Multi-Objective Shape Optimization of a Boundary Layer Ingesting Engine Intake

by

Ayesh Sudasinghe, B. Eng

A thesis submitted to the
Faculty of Graduate and Postdoctoral Affairs
in partial fulfillment of the requirements for the degree of

Master of Applied Science in Mechanical Engineering

Carleton University
Ottawa, Ontario

© 2021
Ayesh Sudasinghe

Abstract

The growth of the airline industry has highlighted the need for more environmentally conscious aviation leading to the conceptualization of more fuel efficient aircraft. One concept that has received significant attention and has been associated with improved fuel efficiency is the Boundary Layer Ingesting (BLI) propulsion system, which refers to the ingesting of the aircraft wake by the propulsors. Although BLI has theoretically been proven to reduce fuel burn, this can potentially be offset by the reduced efficiency and stability experienced by the propulsor in the presence of distorted inflow. Therefore, engine intakes must be optimized in order to mitigate the effects of BLI on the propulsion system.

In this work, the shape optimization of a BLI intake is investigated using a free form deformation technique in combination with a multi-objective genetic algorithm, in order to minimize pressure losses and distortion at the engine inlet. The optimization is performed on both a “straight-through” duct and a S-duct intake at a cruise altitude of approximately 37,000 feet and a free stream Mach number of 0.8 and 0.7, respectively. An optimization strategy was developed for the task which was able to produce a Pareto optimal set of designs with improved pressure recovery and distortion for both intakes. The general trend of the optimal designs show that to reduce distortion the optimizer accelerates the flow to reduce the size of the low total pressure region and increase the dynamic pressure at the engine inlet. In contrast, the pressure recovery was increased by reducing velocity as well as shifting the maximum velocity region to the outlet, which reduces the viscous dissipation losses within the intake.

Acknowledgements

First and foremost, I would like to thank my supervisors, Dr. Hamza Abo El Ella and Dr. Edgar Matida, for giving me this opportunity as well as their continuous support and guidance, making the last couple years an enjoyable and enriching learning experience. I am grateful for your encouragement and patience throughout this academic journey. Also, I would like to thank Dr. Mostafa El Sayed and fellow graduate student Padmassun Rajakareyar for their advice and suggestions during our meetings.

I also would like to thank my friends, without whom I would not have been able to complete this journey. Thank you for all the support and being there to push me to be successful in everything I do. Finally, thank you for all the memories and social distractions which truly made the last couple years more manageable.

Last, but certainly not least, my deepest gratitude and thanks to my family for their unwavering support and encouragement, without which I would not be where I am today. A special thank you to my parents, whom I will always be indebted to for their countless sacrifices; providing me with the opportunities and confidence to achieve my aspirations.

Table of Contents

Abstract	ii
Acknowledgements	iii
List of Tables	vi
List of Figures	vii
Nomenclature	xi
1 Introduction	1
1.1 Motivation and Objective	2
2 Literature Review and Background	4
2.1 Boundary Layer Ingestion (BLI)	4
2.1.1 BLI Aerodynamic Analysis	6
2.1.2 BLI Conceptual Studies	9
2.1.3 Effects on Engine Performance	13
2.2 Review of Intake Optimization Studies	18
2.2.1 Non-BLI Intake Optimization	18
2.2.2 BLI Intake Optimization	19
2.3 Numerical Optimization	21
2.3.1 Overview of Terminology and Definitions	22
2.3.2 Gradient-Based Optimization	25
2.3.3 Gradient-Free Optimization	26
2.4 Geometry Parameterization	28
2.4.1 Overview of B-spline Curves	28
2.4.2 B-Spline Surfaces	30
2.5 Free Form Deformation (FFD)	31

3	Methodology	35
3.1	Geometry Creation	35
3.1.1	Cross Section Definition	36
3.1.2	Duct Geometry	37
3.1.3	Pre- and Post-Domain	40
3.2	Geometry Deformation	41
3.3	Optimization Algorithm	44
3.3.1	<i>gamultiobj</i> Process	46
3.4	Intake Optimization	48
3.4.1	Computational Mesh	50
3.4.2	CFD Methods	52
3.4.3	Objective Functions	55
3.4.4	Alternate Distortion Coefficient	57
4	Results and Discussion	59
4.1	“Straight-Through” Duct Optimization	59
4.2	S-Duct Optimization	65
4.2.1	Sensitivity of the Optimal Shape to the Length Limits	73
4.2.2	Effects of the Centerline Curvature	79
4.2.3	Effects of the Distortion Coefficient on the Optimal Shape	84
5	Conclusions and Recommendations	91
5.1	Recommendations	93
	References	94
	Appendix A Genetic Algorithm: Sensitivity Analysis	101
	Appendix B Grid Convergence Study	105
	Appendix C Outlet Pressure Variation	107
	Appendix D Turbulent Profiles	109
	Appendix E Area Constraint	111

List of Tables

1.1	NASA’s initiatives for fuel burn and noise reduction for the next generation of aircraft [Adapted from [3]]	2
2.1	Examples of gradient-free optimization algorithms	27
3.1	Optimization parameters	49
3.2	Boundary conditions used for the simulations	55
4.1	Comparison of the baseline performance with the Pareto front designs	61
4.2	Boundary conditions for CFD simulations	65
4.3	Comparison of the Pareto optimal designs	67
4.4	New optimization parameters	74
4.5	Comparison of the Pareto optimal designs	75
4.6	New optimization parameters for centerline deformation	80
4.7	Comparison of the Pareto optimal designs of the centerline deformation study	81
4.8	Comparison of the Pareto optimal designs for the alternate distortion coefficient	86
B.1	CFD simulation times with respect to the mesh density	106

List of Figures

2.1	Schematic of BLI for an embedded engine [Adapted from [6]]	5
2.2	Comparison between the podded and BLI configuration [4]	6
2.3	Visual representation of the losses associated with podded and BLI configurations [15]	8
2.4	HWB aircraft & S-duct intake for an embedded configuration [5] . . .	10
2.5	The novel D8 “double-bubble” aircraft concept [5]	10
2.6	Conceptual rendering of NASA’s STARC-ABL aircraft with turboelectric BLI engine [20]	12
2.7	Example of BLI inlet pressure distortion [9] Red = Low total pressure, Blue = High total pressure	13
2.8	Effect of fan efficiency loss on TSFC [27]	15
2.9	Effect of inlet total pressure losses on TSFC [27]	15
2.10	Comparison of the total pressure ratio with and without the fan [30] .	16
2.11	Illustration of the bleed slot locations on the S-duct [36]	20
2.12	The design space corresponding to the objective function	22
2.13	Example of linear & non-linear constraints delimiting the feasible design space [Adapted from [39]]	24
2.14	Example of a Pareto front [Adapted from [40]]	25
2.15	Illustration of the components required to define a B-spline curve(knot vector = [0,0,0,0,0.5,1,1,1,1])	30
2.16	Example of a surface created using B-spline concepts	31
2.17	FFD: Undeformed & deformed plastic region [48]	32
2.18	FFD control points defining the deformation lattice [48]	33
3.1	Control polygon used to define the shape of the duct cross-section ($\mathbf{U} = [0, 0, 0, 0.25, 0.25, 0.5, 0.5, 0.75, 0.75, 1, 1, 1]$)	36
3.2	Control net for the “straight-through” duct	37

3.3	Control point locations for the definition of the S-duct centerline curvature [red points = control points]	38
3.4	Control net defining the S-duct surface	39
3.5	Surface points generated for the S-duct & “straight-through” duct using B-splines	39
3.6	3D rendering of the S-duct domain for CFD simulations	41
3.7	“Straight-through” duct embedded inside a FFD volume	42
3.8	2D representation of the hypothetical FFD volume for a S-duct	43
3.9	General GA flowchart [Adapted from [52]]	44
3.10	Example of how individuals are ranked within a population [Adapted from [54]]	47
3.11	Variables and terminology defining the S-duct geometry [Green: varied during optimization; Red: fixed]	49
3.12	Optimization strategy architecture	50
3.13	Structured mesh of the “straight-through” duct ($L = 6$ m)	51
3.14	Structured mesh of the S-duct geometry ($L = 6$ m)	52
3.15	Inlet total pressure profile for a Mach number of 0.8 at 37,000 ft	54
3.16	Illustration of the DC(60) distortion coefficient calculation procedure; $\alpha = 60^\circ$	56
3.17	Alternate distortion coefficient calculation procedure	58
4.1	Evaluation history of the “straight-through” duct optimization (Including thumbnails of the normalized total pressure distortion at the engine inlet for a subset of the Pareto front)[$3\text{ m} \leq L \leq 8\text{ m}$]	60
4.2	A comparison of the symmetry plane Mach contours and the total pressure distribution at the engine inlet [Top = optimal distortion ($L = 5$ m); Bottom = reference duct of equal length ($L = 5$ m)]	62
4.3	Comparison of the cross sectional shapes of the optimal distortion duct and an undeformed duct of equal length	63
4.4	A comparison of the symmetry plane Mach contours and the total pressure distribution at the engine inlet [Top = optimal PR ($L = 4$ m); Bottom = reference duct of equal length ($L = 4$ m)]	64
4.5	Comparison of the cross sectional shapes of the optimal PR duct and an undeformed duct	65
4.6	Evaluation history and Pareto front for the S-duct optimization	66

4.7	Comparison of the symmetry plane Mach number contours and normalized total pressure distribution at the engine inlet [Top = optimal distortion (II), Middle = optimal PR (VI), Bottom = reference duct of equal length (I)]	69
4.8	Comparison of the symmetry plane curvature of the optimal distortion duct, optimal PR duct, and a reference duct of equal length	70
4.9	Comparison of the cross sections of the optimal distortion duct and an undeformed duct	71
4.10	Comparison of the cross sections of the optimal PR duct and an undeformed duct	72
4.11	Symmetry plane Mach number contours and normalized total pressure distribution at the engine inlet of the dominated solution (VII; $L = 6$ m)	73
4.12	Evaluation history of the S-duct optimization with increased length limits (thumbnails show the normalized total pressure distribution at the engine inlet)	74
4.13	Comparison of the symmetry plane Mach number contours and the normalized total pressure distribution (at the engine inlet) of the optimal distortion ducts [Top = current optimal design ($L = 12$ m); Bottom = original optimal design ($L = 8$ m)]	76
4.14	Comparison of the cross sectional shapes of the current and original optimal distortion ducts [Left = current optimal design ($L = 12$ m); Right = original optimal design ($L = 8$ m)]	77
4.15	Comparison of the symmetry plane Mach number contours and the normalized outlet total pressure distribution of the optimal PR ducts [Top = current optimal design ($L = 9$ m); Bottom = original optimal design ($L = 8$ m)]	78
4.16	Comparison of the cross sections of the current and previous optimal PR duct with an undeformed duct [Left = current optimal design ($L = 9$ m); Right = original optimal design ($L = 8$ m)]	79
4.17	Evaluation history of the centerline deformation optimization with illustrations of the normalized total pressure distribution for a subset of the Pareto optimal designs	80
4.18	Comparison of the symmetry plane Mach number contours and the normalized total pressure distribution at the engine inlet [Top = Opt. PR (Design II); Middle = Opt. Dist. (Design VIII); Bottom = original (Design I)]	82
4.19	The flow solution of the duct with the best trade-off between PR and distortion (Design V in Figure 4.17) including a comparison of the centerline curvatures	84
4.20	Evaluation history and Pareto front for the new distortion coefficient	85

4.21	Mach number contours (symmetry plane) and normalized total pressure distribution (engine inlet; A-A) of optimal distortion ducts [Top = new distortion coefficient (D); Bottom = DC(60)]	87
4.22	Comparison of the cross sectional curvature of the optimal distortion ducts [Left = new distortion coefficient (D); Right = DC(60)]	88
4.23	Mach number contours (symmetry plane) and normalized total pressure distribution (engine inlet) of optimal PR ducts [Top = new distortion coefficient (D); Bottom = DC(60)]	89
4.24	Comparison of the cross sectional curvature of the optimal PR ducts [Left = DC(60); Right = new distortion coefficient (D)]	90
A.1	Evaluation history and Pareto front using a population size of 50 and a maximum of 20 generations	102
A.2	Evaluation history and Pareto front using a population size of 50 and a maximum of 40 generations	103
A.3	Evaluation history and Pareto front using a population size of 50 and a maximum of 50 generations	103
A.4	Evaluation history and Pareto front using a population size of 60 and a maximum of 40 generations	104
A.5	Evaluation history and Pareto front using a population size of 70 and a maximum of 40 generations	104
B.1	Pressure recovery (PR) and distortion [DC(60)] for the “straight-through” duct (duct length = 6 m)	105
B.2	Pressure recovery (PR) and distortion [DC(60)] for the S-duct (duct length = 6 m)	106
C.1	Outlet pressure variation with respect to inlet area for “straight-through” duct	107
C.2	Outlet pressure variation with respect to inlet area for S-duct	108
D.1	Turbulent kinetic energy profile from 2D flat plate simulation	109
D.2	Turbulent eddy frequency profile from 2D flat plate simulation	110

Nomenclature

Acronyms

B-Spline	Basis Spline
BLI	Boundary Layer Ingestion
BWB	Blended Wing Body
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CRM	Common Research Model
FAA	Federal Aviation Administration
FFD	Free Form Deformation
GA	Genetic Algorithm
HWB	Hybrid Wing Body
IATA	International Air Transport Association
LTO	Landing/Takeoff
MIT	Massachusetts Institute of Technology
NASA	National Aeronautics and Space Administration
NSGA	Non-Dominated Sorting Genetic Algorithm
PSO	Particle Swarm
RANS	Reynolds-Averaged Navier Stokes
S-duct	Serpentine duct

SA	Simulated Annealing
SST	Shear Stress Transport
TSFC	Thrust Specific Fuel Consumption
UHB	Ultra High Bypass

Variables

$\dot{m}$	Mass flow rate	$kg \cdot s^{-1}$
A_i	Element area	m^2
A_j	Ring area	m^2
D	Alternate distortion coefficient	-
$DC(60)$	Distortion coefficient	-
F_{engine}	Thrust generated by the engine	N
KE	Kinetic energy	J
M_{inlet}	Inlet free stream Mach number	-
$P_{t,60,min}$	60° sector with the worst area-averaged total pressure	Pa
$P_{t,avg,inlet}$	The area-averaged total pressure at the duct inlet	Pa
$P_{t,avg,j}$	Average total pressure within the ring	Pa
$P_{t,avg,outlet}$	The area-averaged total pressure at the duct outlet (engine inlet)	Pa
$P_{t,i}$	Total pressure within the element	Pa
PR	Pressure recovery	-
q	Dynamic pressure at the duct outlet (engine inlet)	Pa
Re_x	Reynolds number	-
u_∞	Free stream velocity	$m \cdot s^{-1}$
u_j	Jet velocity	$m \cdot s^{-1}$
u_w	Wake velocity	$m \cdot s^{-1}$

y^+	Non-dimensional wall distance	-
-------	-------------------------------	---

Greek Symbols

α	DC(60) sector angle
δ	Boundary layer thickness
Φ	Dissipation

Chapter 1

Introduction

An appreciation and awareness of the detrimental impact of climate change have shifted the technological industry towards developing greener alternatives. With billions of people being transported by airlines each year, the aviation industry is a major contributor of carbon emissions. The International Air Transport Association (IATA) is projecting that by the year 2037 the number of passengers transported by airlines will reach 8.2 billion, almost double the number of passengers from 2019 [1, 2]. The growth of the airline industry at this magnitude is accompanied by the increased detrimental impact on the environment and the increased noise pollution. As a result there is a greater focus on environmentally conscious aviation, rapidly growing towards more fuel efficient and quieter aircraft. Therefore, NASA has set aggressive goals to drastically reduce emissions and noise for the next generation of aircraft which are summarized in Table 1.1. In addition to a 70% reduction in aircraft fuel burn with respect to a “state-of-the-art reference aircraft”, NASA has also set goals to attempt to reduce the Landing/Takeoff (LTO) NO_x emissions by 75% relative to the standards set out in the sixth meeting of the Committee on Aviation Environmental Protection (CAEP/6) as well as reducing the cumulative noise (lateral, flyover, and approach noise) by 71 decibels (dB) relative to the FAA Stage 4 regulations [3]. These initiatives require ingenuity for conceptualizing novel aircraft configurations and/or implementing existing technology in a way to provide improved

performance.

