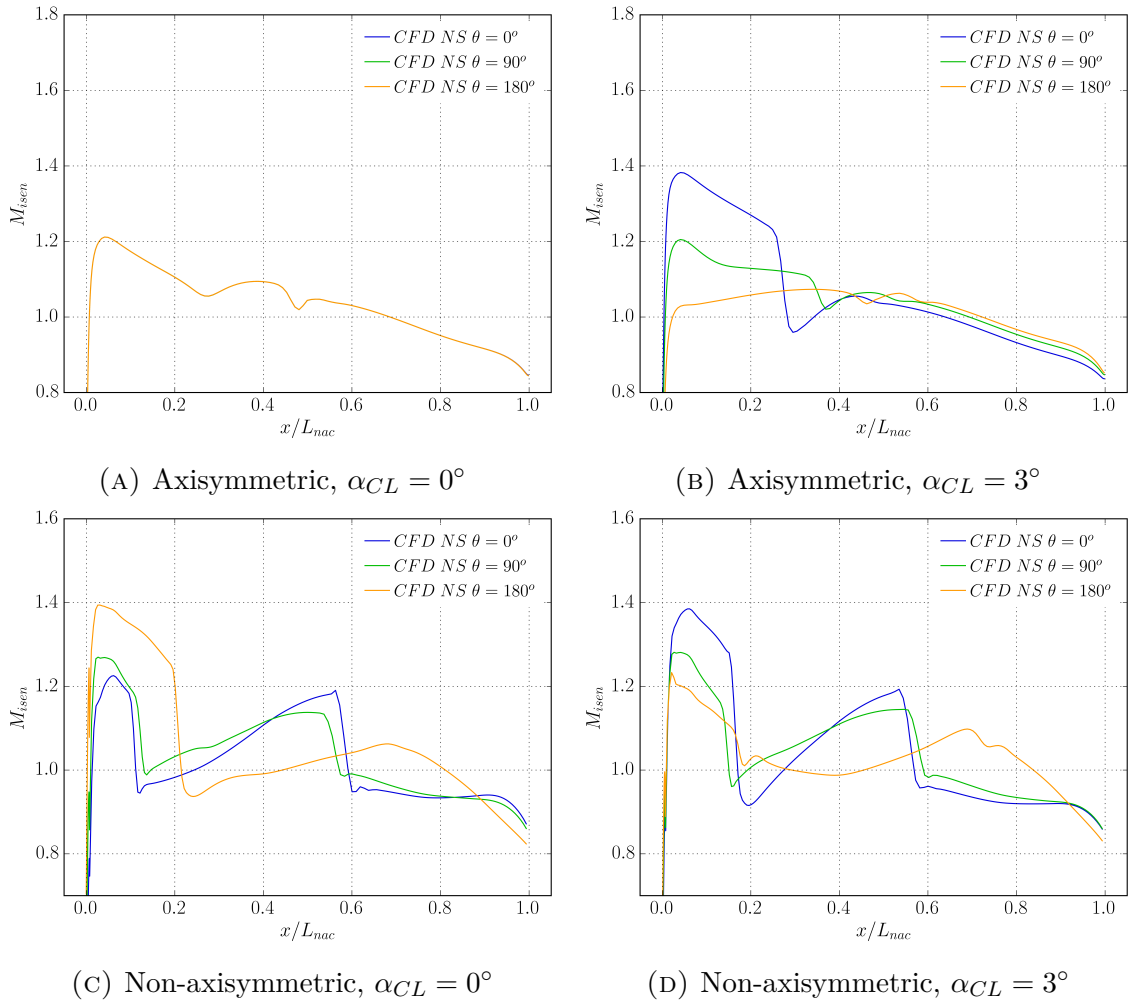
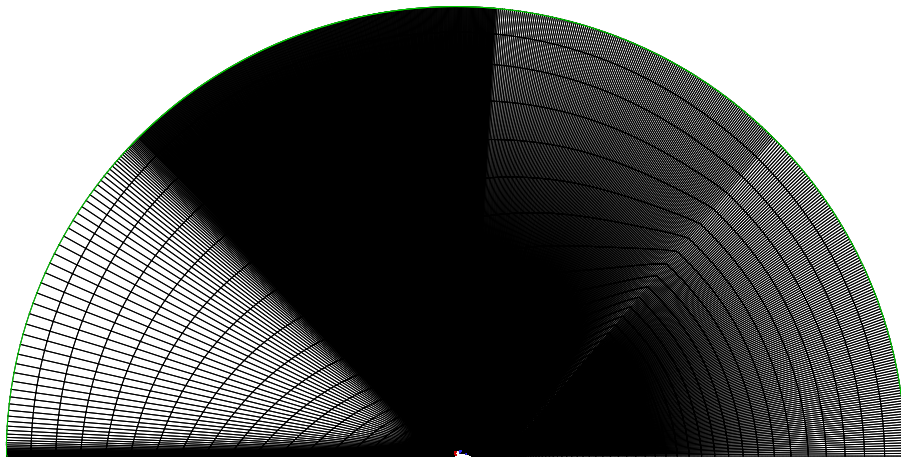


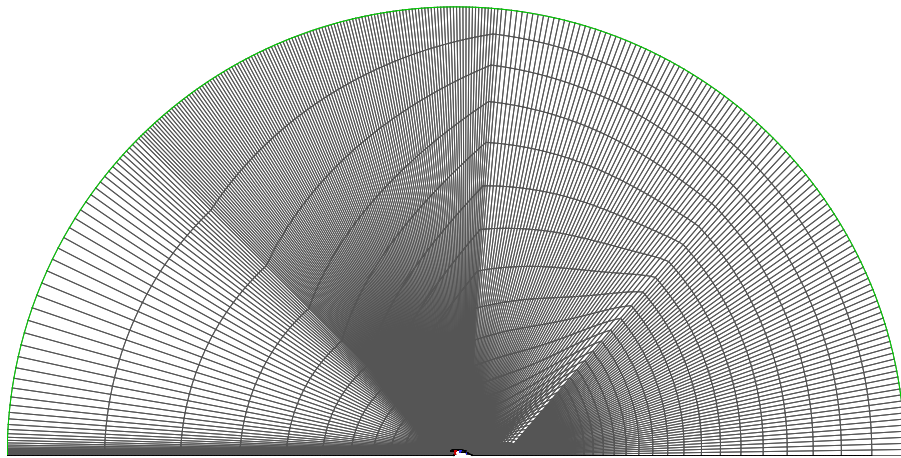
FIGURE 6.1: Comparison of axisymmetric and sub-optimal non-axisymmetric nacelle designs at $M_\infty = 0.85$, $MFCR = 0.7$

6.2 Fidelity reduction to enable further studies

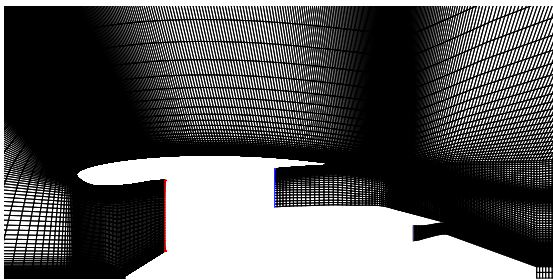
Studies of optimisation and nacelle design which have been discussed in chapter 5 used the same mesh fidelity as the validated method (Chapter 4). As a result of this the process of performing the optimisation would take a prohibitive amount of time to enable a worthwhile analysis of non-axisymmetric nacelles as well. It was therefore decided to further develop the optimisation method in order to use lower fidelity simulation techniques. Mesh size was reduced by approximately 87% relative to the mesh used for all optimisations in chapter 5 (Figure 6.2). This also resulted in an increase in the y^+ from 1 to around 50, such that the $k - \omega$ SST turbulence model used wall functions to model the boundary layer. Regions in the mesh which were expected to have the greatest influence on the nacelle drag, such as the highlight were reduced less (Figure 6.2) than areas which weren't expected to influence greatly, such as the farfield.



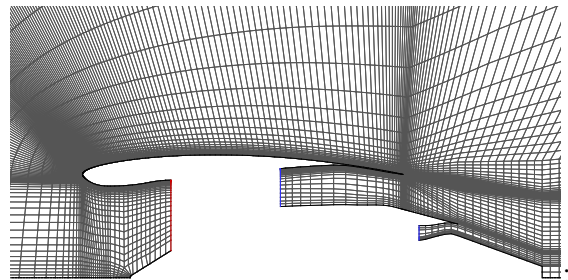
(A) High fidelity overall mesh



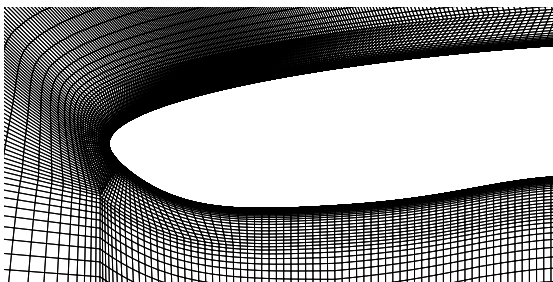
(B) Low fidelity overall mesh



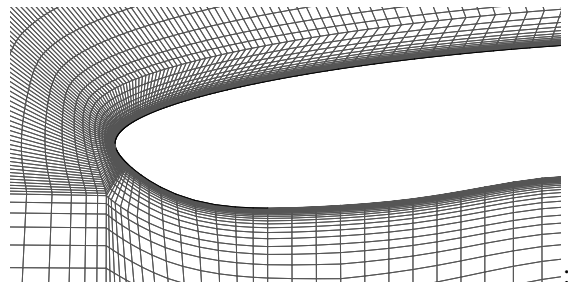
(C) High fidelity nacelle mesh



(D) Low fidelity nacelle mesh



(E) High fidelity highlight mesh



(F) Low fidelity highlight mesh

FIGURE 6.2: Mesh adaptations for reduced fidelity strategy

Verification tests

To ensure that the reduced mesh fidelity did not have too detrimental an effect on the overall optimisation process, two tests were carried out. Firstly, the aerodynamics and performance metrics were compared for a previously optimised design. Secondly, a full generation of designs was compared with low and high fidelities to establish whether the relative performances were correctly predicted.

An optimal design was chosen from the previous set of optimisation results and remeshed with the low fidelity approach. This reduced mesh size exhibited a 10% change in the estimated drag compared to the full fidelity mesh. The prediction of the pressure distribution on the cowl was very similar between the two fidelity meshes (Figure 6.3). The response to freestream Mach number was similar (Figure 6.3) however the low fidelity mesh predicted a higher positive gradient across the range tested and this resulted in a slightly lower drag rise Mach number by around $\Delta M_{DR} = 0.002$.

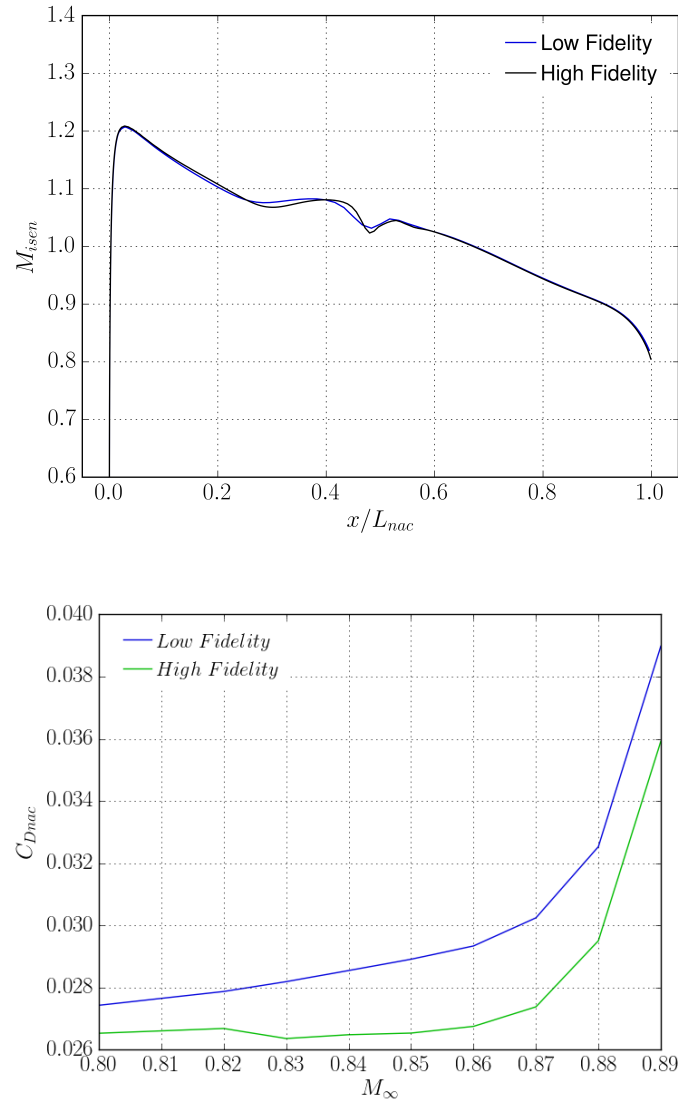


FIGURE 6.3: Comparison of high fidelity and low fidelity meshes for a single nacelle design

This offset was tested extensively in 2D to determine if it was constant across different designs and therefore if the optimisation would tend towards the same area of the design space. A previously computed generation of 50 designs was re-evaluated with the lower fidelity mesh (Figures 6.4,6.5,6.6), the offset was broadly the same at around 10-12% for most of the designs. The designs which did not follow the trend were all found to have unfavourable aerodynamics, typically predicted to cause multiple shock waves at cruise, and would therefore not be found to be optimal anyway. Similarly for the spillage metric a roughly constant offset was observed between the high and low fidelity mesh solutions (Figure 6.6). For the

designs which were particularly poor in the cruise drag, a notable disparity between the low fidelity and high fidelity designs exists but these would not be favoured by an optimisation algorithm due to their poor performance in the cruise drag metric. There is slightly more scatter in the high fidelity to low fidelity offset for the drag rise metric however over 95% had a negative offset from the reduced fidelity.

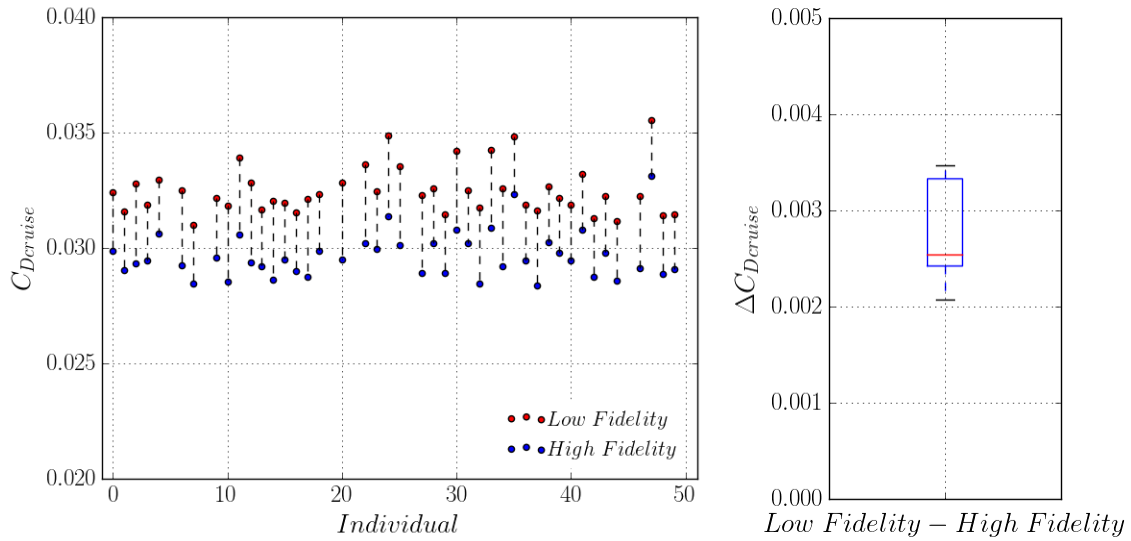


FIGURE 6.4: Comparison of high fidelity and low fidelity meshes for a full generation in terms of cruise drag

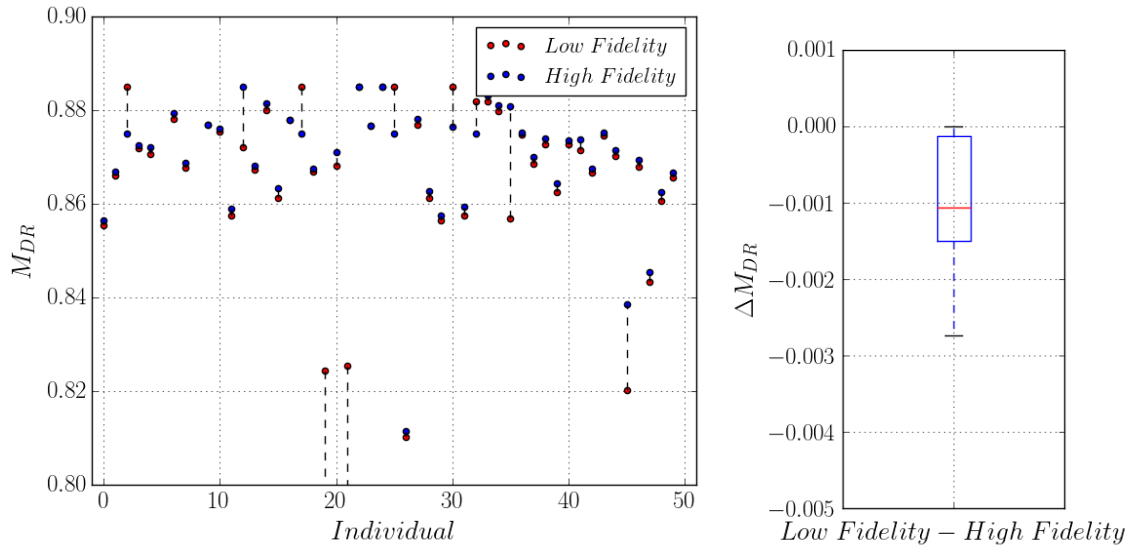


FIGURE 6.5: Comparison of high fidelity and low fidelity meshes for a full generation in terms of drag rise Mach number

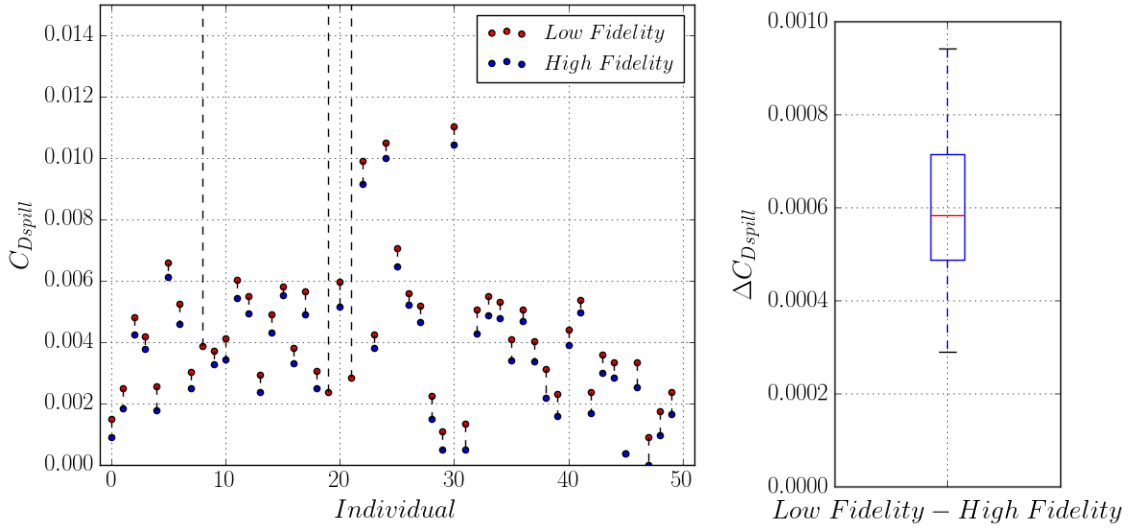


FIGURE 6.6: Comparison of high fidelity and low fidelity meshes for a full generation in terms of spillage drag

6.3 Non-axisymmetric optimisation

With the reduction in fidelity which had been achieved in the mesh, it was possible to reduce the time required for the optimisation procedure to within a viable timeframe to complete 3D CFD in the optimisation loop. A 3D optimisation was set up which consisted of a 3D geometry tool⁹⁹ in place of the 2D parameterisation used previously. This was parameterised along four azimuthal aerolines and the geometry assumed to have planar symmetry normal to the x-axis. The sidelines ($\theta = 90^\circ$ and $\theta = 270^\circ$) were fixed to a design which had been optimised in the 2D axisymmetric approach for the nacelle dimensions of $l_{nac}/r_{hi} = 3.1$ and $r_{TE}/r_{hi} = 1.0$. This left 3 parametric aerolines which were used as degrees of freedom (Figure 6.8). In addition to reduce the dimensionality of the problem, the control of y''_{TE} was removed as it was not shown to be very important to the designs (section 5.6). This resulted in an optimisation problem with 12 degrees of freedom; four on three aerolines. In addition, to minimise the computational cost, the drag rise metric used was substituted from a gradient based metric (Equation 3.44) to an offset between two freestream Mach numbers (Equation 6.1). In contrast to the gradient based drag rise Mach number, the offset metric (D_{Mach}) was to be minimised rather than maximised.

$$D_{Mach} = D_{nac}|_{M_\infty=0.87} - D_{nac}|_{M_\infty=0.85} \quad (6.1)$$

Since this optimisation was much more costly than previous optimisations and it had previously been determined that inflecting designs were poor aerodynamically an additional filter was added to this optimisation. The initial generation was 4000 designs of which 437 did not have inflections on any of the aerolines which were controlled (Figure 6.7). This filtering process removed different regions of the design space dependent upon which aeroline was considered. For instance, on the topline ($\theta = 0^\circ$) there were no viable designs with an $f_{max} < 0.27$ (Figure 6.7a). In contrast, the bottomline could be produced with these low values of f_{max} whilst there were greater restrictions on the combination of high f_{max} and high r_{max}/r_{hi} . This difference is due to the transformed length of the bottomline and topline and the transformed highlight position which will tend to favour a flatter bottomline and require a more curved topline forebody.