Table 1.1: NASA’s initiatives for fuel burn and noise reduction for the next generation of aircraft [Adapted from [3]]

Corners of the trade space	N+3 (2025) Technology Benefits
Noise	-71 dB
Landing/Takeoff (LTO) NO _x Emissions	better than -75%
Performance Aircraft Fuel Burn	better than -70%

The pylon-mounted under wing engine configuration has long remained the most popular arrangement in civil aviation. However, given the need for a more fuel efficient generation of aircraft, the research community has recently investigated alternate configurations for the potential improvements of reducing drag and increasing propulsive efficiency. One concept at the forefront for reduced fuel burn and noise reduction is the boundary layer ingestion (BLI) propulsion system. A BLI propulsion system is one in which the boundary layer developed over the fuselage is ingested into the propulsor and re-energized. The benefit in BLI system configuration lies in the fact that ingesting a lower momentum fluid increases the propulsive efficiency [4]. However, ingesting the boundary layer raises issues regarding the system’s compression components, which tend to operate with reduced efficiency and stability when subjected to distorted inflow.

1.1 Motivation and Objective

Intakes help deliver air to the the propulsion system and are integral to the efficiency of the engine. This is especially true for embedded propulsion systems, particularly the ones that ingest the boundary layer, which require intakes to help direct the flow to the propulsor. Otherwise, the engines may experience a reduced mass flow which can lead to stall or surge of the compression system leading to reduced nominal thrust

output. Thus, the performance of the intakes is crucial to the operation of the engine and must be designed to minimize the pressure losses and distortion at the engine inlet as both conditions lead to engine efficiency issues and increased fuel burn.

Therefore, the objective of this thesis is to implement a shape optimization strategy using commercially available software in an attempt to minimize the total pressure losses and distortion present at the engine inlet due to BLI. Additionally, the optimization strategy is applied to both a “straight-through” duct and a S-duct intake.

Chapter 2

Literature Review and Background

In this chapter, the boundary layer ingestion theory is presented in detail including an aerodynamic analysis of the potential benefits of BLI and its effects on the engine performance. Then an overview of numerical optimization will be presented, followed by a review of the concepts of B-splines and Free Form Deformation (FFD).

2.1 Boundary Layer Ingestion (BLI)

Civil aircraft are currently propelled using jet engines mounted under the wings, allowing the engine to be treated as a separate system from the rest of the aircraft and to ingest undisturbed and uniform flow. Conversely, BLI is an aeropropulsive concept that requires the integration of the propulsion system with the fuselage for reduced fuel burn and efficient propulsion of the aircraft. Boundary layer ingestion (BLI), in this context, means the ingesting of the boundary layer developed over the fuselage into the propulsor as depicted in Figure 2.1. This offers a combination of aerodynamic and system level benefits as ingesting and re-energizing the low momentum boundary layer flow that has formed over the fuselage can substantially reduce the aircraft's drag and increase the propulsive efficiency [5].

Figure 2.1: Schematic of BLI for an embedded engine [Adapted from [6]]

Smith and Roberts [7] first proposed the concept of BLI back in 1946 when they studied the affects of continuous boundary layer removal using suction. The BLI concept can be found in marine applications as it has been integrated into ship propulsion and torpedoes [4, 8]. However, BLI has not been implemented in civil aircraft due to a number of challenges yet to be resolved. For instance, the necessary integration of the engine with the fuselage for BLI begets design complications [9]. As well, the non-uniform velocity profile of the boundary layer leads to pressure distortions at the engine inlet resulting in fan and compressor stability and engine efficiency issues [5].

BLI deals with reducing the total drag acting on the aircraft, by re-energizing the slower moving air within the boundary layer, effectively reducing the drag and wasted kinetic energy for a given thrust requirement [10]. The drag experienced by the aircraft is attributed to the viscous losses present within the flow field, which is the catalyst for the formation of the boundary layer and in extension, the wake. These losses create a momentum deficit within the wake which contributes to the pressure drag of the aircraft. The pressure drag acting on the aircraft is equal to the rate of change of the momentum between the front of the aircraft and the wake. Thus, the greater the momentum deficit within the wake, the greater the drag experienced by the aircraft. Therefore, ingesting the boundary layer re-energizes the wake flow reducing the momentum deficit and thus, the drag on the aircraft [10]. This conceptualization is for the condition of zero net momentum and cruise flight [11]. The following section will show a more detailed analysis of the BLI concept.

2.1.1 BLI Aerodynamic Analysis

Force Analysis

An assessment of the aerodynamic performance of boundary layer ingestion was conducted by Plas [4], comparing the power requirements between a podded and BLI configuration.

Figure 2.2: Comparison between the podded and BLI configuration [4]

Figure 2.2 shows the comparison between podded and the ideal BLI configuration, where 100% of the aircraft wake is ingested. In the podded configuration, the inflow is at the free stream velocity, u_∞ , and is accelerated to a velocity, u_j , creating a momentum excess which balances the momentum deficit caused by the airframe drag. In contrast, the ideal BLI configuration has an inflow velocity of that equal to the aircraft wake, u_w . In cruise conditions, there is a zero net force acting on the aircraft and thus the thrust required by the engine is equal to the airframe drag which is the mass flow times the increase in velocity as shown by Plas [4].

$$F_{engine, no BLI} = \dot{m}(u_j - u_\infty) = \dot{m}(u_\infty - u_w) = D_{airframe} \quad (2.1)$$

$$F_{engine, BLI} = \dot{m}(u_j - u_w) = \dot{m}(u_\infty - u_w) = D_{airframe} \quad (2.2)$$

In addition, the energy added to the flow by the engine is equal to the change in kinetic energy (KE).

$$KE_{no\ BLI} = \frac{1}{2}\dot{m}(u_j^2 - u_\infty^2) \quad (2.3)$$

$$KE_{no\ BLI} = \frac{F}{2}(u_j + u_\infty) \quad (2.4)$$

In comparison, the energy added to the flow for BLI is shown in Equation 2.5.

$$KE_{BLI} = \frac{1}{2}\dot{m}(u_j^2 - u_w^2) \quad (2.5)$$

Assuming the ingested wake flow is accelerated back to free stream velocity, $u_j = u_\infty$, the power added becomes:

$$KE_{BLI} = \frac{F}{2}\dot{m}(u_\infty + u_w) \quad (2.6)$$

The comparison of Equations 2.4 and 2.6 show that since $u_j > u_w$, less power is required to maintain cruise flight conditions for the BLI configuration and therefore, less fuel consumption. Since both configurations are required to overcome the same drag and less power is added to the BLI configuration, there is less wasted energy with BLI.

However, the analysis performed in [4] assumes both configurations are subject to the same drag. The accuracy of this assumption is difficult to predict as embedding the engine within the airframe changes the pressure distribution around the aircraft. Also, BLI essentially “fills in” the wake, reducing the momentum deficit and thereby reducing the required thrust (drag) [12]. Therefore, the assumption that the drag the engine must overcome is equal for both cases is dependant on the configuration of the BLI aircraft and is difficult to assess.

Power Balance Method

An alternative aerodynamic assessment of BLI is using the power balance method developed by Drela [13]. The advantage of the power balance approach is that it does

not require any assumptions regarding the predictions of thrust and drag, which is especially important given the ambiguity of these forces in an integrated configuration. Hence, the objective of the power balance method is not to compare the power required for a given thrust but the comparison of engine power required to maintain a net zero stream-wise force for a given configuration. This method is analogous to the principle of conservation of energy, it identifies the power sources and sinks within a flow field in order to compare the potential areas of reduced losses for the BLI and non-BLI configurations [14]. Figure 2.3 is a visual overview of the losses present within an aircraft flow field.

Figure 2.3: Visual representation of the losses associated with podded and BLI configurations [15]

Drela [13] derives the mechanical power balance equation, given as Equation 2.7, which states that the mechanical power supply, P_k is equal to the total dissipation, Φ . Figure 2.3 shows that the total dissipation is attributed to the mixing within the wakes (Φ_{wake}), vortices (Φ_{vortex}), and jets (Φ_{jet}) due to the presence of non-uniform velocities as well as the losses due to surface dissipation ($\Phi_{fuselage}$) [13, 14].

$$P_k = \Phi_{wake} + \Phi_{jet} + \Phi_{fuselage} + \Phi_{vortex} \quad (2.7)$$

The difference between the mechanical power supply and the wasted kinetic energy within the jet ($P_k - \Phi_{jet}$) is the net propulsive power, in other words, this remaining power must balance the energy consumed by the aircraft. Therefore, the net propul-

sive power must balance the dissipation associated with the wake, fuselage, and the vortices. For the BLI case, wake dissipation and jet dissipation are both less than for the non-BLI configuration. The wake dissipation is less by the factor of ingested wake. For example, if 100% of the wake is ingested then the wake dissipation term disappears completely. Equation 2.8 gives the mechanical power balance equation for a BLI configuration, assuming a fraction of the boundary layer is ingested [5].

$$(P_k - \Phi_{jet}) = (1 - f_{BLI})\Phi_{wake} + \Phi_{fuselage} + \Phi_{vortex} \quad (2.8)$$

Assuming the fuselage and vortex dissipation is equal in both configurations, the BLI configuration requires less mechanical power supply and thus results in less fuel burn.

The aerodynamic analysis presented makes assumptions which cannot be verified without an established aircraft configuration. Therefore the true aerodynamic benefit of BLI and the magnitude of the benefit is still debated as the configuration could significantly alter the dissipation associated with the fuselage. The following section will review studies performed to evaluate the benefit of BLI.

2.1.2 BLI Conceptual Studies

Although the concept of BLI dates back nearly as far as the advent of the jet engine, it is yet to be implemented in civil aircraft. However, recent initiatives to reduce emissions, noise, and fuel consumption for a more sustainable future has pushed companies to develop technologies with a reduced detrimental impact on the environment. This has led to the conceptualization of unconventional aircraft configurations such as the blended wing body (BWB) and hybrid wing body (HWB) aircraft. Generally, HWB and BWB aircraft are designed with the engines integrated into the airframe. This begets the need for a serpentine duct (“S-duct”) intake to direct the airflow into the propulsor and has accelerated the growth in the research of BLI. Figure 2.4 shows an example of a HWB aircraft and a proposed S-duct inlet.

Figure 2.4: HWB aircraft & S-duct intake for an embedded configuration [5]

In recent years, research has been conducted by NASA, MIT, and other research organizations on BLI concepts for commercial aircraft. In 2009, NASA had set goals towards developing the next generation of aircraft with reduced fuel burn, noise, and emissions [5]. NASA, in collaboration with MIT, Aurora Flight Sciences, and Pratt & Whitney proposed the concept of a super-wide fuselage with flush-mounted engines on the top aft of the fuselage, which they have termed the D8 “double bubble” [5]. Figure 2.5 shows the conceptual rendering of the “double bubble” aircraft designed by MIT and NASA.

Figure 2.5: The novel D8 “double-bubble” aircraft concept [5]

Multiple studies have been conducted on the benefit of BLI using the D8 aircraft. Pandya et al. [16] performed a CFD analysis on a 1:11 scale D8 aircraft using an actuator disk model for the fan with a uniform pressure jump and predicted a 9% reduction in required power for the “double bubble” compared to the conventional

podded configuration of the D8 aircraft. In addition, Uranga et al. [17] performed wind tunnel testing on the scaled (1:11) D8 aircraft at a zero-degree angle of attack. The testing was conducted for the the podded and BLI configuration of the D8 aircraft using the same propulsor (i.e. same internal equipment and same inlet area). A comparison between the two configurations showed a 6% reduction in cruise power, determined based on the electrical power drawn by the BLI and non-BLI configurations. Later, this work was extended in [18] to conduct a more detailed analysis of the BLI benefit for the D8 aircraft using the wind tunnel data for the scaled model. Using the power balance method, parametric equations were develop for the input of mechanical flow power as well as the net streamwise force. Also, expressions for the dissipation were determined allowing for the analysis of the sources contributing to the benefit. This study provided the benefit of BLI decoupled from the D8 aircraft configuration. The results showed that for an equal nozzle area propulsor (same propulsor used for both configurations), the reduction in required power ranged between 7.6% and 8.5% depending on the amount of airframe boundary layer ingested. Also, a comparison between equal mass flow rates showed a 9% reduction in power requirement.