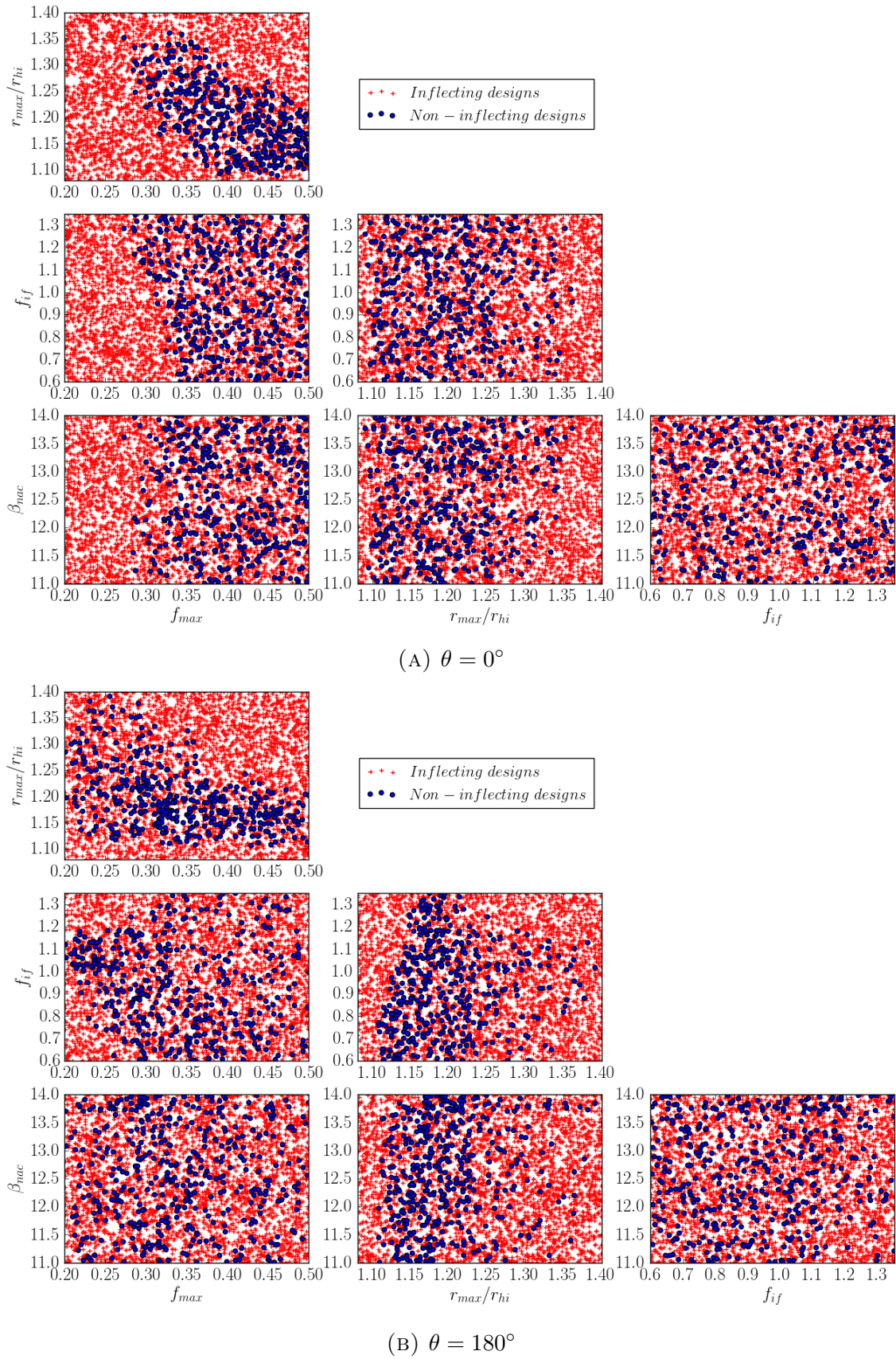


FIGURE 6.7: Filtered design space for the 3D optimisation for topline and bot-tomline

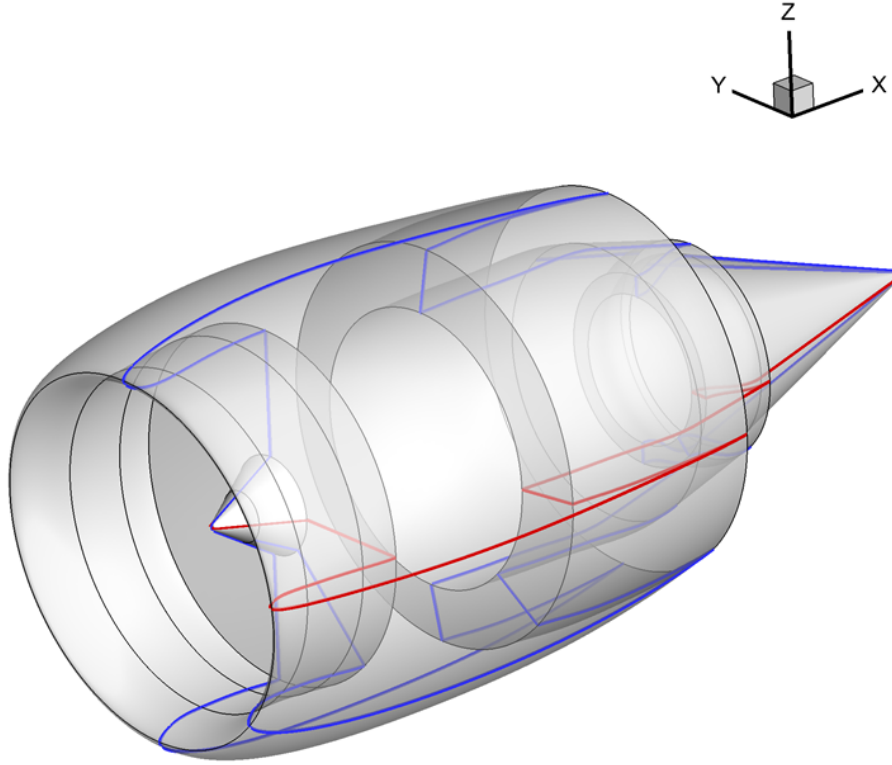


FIGURE 6.8: Sketch of aerolines where red is fixed, blue is variable in the optimisation

The result of this optimisation was again a Pareto front which corresponded to the best designs found in the search space (Figure 6.9). This Pareto set demonstrated the same trends as were observed in the axisymmetric optimisations of single aerolines. Specifically, that the drag incurred by an increase in Mach number could be reduced at the expense of the drag incurred by a reduction in MFCR.

Though the sideline geometry was fixed, the resultant aerodynamics are also a function of the full 3D design. There was therefore a considerable amount of variation from the axisymmetric sideline distribution and the sideline distribution in the non-axisymmetric designs created in the optimisation (Figure 6.10a). This reduced as the optimisation process was continued and the number of designs was reduced (Figure 6.10b). Four optimal designs were taken from the Pareto set to demonstrate the optimal aerodynamics based on the different performance metrics (Figure 6.12). The lowest drag result ($D_{cruise}/F_{Nref} = 0.064$, $D_{Mach}/F_{Nref} = 0.009$, $D_{spill}/F_{Nref} = 0.005$, Figure 6.12a) exhibited a high topline peak Mach number however this provides a suction force which acts on the forward facing area of the forebody. The lowest spillage design ($D_{cruise}/F_{Nref} = 0.067$, $D_{Mach}/F_{Nref} = 0.012$, $D_{spill}/F_{Nref} = 0.001$, Figure 6.12c) has a $\Delta M_{peak} = 0.07$ reduced topline peak Mach

number which improves the off-design spillage performance relative to the lowest drag design. The lowest D_{Mach} design ($D_{cruise}/F_{Nref} = 0.070$, $D_{Mach}/F_{Nref} = 0.004$, $D_{spill}/F_{Nref} = 0.013$, Figure 6.12b) has a very high peak Mach number as the topline design has a lower r_{max} than the other two designs and this increases the local curvature resulting in high accelerations. This results in a very poor spillage performance which makes the design untenable. A fourth design chosen from the Pareto set ($D_{cruise}/F_{Nref} = 0.068$, $D_{Mach}/F_{Nref} = 0.010$, $D_{spill}/F_{Nref} = 0.009$, Figure 6.12d) had the lowest overall peak Mach number of any design and represents a trade-off between the three performance metrics. All designs have similar performance on the $\theta = 135^\circ$ and $\theta = 180^\circ$ aerolines which indicates that the optimal designs are all very similar on the lower half of the nacelle and that the upper half is more instrumental in changing the performance metrics. The upper section has a higher forward facing area due to the droop angle of the nacelle and the topline is at a higher local incidence so is more sensitive to the aerodynamics.

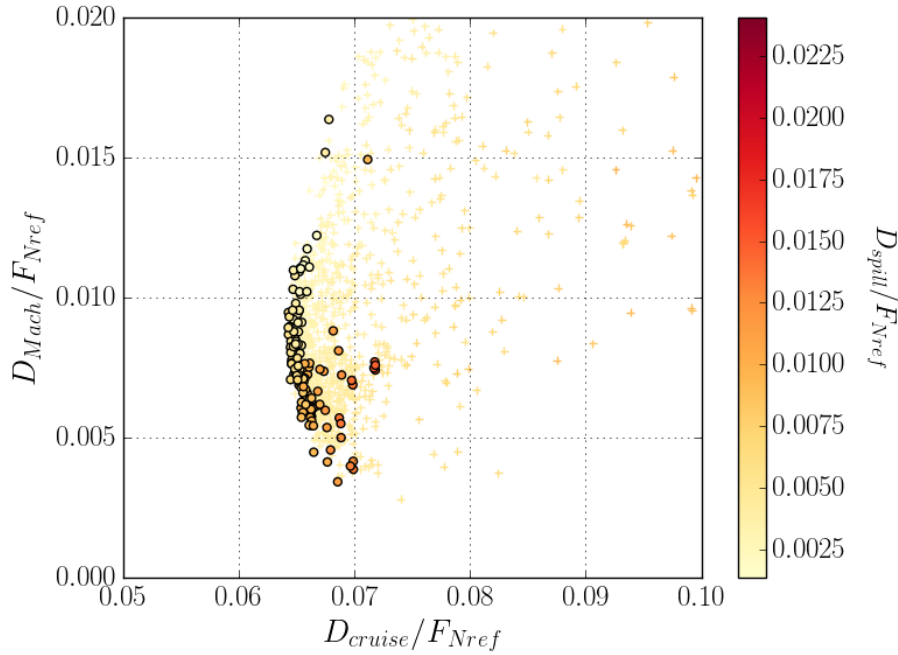
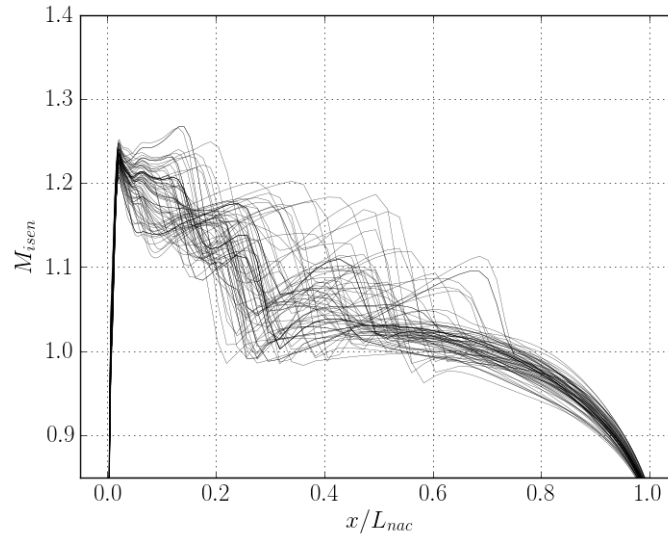
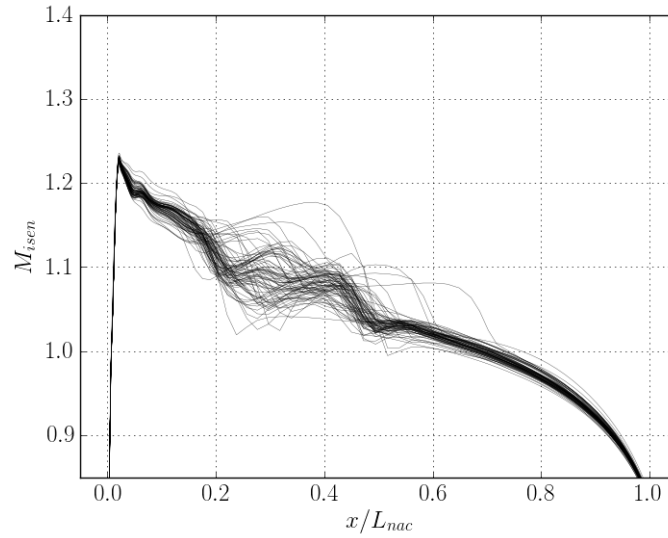


FIGURE 6.9: Final Pareto set for the 3D non-axisymmetric optimisation in objective space



(A) Generation 0



(B) Generation 19

FIGURE 6.10: Comparison of sideline isentropic Mach number distributions for all designs in generation 0 and 19

Certain trends were evident in the design variables of the optimal individuals (Figure 6.11). For instance, the optimal initial forebody radius (f_{if}) varied considerably in the azimuthal distribution. The $\theta = 0^\circ$, which due to the addition of droop and scarf has a higher radial offset between highlight and trailing edge than the sideline, tends towards a higher f_{if} , whilst the lower two lines tend to lower values. This higher value on the topline should also aid in reducing the topline peak Mach number as it results in lower curvature values. The values of r_{max}/r_{hi} for all three aerolines were relatively close to the sideline value, which may indicate that the

optimal designs are constrained by the fixed sideline geometry. The values of f_{max} show a similar trend which again may indicate that the azimuthal variation in this parameter is constrained to the original sideline design. For eleven of the twelve design parameters the bounds were not hit by the optimisation process, however optimal designs existed at the edge of the f_{max} variable for $\theta = 0^\circ$ which suggests that this bound should be expanded.

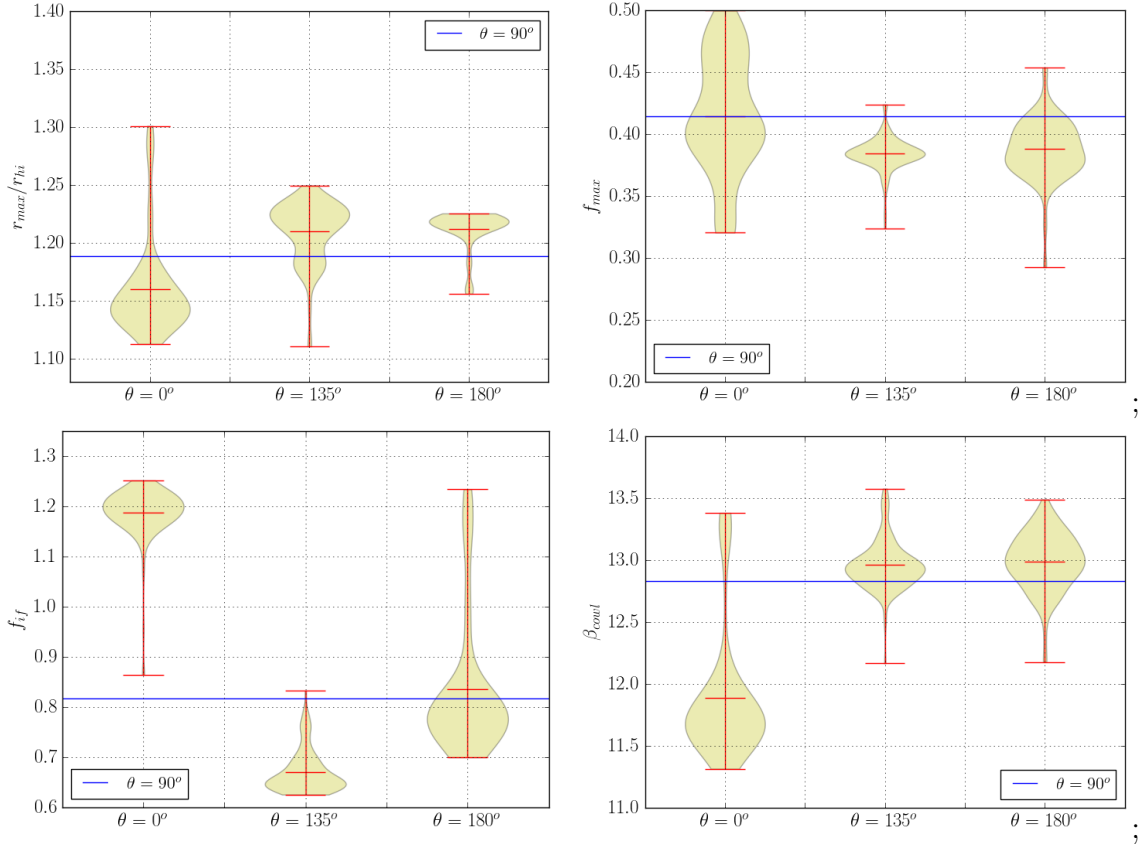


FIGURE 6.11: Final Pareto set for the 3D non-axisymmetric optimisation in design variable space. Violin plots indicate the distribution of the population, as well as maximum, minimum and median values

An optimal design (Figure 6.12d) was chosen from the Pareto set which had a relatively good distribution of isentropic Mach numbers. This design was chosen over the alternative Pareto individuals as it exhibited a favourable isentropic Mach number distribution which provided lower peak Mach numbers than the axisymmetric design at the same incidence and the installation impact was therefore expected to be lower. This design was then meshed and computed with the high fidelity mesh used in the previous studies. The design (Figure 6.13) had a topline peak Mach number at $M_\infty = 0.85$, $MFCR = 0.7$ and $\alpha_{CL} = 3^\circ$ of $M_{peak} = 1.24$. The axisymmetric

nacelle at the same length ($l_{nac}/r_{hi} = 3.1$) and trailing edge ($r_{TE}/r_{hi} = 1.0$) had a topline peak Mach number of $M_{peak} = 1.38$ indicating an improvement from the lengthened non-axisymmetric topline. The highest Mach number on the surface of the non-axisymmetric cowl however occurred around $\theta = 45^\circ$ near the highlight and was $M_{peak} = 1.35$. Since this was not an aeroline which was directly controlled it indicated that further degrees of freedom were required. This is addressed in Appendix E. The drag calculated from the high fidelity solution was $C_{Dcruise} = 0.0237$, whilst for the axisymmetric nacelle the drag was $C_{Dcruise} = 0.0235$. Whilst this is a slightly higher drag from the non-axisymmetric nacelle the reduction in topline peak Mach number should improve the other aspects such as incidence response (section 6.4) and installation (section 6.5). In addition, the isolated spillage drag at $\alpha_{CL} = 3^\circ$, $M_\infty = 0.85$ between $MFCR = 0.7$ and $MFCR = 0.63$ was $C_{Dspill} = 0.0039$ for the axisymmetric nacelle whilst it was $C_{Dspill} = 0.0033$ for the non-axisymmetric nacelle. In addition, the optimised non-axisymmetric nacelle compared favourably to the baseline non-axisymmetric nacelle (section 6.1) which had a cruise drag of $C_{Dcruise} = 0.0267$ at $M_\infty = 0.85$, $MFCR = 0.7$ and $\alpha_{CL} = 3^\circ$.

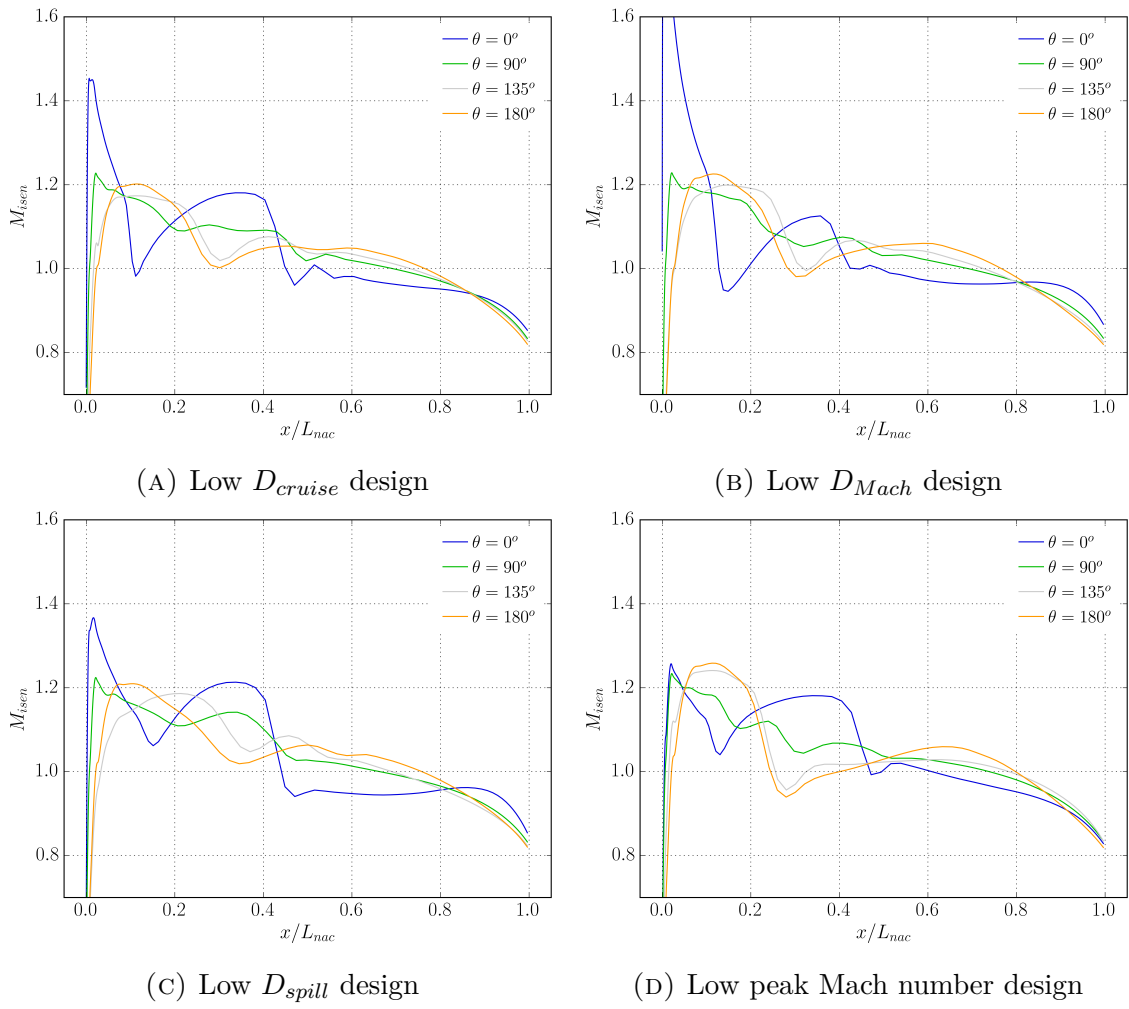
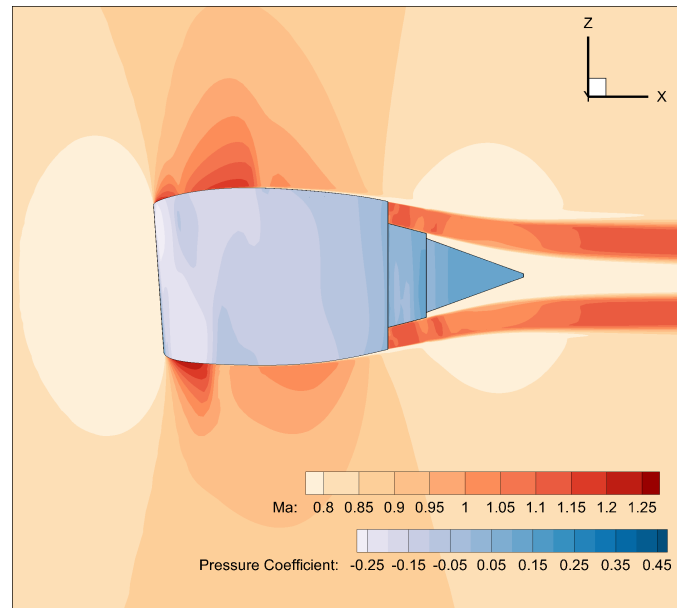
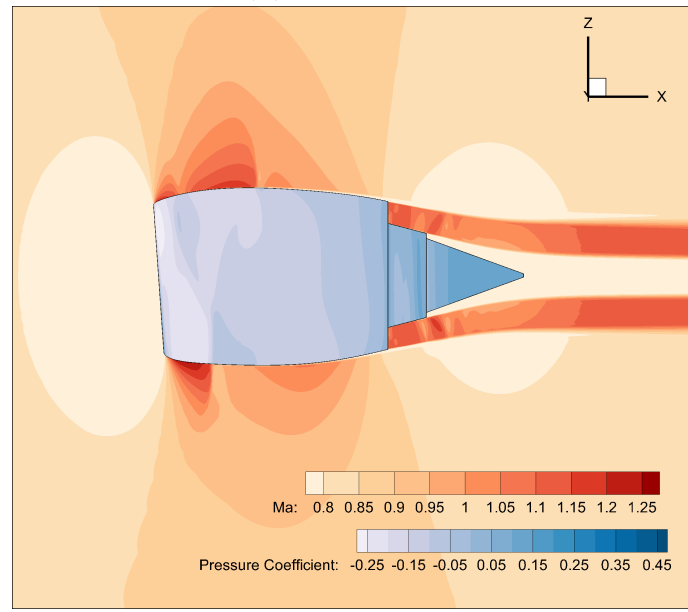