Blumenthal et al. [19] investigated the potential benefits of integrating a BLI propulsion system into a Common Research Model (CRM) geometry. The CRM model is a baseline aircraft geometry developed in collaboration with both public and private sector groups for the purposes of computational fluid dynamics analysis (CFD) and validation. As such, CFD simulations were performed at a cruise flight condition of Mach 0.85 at an altitude of 38,500 feet and an angle of attack of 2 degrees. The results showed a 8% reduction in engine power requirements at cruise flight for the BLI configuration as compared to the baseline geometry. The authors also performed small geometric variations to the aft portion of the fuselage to optimize the benefits of BLI which saw marginal improvements in engine power requirements, resulting in an 8.7% reduction in engine power. It is noted that the actuator disk model used to represent the BLI propulsor imparts a uniform pressure jump across the disk, however, a fan which encounters distorted flow will impart a smaller pressure rise to

the high momentum flow as compared to the low momentum flow which can affect the flow behaviour and accuracy of benefit of BLI.

A few other conceptual designs which leverage the benefits of BLI are NASA's N3-X BWB and STARC-ABL, and Cambridge-MIT's Silent Aircraft (SAX). The STARC-ABL concept is an alternative to the BWB aircraft which does not require a complete redesign of the airframe. Figure 2.6 shows a conceptual rendering of NASA's STARC-ABL which is a conventional tube-and-wing aircraft with a BLI engine mounted on the aft of the fuselage and is similar to the configuration used to carry out the assessment in this thesis.

Figure 2.6: Conceptual rendering of NASA's STARC-ABL aircraft with turboelectric BLI engine [20]

The under-wing engines provide power to an electric generator to power the BLI propulsor. Welstead and Felder [21] have concluded that the STARC-ABL can achieve up to 12% reduction in fuel consumption, however, it is not clear how much of the benefit is directly related to BLI. NASA's N3-X BWB and Cambridge-MIT's SAX aircraft both implement BLI engines, however, the author has not found any work related to the direct benefit from BLI.

The conceptual studies reviewed in this section show the potential for large gains from BLI, generally resulting in reduced engine power requirements leading to a reduction in fuel consumption. However, BLI is not without its potential concerns and the following section will discuss these issues.

2.1.3 Effects on Engine Performance

BLI engines suffer from inlet total pressure non-uniformity and reduced total pressure at the engine inlet, thus the response of fans and compressors to these conditions has also received significant attention. Inlet total pressure distortions is not unique to BLI configurations, aircraft can experience distorted inflow due to separation of the flow within the engine inlet caused by a blockage, crosswinds or rapid maneuvers such as those performed by military aircraft [9, 22]. The fan and compressor are integral components and their performance has a significant impact on the specific fuel consumption and overall efficiency of the engine. As previously mentioned, BLI has gained traction due to the conceptual designs for BWB and HWB aircraft which incorporate a S-duct inlet and thus, the distortion present at the engine inlet is a combination of the boundary layer flow and the secondary flow losses inherent to S-ducts which exacerbate the distortion at the fan face. Figure 2.7 shows an example of the possible BLI-type inlet pressure distortion.

Figure 2.7: Example of BLI inlet pressure distortion [9]
Red = Low total pressure, Blue = High total pressure

Due to the local pressure variation shown in Figure 2.7 there is the potential for engine performance degradation and fan stability issues [23]. Gunn and Hall [24] found that the presence of BLI distortion can lead to a 1% to 2% reduction in stage efficiency with the distortion having a greater impact on the rotor than the stator. In addition,

the distorted total pressure presents challenges with respect to the aeromechanical integrity of the components. For example, the fan and compressor blades will be continuously subject to high and low total pressure regions, which introduce fatigue issues due to the cyclic loading on the turbomachinery components. Finally, the distorted inflow is capable of producing potential blade vibrations [25, 26]. Hence, it is possible that propulsive efficiency gains from BLI are offset by the engine performance degradation and the effect on component lifetime. Therefore, it is prudent that the impact of BLI on the engine be thoroughly investigated and thus, the following will present a review of previous studies on the consequences of BLI-type flow distortion on engine performance and fan stability. However, it is important to note that the impact of BLI is dependent upon the severity and strength of the distortion which is related to several factors: the assumed percentage of boundary ingestion, the diffusion rate of the flow through the duct, and the aircraft configuration. As such, it is difficult to assess the true impact of BLI, but it is beneficial to review these studies to better understand the underlying issues.

Lucas [9] conducted experimental testing on the effects of ingesting a boundary layer on a modified Pratt & Whitney JT15D-1 turbofan engine using distortion screens designed to replicate the total pressure distortion produced by NASA's Inlet A and is shown in Figure 2.7. The results indicated a 15.5% decrease in stream thrust and a 14% increase in thrust specific fuel consumption. Therefore, depending on the trade-off between the reduced thrust requirement due to BLI and the loss in stream thrust, the potential benefit of BLI may be negligible or non-existent.

Hardin et al. [27] used the Numerical Propulsion System Simulation (NPSS) computational model to study the effects of boundary layer ingestion on an Ultra-High-Bypass (UHB) turbofan propulsion system. The simulations used the boundary layer profiles and flow field associated with the Boeing N2A-EXTE blended wing body (BWB) aircraft. Using NPSS computational model, the sensitivity of the thrust specific consumption (TSFC) to reduced fan efficiency and increased inlet total pressure losses were determined and are shown in Figures 2.8 & 2.9.

Figure 2.8: Effect of fan efficiency loss on TSFC [27]

Figure 2.9: Effect of inlet total pressure losses on TSFC [27]

As evident from Figures 2.8 & 2.9, the loss in total pressure at the inlet has a markedly more significant impact on the TSFC. The sensitivity studies were then implemented within the computational analysis to account for the performance losses associated with BLI and applied to a 5-engine configuration with individual inlets embedded within the upper surface of the airframe. The results show that due to the distorted flow seen by the fan, there would be a 1.3% decrease in fan efficiency, which is responsible for a 1.1% increase in fuel burn. The results also show a 0.35% loss in

inlet pressure which is also responsible for a 1.1% increase in fuel burn. Therefore, it leads to an overall increase of 2.2% in fuel consumption in the presence of a BLI-type distortion.

Gladin et al. [23] used simulation software to model an HWB 300 passenger aircraft, derived from the Boeing N2A with ultra-high-bypass (UHB) turbofan engines in both podded and BLI configurations. The simulations were performed for a cruise Mach number and altitude of 0.8 and 35,000 feet, respectively. Based on previous research [28, 29], the authors assumed a 1% to 2% reduction in inlet total pressure recovery and a 1% to 3% reduction in fan efficiency as the nominal values of the system level parameters. Using these estimated losses, the model concluded a 3% to 4% increase in thrust specific fuel consumption (TSFC), depending on the design fan pressure ratio (FPR).

The presence of the distortion at the engine inlet also effects the fan performance and can lead to reduced pressure rise, stability margin, and efficiency as well as increased blade loading. Giuliani et al. [30] studied the effects of BLI, through an S-duct intake, and fan interaction using numerical simulations for a free stream Mach number of 0.4. The numerical simulations were performed on a BLI inlet designed and tested at the NASA Langley Research Center 0.3-meter cryogenic wind tunnel as possible geometries for blended wing body aircraft. The simulations were first performed without the presence of a fan and compared to the results with BLI and fan interaction.

Figure 2.10: Comparison of the total pressure ratio with and without the fan [30]

Figures 2.10 show the total pressure distortion at the engine inlet with (left) and without (right) the presence of the fan. As evident from the above figures, the presence of the fan slightly reduces the height of the distortion and also reduces the losses in total pressure (i.e. higher total pressure recovery at the inlet). However, the counter-rotating vortices causes high incidence angles at the blades leading to higher loading [30]. The results also show that the distortion does not shift in the circumferential direction and is attenuated naturally as it propagates through the fan. The attenuation of the distortion through the fan indicates that there is more energy input into the low total pressure region than the rest of the flow. There was no report of the presence of stall in this paper.

The potential impact of inlet swirl distortion on the fan was experimentally investigated by Frohnapfel et al. [31]. The experiments were conducted on a JT15D-1 Pratt & Whitney turbofan engine. The engine inlet swirl was produced using a swirl generator which replicated the swirl profile at the engine inlet of a conceptual BWB aircraft. Frohnapfel et al. compared the results for the distorted flow conditions to a non-distorted, clean inlet flow and determined that the presence of the distortion reduced the fan outlet total-to-atmospheric pressure ratio by 1.5% indicating a reduction in total work input to the flow. Also, the results showed $\pm 10\%$ deviation in outlet flow angle leading to off-design conditions at the fan exit guide vanes. The results indicate a reduction in the effectiveness of the propulsor. It should be noted that all tests were conducted at 65% of the maximum fan speed to avoid the risk of fan stall.

Finally, Perovic et al. [32] conducted an experimental study on a low-speed single stage fan to assess the effects of a BLI-type total pressure distortion on the fan stall. The distortion profile was representative of the intakes used in the CFD analysis of the Cambridge-MIT Institute Silent Aircraft and were produced using a distortion gauze. It was determined that although the fan is continuously subject to distorted inflow, the effect on fan stability was small. Perovic et al. noted that even though there were local regions in which the stability margin was below that of non-distorted inflow,

the fan was able to operate stably. Much like the previous studies, it was found that high incidence angles were present typically at the rotor blade tip which can lead to stall, however, these regions are dampened out as the blades rotate through the high momentum region. Fan stall can occur if the disturbances persist as it rotates, however, this would only occur if a large portion of the annulus was subject to distorted flow.

As shown from the above literature survey, BLI leads to increased fuel consumption due to inlet total pressure losses and total pressure distortion at the engine inlet. However, there seems to be no concerns regarding the stability of the compression system as the studies have found there is little evidence of stall associated with BLI-type distortions and more importantly the distortion is attenuated through the fan. Although, the distortions do present high incidence angles which lead to higher loading on the blades. Therefore, it is crucial to mitigate the impact of BLI on the engine and fan performance to ensure the maximum benefit is achieved through its implementation. The author has not found any references with respect to the fatigue issues related to BLI as well as the trade-off between the reduced thrust requirement and the reduced thrust generated by the engine.

2.2 Review of Intake Optimization Studies

The performance of intakes are crucial to the successful operation of the engines. As such, this section will highlight optimization studies of intakes which help reduce the pressure losses and distortions present at the engine inlet. The section will be broken into two parts; non-BLI intake optimization followed by a review of BLI intake optimization studies.

2.2.1 Non-BLI Intake Optimization

Chiereghin et al. [33] coupled a free form deformation (FFD) method with a multi-objective genetic algorithm to reduce the total pressure losses and flow distortion

(i.e. the swirl angle at the engine inlet) within a geometrically diffusing S-duct. The geometry deformation was parameterized using 18 design variables with the optimization parameters set to population size of 40 and 5 generations. The designs were evaluated using a steady-state Reynolds-Averaged Navier Stokes (RANS) simulation with a Shear Stress Transport (SST) turbulence model. The optimization was able to produce a maximum total pressure reduction of 20% and a maximum swirl reduction of 10%.

D'Ambros et al. [34] performed shape optimization of the same diffusing S-duct as [33]. This study also implemented a FFD technique, however, slightly modified to use a 2D lattice. The geometry deformation was parameterized using 36 design variables and coupled with a multi-objective Tabu search algorithm. This work is an attempt to compare the results from different optimization algorithms and is thus, an extension of the study by Chiereghin et al [33]. The study adopted a steady-state RANS model with a $k-\omega$ turbulence model, evaluating the performance of the S-duct with respect to the total pressure losses and stream-wise vorticity. The optimization algorithm performed 1300 evaluations resulting in a 14% reduction in total pressure losses and a 71% reduction in swirl angle. In comparison to the study in [33], the current optimization algorithm has provided greater reduction in swirl angle, but at the sacrifice of the total pressure losses.

2.2.2 BLI Intake Optimization

In contrast to the studies presented in the previous section, the studies for the optimization of BLI intakes use techniques such as active flow control methods as well as shape optimization. As such, a review of studies which implement either method will be presented in this section.

Harrison et al. [35] analyzed active flow control methods to redistribute the low momentum flow around the annulus of the outlet for a NASA inlet A configuration with 30% BLI at a flight speed of Mach 0.85 and an altitude of 39,000 ft. The DC(60) distortion metric was implemented to evaluate the distortion at the engine inlet.

This study examined a variety of blowing jets and suction hole configurations using CFD with a subset of the configurations being further investigated experimentally. It was determined that suction holes alone were ineffective in reducing the distortion, however, the combination of both suction holes and blowing jets was able to provide a 75% reduction in total pressure distortion. In comparison, the blowing jets alone were only able to improve the distortion by 28% at a mass flow rate equal to at least 1% total inlet mass flow rate.

An investigation of the improvements associated with active flow control on serpentine inlets was carried out by Ferrar et al. [36]. This study experimentally investigated the effects of employing two bleed slots on the distortion at the engine inlet. The bleed slots were located near the entrance of the intake and near the exit as shown in Figure 2.11.

Figure 2.11: Illustration of the bleed slot locations on the S-duct [36]

The intake was tested for boundary layer thickness ranging from 30% to 52% of the inlet height and a free stream Mach number range of 0.2 to 0.43. The bleeds flow control system was connected to a vacuum pump which was used to ensure that no more than 2% of the inlet mass flow was bled from the flow. The experiments using the maximum bleed flow of 2% of the total inlet mass flow produced reductions in the distortion intensity ranging from 10.8% to 23.4%. However, this does effect the stall margin of the compression system and therefore, this would need to be investigated

before bleed slots could be employed.

Koo et al. [37] performed shape optimization of a BLI S-duct intake using CFD and gradient based optimization techniques. The deformation of the duct was carried out using free form deformation to deform the mesh. The optimization was performed for a inlet Mach number of 0.7 with 60% BLI while attempting to minimize the total pressure losses and distortion at the outlet. The objective function was a weighted combination of the total pressure losses and a circumferential distortion metric, resulting in different optimized shapes for each corresponding weight. The results showed that the pressure recovery could be improved by as much as 2% based on the weighting and also showed significant improvement in distortion with an order of magnitude reduction in the distortion coefficient.

The literature survey presented deals exclusively with S-ducts as engines integrated closely with the fuselage or airframe will likely require S-duct intakes to direct the flow into the propulsor. The literature review highlights the significant improvements which can be made to the pressure recovery and distortion at the engine inlet through the use of flow control techniques and optimizing the intake geometry.

2.3 Numerical Optimization

To push the limitations of the engineering design process requires a combination of human ingenuity and the development of mathematical techniques which allows for the exploration of a vast design space. Over the decades, numerical optimization has been introduced into the engineering design and analysis process, allowing for the conceptualization of designs which may have not been possible through human ingenuity alone. The role of optimization, in engineering, is to produce designs with optimal performance under a given set of design constraints.

This section serves as an introduction to the theory of numerical optimization. A general overview of numerical optimization terminology and definitions will be presented followed by a review of the classification of optimization algorithms.

2.3.1 Overview of Terminology and Definitions

In a mathematical context, the goal of numerical optimization is to find a value or a set of values which either minimizes or maximizes a given function. The function given below and represented in Figure 2.12 will be used to illustrate the key elements and terminology of optimization problems.

Using the general notation found in literature [38], the optimization problem is written as follows:

$$\text{Minimize } J(x, y) = 3(1 - x)^2 e^{-(x^2) - (y+1)^2} - 10\left(\frac{x}{5} - x^3 - y^5\right) e^{(-x^2 - y^2)} - \frac{1}{3} e^{(-(x+1)^2 - y^2)}$$

w.r.t x, y

s.t $-3 \leq x, y \leq 3$

Figure 2.12: The design space corresponding to the objective function

Optimization problems are defined based on several salient features including the

objective function, the design variables, and the constraint functions which are defined as follows:

Objective Function: The objective function, also referred to as the fitness function, is the function which is to be minimized or maximized. In the example given above, the objective function is denoted by $J(x, y)$ [38].

Variables: Also referred to later as design variables, are the parameters which can be varied in order to change and/or minimize the objective function, which would be variables x and y in the given example function. The variables can be multi-dimensional and represented by a design variable vector; $\mathbf{X} = x_0, \dots, x_n$ [38].

Design Space: The design space, as shown in Figure 2.12, is the multi-dimensional space occupied by the objective functions and design variables.

Local/Global Minima: In a mathematical context, minima is the singular form of minimum and is the smallest value of a function within a given interval or the entire domain. There are multiple “valleys” located within the design space, as depicted in Figure 2.12; *Local minima* refers to the minimum value within a given region, but it is not the smallest value of the function within the entire domain. The smallest value within the entire domain is referred to as the *global minima* [38].

Constraints: Optimization problems can be classified as either unconstrained or constrained. For constrained optimization, the feasible region in which the optimal point exists is bounded by constraints. In the given example, the limits on the variables x and y are known as *bound constraints* which limit the value that the variables can take and delimit the design space. Constraints can also take the form of linear and non-linear functions which bound the design space as shown in Figure 2.13.

Figure 2.13: Example of linear & non-linear constraints delimiting the feasible design space [Adapted from [39]]

Multi-Objective Optimization: In the given example there is only one objective function, however, many engineering applications require multiple objectives to be optimized simultaneously, this is known as *multi-objective optimization*. For example, in this thesis the intake is optimized with respect to two objectives, minimizing total pressure losses and the total pressure distortion.

Pareto Front: Multi-objective optimization problems generally do not have one optimal solution, but instead a set of solutions which is a compromise between the two objectives. Figure 2.14 illustrates a minimization problem where the optimal solution exists as both objective values decrease. As the solution approaches the optimum it becomes infeasible to improve one objective without compromising the other, thus leading to a family of solutions which form the *Pareto front*.

Figure 2.14: Example of a Pareto front [Adapted from [40]]

2.3.2 Gradient-Based Optimization

There are thousands of optimization techniques which can be employed to solve various problems. However, all methods can be divided into two categories: algorithms which require the gradients (gradient-based) and the ones that do not (gradient-free). This section will review the working principle of gradient-based optimization algorithms including its advantages and disadvantages.

Gradient-based optimization algorithms, much like all other optimization algorithms, are iterative and require an initial guess of the design variables, $\mathbf{X}$, then proceed to generate improved estimates until the optimal values of $\mathbf{X}$ are obtained [41]. For gradient-based optimization, the determination of the improved estimates require both the first and second derivative of the objective function defined by the gradient and the Hessian matrix, respectively. The most commonly used analogy to illustrate gradient-based optimization is to consider hiking down a mountain or hill. The goal is to reach the bottom of the mountain, the minimum, from an initial starting position. Suppose, the path down is not visible and therefore, the hiker must use local information such as the slope (gradient) and curvature (Hessian) to determine

in which direction to move. To reach the bottom as quickly as possible the hiker would have to minimize the number of times required to stop and measure the slope and curvature [42]. For gradient-based optimization, there are two methods that can be employed to improve the current estimate to a new one: Line Search and Trust Region. Both methods require the Hessian matrix, which is generally expensive to evaluate and often not possible to obtain [41]. To circumvent this issue, algorithms implement what is known as the quasi-Newton method, which uses the approximate Hessian calculated based on gradient evaluations of previous iterates. Quasi-Newton methods provide rapid convergence which is not attainable from gradient-free methods with many variables [43]. The gradient and Hessian allow for the calculation of the search direction which is repeated until the solution arrives at the optimum.

Gradient-based algorithms are extremely fast and have no ambiguity regarding its stopping criteria. A disadvantage of the gradient-based methods is the likelihood of converging to a local optima and the dependency upon the initial starting position. This is problematic, since Hicken [43] suggests that the design space for aerodynamic optimization of 3-dimensional flows is likely to be multi-modal, which would increase the likelihood of the optimizer converging to a local optima. However, Hicken also notes that this can be avoided by using multiple different initial starting positions/designs. Another important observation is that gradient-based algorithms have difficulty dealing with a discontinuous design space [44].

2.3.3 Gradient-Free Optimization

The principle challenge arising from optimization problems of complex engineering applications is they often lack algebraic models with an explicit relationship between the design variables and the objective function(s). This necessitates the use of gradient-free optimization packages, as these algorithms do not require the gradient and Hessian information. The objective function values are only obtained as outputs of a “black-box” (black-box optimization), seldom providing derivative information [45]. An example of where black-box optimization is necessary is when a simulation or

experimental test must to conducted in order to evaluate the objective function. Gradient-free algorithms employ various methods to progress the solution towards optimality; many of these methods imitate processes found in nature. The most commonly used gradient-free algorithms are presented in Table 2.1, highlighting the natural processes in which they are derived from.

Table 2.1: Examples of gradient-free optimization algorithms

Optimization Algorithm	Description
Simulated Annealing (SA)	The process of SA is based on the principles of thermodynamics.
Genetic Algorithm (GA)	GAs are analogous to the natural process of biological evolution and employs a survival of the fittest method.
Particle Swarm (PSO)	PSO algorithm is derived from the basic behaviour of swarms of birds.

Also, most gradient-free algorithms can have a stochastic nature to them. Purely stochastic algorithms randomly vary the design variables in hopes of arriving at an optimal solution, however, this is extremely inefficient and unlikely to arrive at the optimum. Gradient-free algorithms sometimes exploit the random perturbations to the design variables to ensure the optimizer does not get “stuck” in a local optimum.

Gradient-free algorithms are able to address the shortcomings of gradient-based methods as they are capable of handling multi-modal design spaces as they are less likely to be trapped in a local optima. Also, gradient-free methods are able to deal with noisy and discontinuous design spaces as well as discrete and mixed discrete-continuous design variables. A major advantage of gradient-free algorithms is their ability to work with black-box optimizations and objective functions which are implicitly related to the design variables thus allowing disparate commercial software to be implemented within the optimization process. However, these advantages are at the expense of increased computational cost and therefore, are limited to working within lower dimensional design spaces [45].

2.4 Geometry Parameterization

The design optimization process is generally comprised of the following steps: the definition of the CAD geometry; mesh generation; evaluation of the performance; update the geometry based on performance; iterate [43]. The integration of numerical optimization into the process begets complications, requiring the coupling of the various software used in each step. Numerical optimization requires a discrete set of variables, making it difficult to rely on CAD-based geometry. This leads to the investigation of alternative approaches for geometry creation or parameterization. One concept that is used for numerical optimization of complex engineering designs is B-spline curves and surfaces, which were first introduced into computer-aided design by Gordon and Riesenfeld [46]. The implementation of B-splines avoids the need to couple commercial CAD software into the optimization process, which can be difficult, if not impossible. The following sections will provide a review of the B-spline concepts such as control points and knot vectors as well as how these are used to define curves and surfaces.

2.4.1 Overview of B-spline Curves

Basis splines, also known as B-splines, are piecewise polynomial functions whose points are defined by an explicit relationship to an independent variable, u [47]. A general p th-degree B-spline curve is the weighted sum of the basis functions and is defined as follows [47],

$$\mathbf{C}(u) = \sum_{i=0}^n N_{i,p}(u) \mathbf{P}_i \quad a \leq u \leq b \quad (2.9)$$

where $\mathbf{C}(u)$ is a vector containing the coordinates of the curve in physical space, n is the number of control points minus one, $\mathbf{P}_i$ is a vector of control points (i.e. weights), and $N_{i,p}(u)$ are the p th-degree B-spline basis functions. The parameter, u , is defined in parametric space on an arbitrary interval $[a,b]$, however, it is generally normalized

to $[0,1]$. Equation 2.9 is used to map the parameter value from parametric space to physical space, $[0,1] \rightarrow \mathbf{R}^3$. In order to define a B-spline curve there are several requirements: the control point locations, the knot locations, and the discretization of the parameter space. The parameter space can be discretized uniformly, however, due to the asymmetry of the basis functions, the resulting points in physical space will not be uniform [43]. Figure 2.15 depicts a cubic polynomial curve consisting of two segments; the full curve $\mathbf{C}(u)$ is the combination of the piecewise polynomials $\mathbf{C}_1(u)$ and $\mathbf{C}_2(u)$ joined together with a specified level of continuity at points called *breakpoints* [47].

The basis functions are piecewise polynomials defined in parametric space and form the basis of the vector space of the required degree and continuity. The basis functions can be defined using the Cox-De Boor recursion formula given the parameter value, u and the knots, t [47].

$$N_{i,0}(u) = \begin{cases} 1 & \text{if } t_i \leq u < t_{i+1} \\ 0 & \text{otherwise} \end{cases}$$

$$N_{i,p} = \frac{u - t_i}{t_{i+p} - t_i} N_{i,p-1}(u) + \frac{t_{i+p+1} - u}{t_{i+p+1} - t_{i+1}} N_{i+1,p-1}(u) \quad (2.10)$$

As shown by the zeroth-degree conditions, the basis functions are non-zero only within the given interval and is therefore why B-splines are piecewise polynomials with local curvature control capabilities. It should be noted, that it is possible for the denominator of either term, on the right hand side, in Equation 2.10 to be zero, in which case the whole term is taken to be zero.

As previously mentioned, the polynomials defining the spline are joined together at knot locations (or breakpoints). The location of the knots are defined in the parametric space and contained within a knot vector, $\mathbf{U}$ which consists of $k + n + 1$ knots, where k is the order of the curve. A knot vector is a non-decreasing sequence of real numbers in the parameter space [47]. Consider the knot vector given in Figure 2.15. The knot vector defines the level of continuity at each knot location. As in the

given example, there is a knot at u equals 0.5 and since the curve is of degree three, the level of continuity at the knot is, C^{3-1} ; the three refers to the degree of the curve while the one refers to the multiplicity (i.e. how many times the knot value is repeated within the knot vector). Also, for all B-splines created in this thesis, the knot vector will be said to be open. An open knot vector ensures the curve interpolates the first and last control points and is said to be discontinuous at the ends as illustrated in Figure 2.15. In order to use an open knot vector the multiplicity of the first and last knot must be equal to the order of the curve.

Figure 2.15: Illustration of the components required to define a B-spline curve(knot vector = $[0,0,0,0,0.5,1,1,1,1]$)

2.4.2 B-Spline Surfaces

A B-spline surface is simply an extension of the definition of the curves in a second parametric direction, v . B-spline surfaces are defined as follows [47],

$$\mathbf{S}(u, v) = \sum_{i=0}^n \sum_{j=0}^m N_{i,p}(u) N_{j,q}(v) \mathbf{P}_{i,j} \quad a \leq u, v \leq b \quad (2.11)$$

Where both parameter values are normalized to $[0,1]$ and similar to B-spline curves, the control points form a control net in which the degree of the polynomial does not have to be the same in both the u- and v-direction. The control points, shown as red dots, define the control net (blue dashed lines) and is used to produce the surface.

Figure 2.16: Example of a surface created using B-spline concepts

2.5 Free Form Deformation (FFD)

FFD was first introduced by Sederberg and Parry [48] as a universal technique to deform solid geometric models. Consider the following analogy given by Sederberg and

Parry to describe FFD. The object to be deformed is embedded within a clear, flexible plastic region and if object itself is presumed to be flexible then as the plastic region is deformed, the embedded object will deform as well. This analogy is illustrated in Figure 2.17 using several objects. The figure on the left shows the undeformed plastic region with several cubes and spheres embedded inside and the figure on the right illustrates how the embedded objects deform with the deformation of the plastic region.

Figure 2.17: FFD: Undeformed & deformed plastic region [48]

The surrounding plastic region is a Bézier volume defined as a product of the trivariate Bernstein polynomial. Therefore, FFD is essentially a mapping from $\mathbf{R}^3$ to $\mathbf{R}^3$. The deformation region is a parallelepiped defined by a set of control points $(\mathbf{P}_{ijk})$ similar to that of B-splines, as depicted in Figure 2.18. A local coordinate system is imposed on the parallelepiped such that any point $\mathbf{X}$ within the volume has coordinates s, t, u in the local system.