FIGURE 6.12: Four Pareto optimal designs from the non-axisymmetric optimisation



(A) Low Fidelity



(B) High Fidelity

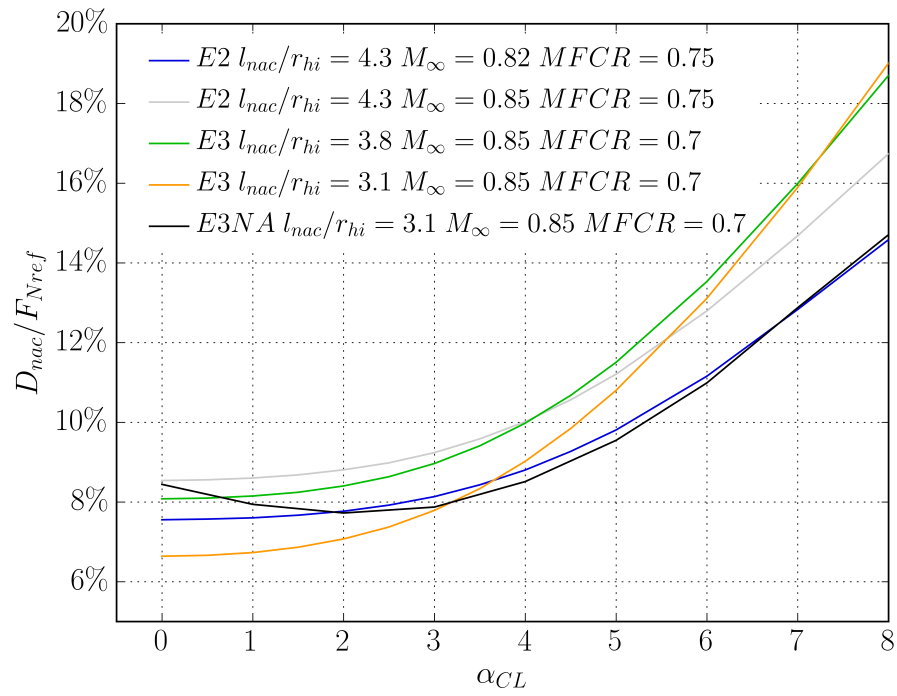
FIGURE 6.13: Comparison of low and high fidelity solutions for the chosen solution

6.4 Influence of Incidence

It was previously shown that the impact of incidence was greatest on a short nacelle (section 5.7). The impact of this was that the benefit in drag from a shortened nacelle can be lost as the operating incidence is increased. Specifically it was shown

that above $\alpha_{CL} = 4.0^\circ$ the nacelle with $l_{nac}/r_{hi} = 3.1$ had more drag than the nacelle with $l_{nac}/r_{hi} = 4.3$ and above $\alpha_{CL} = 7.0^\circ$ more drag than the nacelle with $l_{nac}/r_{hi} = 3.8$.

The addition of three degrees of droop and four degrees of scarf resulted in a greatly improved response to incidence (Figure 6.14). Since the design was optimised at $\alpha_{CL} = 3.0^\circ$ the drooped nacelle had the lowest drag at this incidence (Figure 6.14). This resulted in higher drag at lower incidences, $\alpha_{CL} < 3.0^\circ$, as the bottomline effectively saw a higher incidence which resulted in stronger shock on the bottom of the nacelle (Figure 6.15a). At the design incidence of $\alpha_{CL} = 3.0^\circ$, the loading on the different azimuthal aerolines have a roughly even peak suction (Figure 6.15b). This balance between the azimuthal lines is what results in the minimum drag at this incidence. In contrast, the axisymmetric nacelles all had a balanced peak suction across the azimuthal aerolines at $\alpha_{CL} = 0.0^\circ$ and the topline progressively worsened as the incidence was increased, whilst the bottomline improved (Figure 5.43). The application of the optimisation process to a nacelle with a non-zero design incidence and non-axisymmetric aerolines can therefore allow the design to be suited to a required cruise incidence. This higher design incidence not only ensures the lowest drag at the given angle of attack but also improves the response to higher incidences (Figure 6.14). Effectively, this improvement is also as a result of the topline peak suction (Figure 6.15) being reduced relative to the axisymmetric case of equivalent nacelle length (Figure 5.43). The outcome of this is that in the region of incidences which the nacelle might operate once installed ($\alpha_{CL} = 3.0^\circ$ - $\alpha_{CL} = 5.0^\circ$) the non-axisymmetric nacelle has the lowest levels of drag, lower even than the conventional nacelle operating at the lower Mach number and higher MFCR. The non-axisymmetric $l_{nac}/r_{hi} = 3.1$ nacelle at $\alpha_{CL} = 4.0^\circ$ has 3% less drag than the axisymmetric $l_{nac}/r_{hi} = 4.3$ nacelle, 6% less than the axisymmetric $l_{nac}/r_{hi} = 3.1$ nacelle and 15% less than the axisymmetric $l_{nac}/r_{hi} = 3.8$ nacelle (Figure 6.14).

FIGURE 6.14: C_{Dnac}/F_{Nref} as a function of α_{CL} for four different nacelle designs

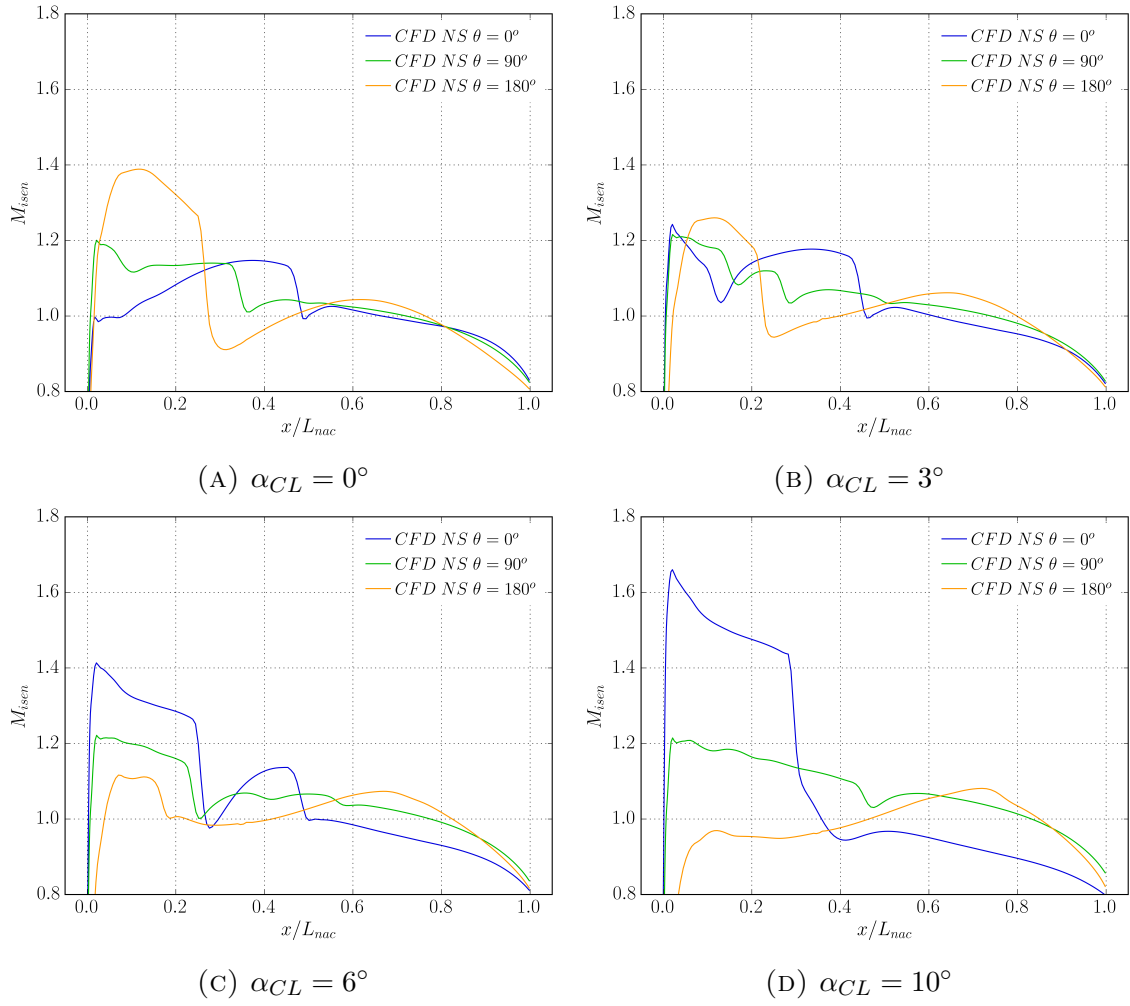
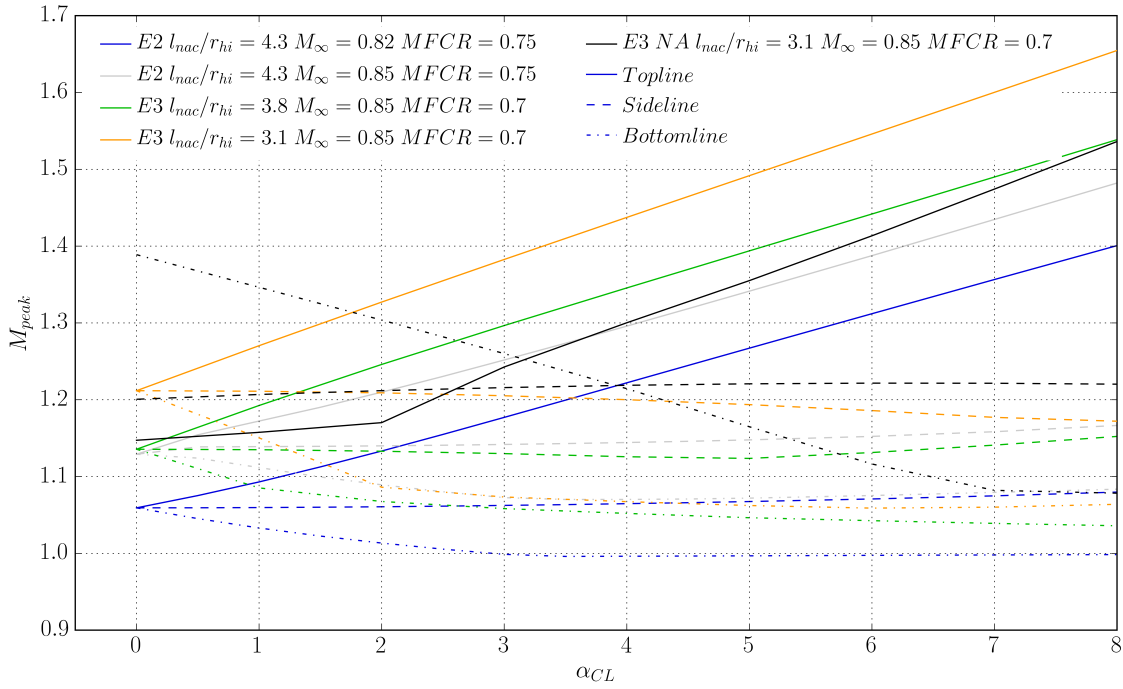


FIGURE 6.15: Isentropic Mach number distributions for the optimal non-axisymmetric engine designs as a function of incidence (α_{CL}) in the cruise condition

FIGURE 6.16: M_{peak} as a function of α_{CL} for four different nacelle designs

6.5 Influence of Installation

6.5.1 Baseline Position

The non-axisymmetric E3 engine nacelle (E3NA) was installed at an installation angle of $\alpha_{inst} = 1.0^\circ$ in the datum position. At the lower values of incidence, the E3NA nacelle had a worse drag performance than the axisymmetric E3 nacelle in isolation, and this is mirrored in the installed case (Figure 6.17). The breakdown of interference between the superposition model and the installed cases was very similar between the axisymmetric and the non-axisymmetric nacelles (Figure 6.18). However, the E3NA had a $0.6\% F_{Nref}$ increase in the beneficial airframe interference (Figure 6.18). This is caused by an increase in the suction of the lower surface wing which is marginally increased for the E3 NA configuration (Figure 6.19). Conversely there was no evident impact on the upper surface of the wing (Figure 6.19). This improvement on the airframe was partially balanced by an increase in the detrimental nacelle interference (Figure 6.18). The influence of the wing pressure field on the nacelle is again to increase the back pressure and provide a reduction in the Mach number on the afterbody of the topline of the nacelle (Figure 6.20). In addition, the

double shock pattern which is present in the isolated case is influenced more (Figure 6.20) than the single shock pattern on the axisymmetric nacelle (Figure 5.70). This result in the acceleration moving from aft shock to forward shock.

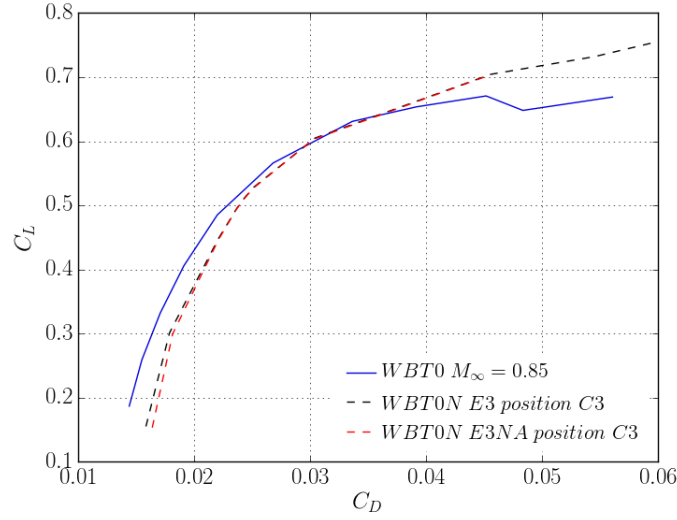


FIGURE 6.17: Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E3 and E3NA engines installed in the datum position

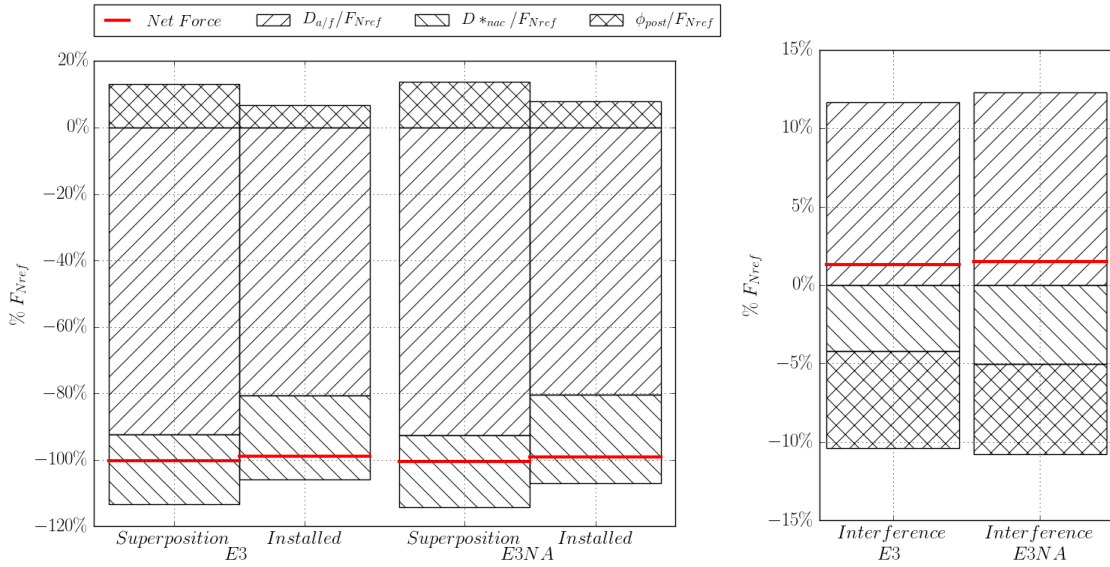


FIGURE 6.18: Stackup of forces acting on the isolated and installed E3NA engine and airframe in the datum position

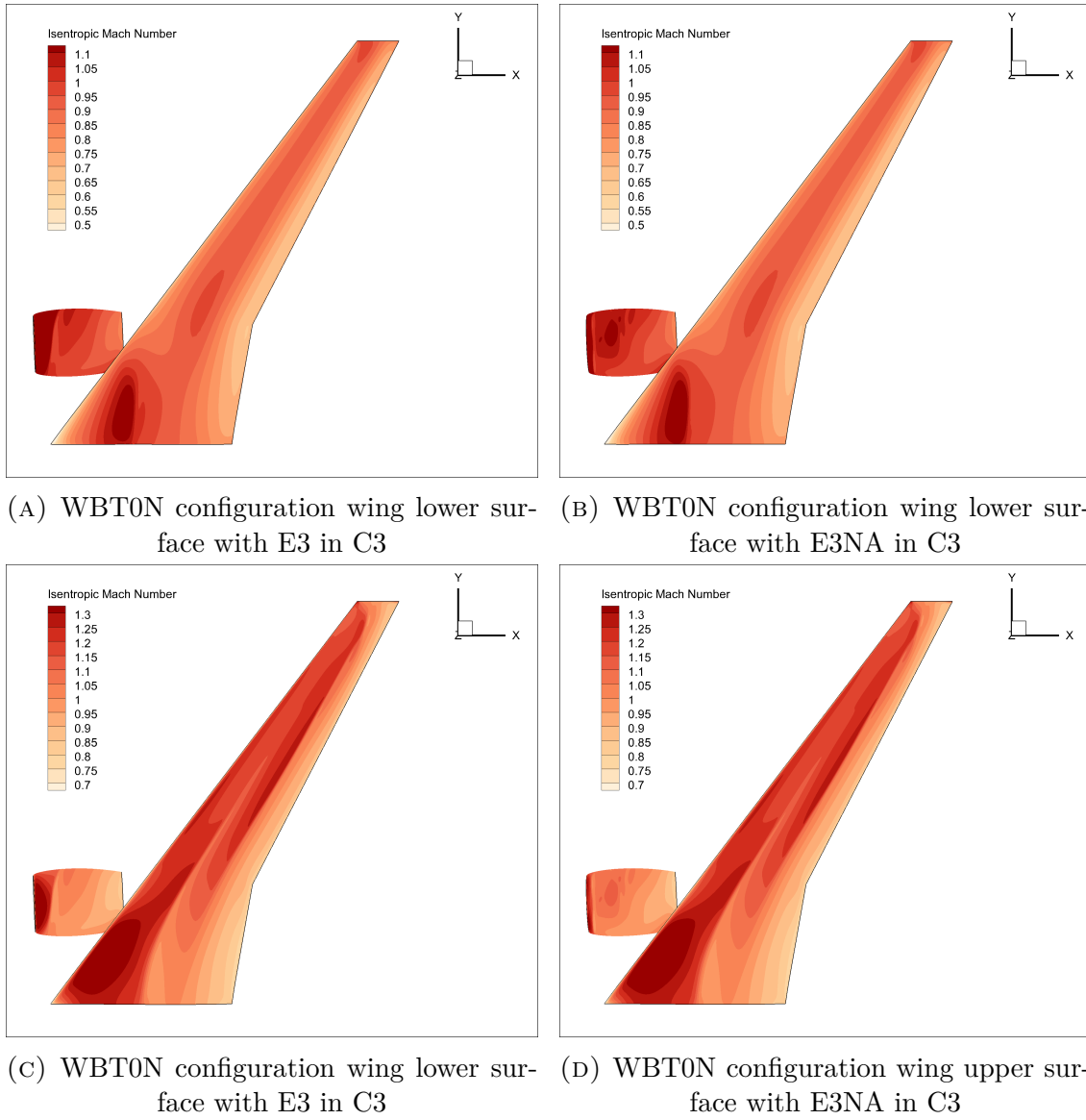


FIGURE 6.19: Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the CRM with the E3 and E3NA engines installed in the datum installation position

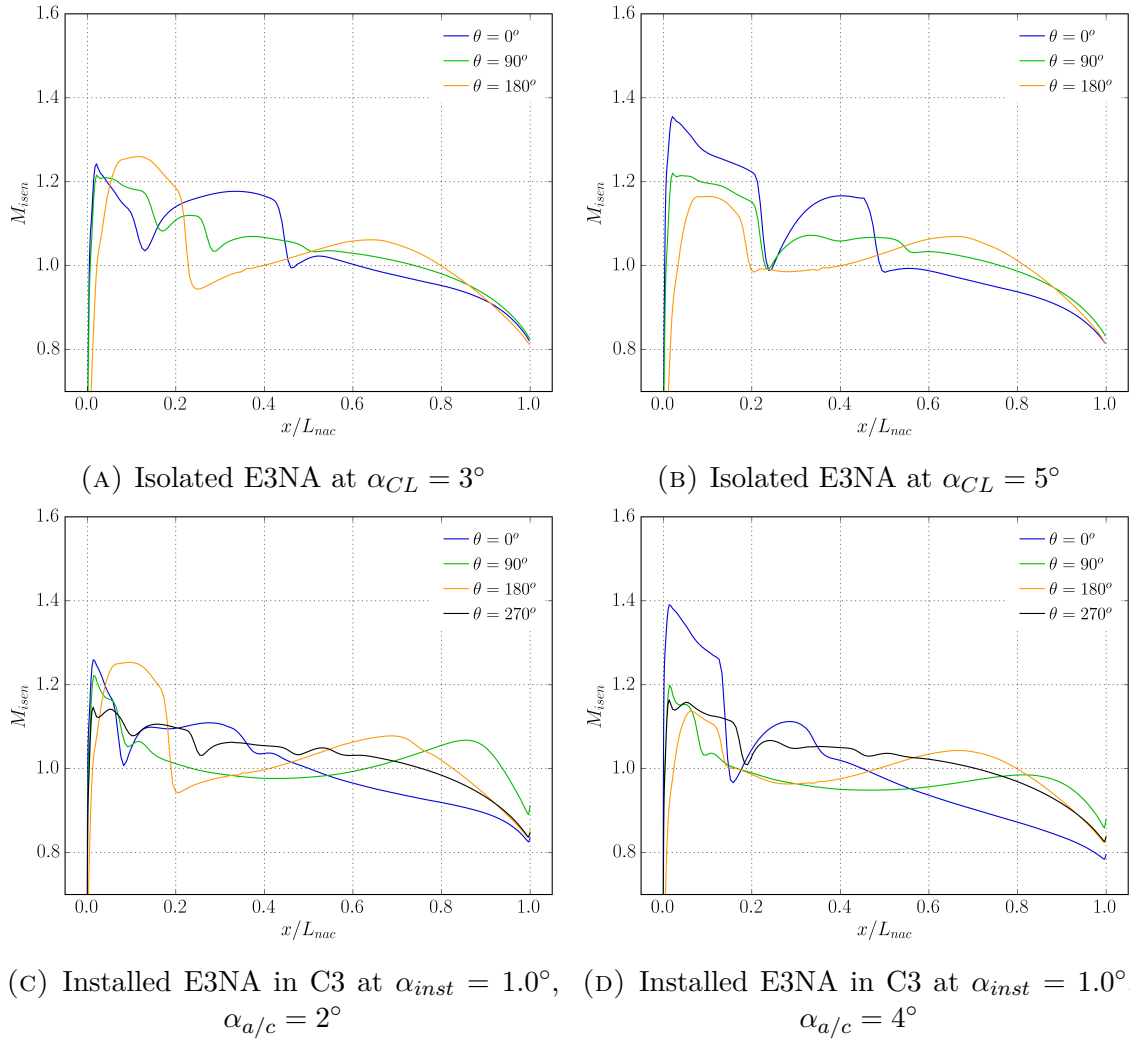


FIGURE 6.20: Isentropic Mach distributions on the E3NA nacelle in isolation and installed in datum position

6.5.2 Close-coupled Position

To establish if a similar trend between the axisymmetric E3 nacelle and the non-axisymmetric E3 NA nacelle was present in a more closely coupled configuration, the two engines were installed in the close-coupled position (Figure 3.6). This position corresponds to the same axial displacement from the wing leading edge but the nacelle is positioned $0.1c_{local}$ closer to the wing vertically. It is likely that larger engines will need to be closer coupled in this manner to avoid difficulties with ground clearance.