Figure 2.18: FFD control points defining the deformation lattice [48]

The white triangles in Figure 2.18 are the control points defining the deformation lattice (Bézier volume) and their locations are given by:

$$\mathbf{P}_{ijk} = \mathbf{X}_0 + \frac{i}{l}\mathbf{S} + \frac{j}{m}\mathbf{T} + \frac{k}{n}\mathbf{U} \quad (2.12)$$

Where $l + 1, m + 1, n + 1$ are the number of control points in each respective direction. For example, in Figure 2.18, $l = 1, m = 2$, and $n = 3$. A transformation must be applied to the physical coordinates of the embedded object to determine its local coordinates. The relationship between the physical coordinate $\mathbf{X}$ and the local coordinate system is given in Equation 2.13.

$$\mathbf{X} = \mathbf{X}_0 + s\mathbf{S} + t\mathbf{T} + u\mathbf{U} \quad 0 \leq s, t, u \leq 1 \quad (2.13)$$

The deformation process is accomplished by the movement of the control points, $\mathbf{P}_{ijk}$. Then the new deformed coordinates of the object, $\mathbf{X}_{ffd}$, are determined through the application of the trivariate Bernstein polynomials on the local coordinates.

$$\mathbf{X}_{ffd} = \sum_{i=0}^l \binom{l}{i} (1-s)^{l-i} s^i \left[\sum_{j=0}^m \binom{m}{j} (1-t)^{m-j} t^j \left[\sum_{k=0}^n \binom{n}{k} (1-u)^{n-k} u^k \mathbf{P}_{ijk} \right] \right] \quad (2.14)$$

The $\mathbf{X}_{ffd}$ vector contains the cartesian coordinates of the deformed object and $\mathbf{P}_{ijk}$ is a vector containing the coordinates of the control points.

FFD is a versatile technique allowing for deformation of arbitrary geometry definitions, large shape changes and focuses on parameterizing the perturbations rather than the geometry itself [43]. For these reasons, FFD has been favored in design optimization studies and has been successfully applied in several research papers [34, 37, 39]. Although, FFD is favored for global, large-scale shape changes, it does present problems with local deformations [43]. Bézier functions are interpolating functions and therefore the movement of one control point affects the whole curve, making it almost impossible to perform local shape changes.

Chapter 3

Methodology

This chapter discusses the implementation of the intermediate calculations and procedures necessary for optimization of the intakes. It will start with a detailed overview of geometry creation leading to a review of the setup of the optimization strategy including the deformation process, CFD methods, and objective functions using commercial software.

3.1 Geometry Creation

This thesis investigates the optimization of both a “straight-through” duct and an S-duct, thus this section will outline the procedure used to create both geometries using B-spline surfaces. The initial geometry is of an arbitrary shape with the only constraint being that the outlet cross section must be circular with a diameter of two meters. The ducts have been designed based on the Airbus A320 aircraft since it is one of the most widely used aircraft worldwide; the Airbus A320 aircraft engines have a fan diameter of two meters which is the reason for the outlet diameter constraint. As such, the initial geometry will simply consist of circular cross sections with constant diameter. The intake optimization is carried out for cruise conditions with a zero-degree angle of attack and since the flow and geometry are symmetrical about the vertical plane only half the duct will be used during the optimization process.

3.1.1 Cross Section Definition

The semi-circular cross section of both geometries is defined using the nine control points and knot vector given in Figure 3.1, to produce a curve of degree two (quadratic). The parameter space is discretized uniformly into hundred points which are mapped from the parametric space to the physical space using Equation 2.9.

Figure 3.1: Control polygon used to define the shape of the duct cross-section ($\mathbf{U} = [0, 0, 0, 0.25, 0.25, 0.5, 0.5, 0.75, 0.75, 1, 1, 1]$)

The control points are defined such that each segment of the control polygon (blue lines) are tangent to a segment of the curve. The given knot vector separates the curve into four segments, with the knots located at a quarter, half, and three-quarters along the curve. Each internal knot has a multiplicity of two, therefore, the level of continuity at the knots is C^0 . However, given the current placement of the control points, there is C^1 continuity, but if the curve is deformed it will not be possible to maintain this level of continuity. Finally, the first and last knots have a multiplicity of $p + 1$ and therefore the curve interpolates the first and last control points.

3.1.2 Duct Geometry

To generate the surface a control net must first be established as shown in Figure 3.2. The control net for the initial constant cross-section “straight-through” duct was generated by placing the control points at equidistant along a straight path. Since the geometry is relatively simple and of a constant cross-sectional shape, the number and location of the control points along the length of the duct is inconsequential.

Figure 3.2: Control net for the “straight-through” duct

The generation of the S-duct surface is more complicated. First, the centerline path defining the curvature of the S-duct must be defined as shown in Figure 3.3. The centerline is defined as two arc segments of degree two that are tangent at the midpoint as well as tangent to the horizontal line at the ends and is derived from [49]. A comparison of the two curves is illustrated inset Figure 3.3. It should be noted that the offset is different from the one used to define the S-duct since the centerline formula within [49] only allows for the definition of either the length or the offset. The other feature defining the centerline curvature is the offset, which is defined as the vertical distance between the center of the inlet and the center of the outlet and as shown in Figure 3.3, the offset is set to four meters. This was determined by analysing

the Boeing 727 tri-jet, which incorporates a center engine that was fed through an S-duct. The ratio of the Boeing 727 fan diameter, D , to the offset, H , (D/H) is 0.5 and thus applying this ratio to the Airbus A320 aircraft engines leads to an offset of four meters. A drawing of the Boeing 727 tri-jet can be found in [50].

Figure 3.3: Control point locations for the definition of the S-duct centerline curvature [red points = control points]

Once the centerline definition has been established, it is important to consider the orientation of the cross-section (control points) with respect to the centerline curve. The cross sections must be perpendicular to the local curvature and therefore, it is necessary to determine the slope at each location where the control points are placed. Thus, derivative of the B-spline curve is determined at each location, which is used to project the control points to a plane perpendicular to the local curvature, forming the control net shown in Figure 3.4.

Figure 3.4: Control net defining the S-duct surface

After the control net has been defined the B-spline functions are able to generate the surface points as illustrated in Figure 3.5. These surface points are imported into ICEM CFD, which uses B-spline approximations to generate a surface that interpolates the points. The B-spline surfaces only define the outer surface of the ducts, the walls enclosing the geometry are created in ICEM CFD.

Figure 3.5: Surface points generated for the S-duct & “straight-through” duct using B-splines

3.1.3 Pre- and Post-Domain

The geometry used to perform the CFD analysis contains an upstream (pre-domain) and/or downstream (post-domain) section. The upstream section is present in both ducts, but only the S-duct requires the downstream portion as well. The pre- and post-domain are simply extensions of the duct surface; once the optimizer has deformed the duct surface the domains are created by extruding the inlet and outlet cross sections.

The upstream section is used to ensure the flow is developed at the inlet which includes a well defined boundary layer profile and the required free stream Mach number. The upstream section is also required for convergence of the solution, especially once the ducts has been deformed. Steady-state flows are classified as elliptical in nature and therefore perturbations induced within the flow field can propagate upstream. This can result in difficult or no convergence if there is no upstream section to resolve these disturbances. The cross sectional shape of the pre-domain is the same as that of the duct inlet cross section. As such, once the duct inlet has been deformed, it is extruded two meters to form the pre-domain. The length of the pre-domain was determined through trial and error; it was determined that a two meter length was sufficient to resolve the disturbances upstream without significantly increasing the computational time.

The post-domain for the S-duct serves a similar purpose. The streamline curvature and deformation of the S-duct can result in flow separation, the post-domain ensures the outlet is sufficiently far from the region of separation as not to influence the results and providing numerical stability. Also, the post-domain is helpful in visualizing the flow behaviour as it would interact with the propulsor. Figure 3.6 depicts the geometry with the upstream and downstream sections for S-duct as well as the symmetry plane which are all created using a replay script within ICEM CFD.

Figure 3.6: 3D rendering of the S-duct domain for CFD simulations

3.2 Geometry Deformation

The deformation process for both the “straight-through” duct and S-duct was accomplished using the free form deformation (FFD) technique. The FFD control points are limited to move normal to the duct surface, thereby reducing the number of degrees of freedom to one. This is required due to the limitations of gradient-free algorithms, since the computational time increases significantly with increasing number of variables and therefore limits the number of design variables which can be used to control the deformation of the duct. Figure 3.7 shows the “straight-through” duct embedded inside the FFD lattice defined by the blue control points.

Figure 3.7: “Straight-through” duct embedded inside a FFD volume

The FFD volume is defined using six control points in the z -direction, three control points in the y -direction, and two control points x -direction. These control points serve as the design variables and are restricted to stay within $\pm 20\%$ of their original location and are fixed in the z -direction. This provides sufficient deformation control to the optimizer while ensuring the design space is not exceedingly large. In addition, the control points on the symmetry plane are fixed (i.e. the control points with coordinates $[0,0,z]$). Also, the set of control points at the outlet of the duct are fixed to maintain a constant diameter. Hence there are five sections consisting of five control points which control the deformation of the duct (i.e. 25 design variables); the length is the 26th design variable. The six planes defining the FFD are equidistant and established after the length has been deformed. The deformation procedure can be performed on either the surface points or the B-spline control points which ultimately define the surface points. The advantage of applying the deformation procedure to the B-spline control points rather than the surface points is the ability to maintain the analytical definition of the duct and is therefore the technique implemented here.

The FFD technique is also applied to the deformation of the S-duct. However, because of the offset between the inlet and outlet of the duct and the parallelepiped

nature of FFD volumes, this would require a sufficiently large FFD lattice with some control points providing minimal deformation and therefore resulting in some cross-sections with a greater geometrical variation capability than others as depicted in Figure 3.8; the control points highlighted by the green circles are sufficiently far from the geometry and thus would not provide much deformation.

Figure 3.8: 2D representation of the hypothetical FFD volume for a S-duct

Also, in order to maintain consistent deformation capability and to ensure deformation normal to the surface of the duct, the FFD embeds and deforms the “straight-through” duct then reorients the deformed cross-sections to form the S-duct. In other words, the “straight-through” duct is deformed and then morphed into the S-duct. Thus, the deformation procedure is the same as for the “straight-through” duct.

3.3 Optimization Algorithm

MATLAB's robust multi-objective genetic algorithm (GA), *gamultiobj* is used in this thesis. This algorithm is able to provide a Pareto optimal set of solutions using a variant of the non-dominating sorting genetic algorithm (NSGA-II), which is an elitist GA that favors individuals with a better fitness value (i.e. lower objective function value). However, as a variant of NSGA-II, *gamultiobj* also favors individuals whom can maintain or improve the population diversity even if their fitness value is lower [51]. Figure 3.9 shows the general workflow of GAs.

Figure 3.9: General GA flowchart [Adapted from [52]]

Genetic algorithms (GAs), also referred to as Evolutionary Algorithms (EA), are analogous to the natural process of biological evolution and was first developed by John Holland [53]. GAs start with an initial population, generally created randomly within the bound and linear constraints of the problem, which attributes to its stochastic nature. These algorithms start with a initial set of designs rather than

a single starting position or design point. During the generation of the initial or subsequent populations, there is no guarantee that the non-linear constraints will be satisfied [51]. The random generation of the initial population is a key element to ensure a diverse set of individuals containing an adequate sample of the design space. However, it is also possible for the user to input an initial population or part thereof. The algorithm will evaluate the performance (i.e. the objective function value) of each individual within the population. Then through a series of processes known as selection, crossover, and mutation advances the solution creating individuals in subsequent populations. Each population is called a generation with the initial population commonly being referred to as generation zero. As the algorithm progresses, the individuals within each population become more elite while some non-elite individuals are kept to maintain the diversity [51].

One of the key elements of genetic algorithms is the number of generations. The initial population is generation zero and each new subsequent population is a new generation. The idea being that as the number of generations progresses, the algorithm should converge to a solution. The maximum number of generations is specified by the user and is a stopping criterion for the algorithm. Intuitively, a high number of generations would guarantee a sufficient number of iterations to converge to a solution, however, this is at the expense of drastically increasing the computational time and with insignificant improvements to the solution. Another key element of GAs is the population size. The population size is important to ensure a diverse set of individuals within each population. If the population size is too small, there will be an inadequate representation of the design space, however, if the population size is large, the computational time will increase dramatically. A sensitivity analysis is conducted in order to determine the sensitivity of the optimized solutions to the population size and maximum number of generations, which can be found in [Appendix A](#).

3.3.1 *gamultiobj* Process

With the general overview and the key elements of GAs defined, the following will describe the iterative process of the *gamultiobj* algorithm as it progresses towards optimality.

Once the initial population has been created, the fitness of each individual is evaluated. To progress to subsequent generations, the algorithm has to go through the process of “evolving” the current population to produce a new generation. There are three methods the algorithm employs to populate the next generation. First, a certain fraction of the elite individuals within the current population is automatically passed onto the next generation, this fraction can be changed by the user, but the default is the top 35% which is used in this thesis. The remaining two methods includes the crossover and mutation functions [51].

The crossover function is analogous to the reproduction stage, where two individuals (parents) from the current population are combined to produce a new individual in the next generation (child). In *gamultiobj*, the parents are selected using a binary tournament in which the rank of each individual within the population is determined. The rank determines the Pareto-optimal and the non-dominated individuals. As depicted in Figure 3.10, rank 1 individuals are not dominated by any other individuals and are the ones located on the Pareto front [54].

Figure 3.10: Example of how individuals are ranked within a population [Adapted from [54]]

Once the individuals on the Pareto front have been ranked, they are temporarily removed and a new resultant Pareto front is found based on the remaining individuals. The individuals within the new Pareto front are ranked 2 and the process continues until all individuals in the given population have been ranked. Individuals with a lower rank (e.g. rank 1 is considered a lower rank than rank 2) a more likely to be chosen as parents for the next generation. Each individual within the same rank is also further ranked based on the *crowding distance*. The crowding distance is a measure of the closeness of the neighboring individuals. During the tournament, four individuals are randomly selected with the two individuals with the highest rank being selected as parents. Individuals with a lower rank and higher distance have a higher chance of being selected as this helps progress the solution towards optimality while maintaining population diversity. Once all individuals in the population have been through the selection tournament process, the crossover function can combine the parents to create the child for the subsequent population [51].

gamultiobj uses the intermediate crossover function given in Equation 3.1 to produce the child [51].

$$child = parent_1 + random \times ratio \times (parent_2 - parent_1) \quad (3.1)$$

The children are created as a weighted average of the parents. The weight can be user specified by adjusting the *ratio* parameter, which can be a scalar or a vector. By default, the value of *ratio* is a vector of all ones. The *random* parameter in Equation 3.1 is a randomly generated value between 0 and 1 which dictates how the parent variables are combined. This parameter gives an inherent randomness to the crossover function, but as the algorithm progresses towards optimality the difference between the parent variables will be insignificant thereby mitigating the effect of the random nature. The fraction of the population created using the crossover function is by default 0.8, but can be adjusted by the user. The remaining individuals of the population are created using the mutation function [51].

The final process is the mutation function which ensures a diverse population and a thorough exploration of the design space. The mutation rules apply random changes to one or more of an individual's variables to form a child. When the optimization problem consists of bound and linear constraints, *gamultiobj* uses an adaptive mutation function which randomly generates the mutation direction and magnitude. The direction and magnitude are adapted until the bound and linear constraints are satisfied [51].

With the above processes completed the algorithm is able to produce the subsequent population (i.e. new generation), for which these same steps will be applied until the stopping criteria is reached. The *gamultiobj* algorithm is set to terminate when the number of generations has reached 50 based on the sensitivity analysis which can be found in Appendix A.

3.4 Intake Optimization

A numerical optimization strategy was developed to optimize two BLI intake geometries, a “straight-through” duct and a S-duct. The length and cross sectional shape are allowed to vary during the optimization while the offset and exit diameter are fixed. For the S-duct optimization, the centerline curvature is initially fixed, but

a sensitivity analysis was conducted. An overview of the optimization problem is represented in Figure 3.11 and Table 3.1 using a generic S-duct.

Figure 3.11: Variables and terminology defining the S-duct geometry [Green: varied during optimization; Red: fixed]

Table 3.1: Optimization parameters

Variable	Range
Centerline Curvature	fixed
D_{exit} (Airbus A320 fan diameter)	2 m (fixed)
Offset	fixed
Control Points (cross-sectional shape)	$\pm 20\%$ from original location
Length	$3\text{ m} \leq L \leq 8\text{ m}$ (integer values only)

MATLAB was used to couple the various software used in each step of the optimization process with the multi-objective genetic algorithm. Figure 3.12 depicts the implementation of the optimization strategy architecture which is fully automated.

Figure 3.12: Optimization strategy architecture

The optimization loop can be viewed as consisting of two major modules; the optimization algorithm and evaluation sequences. The optimization algorithm provides the design variable vector which determines the shape of duct and progresses the design towards optimality. The evaluation sequences use the design variable vector to generate the geometry and perform the intermediate calculations necessary to evaluate its performance. The previous sections have discussed the optimization algorithm and the deformation process, therefore, the following sections will delve into the meshing procedure, CFD setup, and the objective functions implemented to evaluate the performance of the designs.

3.4.1 Computational Mesh

ICEM CFD was used to generate a structured mesh for both ducts as shown in Figure 3.13 and 3.14. The grid consisted of a H-O topology to help capture the curvature of the ducts and the detail within the boundary layer. Since computational time is a major factor in the optimization study, a coarse-medium mesh is generated for each design. The mesh for the “straight-through” duct consisted of approximately 300,000 elements and a first layer height of $2.5 \times 10^{-5} \text{ m}$, in comparison to the S-

duct mesh which was generated with approximately 370,000 elements and a first layer height of $2 \times 10^{-5} \text{ m}$. The slightly large mesh size for the S-duct is due to the incorporation of the downstream section. Both meshes resulted in a y^+ value of around 10-20 depending on the flow field within the duct. A mesh convergence analysis was performed to ensure that the grid provided adequate representation of the flow field and can be found in Appendix B. Although, it should be noted that the grid convergence study is performed with respect to an undeformed duct and as such, it would be difficult to fully resolve regions of flow separation, supersonic flow, and shock waves. However, the presence of shock waves would result in pressure losses that would make the duct undesirable and thus, would be filtered out through the optimization process. Therefore, the resolution of shock waves is not of interest in the present study. In terms of flow separation and supersonic flow conditions, the mesh is adequate to capture the presence of these flow features, although, the optimized designs should be computed on a finer computational mesh to fully resolve these features further.

Figure 3.13: Structured mesh of the “straight-through” duct ($L = 6 \text{ m}$)

Figure 3.14: Structured mesh of the S-duct geometry ($L = 6$ m)

The mesh was generated for each design using a replay file called from MATLAB. The replay file is equivalent to a macro in which the series of steps taken to generate the initial mesh is recorded and then “played” back to generate the mesh for subsequent designs. The characteristics of the computational mesh were consistent across all designs since if the number of cells change between iterations the objective function will become discontinuous. This is because the mesh defines the computational space which ultimately determines the flow solution produced by the CFD analysis and thus, the objective function values. Therefore, changes to the computational space can result in objective function value changes due to mesh characteristics rather than the design changes [44].

3.4.2 CFD Methods

The performance of each design is evaluated using the steady-state Reynolds-Averaged Navier Stokes (RANS) model of the commercial CFD software, ANSYS CFX. The “straight-through” ducts are simulated at the approximate cruise altitude and Mach number of the Airbus A320 aircraft, 37,000 feet and 0.8, respectively. However, the

free stream Mach number was reduced to 0.7 for the simulation of the S-duct since it became difficult to maintain a Mach number of 0.8 as the low viscosity of air at the cruise altitude led to pockets of supersonic flow and it was conjectured the tendency for the flow to reach supersonic conditions at this altitude limited the deformation. Finally, the shear stress transport (SST) turbulence model is used in this optimization study as it provides an excellent balance between near-wall treatment of the flow and the shear-free layer. Also, this turbulence model has been found to provide accurate predictions of the onset of flow separation, which is especially crucial for this optimization study [55].

In order to simulate BLI, the turbulent boundary layer thickness formula, Equation 3.2, is applied to the flow over a flat plate with a length equal to that of the Airbus A320 aircraft. This calculation results in a boundary layer thickness of 0.24 meters, however, since the formula assumes a smooth surface and does not account for boundary layer transition, the thickness was increased to 0.3 meters or 15% of the initial inlet height. A 0.3 meter thickness is consistent with the value found in literature [56], which states that the boundary layer thickness is a foot or more at the rear of the aircraft. Changing the boundary layer thickness from 0.24 meters to 0.3 meters may have a impact on the optimal shapes, however, it is assumed the general cross sectional deformation trend will remain the same and only the degree to which the cross section is deformed may be impacted.

$$\delta = \frac{0.37x}{Re_x^{1/5}} \quad (3.2)$$

Then the flow over a two-dimensional flat plate is solved to determine the total pressure distribution of a turbulent boundary layer and scaled to give the required boundary layer thickness. This total pressure distribution representing the boundary layer, as shown in Figure 3.15 is applied as the inlet boundary condition to simulate BLI. In addition to the total pressure profile, the associated turbulent kinetic energy and turbulent eddy frequency profiles, found in Appendix D, are also applied to ensure proper representation of a turbulent boundary layer. Also, in order to solve

the energy equation, the total temperature at the inlet is set to the temperature at the cruise altitude of the Airbus A320 aircraft.

Figure 3.15: Inlet total pressure profile for a Mach number of 0.8 at 37,000 ft

Initially, the outlet is set to a static pressure equal to that of the atmospheric pressure at the cruise altitude. However, due to the deformation of the inlet, it becomes necessary to iteratively set the outlet static pressure to maintain the required inlet free stream Mach number. Although, this would increase the computational time substantially and therefore, a relationship between the inlet area and outlet static pressure was developed through trial and error. It is important to note that changing the outlet static pressure does alter the altitude in which the flow is simulated. An alternative method would be to alter the total pressure profile at the inlet, but it was found to yield diverging solutions on some occasions or non-convergence. Figures C.1 and C.2, in Appendix C, show the trend determined to maintain a Mach number of 0.8 and 0.7 for the “straight-through” duct and the S-duct, respectively. In other words, once the optimizer has deformed the inlet, the outlet static pressure is determined using the trend line equation found in Figures C.1 and C.2 and applied as the outlet static pressure. Although, the actual inlet Mach numbers varied between 0.78 - 0.83

and 0.67 - 0.72 for the respective free stream Mach number. The limitation with this method is that change in the outlet static pressure may influence the optimal design, as the optimizer may attempt to determine a design which increases the outlet static pressure thus increasing the total pressure at the duct outlet. This leads to the optimizer attempting to optimize the outlet static pressure. A summary of the boundary conditions can be found in Table 3.2.

Table 3.2: Boundary conditions used for the simulations

Boundary	Boundary Condition	Description
Inlet (Pre-domain)	Total Pressure Profile	Total pressure profile with a boundary layer thickness of 0.3 meters and a free stream Mach number of 0.8
Wall (Upstream, Downstream, & Duct)	No-slip	Smooth wall with viscous dissipation
Outlet	Static Pressure	Outlet static pressure varies with changing inlet area to maintain free stream Mach number
Symmetry	Symmetry	The flow field is symmetrical about the ZY plane

The CFD simulations for both ducts are set to run for a maximum of 200 iterations with a residual value of 10^{-4} and a system imbalance of $\leq 1\%$. A high resolution scheme is used for spatial discretization, which is a 2^{nd} -order upwind biased-scheme. These parameters ensure an adequate convergence of the simulation for the optimization study. It should be noted that due to the increase in computational time for the S-duct geometries, especially when flow separation occurs, the timescale was set to four.

3.4.3 Objective Functions

The optimization algorithm needs the values of the objective functions to assess the performance of each design and progress towards optimality. As previously discussed, the two major factors affecting the performance of engines is the loss in inlet pressure and the total pressure distortion at the engine inlet. Therefore, the performance metrics for the intakes assess the total pressure losses within the duct and the total pressure distortion at the duct outlet.

Equation 3.3 is a common performance metric, known as the pressure recovery (PR), used to determine the total pressure losses present in a duct [34, 39, 37].

$$PR = \frac{P_{t,avg,outlet}}{P_{t,avg,inlet}} \quad (3.3)$$

Maximizing Equation 3.3 is equivalent to minimizing the total pressure losses. However, the optimization algorithm attempts to minimize its objective functions and therefore, the reciprocal of the PR equation is implemented. The total pressure at the inlet and outlet is calculated as an area-averaged value to account for the non-uniformity of the mesh.

The second objective is to minimize the total pressure distortion at the engine inlet or in other words, the objective is to obtain a uniform total pressure profile. The DC(60) distortion coefficient is commonly found in literature as a metric to determine the uniformity at the engine inlet [57].

$$DC(60) = \frac{P_{t,avg,outlet} - P_{t,60^\circ,min}}{q} \quad (3.4)$$

Minimizing Equation 3.4 is equivalent to reducing the distortion at the outlet. The term ‘q’ is the dynamic pressure and $P_{t,60^\circ,min}$ term refers to the 60° sector with the worst area-averaged total pressure. Figure 3.16 shows how the DC(60) distortion coefficient is calculated.

Figure 3.16: Illustration of the DC(60) distortion coefficient calculation procedure; $\alpha = 60^\circ$

During the post-processing stage, the static pressure, total pressure, and element area are extracted from the duct outlet plane. Starting at the 6-o'clock position, the total pressure in the 60° sector is calculated then the sector is shifted by 5° , as shown by the red dashed lines in Figure 3.16, and the total pressure for the new sector is calculated. This continues until the sector has traversed the outlet, then the lowest calculated area-averaged total pressure is used to calculate the distortion coefficient.

3.4.4 Alternate Distortion Coefficient

In numerical optimization, the objective functions define the design space and thus have a great deal of influence on the optimal solutions. In particular for intake optimization, the PR objective function is consistent throughout literature, however, the distortion metric tends to vary. Therefore, in order to explore the influence the distortion objective function has on the optimal design an alternate distortion metric is implemented. This distortion metric was developed by Koo et al. [37] and is derived from the sum of squared error statistical formula as shown by Equation 3.5.

$$D = \sum_j^m \sum_i^n \frac{(\frac{P_{t,i} - P_{t,avg,j}}{P_{t,avg,j}})^2 dA_i}{A_j} \quad (3.5)$$

This equation is a circumferential distortion metric and thus measures the circumferential variation in total pressure at the duct outlet. The outlet is divided into $j = 1, 2, \dots, m$ rings with each ring being composed of n elements as shown in Figure 3.17.

Figure 3.17: Alternate distortion coefficient calculation procedure

The distortion coefficient measures the area-averaged sum of squared error within each ring by comparing the total pressure within each element to the average pressure within each ring [37]. The difference in pressure between the element and ring is normalized with respect to average ring pressure which is an additional term added to the formula provided in [37] as the CFD software used in that work, Jetstream, is non-dimensional.

Chapter 4

Results and Discussion

This chapter will present the results for the optimization of both the “straight-through” duct and the S-duct determined using the methods described in Chapter 3. However, only a detailed analysis of the optimal distortion and pressure recovery (PR) ducts will be presented. The optimization of the “straight-through” duct was performed on a workstation using 16-cores with 128-gigabytes (GB) of RAM and took approximately seven days to complete. The S-duct optimization was performed on a different workstation using 16 cores with 64 GB of RAM, however, it was able to carry out the process within four days. This change in computational speed between the two workstations is likely due to the different processors.

In the following sections, the duct outlet (also referred to as the engine inlet; Figure 3.6) total pressure distribution is presented as a dimensionless value. The outlet total pressure calculated at each node has been normalized with respect to the area-averaged total pressure at the outlet. This is in order to allow for comparisons between ducts with different outlet static pressures.

4.1 “Straight-Through” Duct Optimization

The “straight-through” duct analysis carried out in this section is based on the results obtained from the optimization using a population size of 50 and 50 generations,

as the best PR and distortion were obtained using these parameter values. The evaluation history and Pareto fronts established for the varying number of generations and population size can be found in Appendix A.

The optimization of the “straight-through” duct is performed at a free stream Mach number of 0.8 and a boundary layer thickness of 0.3 meters. The outlet static pressure is dependent on the inlet area and thus calculated based on the trend shown in Figure C.1. In order to progress the designs towards optimality, the optimizer is free to vary the length and control points as indicated in Table 3.1. The final results from the optimization study are shown in Figure 4.1, which performed over 1700 simulations leading to the Pareto front highlighted by the red dots. The thumbnails shown in Figure 4.1 illustrate the normalized total pressure distributions.

Figure 4.1: Evaluation history of the “straight-through” duct optimization (Including thumbnails of the normalized total pressure distortion at the engine inlet for a subset of the Pareto front)[$3 \leq m \leq 8$]

Since the length of the duct is considered a design variable it is difficult to provide comparisons between an undeformed baseline geometry and the optimized geometry because the distortion (Equation 3.4) and PR (Equation 3.3) varies between ducts of different length. However, for the sake of comparison an undeformed duct of five meters is plotted in the above figure. The distortion present at the engine inlet is

shown for a subset of the Pareto front in Figure 4.1. The performance of the Pareto optimal designs are summarized in Table 4.1.