In the close-coupled position both engines experienced an increase in the overall aircraft drag of $\Delta C_{D,a/c} = 0.003$ at $C_L = 0.5$ relative to the datum position. This

indicates that they both experienced a similar detriment in the aerodynamics due to the closer coupling. This is partly caused by an increase in the effective nacelle incidence of $\Delta\alpha_{CL} = 0.2^\circ$ due to the increased upwash in the close coupled position. However this has less than a $C_{D,a/c} = 0.0001$ impact on the superposition model (Figure 6.22) relative to the datum position (Figure 6.18). As such, the increase in drag is due to the interference effects, with the largest impact on the airframe (Figure 6.22). The main interaction which caused the reduction in beneficial airframe interaction relative to the datum position was an increase in the wave drag on the lower surface of the wing which is present for both E3 and E3 NA (Figure 6.23).

The close-coupling between nacelle and wing results in a smaller gully space between the wing and the nacelle (Figure 6.24) relative to the datum case. This results in a flow acceleration through this gully space. In addition, the gully between the fuselage reduces due to the vertical displacement and this results in an increase in the velocity in the sideline plane of the nacelle and underside the wing (Figure 6.24). This increase of Mach number on the lower surface of the wing (Figure 6.23d) relative to the datum position (Figure 6.19) causes the relative increase in drag on the airframe.

The spillage performance of the installed E3 and E3 NA were also assessed at $\alpha_{CL} = 2.3^\circ$. At this incidence both configurations generated lift of $C_L = 0.49$, which was approximately the design lift coefficient. At MFCR=0.7, $M_\infty = 0.85$, both the E3 configuration and E3 NA configuration had $C_{D,a/c} = 0.0238$. In the spillage condition of MFCR=0.63, $M_\infty = 0.85$, the E3 configuration drag increased by $\Delta C_{D,a/c} = 0.0005$ whilst the E3 NA configuration increased by $\Delta C_{D,a/c} = 0.0004$ which indicated a spillage improvement on the non-axisymmetric nacelle.

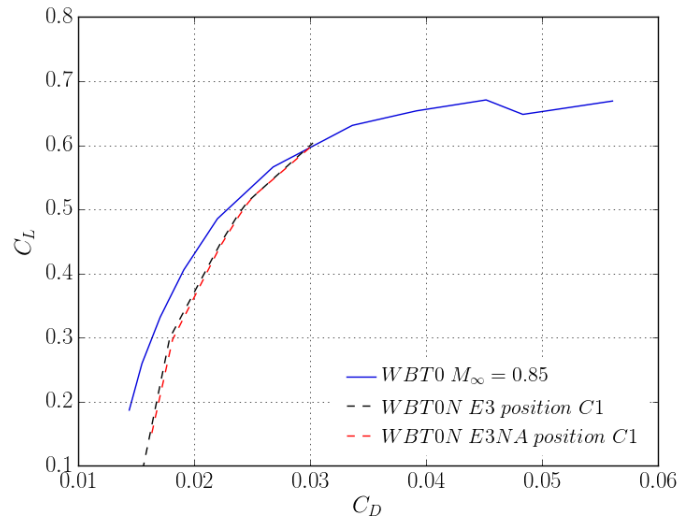


FIGURE 6.21: Drag polar for the CRM clean wing aircraft (WBT0) at $M_\infty = 0.85$ and for the CRM with the E3 and E3NA engines installed in the close-coupled position

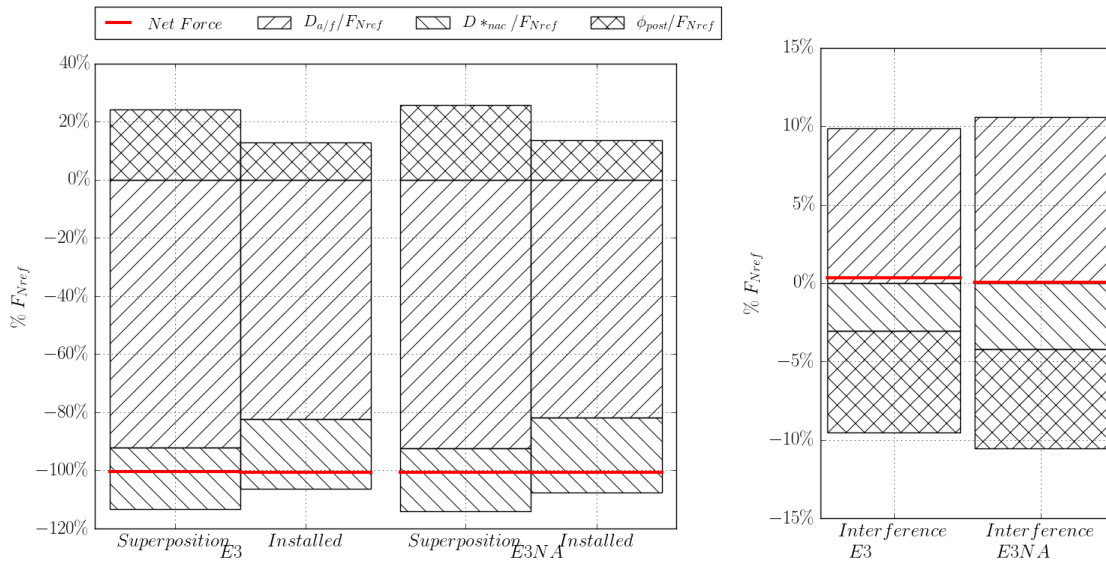


FIGURE 6.22: Stackup of forces acting on the isolated and installed E3NA engine and airframe in close-coupled position

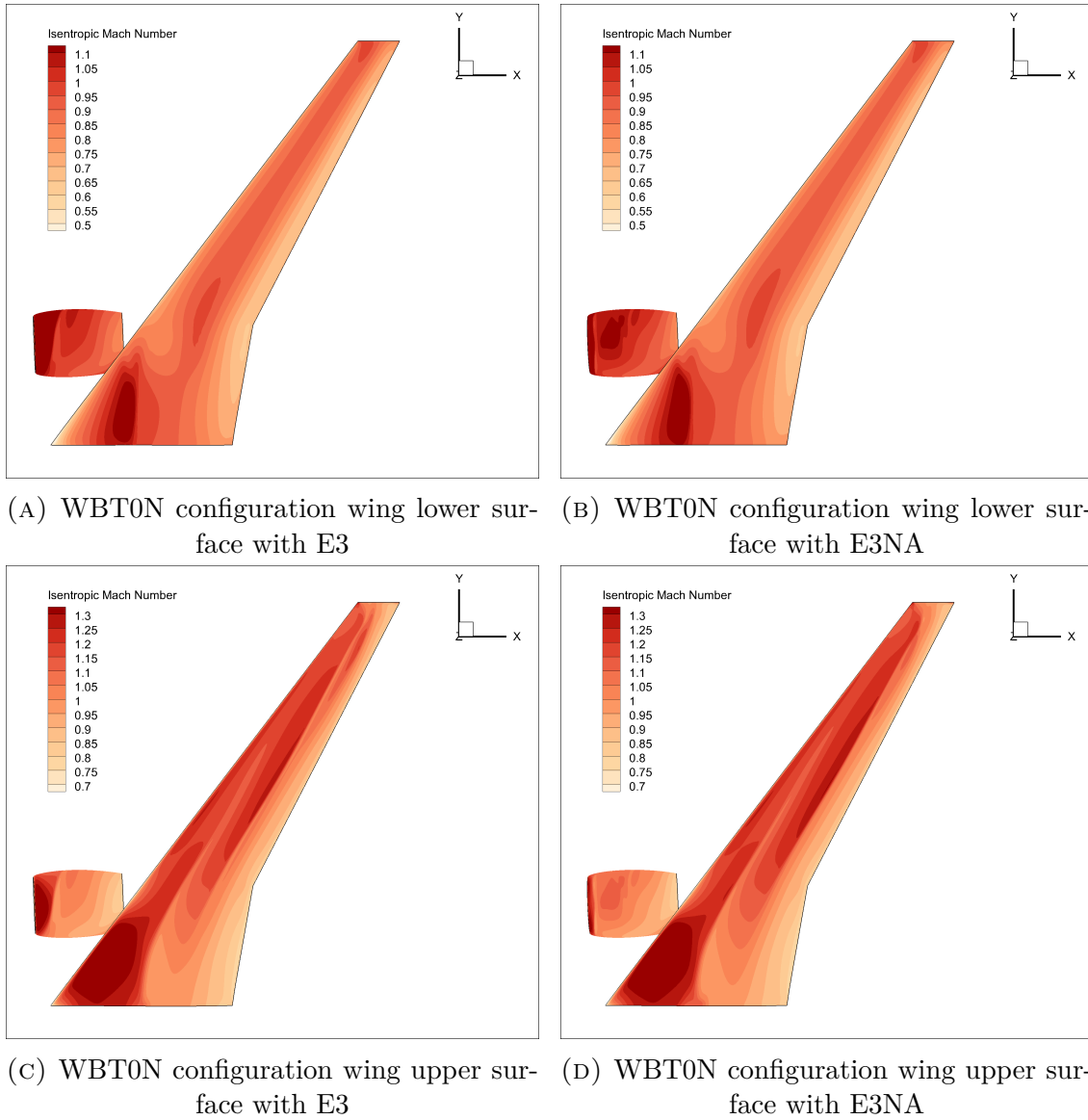


FIGURE 6.23: Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the CRM with the E3 and E3NA engines installed in close-coupled configuration

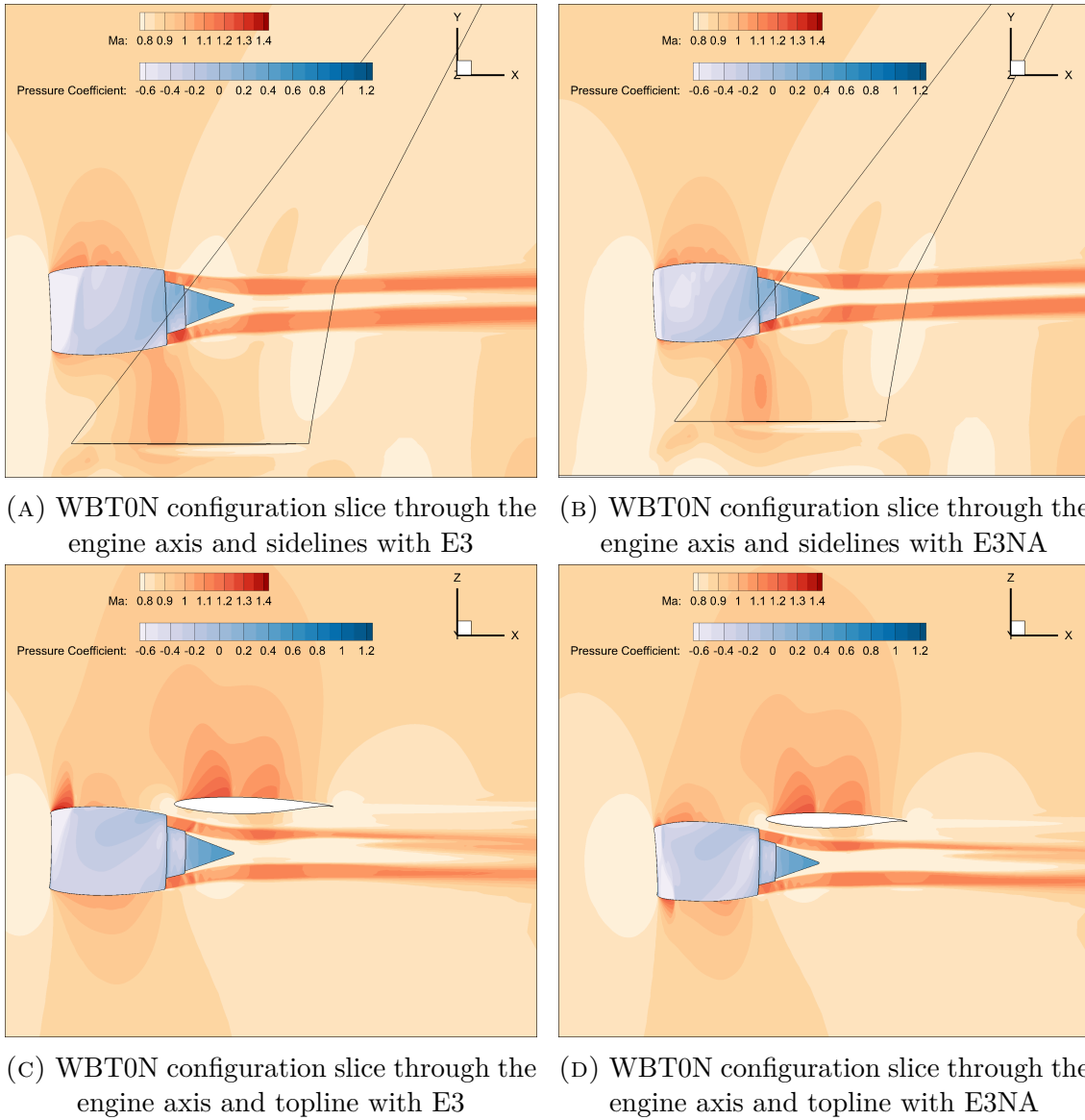


FIGURE 6.24: Isentropic Mach contours on the upper surface of the wing at two Mach numbers for the CRM with the E3 and E3NA engines installed in close-coupled configuration

6.6 Conclusions

The work undertaken in this chapter has focussed on non-axisymmetric nacelle design and interaction with incidence and installation. The key outcomes and conclusions were as follows:

- The optimisation method can be expanded to non-axisymmetric nacelles through the control of azimuthal aerolines.

- It was established from this work that the nacelle should be parameterised by at least five aerolines to ensure adequate control over the peak Mach numbers.
- The use of the optimisation provides a viable means to produce a design which is an improvement over the simple approach of using the same design variables for each section.
- The use of droop and scarf on the nacelle can reduce the topline peak Mach number relative to an axisymmetric configuration. This reduces the drag at higher incidences relative to the axisymmetric case.

CHAPTER 7

Conclusions and Future Work

7.1 Conclusions

7.1.1 Conclusions on drag evaluation

This thesis has outlined the progress made through a project which aimed to assess the aerodynamic impact of novel nacelle designs for ultra-high bypass ratio engines. The initial research focussed on the validation of an existing CFD process^{5,98} which provided validation data for the aerodynamic drag of an aero-engine nacelle. The method was determined to be able to predict an indicative mid-cruise drag within around 4% of the experimental data. In addition, sensitivity to freestream Mach number, MFCR and incidence were determined. The drag rise Mach number (M_{DR}) was predicted within $\Delta M_{DR} = 0.003$ at a typical mass flow capture ratio of 0.70 for the nacelle most representative of those considered in the project. The wind tunnel data which was validated against included sting interference. The use of a sting in experimental work produces an increase in the static pressure at the trailing edge of the nacelle which decreases the measured drag ($C_{D'}$) and increases the drag rise Mach number (M_{DR}). Due to a similar effect, the inclusion of an exhaust system with the nacelle, as well as representative efflux flows generating a realistic exit streamtube, greatly increases the modified nacelle drag (C_{D*nac}) and significantly lowers the drag rise Mach number (M_{DR}). Relative to a typical civil aero-engine configuration, with separate jets, the typical wind tunnel configuration of a sting mounted nacelle results in a 20% reduction in cruise drag (C_{Dnac}), a $\Delta M_{DR} = 0.007$ increase in drag rise and a 12% of cruise drag increase in the spillage drag (C_{Dspill}). Conversely, relative to the separate jets exhaust, the computational

modelling approach of using a simple idealized exhaust results in a 18% increase in cruise drag (C_{Dnac}), a $\Delta M_{DR} = 0.002$ increase in M_{DR} and an increase in the spillage drag of 13% of the cruise drag. This indicates that the inclusion of a sting for wind tunnel drag testing is likely to result in an optimistic drag prediction relative to a full configuration whilst the simplified model without exhaust system will overpredict the drag. For an estimation of drag of a nacelle, the influence of the exhaust system therefore must be included. In addition, prediction of the sting drag with the use of a flat plate correction is not accurate as the boundary layer after the nacelle is significantly different from that of a flat plate. An alternative formulation of this correction has been demonstrated in Appendix C. However, the accurate drag estimation of the nacelle would need a correction for the efflux.

7.1.2 Conclusions on optimisation method

A novel geometric design space was identified based on analysis of existing engine nacelle designs. An initial attempt to understand this design space was carried out with a full factorial design of experiment. The designs determined by this approach were assessed with the validated CFD method. It was evident from the results of this analysis that the designs required additional control to enable viable aerodynamics and geometries without convex sections. In order to assess a higher dimensional design space an alternative approach was developed since a full factorial would be costly. Based on NSGA-II, this optimisation method has been developed and tested on a series of analytical functions. Separately the ability of the tool was demonstrated to converge well to single and multi-objective problems. Since the optimal set which a genetic algorithm converges to is dependent on the settings and initial conditions of the optimisation a series of different configurations were tested. The influence of the initial random sample size used to start the generational process as well as the number of generations used was assessed in terms of the volume of the final Pareto set. It was demonstrated for an analytical function that with appropriate settings the solution could converge to 99.5% of the analytical solution. An alternative means of initialising a genetic algorithm was also proposed and demonstrated in this thesis, whereby the initial population is not fully random. Instead, a Latin Hypercube set of individuals is created within the bounds of the search space. This sample is used as the initial population and the process continued in the same manner as the original NSGA-II. This has the benefit of greatly improving

the probability that a single optimisation procedure will converge to a maximal hypervolume given appropriate mutation, population and generation settings without increasing the number of evaluations. This should be adopted for the initialisation of a genetic algorithm as it is unlikely to increase the optimisation time excessively when CFD is used as the evaluation method. The optimisation method demonstrated that to enable a well designed nacelle, the objective functions minimised should include control over the pressure distributions. The method, demonstrated for both 2D and 3D geometries allows a reliable exploration of the design space of novel, short and slim, nacelles.

7.1.3 Conclusions on short nacelle performance

A short nacelle ($l_{nac}/r_{hi} = 3.1$) was demonstrated to have a 22% reduced drag relative to a more conventional length ($l_{nac}/r_{hi} = 4.3$) at $\alpha_{CL} = 0^\circ$. A study of nacelle installation was undertaken for underwing positions. It was demonstrated that an engine with a short nacelle ($l_{nac}/r_{hi} = 3.1$) had a much more coupled relationship with the aircraft relative to a more conventional length nacelle ($l_{nac}/r_{hi} = 4.3$). Overall, the benefit in drag for the aircraft with the short nacelle was 3% of the overall drag however this benefit was largely due to beneficial pressure forces on the airframe caused by increased suction on the forward facing section of the wing. This indicates that the aerodynamics for short nacelles must be considered in conjunction with the airframe to maximise the drag benefit. The sizeable improvement in aircraft drag could potentially then be worth \$36m per year, for an aircraft.²⁴ If designed in conjunction with a wing, exhaust system and suitable intake, the drag improvements could form part of a plausible solution for future ultra-high bypass ratio engines. However, due to the generally higher peak Mach numbers which were present on the $l_{nac}/r_{hi} = 3.1$ nacelle, the drag increased more rapidly with incidence in isolated engine simulations relative to the $l_{nac}/r_{hi} = 4.3$ nacelle. This meant that by $\alpha_{CL} = 6^\circ$ the drag was equal between the $l_{nac}/r_{hi} = 3.1$ nacelle and the $l_{nac}/r_{hi} = 4.3$ nacelle. Whilst the incidence of the nacelle in cruise was not above $\alpha_{CL} = 4.5^\circ$ in any of the configurations within this thesis, if it were required to increase the installation angle or the coupling with the wing then this trade-off must be considered. In addition, the $l_{nac}/r_{hi} = 3.1$ had prohibitively high drag in two separate windmill conditions which were evaluated. For a diversion mission, the $l_{nac}/r_{hi} = 3.1$ nacelle had an optimal diversion Mach number of $M_\infty = 0.45$ whilst the $l_{nac}/r_{hi} = 4.3$ nacelle had an optimal diversion Mach number of $M_\infty = 0.65$.