Table 4.1: Comparison of the baseline performance with the Pareto front designs

Optimized Geometry	PR	% Improvement	DC(60)	% Improvement
Design I: Reference (L = 5 m)	0.991	-	0.0312	-
Design II: Optimal PR (L = 4 m)	0.998	+ 0.70	0.029	+ 7.05
Design III: Optimal Dist. (L = 5 m)	0.994	+ 0.30	0.0135	+ 56.7
Design IV (L = 5 m)	0.992	+ 0.10	0.0140	+ 55.1
Design V (L = 6 m)	0.990	- 0.10	0.0141	+ 54.5
Design VI (L = 8 m)	0.988	- 0.30	0.0139	+ 55.4

As evident from Table 4.1, the designs on the Pareto front have substantially better distortion performance compared to the reference geometry. The PR values are relatively high, which can be explained by the low viscosity characteristics of air at the cruise altitude. Although the differences in pressure loss are small compared to the distortion, these small PR losses equate to different fuel consumptions. This is shown in [27] as Hardin et al. stated that a 0.35% increase in pressure losses was responsible for a 1.1% increase in fuel burn. Also, as observed from Table 4.1, the DC(60) distortion coefficient improvement jumps significantly between Design II and Design III while staying relatively constant between the other designs. This improvement is a result of the markedly different cross-sectional shapes as seen from Figures 4.3 and 4.5. As such, the best compromise is actually the optimal distortion duct (Design III), which offers improved PR and significant reductions in distortion. To assess how the performance of the optimal designs have been achieved, the following will provide an analysis of the optimal distortion and PR ducts. Figures 4.2 and 4.4 illustrate the comparison of the symmetry plane Mach number contours and the normalized total pressure distribution at the engine inlet of the geometries with the best distortion coefficient and PR, respectively.

The low total pressure region at the engine inlet, of the optimal distortion duct, was reduced in size as shown in Figure 4.2, especially towards the lower sides of

the distortion. The symmetry plane Mach number contours illustrate how the duct has been deformed resulting in the modification of the flow through the duct. This modification of the flow causes the inertial forces to dominate the viscous dissipation and thus reduces the size of the distortion. The optimizer has determined that a duct length of five meters is optimal and it is hypothesised that the optimizer is attempting to balance between the losses due to viscous dissipation (i.e. increasing the length) and providing enough length to reduce the size of the distortion as much as possible.

Figure 4.2: A comparison of the symmetry plane Mach contours and the total pressure distribution at the engine inlet [Top = optimal distortion ($L = 5$ m); Bottom = reference duct of equal length ($L = 5$ m)]

In order to assess how the deformed duct was able to reduce the distortion at the engine inlet, Figure 4.3 shows the comparison of the cross-sectional shapes between the optimal distortion duct and an undeformed duct of equal length. The optimizer has reduced the area at the bottom of the inlet cross section and widened the area at the top. It seems that this serves two purposes, first, the reduced area at the bottom means there is less BLI since the total pressure profile is interpolated onto the inlet mesh and second, since the bottom is “pulled down” it helps to accelerate

the flow as it progresses towards the outlet. Also, the widened area at the bottom of the following cross section serve a similar purpose. Since, the area is wider than that of the outlet, it accelerates the boundary layer flow and since the lack of uniformity is one due to reduced momentum, this increases the dynamic pressure at the engine inlet resulting in improved distortion.

Figure 4.3: Comparison of the cross sectional shapes of the optimal distortion duct and an undeformed duct of equal length

In contrast to the optimal distortion duct, the optimal PR duct (thumbnail II in Figure 4.1) has a slower velocity throughout portions of duct, but a faster exit velocity. Although the exit velocity is greater than compared to the optimal distortion duct, the acceleration occurs at the outlet, thus having minimal impact on the pressure losses within the duct. The optimizer has found the optimal length of the duct to be four meters compared to the five meters for the optimal distortion duct. The reduced velocity in combination with the decreased length are responsible for the lower frictional losses within the duct and thus, the increased PR. Since the minimum length set during the optimization is three meters, it is intuitive to assume that the shortest length would provide the best PR and although the optimizer was able to

produce improved PR by shortening the length to three meters, the optimal length is four meters. This is likely because the optimizer is attempting to balance between both distortion and PR, thus a length of four meters provides the best compromise.

Figure 4.4: A comparison of the symmetry plane Mach contours and the total pressure distribution at the engine inlet [Top = optimal PR ($L = 4$ m); Bottom = reference duct of equal length ($L = 4$ m)]

To assess how the optimal PR duct was able to reduce the pressure losses, Figure 4.5 shows the comparison of the cross sectional shapes of the optimal PR duct with that of an undeformed duct of equal length. The cross sections of the optimal PR duct does not present significant deformation due to the limits set by the area constraints (Appendix E). However, the optimizer has widened the cross sections, especially at the bottom of the cross section, decreasing the flow velocity and thus the losses due to friction and providing the improvements on the pressure recovery. The combination of the fixed outlet and the widened intermediate cross sections is the reason the flow accelerates at the outlet.

Figure 4.5: Comparison of the cross sectional shapes of the optimal PR duct and an undeformed duct

4.2 S-Duct Optimization

A discussion of the results from the optimization of the S-duct are presented here. The population size and the maximum number of generations used in the optimization of the “straight-through” duct is also implemented for the S-duct.

The optimization of the S-duct is carried out using the boundary conditions given in Table 4.2, with the optimizer free to vary the length of the duct as well as the control points as shown in Figure 3.11 and Table 3.1.

Table 4.2: Boundary conditions for CFD simulations

Variable	Range
Free stream Mach number, M_{inlet}	0.7
Boundary layer thickness, δ	0.3 meters
Outlet static pressure	Based on Figure C.2

The final results of the optimization study are shown in Figure 4.6 with a subset

of the Pareto front highlighted by the red dots and the thumbnails illustrating the normalized total pressure distributions at the engine inlet. It should be noted that the reason the evaluations shown in Figure 4.6 seem to appear in columns is likely due to the lack of precision of the pressure recovery values when they are extracted during the post processing step.

Figure 4.6: Evaluation history and Pareto front for the S-duct optimization

Analyzing the optimization history presented in Figure 4.6 indicates the presence of a different Pareto front when compared against the “straight-through” duct. It is interesting to notice that the subset of the Pareto highlighted by the red dots all have a length of eight meters. This is expected for a duct with minimal distortion, since the longer length reduces the adverse pressure gradient experienced by the flow and thus, reduces the growth of the boundary layer and possibility of separation. However, it is conjectured that for the optimal PR duct the optimizer is attempting to balance between the acceleration of the flow over an aggressive secondary bend of the outer curvature and the increased viscous dissipation due to the lengthening of the duct as well as avoiding flow separation. The Pareto optimal solutions all provided relatively significant improvements in PR and distortion as presented in Table 4.3. The optimal PR duct provides a 0.61% increase in PR compared to an undeformed

duct of equivalent length which would equate to significant decrease in thrust specific fuel consumption (TSFC) according to [27]. Also, it should be noted that the difference in PR values is with respect to a duct of eight meters and thus, the optimal PR duct would substantially outperform shorter ducts. On the other hand, the optimal distortion duct was able to improve the distortion metric by 27.3%, significantly reducing the overall pressure variation of the low total pressure region with respect to the outlet total pressure. However, based on the results in Table 4.3, Design IV seems to provide the best compromise between PR and distortion, especially since the PR has a greater impact on the fuel consumption than the distortion.

Table 4.3: Comparison of the Pareto optimal designs

Optimized Geometry	PR	% Improvement	DC(60)	% Improvement
Design I: Reference (L = 8 m)	0.9824	-	0.0759	-
Design II: Optimal Distortion (L = 8 m)	0.9866	+ 0.43	0.0552	+ 27.3
Design III (L = 8 m)	0.9868	+ 0.45	0.0554	+ 27.0
Design IV (L = 8 m)	0.9874	+ 0.51	0.0566	+ 25.4
Design V (L = 8 m)	0.9880	+ 0.57	0.0574	+ 24.4
Design VI: Optimal PR (L = 8 m)	0.9883	+ 0.60	0.0588	+ 22.5
Design VII: dominated solution (L = 6 m)	0.9803	-0.21	0.16	-111

Figure 4.7 illustrates the comparison of the flow field and symmetry plane curvature between the optimal distortion duct, optimal PR duct, and an undeformed of equal length. As depicted by the total pressure distribution, the structure of the low total pressure region is similar, but the area-averaged total pressure is greater for the optimal distortion duct. Thus raising the average total pressure in the sector and providing reduced distortion at the engine inlet. As shown by the symmetry plane curvature in Figure 4.8, the second bend in the outer curve has a reduced radius of curvature. This reduces the acceleration over the bend and shifts the region of maximum velocity towards the outlet. It seems that given the distortion metric implemented, the optimizer is attempting to reduce the distortion by increasing the dynamic pressure of the flow at the engine inlet. This is consistent with the description given in [35] that the DC(60) metric is “roughly interpreted as a measure of

the flow kinetic energy” and therefore, a higher kinetic energy would mean a lower distortion. In terms of the effect on the propulsor, the higher kinetic energy of the low total pressure region is beneficial as this would reduce the susceptibility to stall of the compression system.

The flow field within the optimal PR duct is similar to that of the optimal distortion duct as illustrated in Figure 4.7. As shown by the Mach number contours the region of maximum velocity is also shifted towards the engine inlet which reduces the losses present within the flow field due to viscous dissipation, increasing the pressure recovery. Also, the velocity through the duct is generally slower than compared to the undeformed duct which reduces the wall shear stress and thus, the losses due to friction. The structure of the low total pressure region is similar to that of the optimal distortion duct as well with a slightly greater height. It is interesting to note the almost identical symmetry plane curvature of both optimized ducts indicating a high degree of coupling between the two objectives. It seems the difference between the optimal distortion and PR duct is a result of the slight separation seen in the optimal PR duct which is attributed to the inlet symmetry plane curvature being “pulled down” as opposed to optimal distortion duct where it is “pulled up”. This separation reduces the average total pressure within the low total pressure region and thus increasing the distortion.

Figure 4.7: Comparison of the symmetry plane Mach number contours and normalized total pressure distribution at the engine inlet [Top = optimal distortion (II), Middle = optimal PR (VI), Bottom = reference duct of equal length (I)]

Figure 4.8: Comparison of the symmetry plane curvature of the optimal distortion duct, optimal PR duct, and a reference duct of equal length

A comparison of the cross sections of the optimal distortion duct and the undeformed duct is presented in Figure 4.9 where the upper left figure is a magnified depiction of the cross sections close to the inlet. As shown, the optimizer has widened the cross sections, particularly at the bottom. This is similar to the cross sectional changes of the “straight-through” duct; due to the fixed outlet, the widening of the cross sections results in the acceleration of the flow. The change in cross sectional shape is not as aggressive as it was for the “straight-through” duct as it is believed that this would likely lead to flow separation and/or pockets of supersonic flow which deteriorate the performance of the duct. It is also conjectured that majority of the benefit is a result of lengthening the duct as this reduces the adverse pressure gradient acting on the flow rather than the change in cross sectional shape. Also, it should be noted that towards the inlet the bottom of the cross section is “pulled up”, it seems that this modification has led to a increased acceleration over the bend resulting in a higher momentum within the boundary layer and reduced susceptibility to separation. As shown in Figure 4.8, the inner curvature of the optimal distortion duct and

optimal PR duct as almost identical except towards the inlet, which is responsible for the slight separation seen in the optimal PR duct as shown in Figure 4.7.

Figure 4.9: Comparison of the cross sections of the optimal distortion duct and an undeformed duct

Figure 4.10 shows the comparison of the cross sections of the optimal PR duct and an undeformed duct of equal length. As shown, the optimizer has widened the cross sections which has led to the reduced velocity throughout the duct and thus, the increased PR. The bottom of the inlet cross section is “pulled down”, this creates the narrow cross sectional area similar to the “straight-through” optimal distortion duct. The narrow area at the bottom allows for the expansion of the flow in the following sections of the duct resulting in a static pressure rise within the boundary layer flow. This “pulled down” modification is the reason for the slight separation shown in Figure 4.7 for the optimal PR duct as it reduces the momentum of the boundary layer flow over the bend. Hence, if the expansion in area is too sharp the flow will separate and thus, it seems the optimizer has attempted to balance between the pressure rise due to the expansion and the possibility of flow separation. In other words, since the expansion precedes the first bend, if the expansion slows down the flow velocity significantly the adverse pressure gradient will likely cause the flow to

separate.

Figure 4.10: Comparison of the cross sections of the optimal PR duct and an undeformed duct

It is interesting to consider one of the dominated solution shown in Figure 4.6 such as Design VII and how its design differs from the optimal PR and distortion ducts. The symmetry plane Mach number contours and the normalized total pressure distribution at the engine inlet are shown in Figure 4.11. The length of the duct is only six meters compared to the eight meter length of the Pareto optimal designs. The reduced pressure recovery is a result of the flow separation which occurs around the inflection point of the duct centerline curvature and towards the exit marked by the A-A section line. In contrast to the optimal distortion and PR ducts, the region of maximum acceleration occurs over the second bend of the outer curvature leading to the flow separation and increased viscous dissipation. This is likely the reason the optimizer has shifted the region of maximum velocity to the outlet for the optimized designs. Analyzing the normalized total pressure at the engine inlet (A-A view on the right), the separation has led to a region of low total pressure at the top. Also, the aggressive curvature of the duct increases the centrifugal force acting on the flow, which results in the high momentum flow “replacing” the low momentum

flow at the bottom by shift the region towards the center of the duct. This leads to a lower area-averaged total pressure and the increased distortion present at the engine inlet. This highlights the reason why the optimizer has increased the length for the optimal distortion and PR ducts as well as shifting the region of maximum velocity towards the outlet (A-A section), as these modifications help avoid flow separation and increase the dynamic pressure at the outlet.