For an example diversion of the A350¹¹⁹ this would result in a reduced ETOPS range of approximately 32%. Similarly in an end of runway windmill condition, the $l_{nac}/r_{hi} = 3.1$ nacelle had flow separation at $\alpha_{CL} = 22^\circ$ whilst the $l_{nac}/r_{hi} = 4.3$ nacelle did not separate below $\alpha_{CL} = 30^\circ$, which is the requirement.⁴⁹ Overall, the benefit of a short nacelle is significant in the cruise condition and worth further study. The windmill condition should be evaluated in further detail to establish whether the end of runway separation can be mitigated through beneficial installation or whether the increase in drag in a diversion is significant in terms of the overall aircraft system.

7.2 Future Work

Throughout the work several further avenues of work were established. These would each improve the understanding of the aerodynamics of aero-engines if explored. The most notable are as follows:

- The optimisation work outlined was all performed with the NSGA-II algorithm. It has been observed in the literature that this algorithm can sometimes converge to a Pareto set which is not evenly resolved. This could be the reason for much of the unevenness observed in the Pareto sets in this thesis. It is therefore suggested that alternative genetic algorithms be tested for nacelle optimisation. This should allow a more evenly resolved Pareto set which can verify the trends.
- Non-axisymmetric optimisation should be carried out with no fixed aerolines. This would help to determine if the optimal designs shown in this project could be improved if not constrained by a given sideline geometry.
- Additional work should be undertaken to establish the influence of the exhaust system on the installation. This could be done by parametrically varying the geometrical parameters. This would establish whether the observed thrust loss due to installation could be mitigated for the engines tested.
- Further work should be carried out with additional installation positions. Whilst two installation positions were demonstrated in this project, the influence of horizontal position was not established. Closely coupled positions were studied, but next generation aero-engines may require overlap with the wing due to the relative forward position of the centre of gravity in a geared turbofan.
- Additional nacelles should be designed with different values of droop and scarf to determine whether the trends displayed for a single case can be exploited further.
- The method of reduced fidelity optimisation should be expanded to incorporate full aircraft configurations since the short nacelles have demonstrated highly coupled aerodynamics once installed.

- Since the installation assessments here were carried out without a pylon, the influence for such supercritical nacelles should be assessed. The gully flow is likely to become a more difficult area to design as the nacelle becomes more closely coupled and the uninstalled peak Mach numbers increase.
- As the redesign of the exhaust and intake were not considered in this thesis, separate work should establish the extent to which a short intake and an exhaust system with greater wing coupling would allow for nacelle length reduction. This work could take the form of a fully coupled optimisation which would demonstrate the trade offs between different requirements from intake, nacelle, exhaust and installation.

References

- ¹Rolls-Royce media, <https://www.rolls-royce.com/media.aspx#section-press-releases>, Accessed: January 2018.
- ²AirInsight, “Rolls-Royce Advance and UltraFan CTi fan blade flies for the first time,” <https://www.airinsight.com/rolls-royce-advance-ultrafan-cti-fan-blade-flies-first-time/>, Accessed: January 2018.
- ³Whurr, J., “Future Civil Aeroengine Architectures & Technologies,” Invited Lecture, European Turbomachinery Conference in Lappeenaranta, Finland, 2013.
- ⁴ESDU, “Drag of Axisymmetric Cowls at Zero Incidence for Subsonic Mach Numbers,” Tech. Rep. ESDU 81024, ESDU International, London, UK, November 1981.
- ⁵Stańkowski, T. P., MacManus, D. G., Sheaf, C. T., and Christie, R., “Aerodynamics of aero-engine installation,” *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, Vol. 230, No. 14, 2016, pp. 2673–2692.
- ⁶Marshall, E., “Unpublished Internal Report,” Technical report B1A3-101DT, Rolls-Royce plc., 1994.
- ⁷Howe, D. and Wynosky, T., “Energy Efficient Engine Program Advanced Turbofan Nacelle Definition Study,” Technical Report NASA CR-1/4942, NASA, 1985.
- ⁸Dassonville, C., *Exploration of the Design Space for Parametric Nacelle Geometries*, M.Sc. thesis, Cranfield University, Department of Power and Propulsion, 2014.
- ⁹Zimbrick, R. and Colehour, J., “Investigation of Very High Bypass Ratio Engines for Subsonic Transports,” *Journal of Propulsion and Power*, Vol. 6, No. 4, 1990, pp. 490–496.
- ¹⁰Deb, K., *Multi-Objective Optimization Using Evolutionary Algorithms*, Wiley, Chichester, England, 1st ed., 2001.
- ¹¹Stańkowski, T. P., *Aerodynamics of Propulsion System Integration for Modern Aero-Engines*, Ph.D. thesis, School of Aerospace, Transport and Manufacturing, Propulsion Engineering Centre, Cranfield University, 2016.
- ¹²Deb, K., *Multi-objective optimization using evolutionary algorithms*, Wiley, Chichester, UK, 2001, p. 333.
- ¹³ESDU, “Estimation of Windmilling Drag and Airflow of Turbo-Jet and Turbo-Fan Engines,” Tech. Rep. ESDU-81009, ESDU, 1984.
- ¹⁴Mattingly, J. D., *Elements of propulsion: gas turbines and rockets*, AIAA, 2006.
- ¹⁵Haselbach, F., Newby, A., and Parker, R., “Concepts and Technologies for the Next Generation of Large Civil Aircraft Engines, 29th Congress of the International Council of the Aeronautical Sciences,” No. 2014-0078, 2014.
- ¹⁶Daly, M., editor, *Jane’s Aero-Engines*, IHS Global Ltd, 28th ed., 2010, p. 64.
- ¹⁷The High Level Group on Aviation Research, “Flightpath 2050: Europe’s Vision for Aviation, Maintaining Global Leadership & Serving Society’s Needs, Report of the High Level Group on Aviation Research,” European Commission Report, 2011.

- ¹⁸Arguelles, P., Lumsden, J., Bischoff, M., Ranque, D., Busquin, P., Rasmussen, S., Droste, B., Reutlinger, P., Evans, R., and Robins, R., "European Aeronautics: A Vision for 2020," Report, Advisory Council for Aeronautics Research in Europe, 2001.
- ¹⁹Parker, R., "From blue skies to green skies: engine technology to reduce the climate-change impacts of aviation," *Technology Analysis & Strategic Management*, Vol. 21, No. 1, 2009, pp. 61–78.
- ²⁰Salam, A. and Eunus, Z., "Recent Green Aviation Technologies an Overview," *Journal of Modern Science and Technology*, Vol. 1, No. 1, 2013, pp. 61–75.
- ²¹Haines, A., "Aerodynamic Interference - A General Overview, Special Course on Subsonic/-Transonic Aerodynamic Interference for Aircraft," Report AGARD No. 712, AGARD, 1983.
- ²²Peters, A., *Ultra-Short Nacelles for Low Fan Pressure Ratio Propulsors*, Ph.d. thesis, Massachusetts Institute of Technology, February 2014.
- ²³Hoheisel, H., "Aerodynamic aspects of engine-aircraft integration of transport aircraft," *Aerospace science and technology*, Vol. 1, No. 7, 1997, pp. 475–487.
- ²⁴Gilyard, G., Georgie, J., and Barnicki, J., "Flight Test of an Adaptive Configuration Optimization System for Transport Aircraft," Technical Memorandum NASA/TM- 1999-206569, National Aeronautics and Space Administration, Dryden Flight Research Center, Edwards, California 93523-0273, 1999.
- ²⁵Smith, D., "The sustainable and green engine (SAGE) - Aircraft engine or the future?" *The International Journal of Entrepreneurship and Innovation*, Vol. 17, No. 4, 2016, pp. 256–265.
- ²⁶Peters, A., Spakovszky, Z. S., L. W., and Rose, B., "Ultra-Short Nacelles for Low Fan Pressure Ratio Propulsors," *Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition GT2014*, 2014.
- ²⁷Christie, R., Heidebrecht, A., and MacManus, D., "An Automated Approach to Nacelle Parameterization Using Intuitive Class Shape Transformation Curves," *Journal of Engineering for Gas Turbines and Power*, Vol. 139, No. 6, 2017.
- ²⁸Johnson, R., editor, *The Handbook of Fluid Dynamics*, CRC Press, Springer, 2000.
- ²⁹Faust, G. and Mungur, P., "Nacelle Design," *NASA report for General Electric Company*, No. N90-12551/9, NASA, 1987.
- ³⁰MIDAP Study group, "AGARDograph No. 237 Guide to in-flight thrust measurement of turbojets and fan engines," Tech. Rep. AGARD-AG-237, AGARD, 1979, by Ministry-Industry Drag Analysis Panel Study Group and others.
- ³¹ESDU, "Introduction to the Measurement of Thrust in Flight (Air Breathing Ducted-Flow Engines)," Tech. Rep. ESDU-69006, ESDU, 1981.
- ³²Trapp, L. G. and Argentieri, H. G., "Evaluation of nacelle drag using computational fluid dynamics," *Journal of Aerospace Technology and Management*, Vol. 2, No. 2, 2010, pp. 145–153.
- ³³Aeronautical Research Council, "Definitions of the Thrust of a Jet Engine and of the Internal Drag of a Ducted Body," *The Journal of the Royal Aeronautical Society*, Vol. 59, No. 536, August 1955.
- ³⁴Roskam, J. and Lan, C.-T., *Airplane Aerodynamics and Performance*, DAR corporation, Lawrence Kansas, 1997, p. 141.

- ³⁵Shevell, R. S. and Bayan, F. P., "Development of a method for predicting the drag divergence Mach number and the drag due to compressibility for conventional and supercritical wings," Tech. Rep. SUDAAR 522, Department of Aeronautics and Astronautics Stanford University, 1981.
- ³⁶Borradaile, J., "Towards the optimum ducted UHBR engine," *24th Joint Propulsion Conference*, No. AIAA Paper No. 88-2954, 1988.
- ³⁷Farokhi, S., *Aircraft Propulsion*, Wiley, 2009, p. 241.
- ³⁸Street, P. G., "Thrust/drag analysis for a front fan nacelle having two separate co-axial exhaust streams," Tech. Rep. ARC/CP-1311, Aeronautical Research Council, 1975.
- ³⁹ESDU, "Mass flow and momentum functions for one-dimensional flow of gas in ducts," Tech. Rep. 81004, ESDU, 1981.
- ⁴⁰Baals, D., Smith, N., and Wright, J., "Development and application of high-critical-speed nose inlets," Tech. Rep. NACA Rep. 920, NACA, 1945.
- ⁴¹Seddon, J. and Goldsmith, E. L., *Intake aerodynamics*, Blackwell Science Ltd., Oxford, 1998.
- ⁴²Zachos, P., *Gas Turbine Sub-idle Performance Modelling; Altitude Relight and Windmilling*, Ph.D. thesis, Cranfield University, School of Engineering, Department of Power and Propulsion, February 2010.
- ⁴³Dufour, G. and Thollet, W., "Body Force Modelling of the Aerodynamics of the Fan of a Turbofan at Windmill," *ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*, Seoul, South Korea, June 13-17 2016.
- ⁴⁴Prasad, D. and Lord, W. K., "Internal Losses and Flow Behavior of a Turbofan Stage at Windmill," *ASME Turbo Expo 2008: Power for Land, Sea, and Air*, Vol. 6, Berlin, Germany, June 9-13 2008, pp. 349-361.
- ⁴⁵Obert, E., *Aerodynamic Design of Transport Aircraft*, IOS Press BV, 1st ed., 2009, pp. 491-492.
- ⁴⁶Langley, M., "The design of axisymmetric cowls for podded nacelles for high by-pass ratio turbofan engines," Tech. Rep. Reports and memoranda No. 3846, Aircraft Research Association Ltd, Bedford, 1979.
- ⁴⁷Daggett, D. L., Brown, S. T., and Kawai, R. T., "Ultra-Efficient Engine Diameter Study," Tech. Rep. NASA/CR-2003-212309, NASA, 2003.
- ⁴⁸Sokhey, J., "Analysis of installed wind tunnel test results on large bypass ratio engine/nacelle installations," *AIAA/AHS/ASME/ASEE 26th Joint Propulsion Conference*, No. 90-2146, 1990.
- ⁴⁹Motycka, D., Welling, S., and F.A., L.-S., "Comparison of Model and Full Scale Inlet Distortions for Subsonic Commercial Transport Inlets," *AIAA/AHS/ASEE Aircraft Design Systems and Operations Meeting*, No. AIAA-84-2487, San Diego, California, 1984.
- ⁵⁰Van Dam, C., "Recent experience with different methods of drag prediction," *Progress in Aerospace Sciences*, Vol. 35, No. 8, 1999, pp. 751-798.
- ⁵¹Vos, J. and Sanchi, S., "DPW4 Results Using Different Grids Including Near-Field/Far-Field Drag Analysis, 28th AIAA Applied Aerodynamics Conference," Chicago, Illinois, 2010.
- ⁵²Christie, R., Ramirez, S., and MacManus, D. G., "Aero-engine installation modelling and the impact on overall flight performance, Advanced Aero Concepts, Design and Operations," 2014.
- ⁵³Becker, J., "Wind Tunnel Investigation of Air Inlet and Outlet Openings on a Streamline Body," NACA Technical Report 920, NACA, Langley, Virginia, 1945.

- ⁵⁴Re, R., "An investigation of several NACA 1-series inlets at Mach numbers from 0.4 to 1.29 for mass flow ratios near 1.0," Tech. Memor. X-3324, NASA, 1975.
- ⁵⁵Mullender, A. J. and Lecordix, J. L., "The ELFIN II and LARA HLF nacelles: Design manufacture and test," *2nd European forum on laminar flow technology*, Bordeaux, France, 1996.
- ⁵⁶Young, T., "An investigation into potential fuel savings for 110-130 seat passenger transport aircraft due to the incorporation of natural laminar flow or hybrid laminar flow control on the engine nacelles," *Proc IMechE Part G: Journal of Aerospace Engineering*, Vol. 227, No. 8, 2012, pp. 1300–1324.
- ⁵⁷Seddon, J. and Goldsmith, E. L., *Intake Aerodynamics*, Blackwell Science Ltd., Oxford, 1998, p. 12.
- ⁵⁸Krenz, G., "Engine/Airframe Interference, Special Course on Subsonic/Transonic Aerodynamic Interference for Aircraft," Report AGARD No. 712, AGARD, 1983.
- ⁵⁹Grieb, H. and Eckardt, D., "Propfan and Turbofan - Antagonism or Synthesis," Technical Report ICAS-86-3.8.2, MTU Motoren- und Turbinen-Union Munchen GmbH, 1986.
- ⁶⁰Ingraldi, A., "Interference Effects of Very High Bypass Ratio Nacelle Installations on a Low-Wing Transport," *Journal of Engineering for Gas Turbines and Power*, Vol. 114, No. 1, 1992.
- ⁶¹Patterson, J. C., "A Wind-Tunnel Investigation of Jet-Wake Effect of a High-Bypass Engine on Wing-Nacelle Interference Drag of a Subsonic Transport," Technical Report NASA-TN-D-4693, NASA, 1968.
- ⁶²Harris, A. and Paliwal, K., "Civil Turbofan Propulsion System Integration Studies Using Powered Testing Techniques at ARA," Tech. rep., Aircraft Research Association, Bedford, UK, 1984.
- ⁶³Sibilli, T., *Modelling the Aerodynamics of Propulsive System Integration at Cruise and High-Lift Conditions*, Ph.D. thesis, Cranfield University, Department of Power and Propulsion, 2012.
- ⁶⁴Vassberg, J. C., DeHaan, M. A., Rivers, S. M., and Wahls, R. A., "Development of a Common Research Model for Applied CFD Validation Studies," *26th AIAA Applied Aerodynamics Conference*, No. 6919, American Institute of Aeronautics and Astronautics, AIAA, 2008.
- ⁶⁵Rivers, M. B. and Dittberner, A., "Experimental Investigations of the NASA Common Research Model in the NASA Langley National Transonic Facility and NASA Ames 11-Ft Transonic Wind Tunnel (Invited)," *28th AIAA Applied Aerodynamics Conference*, Vol. 1126, AIAA, 2010, p. 2011.
- ⁶⁶Zilliac, G. G., Pulliam, T. H., Rivers, M. B., Zerr, J., Delgado, M., Halcomb, N., and Lee, H., "A Comparison of the Measured and Computed Skin Friction Distribution on the Common Research Model," *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, No. AIAA 2011-1129, AIAA, 2011.
- ⁶⁷Vassberg, J. C., Zickuhr, T., Brodersen, O., Wahls, R., Mavriplis, D., and Murayama, M., "Summary of the Fourth AIAA Computational Fluid Dynamics Drag Prediction Workshop," *Journal of Aircraft*, Vol. 51, No. 4, 2014, pp. 1070–1089.
- ⁶⁸Marx, J., "Installation features of advanced, very high bypass turbofan propulsion systems," *AIAA/AHS/ASME/ASEE Aircraft Design, Systems and Operations Conference*, No. 89-2141, 1989.
- ⁶⁹Zakrzewski, P., "Unpublished Internal Report," Tech. Rep. IAR85059, Rolls-Royce plc., 1998.

- ⁷⁰Airbus, *A330: Aircraft characteristics airport and maintainance planning*, April 2013, Airbus A330 dimensions including nacelles information.
- ⁷¹Daly, M., editor, *Jane's Aero-Engines*, IHS Global Ltd, 28th ed., 2010, p. 505.
- ⁷²Motycka, D., DiSabato, V., and Andersen, L., "The Use of a Powered Model for Subsonic Nacelle Optimization," *ASME International Gas Turbines and Fluids Engineering Conference*, 1972.
- ⁷³Petrusson, A., *Aerodynamic Evaluation of Nacelles for Engines with Ultra High Bypass Ratio*, Master's thesis, Chalmers University of Technology, 2017.
- ⁷⁴Chen, X. and Agarwal, R. K., "Shape Optimization of Airfoils in Transonic Flow Using a Multi-Objective Genetic Algorithm," *31st AIAA Applied Aerodynamics Conference*, No. AIAA 2013-2907, 2013.
- ⁷⁵Wang, W., Mo, R., and Zhang, Y., "Multi-objective aerodynamic optimization design method of compressor rotor based on Isight," *Procedia Engineering*, Vol. 15, 2011, pp. 3699–3703.
- ⁷⁶Toubin, H., Salah El Din, I., and Meheut, M., "Multipoint Aerodynamic High Fidelity Shape Optimization of an Isolated Engine Nacelle," *AIAA SciTech 52nd Aerospace Sciences Meeting*, No. AIAA 2014-0903, 2014.
- ⁷⁷Fang, X., Zhang, Y., and Chen, H., "Transonic Nacelle Aerodynamic Optimization Based on Hybrid Genetic Algorithm," *17th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference*, No. AIAA 2016-3833, 2016.
- ⁷⁸Song, W. and Keane, A. J., "Surrogate-Based Aerodynamic Shape Optimization of a Civil Aircraft Engine Nacelle," *AIAA Journal*, Vol. 45, No. 10, October 2007, pp. 2565–2574.
- ⁷⁹Albert, M. and Bestle, D., "Aerodynamic Design Optimization of Nacelle and Intake," *Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition*, No. GT2013-94857, American Society of Mechanical Engineers, June 2013.
- ⁸⁰Deb, K., *Multi-objective optimization using evolutionary algorithms*, Wiley, Chichester, UK, 2001, p. 201.
- ⁸¹Deb, K., Pratap, A., Agarwal, S., and Meyarivan, T., "A Fast and Elitist Multiobjective Genetic Algorithm: NSGA-II," *IEEE Transactions on Evolutionary Computation*, Vol. 6, No. 6, April 2002, pp. 182–197.
- ⁸²Deb, K., "A fast-elitist non-dominated sorting genetic algorithm for multi-objective optimization: NSGA-II," Technical Report KanGAL Report No. 200001, Kanpur Genetic Algorithms Laboratory, Kanpur, 2000.
- ⁸³Michalewicz, Z., *Genetic Algorithms + Data Structures = Evolution Programs*, Springer, 1992.
- ⁸⁴Magalhaes-Mendes, J., "A Comparative Study of Crossover Operators for Genetic Algorithms to Solve the Job Shop Scheduling Problem," *WSEAS Transactions on Computers*, Vol. 12, 2013, pp. 164–173.
- ⁸⁵ANSYS, I., "ANSYS Fluent Theory Guide," November 2013, Release 15.0.
- ⁸⁶Anderson, J., *Computational Fluid Dynamics: The Basics with Applications*, McGRAW-HILL, 1995.
- ⁸⁷Chou, P., "On velocity correlations and the solutions of the equations of turbulent fluctuation," *Quarterly of Applied Mathematics*, Vol. 3, No. 1, 1945, pp. 38–54.
- ⁸⁸Hinze, J., *Turbulence*, McGRAW-HILL, 1975.

- ⁸⁹Spalart, P. and Allmaras, S., "A One-Equation Turbulence Model for Aerodynamic Flows," *30th Aerospace Sciences Meeting and Exhibit*, 1992.
- ⁹⁰Shih, T., Liou, W., Shabbir, A., Yang, Z., and Zhu, J., "A new k-epsilon eddy viscosity model for high reynolds number turbulent flows," *Computers & Fluids*, Vol. 24, No. 3, March 1995, pp. 227–238.
- ⁹¹Wilcox, D., *Turbulence Modeling for CFD*, DCW Industries Inc., 1998.
- ⁹²Menter, F., "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.
- ⁹³Menter, F., Kuntz, M., and Langtry, R., "Ten Years of Experience with the SST Turbulence Model," *Turbulence, Heat and Mass Transfer 4*, edited by K. Hanjali, Y. Nagano, and M. Tummers, Begell House Inc, 2003, pp. 625–632.
- ⁹⁴Bardina, J., Huang, P., and Coakley, T., "Turbulence Modeling Validation, Testing, and Development," Technical Memorandum TM-110446, National Aeronautics and Space Administration, Ames Research Center, 1997.
- ⁹⁵MacMillan, W. L., *Development of a modular type computer program for the calculation of gas turbine off design performance*, Ph.D. thesis, School of Mechanical Engineering, Cranfield University, 1974.
- ⁹⁶Kulfan, B. and Bussioletti, J., "Fundamental" Parameteric Geometry Representations for Aircraft Component Shapes," *11th AIAA/ISSMO Multidisciplinary Analysis Optimization Conference*, No. AIAA 2006-6948, American Institute of Aeronautics and Astronautics, Sept. 2006, pp. –.
- ⁹⁷Zhu, F. and Qin, N., "Intuitive Class/Shape Function Parameterization for Airfoils," *AIAA Journal*, Vol. 52, No. 1, Oct. 2013, pp. 17–25.
- ⁹⁸Heidebrecht, A., Stankowski, T., and MacManus, D., "Parametric Geometry and CFD Process for Turbofan Nacelles," *Proceedings of ASME 2016 Turbo Expo: Turbomachinery Technical Conference and Exposition*, No. GT2016-57784, 2016.
- ⁹⁹Christie, R., Bruni, G., Ancona, M., and MacManus, D., "NADIT Work Package 1.8, Deliverable 3: Intake and 3D nacelle geometry tool," Technical Report TR-2016-4, Cranfield Rolls-Royce UTC in Performance Engineering, School of Engineering, Cranfield University, Cranfield, Bedfordshire, MK430AL, UK, June 2016.
- ¹⁰⁰Oliveira, G. L., Trapp, L. G., and Puppim-Macedo, A., "Integration Methodology for Regional Jet Aircraft with Underwing Engines," *41st Aerospace Sciences Meeting and Exhibit*, Vol. AIAA 2003-934, AIAA, Reno, Nevada, January 6-9 January 2003.
- ¹⁰¹Fry, K., "Z30/2 Data Reduction Specification," Tech. Rep. AD-94/32, Aircraft Research Association Limited, 1994.
- ¹⁰²Malan, P. and Brownt, E., "Inlet Drag Prediction for Aircraft Conceptual Design," *Journal of Aircraft*, Vol. 31, No. 3, 1994, pp. 616–622.
- ¹⁰³Richards, K., *Aerodynamic Design, Performance and Optimisation of Conventional Nacelles*, M.Sc. thesis, Cranfield University, School of Aerospace, Transport and Manufacturing, 2016.
- ¹⁰⁴Pilat, P., *Multi-Objective Optimisation of Short Slimline Nacelles*, M.Sc. thesis, Cranfield University, School of Aerospace, Transport and Manufacturing, 2016.
- ¹⁰⁵Bruni, G., *Multi-Objective Optimisation of Short Intakes*, M.Sc. thesis, Cranfield University, School of Aerospace, Transport and Manufacturing, 2016.

- ¹⁰⁶While, L., Bradstreet, L., and Barone, L., “A Fast Way of Calculating Exact Hypervolumes,” *IEEE Transactions on Evolutionary Computation*, Vol. 16, No. 1, February 2012, pp. 86–95.
- ¹⁰⁷Patella, R., “Unpublished Internal Report,” Tech. Rep. IAR41132, Rolls-Royce plc., 1994.
- ¹⁰⁸Raymer, D. P., *Aircraft Design: A conceptual Approach*, American Institute of Aeronautics and Astronautics, 1992.
- ¹⁰⁹Alexander, J., “Unpublished Internal Report,” Technical report IAR41177, Rolls-Royce plc., 2007.
- ¹¹⁰Menter, F., Langtry, R., Likki, S., Suzen, Y., Huang, P., and Völker, S., “A correlation-based transition model using local variables-Part I: model formulation,” *Journal of Turbomachinery*, Vol. 128, 2006, pp. 413.
- ¹¹¹Jamil, M. and Yang, X.-S., “A Literature Survey of Benchmark Functions for Global Optimization Problems,” *Int. Journal of Mathematical Modelling and Numerical Optimisation*, Vol. 4, No. 2, 2013, pp. 150–194.
- ¹¹²Kursawe, F., “A variant of evolution strategies for vector optimization,” *Parallel Problem Solving from Nature Proceedings of the 1st Workshop*, Berlin, 1991, pp. 193–197.
- ¹¹³Deb, K., Thiele, L., Laumanns, M., and Zitzler, E., “Scalable Test Problems for Evolutionary Multi-Objective Optimization,” TIK-Technical Report 112, Institut für Technische Informatik und Kommunikationsnetze, ETH, Zurich, Gloriastrasse 35., ETH-Zentrum, CH-8092, Zurich, Switzerland, July 2001.
- ¹¹⁴Wessing, S., *Two-stage Methods for Multimodal Optimization*, Ph.D. thesis, Technische Universität Dortmund, 2015.
- ¹¹⁵Carter, E. C. and Pavitt, M., “The Operation and Performance of the ARA Cowl Inlet Test Rig,” Technical Report Memo No. 181, Aircraft Research Association Limited, 1976.
- ¹¹⁶Goulos, I., Stankowski, T., Otter, J., MacManus, D., Grech, N., and Sheaf, C., “Aerodynamic Design of Separate-Jet Exhausts for Future Civil Aero-engines-Part I: Parametric Geometry Definition and Computational Fluid Dynamics Approach,” *Journal of Engineering for Gas Turbines and Power*, Vol. 138, 2016, pp. 1–14.
- ¹¹⁷Daly, M., editor, *Jane’s Aero-Engines*, IHS Global Ltd, 28th ed., 2010.
- ¹¹⁸Airbus S.A.S., Customer Services, Technical Data Support and Services, 31707 Blagnac Cedex, France, *Airbus A330 Aircraft Characteristics Airport and Maintenance Planning*, 01 2014.
- ¹¹⁹“EASA certifies A350 XWB for up to 370 minute ETOPS,” 2014.
- ¹²⁰Otter, J., “Personal Communication,” August 2017.
- ¹²¹Roache, P., *Verification and validation in computational science and engineering*, Hermosa publishers, 1998.
- ¹²²Branke, J., Kaussler, T., and Schmeck, H., “Guidance in evolutionary multi-objective optimization,” *Advances in Engineering Software*, Vol. 32, 2001, pp. 499–507.
- ¹²³Branke, J. and Deb, K., “Integrating User Preferences into Evolutionary Multi-Objective Optimization,” Tech. Rep. KanGAL Report Number 2004004, KanGAL, 2004.

APPENDIX A

Summary of mesh and domain independence studies

A.1 Mesh independence

Mesh independence was carried out in accordance with the procedure outlined by Roache.¹²¹ The method involves calculating a grid convergence index (GCI) for the different meshes (Equations A.1, A.2) for a value f evaluated for each of the meshes produced. A Richardson extrapolation (Equations A.3, A.4) is used to determine the mesh independent value of the metric of interest. The GCI requires the evaluation of the relative error between two given meshes, a ratio (r) of the two grid sizes (N) (Equation A.5), the order of the convergence (p) (Equation A.6) and a factor of safety (F_S).

$$GCI_{med-fine} = F_s \left| \frac{f_{fine} - f_{med-fine}}{f_{med-fine}} \right| \quad (A.1)$$

$$GCI_{cor-med} = F_s \left| \frac{f_{med} - f_{cor-med}}{f_{cor-med}} \right| \quad (A.2)$$

$$f_{med-fine} \cong f_{fine} + \frac{f_{fine} - f_{med}}{r^p - 1} \quad (A.3)$$

$$f_{cor-med} \cong f_{med} + \frac{f_{med} - f_{cor}}{r^p - 1} \quad (A.4)$$

$$r = \left(\frac{N_1}{N_2} \right)^{1/d} \quad (A.5)$$

$$p = 2 \quad (A.6)$$

A.1.1 2D axisymmetric isolated nacelle studies

Throughout the project various configurations were simulated with different levels of accuracy in terms of what they represented. The first model which was used for a mesh study was the E3 engine in the NE-RE configuration (Figure 5.1f) with a short nacelle ($l_{nac}/r_{hi} = 3.1$). Three meshes were created (Table A.1) and simulated at a typical condition of $M_\infty = 0.85$, $MFCR = 0.7$. The metric which was calculated to assess the mesh independence was nacelle drag ($C_{D_{nac}}$), which includes the influence of post exit terms (ϕ_{post}), and the modified nacelle drag ($C_{D^{*}nac}$). It was determined from this study that the fine mesh should be used to minimise the mesh dependent error as it had around a 1% error from the Richardson extrapolation on both the $C_{D^{*}nac}$ and the $C_{D_{nac}}$.

Configurations	Coarse	Medium	Fine
Cells	58153	72769	88672
GCI $C_{D^{*}nac}$	-	1.43%	0.98%
GCI $C_{D_{nac}}$	-	1.44%	1.02%

TABLE A.1: Results from the CFD simulations of 2-dimensional axisymmetric nacelle meshes produced for independence studies

A.1.2 3D isolated nacelle studies

A mesh independence study was carried out on the E3 engine with an optimal nacelle at a representative cruise condition of $M_\infty = 0.85$, $MFCR = 0.7$. The GCI was evaluated for the D_{nac}^* and D_{nac} for the grids produced. Based on these results (Table A.2) the fine mesh, with a GCI of -0.04% relative to the Richardson extrapolation, was considered independent of the mesh and was used for all studies.

Configurations	Coarse	Medium	Fine
Cells	6209024	8138830	11131029
GCI $C_{D^{*}nac}$	0.08	0.02%	0.01%
GCI $C_{D_{nac}}$	-0.63%	-0.05%	-0.04%

TABLE A.2: Results from the CFD simulations of 3-dimensional nacelle meshes produced for independence studies

A.1.3 3D installed studies

A mesh refinement study was carried out for the CRM aircraft in a previous work.¹¹ Four meshes were produced for a wing/body/tail (WBT0) configuration and three meshes were produced for a wing/body/nacelle/pylon (WBNP) configuration. The WBT0 configuration was computed at a Mach number of $M_\infty = 0.85$. The GCI was computed relative to the Richardson extrapolation for each of these configurations at a range of incidences. At $\alpha_{a/c} = 2.5^\circ$, the GCI for the WBT0 configuration was 4.06% for the $C_{D,a/c}$ of the fine mesh (Table A.3), which was concluded to be satisfactory and the superfine mesh would not provide enough improvement for the increased computational cost.¹¹ The GCI of the C_L of the fine mesh was around a similar level of 4.83%. The WBNP configuration was computed at a Mach number of $M_\infty = 0.83$. The fine mesh was again chosen for this configuration based on a typical cruise condition of 2.3% for the $C_{D,a/c}$ and 0.71% for the C_L (Table A.4).¹¹

Configurations	Coarse	Medium	Fine	Superfine
GCI $C_{D,a/c}$	6.54%	5.99%	4.06%	3.05%
GCI C_L	2.18%	3.19%	4.83%	3.63%

TABLE A.3: Results from the CFD simulations of WBT0 configuration at $\alpha_{a/c} = 2.5^\circ$ ¹¹

Configurations	Coarse	Medium	Fine
GCI $C_{D,a/c}$	4.16%	2.92%	2.30%
GCI C_L	-0.75%	0.90%	0.71%

TABLE A.4: Results from the CFD simulations of WBNP configuration at $\alpha_{a/c} = 2.5^\circ$ ¹¹

A.1.4 3D isolated nacelle studies

The mesh study was carried out on an E3 engine in the nacelle,exhaust (NE) configuration with representative exhaust flow (RE) with an optimal $l_{nac}/r_{hi} = 3.1$ nacelle. Three meshes were created with a spherical domain size of $r_{domain} = 100r_{max}$.

Configurations	Coarse	Medium	Fine
V2 MS/NS	0.02081	0.02042	0.02040
E2 FS/NE/RE	35870	71344	144712

TABLE A.5: Results from the CFD simulations of 3-dimensional nacelle meshes produced for independence studies

A.2 Domain independence

The domains used in the project were all semi-circular, hemi-spherical or a full sphere. In the 2D axisymmetric representation of engines the domain was semi-circular and the radius of the domain determined as a ratio of the maximum nacelle radius.

A.2.1 2D axisymmetric isolated nacelle studies

For the 2D axisymmetric nacelle configurations, a semi-circular domain was used. A domain independence study was carried out for the E3 engine with the baseline nacelle at the cruise condition ($M_\infty = 0.85$, $MFCR = 0.7$). Four domain sizes were tested which defined the radius of the semi-circle based on the maximum radius of the E3 baseline nacelle. There was no change in the drag between the $80r_{max}$ domain and the $100r_{max}$ and as such the $80r_{max}$ domain was used for all further studies.

Domain Size	$40r_{max}$	$60r_{max}$	$80r_{max}$	$100r_{max}$
C_{Dnac}	0.0248	0.0246	0.0242	0.0242

TABLE A.6: 2-dimensional axisymmetric nacelle meshes produced for independence studies

A.2.2 3D isolated nacelle studies

A domain study was carried out on the E3 engine with the optimal nacelle. The drag (C_{Dnac}) was evaluated for three domain sizes to establish at if the result was dependent on the domain size. It was determined that a change of $\Delta C_{Dnac} = 0.0001$

between the medium and large domains indicated that the large domain was suitably accurate (Table A.7). This domain size was therefore used for all further studies.

Domain Size	$60r_{max}$	$80r_{max}$	$100r_{max}$
C_{Dnac}	0.0224	0.0221	0.0220

TABLE A.7: 3-dimensional isolated nacelle domain study

APPENDIX B

Summary of additional validation studies

This appendix provides the details of additional validation activities carried out which were not included in the main text. Additional studies were carried out on the I3 intake model. The key conclusions from this study were that a second order scheme was sufficient to model the intake flow without the additional cost of resolving to a third order accuracy. The third order solution gave spurious results which had a notable impact on the flow topology and had worse agreement with the measurement data.

B.1 Effect of scheme order

To assess the CFD methodology a series of tests were carried out with different CFD settings. A comparison for the $k - \omega$ SST model with 2nd and 3rd order discretisation was made on the datum case. For both 2nd and 3rd order, the peak Mach number is about 0.05M lower than the experimental data. For the 3rd order solution, the flow also remains supersonic up to the fan exit (Figure B.1). However, comparison of the 2nd order results with the 3rd order results showed that the 2nd order results agree closer with the wind tunnel post shock Mach distribution. This suggested that the 3rd order result did not correctly capture the shock interaction which resulted in a large separation and a notable change in the flow topology within the duct.

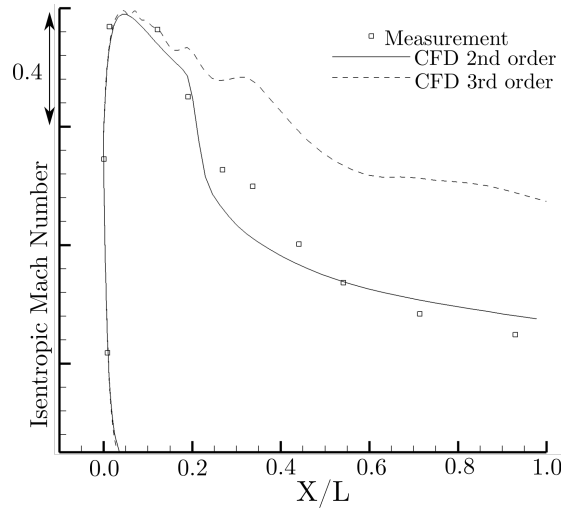


FIGURE B.1: Isentropic Mach distributions for datum mesh at $M=0.25$, $WRR=2133\text{lb/s}$ at 33 degrees at 2nd order discretisation and 3rd order discretisation

B.2 Intake validation with different turbulence models

The Spalart-Allmaras turbulence model was developed for faster calculations of aerodynamic flows than 2 equation models. The datum case was re-calculated with the datum mesh of 6.7 million cells and the Spalart-Allmaras turbulence model. This shows good agreement with both experimental data and the 3rd order $k - \omega$ SST-CC solution (Figure B.2).

The 4 equation transition SST model couples the $k - \tilde{\nu}$ model with equations for intermittency and transition criteria based on the $\gamma Re - \theta$ correlations from Menter and Langtry.¹¹⁰ This model was used with the datum case and datum mesh of 6.7 million cells and a freestream intermittency factor of 1. This turbulence model produced a delayed shock in comparison to both the experimental data and the $k - \omega$ SST models (Figure B.3). However the downstream Mach number values were improved over the prediction given by the $k - \omega$ SST model.

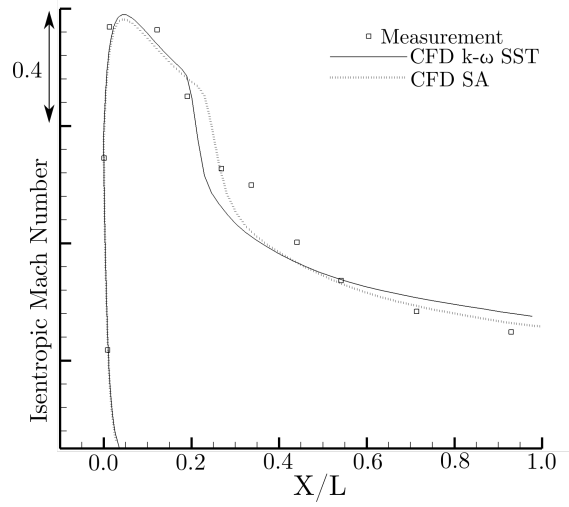


FIGURE B.2: Isentropic Mach number distributions of Spalart-Allmaras results over lower intake lip at centreline for the I1 intake case, with $M_\infty = 0.25$, $\alpha_{CL} = 33^\circ$, WRR=2133 lb/s and with datum mesh (6.7 million cells)

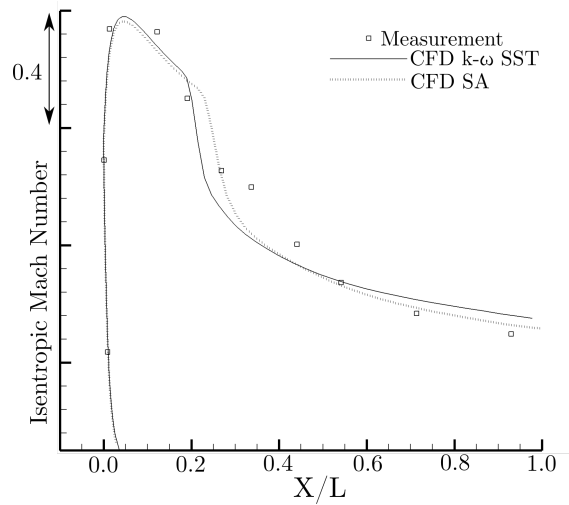


FIGURE B.3: Isentropic Mach number distributions of Transition-SST results over lower intake lip at centreline for the I1 intake case, with $M_\infty = 0.25$, $\alpha_{CL} = 33^\circ$, WRR=2133 lb/s and with datum mesh (6.7 million cells)

APPENDIX C

Prediction of sting drag coefficient

The assessment of the nacelle drag results discussed so far has related to nacelles mounted on a sting, which incorporated two additional drag terms. These were the drag incurred by the sting between the trailing edge of the nacelle and the measurement plane (ϕ_{sting}) and the interference on the nacelle drag caused by the presence of the sting (F_{interf}). For nacelle 1 both the experimental data and the CFD analyses include ϕ_{sting} and F_{interf} in the presented results of c'_D . For nacelles V2 and V3 a correction for ϕ_{sting} has been applied to both the experimental and CFD drag coefficient according to the method presented in section A.1 to calculate c_{Dc} . Although most experimental test configurations depend on the use of a sting in the assessment of nacelle drag, it is important to understand the influence that these inherent measurement requirements may have on reported nacelle drag characteristics. Within that context, the current suite of nacelle test cases is used to assess the influence of these terms. One element is the typical corrections that are applied for the sting drag and the accuracy of this flat plate correction for the sting drag terms was assessed by comparing the analytical flat plate model and the CFD estimate (Figure C.1). This also enables a direct comparison between the drag incurred over the sting region upstream of the measurement rake and the drag calculated from flat plate analytical assessment (equation 3.22). The values of ϕ_{sting} were converted to a coefficient (c_{Dsting}) relative to the freestream dynamic pressure and the nacelle maximum cross-sectional area. The amount of drag was overestimated by the flat plate analysis by between 0.0018 and 0.0021 (Figure C.1), a 25% increase in comparison to the CFD prediction. This resulted in a lower predicted value of the nacelle drag from the flat plate corrected method (c_{Dc}) relative to that calculated by the CFD up to the nacelle trailing edge (c_{D*}). There is also a difference in the gradient of estimated drag with the Mach number with a 2%

larger change in $c_{D_{sting}}$ relative to $c_{D_{sting,ref}}$ over $\Delta M = 0.09$ (Figure C.1). This introduces an inaccuracy into the measured drag rise Mach number where it is overestimated by $\Delta M_{DR} = 0.0009$. The flat plate analogy makes the assumption that the flow develops along the nacelle similarly to how it would along an equal length flat plate. However, there are notable differences between the flow over a nacelle and over a flat plate. For instance, over the afterbody of the nacelle the boundary layer undergoes diffusion which influences the boundary layer characteristics at the start of the sting. This affects the contribution of skin friction drag from the cowl and it is clear that the use of a simple flat plate skin friction estimation overestimates the drag from the sting, which results in a reported measured lower value of c_{Dc} compared with simply measuring the drag to the trailing edge (c_{D*}) (Figure C.2). In order to demonstrate the magnitude of this increase of sting drag calculated, the total drag of the combined nacelle and sting ($c_{D'}$) is also shown in Figure C.2.

In order to address the over-estimation of sting drag from the flat plate analysis, a subsequent assessment was carried out which used an adjusted skin friction method (equation C.1), with coefficients adjusted using least squares regression to the CFD data for the skin friction drag of the sting. The regression was based on the CFD calculations for nacelle V2 between $M_\infty = 0.8$ and $M_\infty = 0.89$ at 0 degrees and $MFCR = 0.7$. An additional factor was also derived to account for the effects of incidence from the CFD predictions of nacelle V2 at 2 and 4 degrees; these effects were not incorporated previously.¹⁰¹ This adjustment of the coefficients of the skin friction equation predicts a lower influence from the sting. This was then used to produce a different corrected drag coefficient (c_{Dc2}) (equations C.2, C.3).

$$C_{f2} = \frac{0.3914}{(\log_{10} Re)^2 \cdot 58(1 + 0.4752M^2)^{0.65}} (0.8942\alpha_{CL})^{0.0214} \quad (C.1)$$

$$c_{D_{sting2}} = \left[\frac{A_{wc} + A_{ws}}{A_{max}} \right] c_{f2,combined} - \frac{A_{wc}}{A_{max}} c_{f2,t} \quad (C.2)$$

$$c_{Dc2} = c_{D'} - c_{D_{sting2}} \quad (C.3)$$

The inclusion of these coefficients brought the RMS difference between the flat plate estimated $c_{D_{sting2}}$ and the sting drag measured from CFD to within 10^{-10} at the three incidences tested (Figures C.3A,B). This new correction factor (equation C.2) was then applied to the CFD data for nacelle V3, where the previous flat plate analogy corrected drag ($c_{D_{sting}}$) had overestimated the sting drag by up to

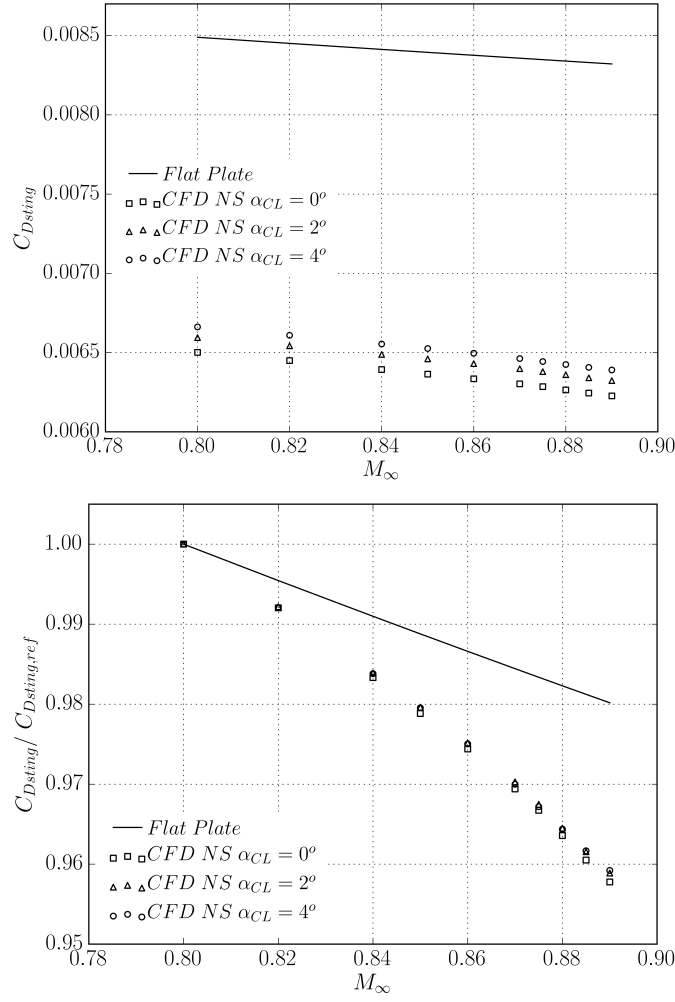


FIGURE C.1: Skin friction drag extracted for sting attached to nacelle V2 CFD data and flat plate estimation

$\Delta c_{Dsting} = 0.0022$ in comparison to the CFD estimate (Figure C.3C). The prediction of drag from this new model ($c_{Dsting2}$) reduced the difference by an order of magnitude to $\Delta c_{Dsting2} = 0.0002$ for all Mach numbers. $c_{Dsting2}$ was also calculated for nacelle V1, which had a shorter sting section and a much longer cowl so provided a much different test case. The calculation of the previous skin friction model (c_{Dsting}) for nacelle V1 had a maximum error relative to the CFD of 0.0015 whereas the maximum difference in $c_{Dsting2}$ reduce to within 0.0005 of the CFD estimation (Figure C.3D).

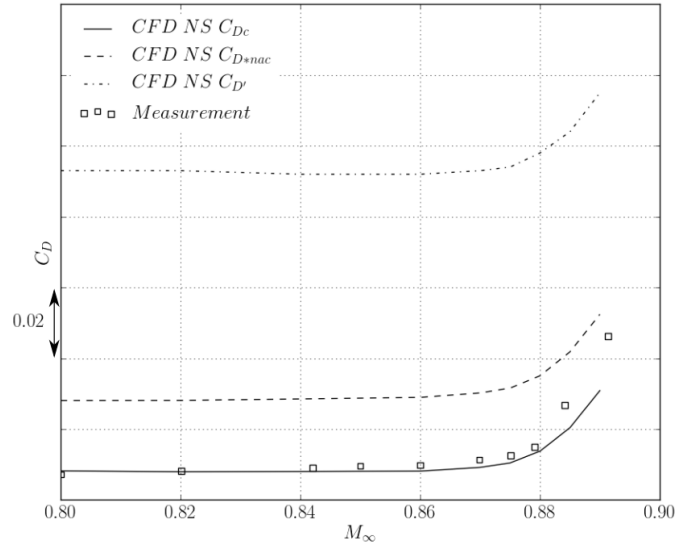
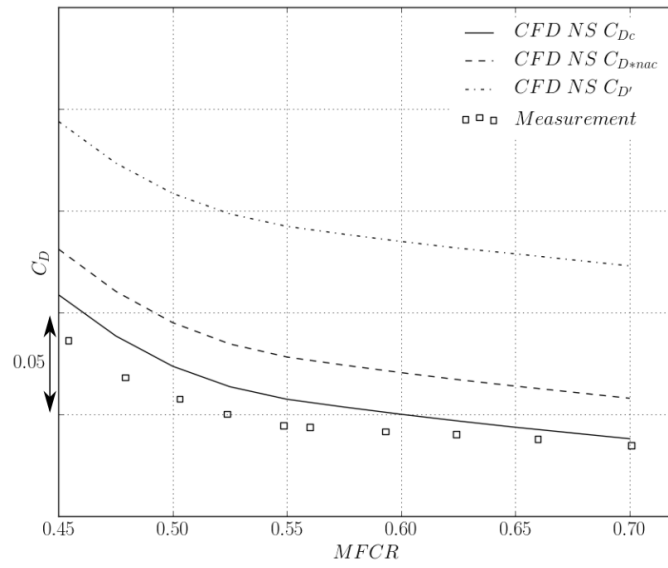
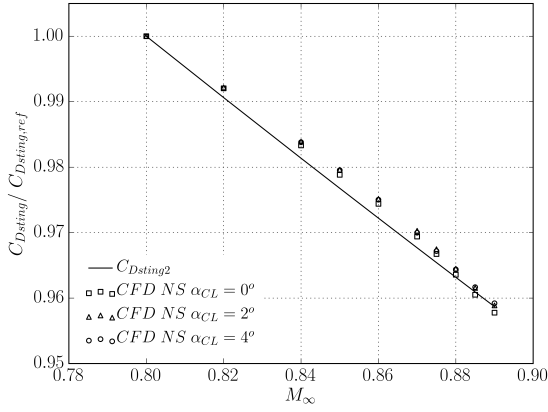
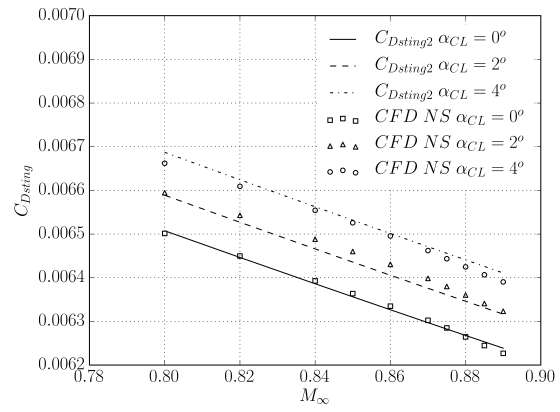
(A) $MFCR = 0.7$ (B) $M_\infty = 0.8$

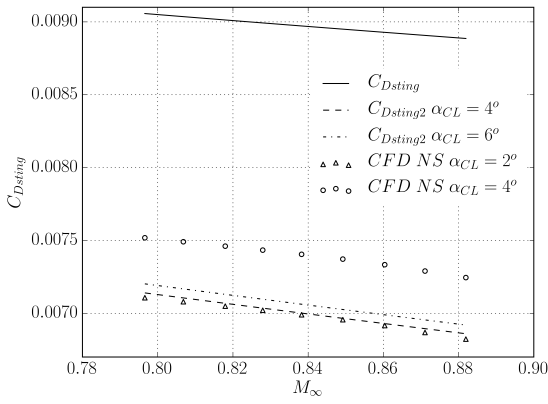
FIGURE C.2: Drag coefficient for CFD analysis and experimental data for nacelle V2



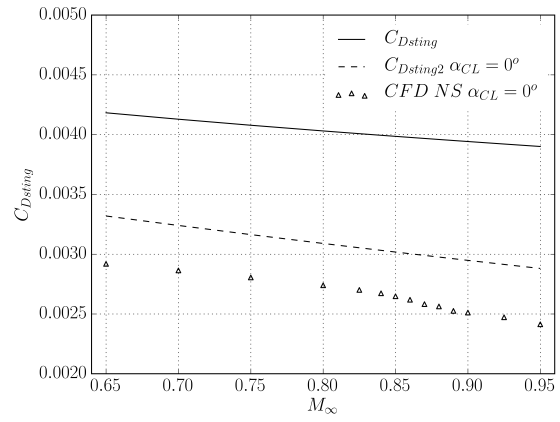
(A) V2



(B) V2



(C) V3



(D) V1

FIGURE C.3: Values of sting drag calculated from CFD and updated flat plate correlations

APPENDIX D

Response surface model based optimisation

This appendix outlines the application of the optimisation method to a response surface model which had been previously produced. The aim was to repeat the optimisation tests carried out on the DTLZ2 function (section 4.2) on a more representative aerodynamic problem without incurring too much computational cost. The purpose was therefore to give greater confidence in the applicability of the method prior to use with CFD in the loop. The optimization routine (Section 3.5) was applied to an RSM of nacelle drag and drag rise Mach number. The aim was to determine the Pareto optimal solutions within the Kriging interpolation for the three objective functions for the nacelle ($c_{Dcruise}$, M_{DR} , Δc_{Dspill}). The optimization on the RSM began with a random initial generation (Figure D.1a) which was then used to produce subsequent, improved generations (Figures D.1b,D.1c,D.1d). After 100 generations and 4000 individual evaluations of the RSM model the non-dominated members of these generations produce an overall optimal Pareto front (Y')(Figure D.2). It can be seen from this optimal set that the design space narrows as the drag rise Mach number is improved until $M_{DR} = 0.917$, above which there are no nacelle designs which have been identified in the optimal set (Y'). The lowest drag found in the random design space (Figure D.1a) was $c_{Dcruise} = 0.0352$ which reduced to $c_{Dcruise} = 0.0285$ in the final Pareto front (Figure D.2) found after 100 generations. Similarly, the highest M_{DR} increased from 0.881 to 0.917 whilst the minimum Δc_{Dspill} reduced from 0.0029 to -0.0002. Overall, the optimization produced a relatively large trade-off surface from which designs could be chosen. A designer will typically design for a M_{DR} which is 0.02 above the desired cruise Mach number³⁶ and the Pareto surface allows a range of designs which meet this criterion. For example, for a typical cruise Mach number of 0.85, the other two performance metrics (M_{DR} , $\Delta c_{D,spill}$) can then be traded against each other with

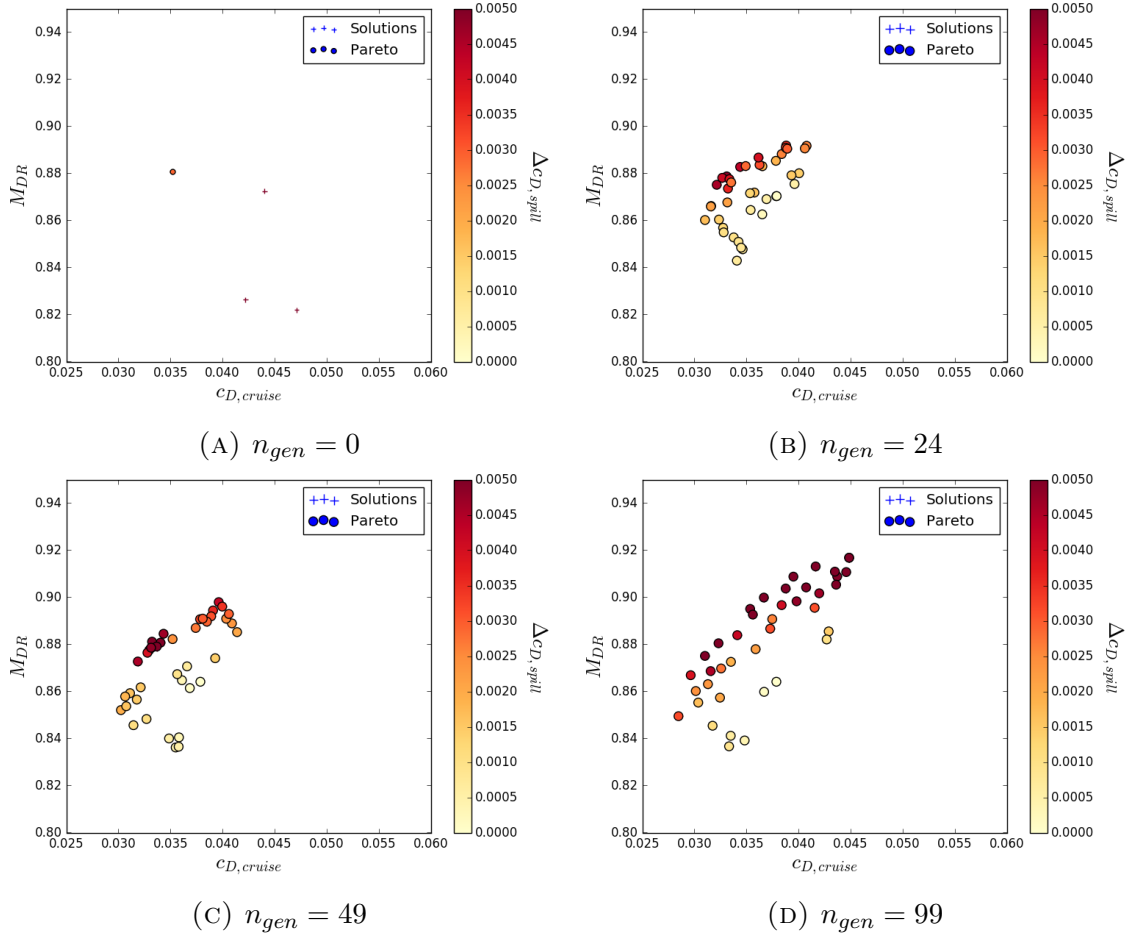


FIGURE D.1: Population in objective space

the lowest drag design ($c_{D,cruise} = 0.0309$) which exceeds $M_{DR} = 0.87$ giving a high spillage of $\Delta c_{D,spill} = 0.0044$. The reverse is true whereby the lowest spillage design which meets the $M_{DR} > 0.87$ criterion ($\Delta c_{D,spill} = -5 \times 10^{-7}$) also has a high cruise drag ($c_{D,cruise} = 0.0382$). This trade off can then be considered across a flight envelope to determine which is the preferred design overall.

D.1 Influence of initial sample size

The design space for nacelle performance in terms of the three functions ($c_{D,cruise}, M_{DR}, \Delta c_{D,spill}$) is expected to have an uneven shape due to different viable designs which produce effective local minima. Consequently it is important to ensure that the implementation of the NSGA-II optimization algorithm is sufficiently calibrated to ensure correct convergence and diversity preservation since the MOO problem of nacelle design may not be represented fully by existing benchmark functions. To determine

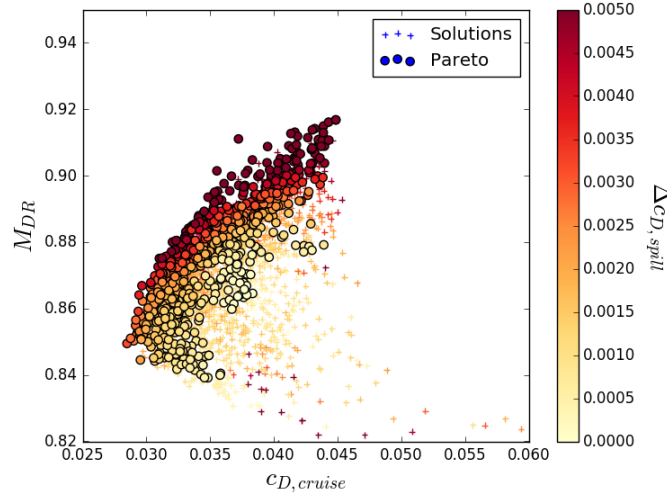


FIGURE D.2: Final Pareto optimal set extracted from the RSM plotted as a function of the 3 objective functions

the dependency of the method on the initial estimate from the random sample, the initial population size was varied. To perform well, an algorithm should be independent of the random sample and should converge to a Pareto set of reasonably equivalent fitness irrespective of the initial data. A statistical approach, with a sample size of 50, was taken to determine the differences in the Pareto optimal set from different zeroth generation sizes over the generations. This was tested for three different mutation standard deviation sizes as was previously investigated for the DTLZ2 test case (section 3.5). As for DTLZ2, a trend was observed in which the hypervolume increases as generations were increased and the lowest value of hypervolume was found at $i_{gen0} = 40$ and $N=10$. Relative to this worst case, there was up to a 7% improvement from increased generations and increased i_{gen0} (Figure D.3). For any given number of evaluations an increase in i_{gen0} gives an improvement in the hypervolume. This is particularly true when only a low number of evaluations are possible. For instance for 1000 evaluations an increase in the i_{gen0} from 40 to 400 increases the average hypervolume from 0.875 to 0.885 with $\sigma_{mutation} = 0.005$ (Figure D.3b). Unlike the benchmark function (DTLZ2) in which the highest hypervolume was achieved with the use of the highest value of $\sigma_{mutation}$, the highest hypervolume in the nacelle RSM was found with and intermediate value of $\sigma_{mutation} = 0.005$ (Figure D.3b). This non-monotonic relationship for the nacelle aerodynamic characteristics suggests that if the mutation size is increased too far then the algorithm is negatively affected as it will tend to mutate at larger magnitudes than the size of the local recessions in the design space. Due to this it will be less likely to find local or global minima. The optimizations

with $mutation = 0.05$, $N=100$, $i_{gen0} = 400$ found on average a maximum M_{DR} of 0.928 (Figure D.3c) whereas with $\sigma_{mutation} = 0.005$ the average maximum M_{DR} was 0.935 (Figure D.3b). This may mean that some prior knowledge of the design space is needed to more effectively calibrate the algorithm. This problem may have arisen in the nacelle design space due to the many different designs which are aerodynamically possible for the considered drag criteria. As a result, disconnected regions of the input variable space may lead to Pareto optimal solutions. It is of interest therefore to be able to limit the solutions to those of particular interest and guide the optimization towards specific areas of engineering interest.

D.2 Influence of constraints

Often in a MOO problem it is desirable to focus the results to a specific region of interest. For example in a nacelle design optimization it is typically desired that M_{DR} be 0.02 above the cruise Mach number³⁶. Since the viable design space for nacelle design is reduced significantly at higher M_{DR} values due to the difficulty of avoiding any transonic effects it is of great interest to produce, at higher M_{DR} values, as detailed map as possible. By applying constraints to the objective functions, it is possible to restrict the objective space which is being resolved and therefore focus the population into a region of interest. This can then reduce the computational cost for finding the entire Pareto front and is for particular concern when a more time consuming optimization procedure incorporating CFD analyses in the routine is being performed. An example analysis was carried out with a constraint of $M_{DR} \geq 0.9$ for 50 individual optimizations to allow mean values of the aerodynamic parameters to be extracted. The use of a constraint forced solutions to be focussed in a smaller region of the design space and higher maximum M_{DR} values achieved. On average across the 50 optimizations the maximum M_{DR} found was 0.939 compared to 0.921 for the equivalent unconstrained cases. Example Pareto fronts from the unconstrained and constrained optimizations demonstrate the additional designs which were found in the constrained case (Figure 13). The average of the minimum drag values found above $M_{DR} = 0.9$ was slightly increased to 0.0335 for the constrained case from 0.0333 for the unconstrained. Similarly, the average of the minimum spillages from the constrained cases was 0.0029 compared to 0.0026 for the unconstrained which demonstrates a penalty which arise from the use of this constraint on M_{DR} . These designs with the lower spillage values tended to also have M_{DR} values close to the M_{DR} constraint. This may mean that the respective

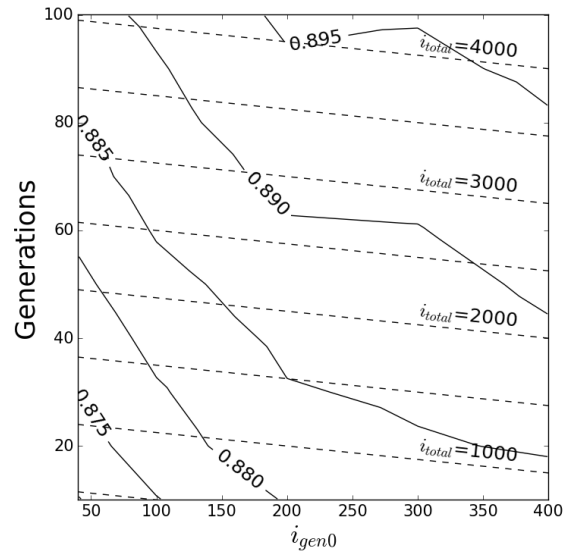
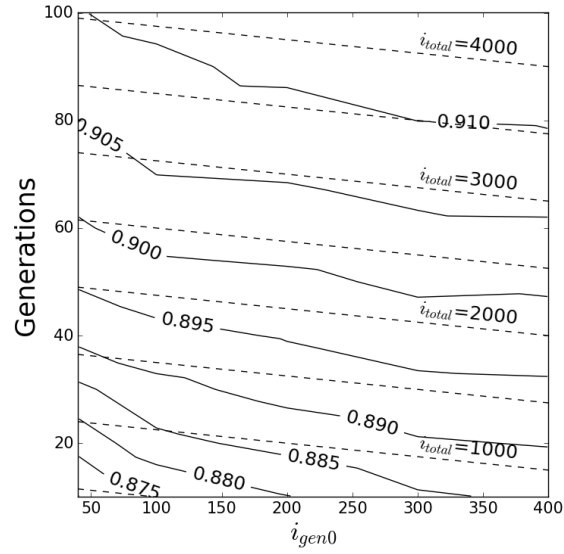
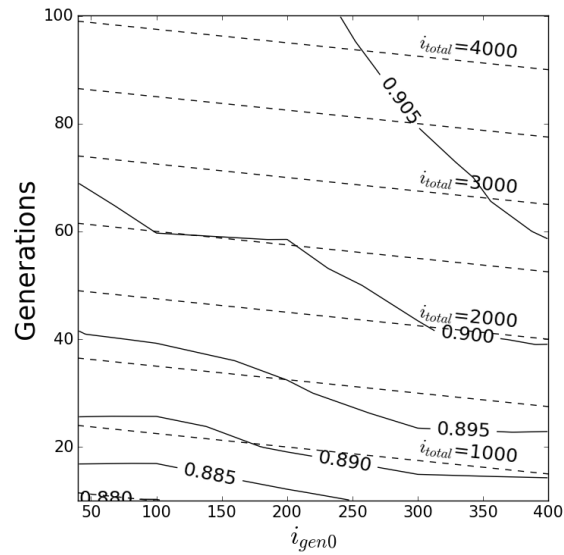
(A) $\sigma_{mutation} = 0.0005$ (B) $\sigma_{mutation} = 0.005$ (C) $\sigma_{mutation} = 0.05$

FIGURE D.3: Mean Hypervolume from 50 individual optimizations with the RSM as a function of zeroth generation size and generation number

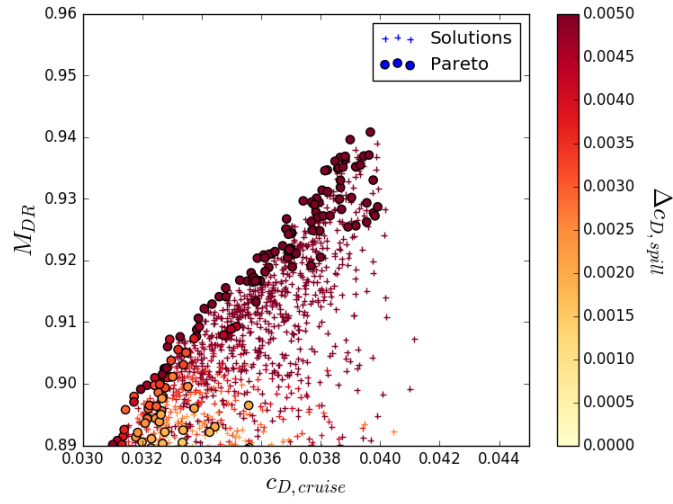
	$\mu_{C_{Dcruise}} _{M_{DR}=0.9}$	$\sigma_{C_{Dcruise}} _{M_{DR}=0.9}$	$\mu_{C_{Dspill}} _{M_{DR}=0.9}$	$\sigma_{C_{Dspill}} _{M_{DR}=0.9}$
Unconstrained	0.0337	0.0016	0.0027	0.0011
$M_{DR} \geq 0.9$	0.0342	0.0021	0.0030	0.0008
$M_{DR} \geq 0.89$	0.0329	0.0013	0.0024	0.0009

TABLE D.1: Average minimum cruise drag and spillage drag at a given drag rise Mach number range for the different constrained optimizations

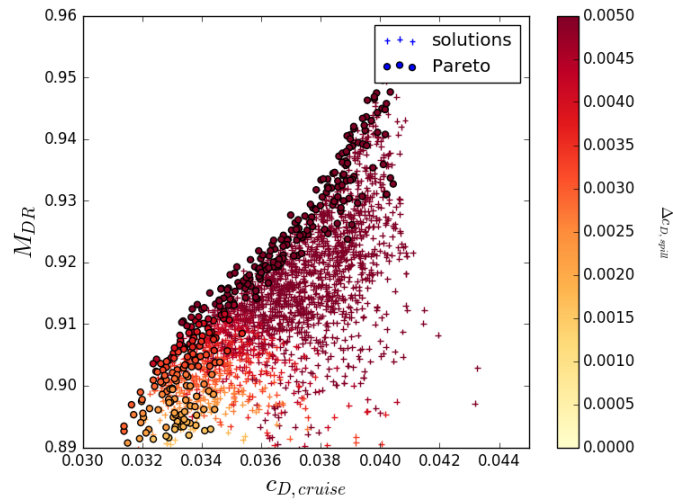
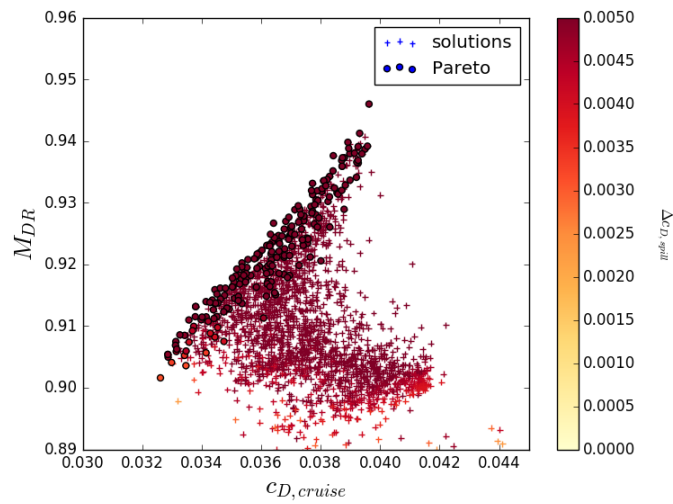
regions in design space are less likely to be explored by the optimizer because of their proximity to regions which do not meet the constraint on M_{DR} and are therefore never populated. To verify this aspect, an additional 50 optimizations were carried out with a constraint of $M_{DR} \geq 0.89$. The result of the adjusted constraint ($M_{DR} \geq 0.89$) was that the average minimum spillage above a $M_{DR} \geq 0.9$ was decreased to 0.0024 compared to 0.0029 for the higher constraint of $M_{DR} \geq 0.9$. At the same time, the average value of the maximum M_{DR} was improved to 0.936 compared to 0.921 for the equivalent unconstrained cases. To assess how these constraints would affect the specific designs found, the values of the lowest drag and spillage drag found from each optimization with a $M_{DR} = 0.9$ were assessed to determine the values around the constraint. An upper bound on the samples was taken at $M_{DR} = 0.901$ to enable more robust sampling of the solutions. The averages across the 50 optimizations (Table D.1) demonstrated that the higher constraint $M_{DR} \geq 0.9$ resulted in higher minimum drag and spillage compared to the unconstrained evaluations. This represents a loss of convergence to the Pareto front caused by the constraint. However, the lower constraint had the opposite effect and provided beneficial reductions in both drag and spillage at $M_{DR} = 0.9005 \pm 0.0005$. In particular, this judicious use of the constraint of $M_{DR} \geq 0.89$ provides a useful reduction of nacelle cruise drag by about 2.4% and spillage by 11% relative to the unconstrained evaluations.

D.3 Influence of weighting the objective functions

In some situations one objective function is more important to the designer than others. This is of particular concern for nacelle design where there is a wide range of operating conditions. For example, it may be required to put a greater importance on the cruise drag coefficient over the spillage drag since the majority of the fuel burn is typically dictated by the cruise drag. In these cases, the option to guide the



(A) Unconstrained

(B) $M_{DR} \geq 0.89$ (C) $M_{DR} \geq 0.9$ FIGURE D.4: Example final Pareto optimal set extracted from the RSM (maximum obtained M_{DR})

optimization into regions of particular interest would be desirable. Several methods of applying weights to the objective functions have been proposed^{122 123 10} and this section demonstrates some of these on the nacelle RSM optimization to determine their efficacy. Objective weightings for evolutionary algorithms can broadly be split into two groups: those which manipulate the dominance function and those which skew the distance algorithm¹²³. Different benefits and downsides have been observed for these two approaches with guided dominance for instance being significantly faster¹²³. For the purposes of this analysis two approaches have been compared: dominance weighting¹²² and distance weighting¹⁰. Adjustment of the dominance function involves redefining the concept of domination¹²². In a standard implementation of NSGA-II, dominance is defined by the equation 2.5 in which the inequality must be strictly true in at least one objective. The weighted dominance approach which was utilised skews this definition to include a larger region in at least one objective dimension. This has the impact that if a solution dominates another in the weighted objective then it is allowed to be slightly worse in another. The domination equation then becomes equation D.1, where $\hat{x}$ dominates $\hat{y}$ if the equation is satisfied for all objectives and the inequality is strictly true in at least one. Values of $a_{12}, a_{13} \dots a_{m(m-1)}$ can then be set to any value to give a weighting of one objective function over another. For instance a value of $a_{12} = 2$ would give a preference of the second objective function over the first objective function of 2 units.

$$\Omega_i(f(x)) \leq \Omega_i(f(y)) // \quad (D.1)$$

$$\begin{aligned} \Omega_1(f_1, f_2) &= f_1 + a_{12}f_2 \\ \Omega_2(f_2, f_1) &= f_2 + a_{21}f_1 \\ \Omega_1(f_1, f_3) &= f_1 + a_{13}f_3 \\ &\dots etc. \end{aligned} \quad (D.2)$$

The weighted dominance operator was implemented into the NSGA-II algorithm and tested on the nacelle RSM to determine the performance of this method to find a specific region of the design space. In this example to demonstrate the approach a higher drag rise was desired at the expense of the other objectives. For these purposes values of $a_{21} = 0.5$, $a_{31} = 1.0$, $a_{12} = 2.0$, $a_{13} = 1.0$, $a_{23} = 0.5$ and $a_{32} = 2.0$ were used. This will give the second function (M_{DR}) a weighting over the other two ($C_{Dcruise}, \Delta C_{Dspill}$) and make the other two equally weighted between each other. These settings were used for an analysis to optimise the three objective functions

in the nacelle RSM. This analysis was again performed 50 times to allow statistical analysis of the optimization. Pareto fronts shown (Figure D.5) were chosen because they found the highest mean M_{DR} from the 50 individual optimizations performed. The application of this weighting showed that the dominance weighted optimization found a higher mean M_{DR} of 0.908 (Figure D.5b), compared to the highest mean M_{DR} found in the unweighted optimizations of 0.890 (Figure D.5a). The results of this (Figure D.5) demonstrated that a weighted dominance operator can give improved values of a particular objective function. For this example case the weighting has reduced the density of the solutions in the lower M_{DR} which meant that the lowest spillage found increased from -0.0008 in the unweighted case to 0.0020 when dominance weighting was applied. Similarly for cruise drag, the lowest value found for the unweighted optimization was 0.0274 which increased to 0.0297 in the dominance weighted optimization.

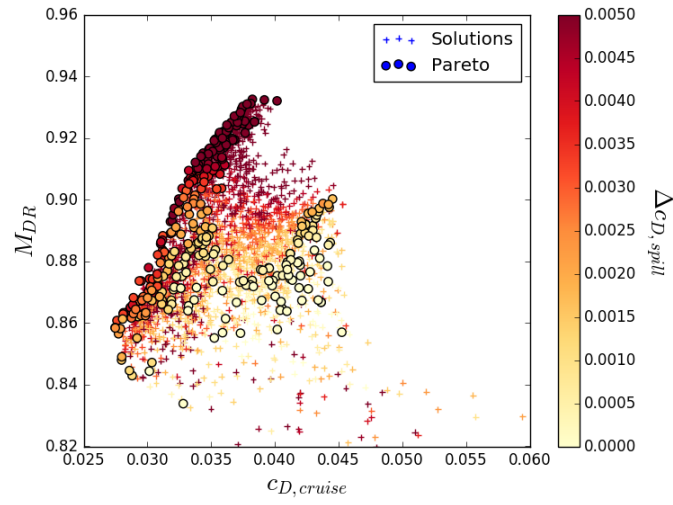
An alternative means of weighting the objective functions in the optimization is to adjust the distance operator¹⁰. In the standard implementation, the Euclidean distance is calculated between each individual in a NDS and used to determine the most isolated individuals. In a weighted distance operator, the Euclidean distance is skewed to give higher values to individuals which perform better in a preferred objective function whilst still including information on the spread of the individuals (equation D.3).

$$d(i, j) = \left[\sum_{k=1}^m w'_k \frac{(f_k^i - f_k^j)^2}{(f_k^{max} - f_k^{min})^2} \right]^{\frac{1}{2}} \quad (D.3)$$

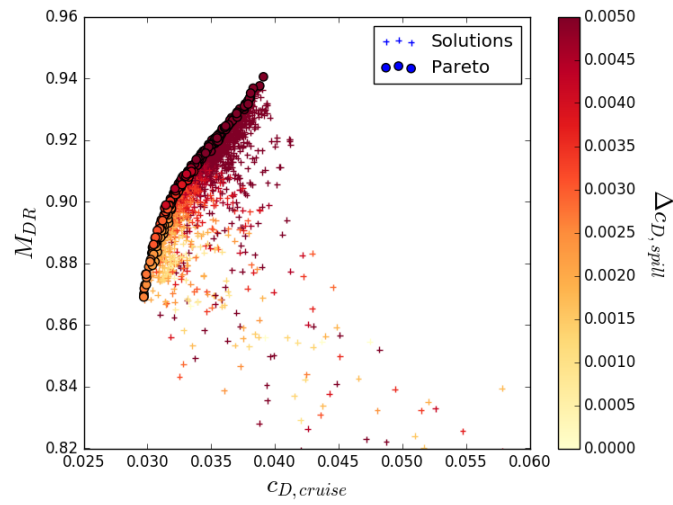
$$\begin{aligned} w'_k &= \frac{(1 - w_k)}{\max_{m=1}^{m=n} (1 - w_m)} \text{ for convex pareto fronts} \\ w'_k &= \frac{w_k}{\max_{m=1}^{m=n} w_m} \text{ for non - convex pareto fronts} \end{aligned} \quad (D.4)$$

$$w_k \in (0, 1)$$

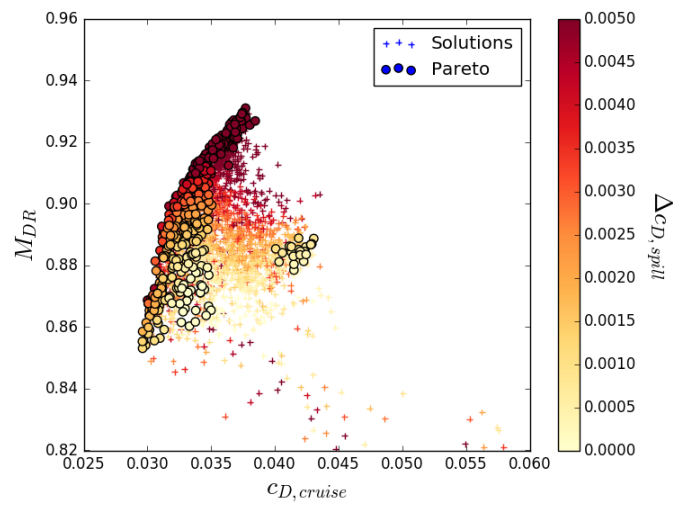
To compare to the performance of dominance weighting, a similar problem was posed for distance weighting. To weight drag rise Mach number over spillage drag and cruise drag, values of $w'_1 = 0.5$, $w'_2 = 1.0$ and $w'_3 = 0.5$ were used. 50 analyses were carried out with these settings for the three objective functions used in the nacelle RSM. 4360 evaluations were undertaken over 100 generations with



(A) Unweighted



(B) Weighted through dominance operator



(C) Weighted through distance operator

FIGURE D.5: Example final Pareto optimal sets extracted from the RSM

$i_{gen0} = 400$. An example result of this analysis (Figure D.5c) demonstrated that the use of a weighted distance operator could provide more beneficial values of a particular objective function at the expense of the others. The highest mean M_{DR} found by the distance weighted optimizations was 0.897 which is higher than the unweighted case but lower than the dominance weighted. In this example Pareto front (Figure D.5c) there was a minimum $c_{Dcruise}$ of 0.0296 and a minimum Δc_{Dspill} of -0.0006. This demonstrates a worse result in these two metrics compared to the unweighted but an improvement over the dominance weighted approach which negatively increased these values to 0.0296 and 0.0020 respectively. To assess the benefit of the weighted optimization methods to a nacelle design lowest drag and spillage values were taken again at a drag rise of $M_{DR} = 0.9$ with an upper bound of $M_{DR} = 0.901$ to enable sufficient sampling. At this drag rise Mach number, both weighted approaches provide a lower value of $c_{Dcruise}$ with reduction on average of 0.0017 from the dominance weighted optimizations (Table D.2). The spillage is conversely affected on average however by weighting (Table D.2). In addition, the most consistent results, i.e. those with the lowest standard deviation within the 50 optimizations, were from the dominance weighted optimizations (Table D.2). The standard deviation from the dominance weighted optimisation also demonstrated that this approach may be more consistent in producing improved designs than the constrained objective approach (Table D.1). In addition a similar sample was taken at $M_{DR} = 0.92$. to determine the representative functions at a more difficult drag rise Mach number. Table D.2 demonstrates that whilst on average the unweighted method finds a minimum $c_{Dcruise} = 0.0367$ with a standard deviation of 0.0011, the dominance weighted optimizations found a lower drag of 0.0355 on average with a lower standard deviation amongst the fifty independent cases. The spillage was concurrently reduced at this drag rise Mach number to on average 0.0069 from 0.0071 in the unweighted cases. The distance weighted optimization did not demonstrate any improvement for this given drag rise Mach number.

	Unweighted	Dominance weighted	Distance weighted
$\mu_{C_{Dcruise}} _{M_{DR}=0.9}$	0.0337	0.0320	0.0335
$\sigma_{C_{Dcruise}} _{M_{DR}=0.9}$	0.0016	0.0005	0.0024
$\mu_{C_{Dspill}} _{M_{DR}=0.9}$	0.0027	0.0033	0.0030
$\sigma_{C_{Dspill}} _{M_{DR}=0.9}$	0.0011	0.0005	0.0009
$\mu_{C_{Dcruise}} _{M_{DR}=0.92}$	0.0367	0.0355	0.0366
$\sigma_{C_{Dcruise}} _{M_{DR}=0.92}$	0.0011	0.0005	0.0012
$\mu_{C_{Dspill}} _{M_{DR}=0.92}$	0.0071	0.0069	0.0071
$\sigma_{C_{Dspill}} _{M_{DR}=0.92}$	0.0006	0.0004	0.0006

TABLE D.2: Average minimum cruise drag and spillage drag at a given drag rise Mach number range for the different weighted optimisations

APPENDIX E

Additional non-axisymmetric optimisation

An additional optimisation was carried out which added a fifth control aeroline to define the cowl geometry at $\theta = 45^\circ$. The lowest three control aerolines ($\theta = 180^\circ, \theta = 90^\circ, \theta = 135^\circ$) were fixed from the optimal design chosen from the first non-axisymmetric optimisation (Figure 6.12d). This reduced the degrees of freedom in the problem to eight. The bounds of this second optimisation (Table E.1) were expanded slightly from the first optimisation as it was noted that the Pareto set included designs which had the highest value of f_{max} for the $\theta = 0^\circ$ line which indicated that the Pareto set likely extended beyond that limit.

The procedure followed for this optimisation was the same as in section 6.3. This process therefore culminated in another Pareto set of equally optimal designs (Figure E.1). The Pareto set from this optimisation found designs which had a minimum drag of $6.4\%F_{Nref}$ which was similar to those found in the previous non-axisymmetric optimisation (Figure 6.9). A design was picked from the Pareto set with the lowest overall peak Mach number to compare with the similarly extracted design from the previous optimisation. This design benefited from the control of the additional aeroline at $\theta = 45^\circ$ and as such had a reduced peak Mach number compared to the previous optimal design (Figure E.2). Relative to the previous optimal

	Lower bound	Upper bound
f_{if}	0.6	1.4
r_{max}/r_{hi}	1.08	1.4
l_{fb}/l_{nac}	0.3	0.6
β_{nac}	11.0°	14.0°

TABLE E.1: Parametric geometry bounds used for the optimisation of the non-axisymmetric nacelle

design all the drag metrics also reduced which indicates an overall better design. The cruise drag was reduced from $D_{cruise}/F_{Nref} = 0.068$ to $D_{cruise}/F_{Nref} = 0.066$, the drag due to the Mach number increase was reduced from $D_{Mach}/F_{Nref} = 0.010$ to $D_{Mach}/F_{Nref} = 0.009$ and the spillage drag was reduced from $D_{spill}/F_{Nref} = 0.009$ to $D_{spill}/F_{Nref} = 0.005$. This study demonstrates that to produce a well designed nacelle the optimisation should have at least 5 aerolines to control the parametric design. The optimal design (Figure E.2b) from the optimisation with $\theta = 45^\circ$ control exhibited a peak Mach number on the cowl of $M_{peak} = 1.29$ which was present between the $\theta = 0^\circ$ and $\theta = 45^\circ$ aerolines, which indicates that further control could add benefit. However, this is a large reduction from the previous peak Mach number of $M_{peak} = 1.35$ for the optimisation without control of the $\theta = 45^\circ$ aeroline, and is generally within acceptable peak Mach number limits.

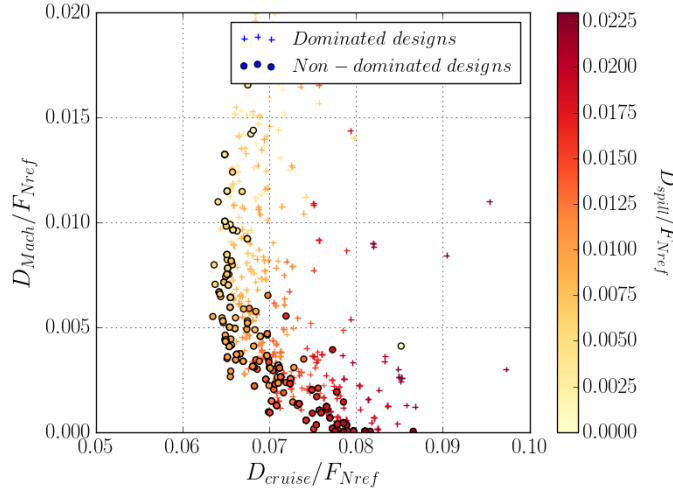
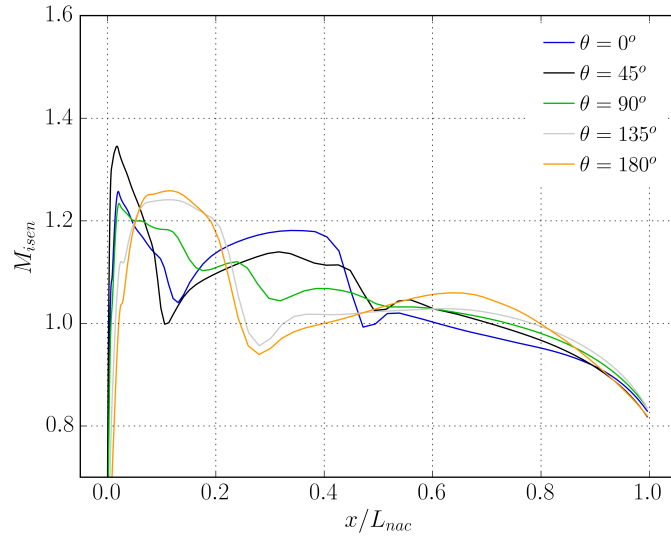
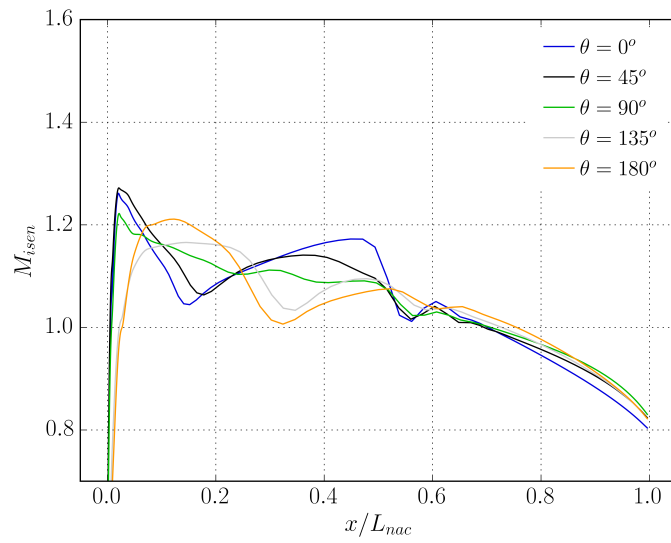


FIGURE E.1: Final Pareto set for the 3D non-axisymmetric optimisation in objective space



(A)



(B)

FIGURE E.2: Optimal designs from (A) previous optimisation (Section 6.3) and (B) optimisation with added aeroline control at $\theta = 45^\circ$