Figure 4.11: Symmetry plane Mach number contours and normalized total pressure distribution at the engine inlet of the dominated solution (VII; $L = 6$ m)

4.2.1 Sensitivity of the Optimal Shape to the Length Limits

The Pareto optimal solutions shown in Table 4.3 all have a length of eight meters, the maximum allowable length set during the optimization. Therefore, the sensitivity of the optimal shape to the limits set on the length are explored in this section. The limits have been increased to vary between eight and twelve meters as shown in Table 4.4. Also, the boundary conditions are the same as those used in the original optimization as shown in Table 4.2.¹

¹“original optimization/optimal” refers to the optimization of the S-duct using the original length limits given in Table 3.1 and presented in Section 4.2. “Current optimization/optimal” will refer to the results presented in this section (increased length limits as shown in Table 4.4).

Table 4.4: New optimization parameters

Variable	Range
Centerline Curvature	fixed
D_{exit} (Airbus A320 fan diameter)	2 m (fixed)
Offset	fixed
Control Points (cross-sectional shape)	$\pm 20\%$ from original location
Length	$8\text{ m} \leq L \leq 12\text{ m}$ (integer values only)

The optimization was carried out using the new length limits which produced the corresponding evaluation history and Pareto front as shown in Figure 4.12. For a subset of the Pareto front the normalized total pressure distribution at the engine inlet is depicted in the thumbnails.

Figure 4.12: Evaluation history of the S-duct optimization with increased length limits (thumbnails show the normalized total pressure distribution at the engine inlet)

Also, the performance of the optimal PR duct (orange triangle) and the optimal distortion duct (black square) from the original optimization of the S-duct are also plotted in Figure 4.12 for reference. As illustrated, the new Pareto front is only marginally improved with respect those previous optimum values. However, the ducts on the new Pareto front all have varying lengths with the length increasing for the

ducts with better distortion. The subset of the Pareto optimal solutions are summarized in Table 4.5 and although the designs have varying lengths a comparison with respect to an undeformed duct of eight meters is also provided for reference. The optimal PR ducts (original and current) have almost identical PR values, but the current optimum design has a lower distortion coefficient due to the increased length. In contrast, the current optimal distortion duct has improved by 9.1% relative to the original design.

Table 4.5: Comparison of the Pareto optimal designs

Optimized Geometry	PR	% Improvement	DC(60)	% Improvement
Design I: Reference (L = 8 m)	0.9824	-	0.0759	-
Original Optimization				
Optimal PR (orange triangle; L = 8 m)	0.9883	+ 0.60	0.0588	+ 22.5
Optimal Distortion (black square; L = 8 m)	0.9866	+ 0.43	0.0552	+ 27.3
Current Optimization				
Design II (L = 12 m)	0.9856	+ 0.32	0.0493	+ 35.0
Design III (L = 12 m)	0.9858	+ 0.35	0.0497	+ 34.5
Design IV: Optimal Distortion (L = 12 m)	0.9871	+ 0.48	0.0483	+ 36.4
Design V (L = 10 m)	0.9877	+ 0.54	0.0511	+ 32.7
Design VI: Optimal PR (L = 9 m)	0.9884	+ 0.61	0.0577	+ 24.0

The comparison of the symmetry plane Mach number contours and the normalized total pressure distributions at the engine inlet for the current and original optimized designs are shown in Figure 4.13. The length of the current duct has been increased to twelve meters (i.e. the maximum allowable limit). The length may seem excessive given that it occupies approximately 30% of the overall length of the A320 aircraft. However, based on the length-to-diameter ratio of the Boeing 727 tri-jet ($\frac{L}{D} = 5.6$), this length value is reasonable for the Airbus A320 aircraft, based on its engine diameter ($\frac{L}{D} = \frac{12m}{2m} = 6$). As previously discussed, the optimizer is attempting to reduce the adverse pressure gradient experienced by the flow by lengthening the duct. This reduces the momentum loss of the boundary layer flow and thus, the distortion present at the engine inlet. Similarly to the optimized ducts from the original optimization, the optimizer has shifted the region of maximum velocity to

the outlet. Comparison of the low total pressure region at the engine inlet shows that the longer duct has a higher area-averaged total pressure and a slightly smaller region.

Figure 4.13: Comparison of the symmetry plane Mach number contours and the normalized total pressure distribution (at the engine inlet) of the optimal distortion ducts [Top = current optimal design ($L = 12$ m); Bottom = original optimal design ($L = 8$ m)]

The cross sectional shapes of the current and original optimal distortion ducts are shown in Figure 4.14. The cross sectional deformation of the current duct is similar to that of the original optimization. The intermediate cross sections have been widened in order to accelerate the boundary layer flow as it progresses towards the outlet, reducing the distortion. The flow velocity at the outlet is less than the original optimized duct indicating that the reduction in adverse pressure gradient is more significant than the increase in dynamic pressure. The slight difference occurs towards the inlet, where the bottom of the cross section has been “pulled down”. It

is hypothesised the optimizer deforms the inlet by narrowing the bottom of the cross section to reduce the amount of BLI. This was not done in the original optimization with a lower length limit, presumably because the following expansion in area would retard the flow acceleration too greatly, which in conjunction with the adverse pressure gradient would lead to flow separation.

Figure 4.14: Comparison of the cross sectional shapes of the current and original optimal distortion ducts [Left = current optimal design ($L = 12$ m); Right = original optimal design ($L = 8$ m)]

Next, a comparison of the Mach number contours and the normalized total pressure distribution of the current and original optimal PR ducts is shown in Figure 4.15. As shown, the length of the original optimal PR duct is eight meters (maximum allowable) compared to a length of nine meters for the current optimal PR duct. The structure of the low total pressure region in both optimal ducts is almost identical with the current optimal PR duct having a slightly smaller width. The increase in length has reduced the flow acceleration produced by the turning of the flow which reduces the wall shear stresses and thus, the pressure losses. However, this has provided minimal improvement in the PR, as seen in Table 4.5 and could be due to numerical inconsistencies rather than from the performance of the duct. Thus, it seems that the reduced losses due to the slower velocity is balanced by the increased viscous dissipation associated with the lengthening of the duct. However, the increased length has aided in reducing the distortion present at the engine inlet

as shown in Table 4.5. Thus, the current design is a better compromise between the PR and distortion objectives and therefore an overall better design.

Figure 4.15: Comparison of the symmetry plane Mach number contours and the normalized outlet total pressure distribution of the optimal PR ducts [Top = current optimal design ($L = 9$ m); Bottom = original optimal design ($L = 8$ m)]

Analyzing the cross sectional deformation of the current and original optimal PR ducts shown in Figure 4.16, the deformation is consistent between both ducts. The optimizer has increased the PR by widening the cross sectional area and reducing the flow velocity. However, it seems that with the lengthening of the duct, it has allowed for a slightly more aggressive deformation since the reduction in adverse pressure gradient means the boundary layer is less likely to separate.

Figure 4.16: Comparison of the cross sections of the current and previous optimal PR duct with an undeformed duct [Left = current optimal design ($L = 9$ m); Right = original optimal design ($L = 8$ m)]

Therefore, it is evident that the limits on the length had a more significant impact on the distortion than the PR. This is because the lengthening of the duct reduces the adverse pressure gradient, which has a more significant impact on the distortion coefficient than the losses due to increased viscous dissipation has on the PR. This is due to the low viscosity characteristics of air at high altitudes leading to reduced frictional losses.

4.2.2 Effects of the Centerline Curvature

The S-duct centerline is defined as shown in Figure 3.3 in Chapter 3 using five control points. The centerline curvature plays an important role in the flow field within the S-duct and thus, the effects of the centerline curvature are investigated here. The centerline deformation is performed by moving the control points shown in Figure 3.3. The control points, CP2 and CP4, are limited to one degree of freedom allowing movement only in the z -direction while CP3 has two degrees of freedom with movement capabilities in both the y - and z -directions. The control points are set to move $\pm 50\%$ from their original location as it was determined that greater freedom resulted in poor mesh quality and non-convergence of the simulations. The cross sections and

length were held fixed while the centerline curvature was deformed as shown in Table 4.6. The deformation of the centerline is applied to the optimal distortion duct from the previous section (Section 4.2.1; Design IV), since this duct provided a good balance between PR and the distortion present at the engine inlet.

Table 4.6: New optimization parameters for centerline deformation

Variable	Range
Centerline Curvature	variable (control point variation: $\pm 50\%$)
D_{exit} (Airbus A320 fan diameter)	2 m (fixed)
Offset	fixed
Control Points (cross-sectional shape)	fixed
Length	fixed (12 m)

The optimization of the centerline produced the evaluation history and Pareto fronts shown in Figure 4.17. The thumbnails illustrate the normalized total pressure distribution at the engine inlet.

Figure 4.17: Evaluation history of the centerline deformation optimization with illustrations of the normalized total pressure distribution for a subset of the Pareto optimal designs

As depicted, there are two Pareto fronts developing. The first Pareto front has sub-

stantially better PR while the second Pareto front favors the distortion as shown in Table 4.7. The thumbnails illustrate the two distinctive outlet total pressure distributions between the two fronts; in the first Pareto front the distortion is at the 6 o'clock position while for the second Pareto front the distortion is shifted towards the center of the outlet. The optimized designs showed significant improvements in both objective functions and Table 4.7 highlights a subset of the Pareto front as shown in Figure 4.17.

Table 4.7: Comparison of the Pareto optimal designs of the centerline deformation study

Optimized Geometry	PR	% Improvement	DC(60)	% Improvement
Design I: Reference	0.9858	-	0.0483	-
Pareto Front 1				
Design II: Optimal PR	0.9916	+ 0.59	0.0624	- 29.2
Design III	0.9888	+ 0.30	0.0463	+ 4.1
Design IV	0.9870	+ 0.12	0.0398	+ 17.6
Design V	0.9854	- 0.04	0.0371	+ 23.2
Pareto Front 2				
Design VI	0.9735	- 1.25	0.0359	+ 25.7
Design VII	0.9709	- 1.51	0.0264	+ 45.3
Design VIII: Optimal Distortion	0.9643	- 2.18	0.0108	+ 77.6

The Mach number contours and outlet total pressure distribution for the optimal PR and distortion ducts are shown in Figure 4.18. As shown, the two ducts have markedly different centerline curvatures which lead to the increased performance metrics.

Figure 4.18: Comparison of the symmetry plane Mach number contours and the normalized total pressure distribution at the engine inlet [Top = Opt. PR (Design II); Middle = Opt. Dist. (Design VIII); Bottom = original (Design I)]

The optimal distortion duct has an extremely aggressive centerline curvature almost resulting in a 90° bend. The aggressive curvature leads to flow separation along the inner curvature of the duct and a pocket of supersonic flow over the outer curvature. The inflection point of the centerline curvature is shifted upstream leading to an almost straight duct section prior to the outlet. The separation in conjunction with these changes in the duct curvature produce a pressure differential which produce counter-rotating vortices, known as Dean vortices, that shift the low total pressure region towards the center of the outlet. This results in a more uniform circumferential total pressure profile which is distributed more evenly across all the sectors of the

outlet and thus, giving an improved distortion at the engine inlet. However, the flow separation and the pocket of supersonic flow both lead to increased loss in PR.

The optimal PR recovery duct does not feature an s-shaped centerline curvature as seen from Figure 4.18. The centerline curvature is representative of a elongated arc; closer to a straight path from the inlet to the outlet rather than a s-shaped curve. The deformation of the convoluted s-shaped centerline curvature to an elongated arc eliminates the acceleration experienced by the flow. Thus, the flow velocity within the duct is relatively constant leading to reduced pressure losses. However, the sharp change of the outer curvature at the outlet preceding the downstream section of the duct results in flow separation and supersonic flow conditions. Since these occur at the outlet and downstream section of the duct, it has minimal impact on the performance of the duct. However, this would negatively impact the propulsor.

The majority of the Pareto optimal solution contain supersonic flow conditions except for Design V, in which the centerline curvature is not as aggressive as the optimal PR and distortion ducts as shown in Figure 4.19. As illustrated, the first half of the duct centerline is almost linear with minimal curvature. This sharp change in flow direction from the inlet results in an increased acceleration of the flow and thus, increasing the boundary layer momentum. The increased momentum within the boundary layer means it is less affected by the adverse pressure gradient and is less susceptible to separation. The resultant centerline deformation has led to increased velocity at the outlet as well, increasing the dynamic pressure and reducing the size of the low total pressure region.

Figure 4.19: The flow solution of the duct with the best trade-off between PR and distortion (Design V in Figure 4.17) including a comparison of the centerline curvatures

The centerline curvature is integral to the performance of the duct as shown by the preceding analysis. The performance of the initially optimized duct was improved significantly through centerline deformation alone. Therefore, it would be interesting to assess the optimization of the duct by exploring the full design space with the deformation of the cross sections and the centerline. However, this optimization is not covered in this thesis, but it is presumed that it would provide significant improvements to the overall performance of the ducts.

4.2.3 Effects of the Distortion Coefficient on the Optimal Shape

The DC(60) distortion coefficient seems to attempt to reduce the distortion by increasing the dynamic pressure at the outlet. However, it seems a more circumferentially distributed low total pressure region should provide low distortion and improved performance as well. Thus, the DC(60) coefficient is replaced with a new circumferential distortion metric that can be found in [37] and the calculation is shown in Section 3.4.3. This distortion metric allows the low total pressure region to redistribute itself around the circumference of the outlet to provide improved uniform total pressure

distribution. Therefore, this section will investigate how a change in distortion metric affects the optimal shape of the duct.

The optimization parameters and ranges are the same as those listed in Table 4.4. The ducts are optimized for a free stream Mach number of 0.7 and a boundary layer thickness of 0.3 meters with the outlet static pressure determined using the trend shown in Figure C.2. This led to the following evaluation history and Pareto front as illustrated in Figure 4.20.

Figure 4.20: Evaluation history and Pareto front for the new distortion coefficient

As illustrated by the thumbnails, there is a distinct difference between the outlet total pressure distribution of the ducts with high PR and low distortion. The low total pressure region of the ducts with low distortion tends to “spread out” circumferentially along the bottom surface to provide a more uniform distribution leading to reduced distortion at the engine inlet. The performance of a subset of the Pareto optimal designs are presented in Table 4.8 with their corresponding DC(60) coefficient as well.

Table 4.8: Comparison of the Pareto optimal designs for the alternate distortion coefficient

Optimized Geometry	PR	D	DC(60)
Optimal Distortion _{DC(60)} (L = 12 m)	0.9858	0.0035	0.0483
Optimal PR _{DC(60)} (L = 9 m)	0.9884	0.0057	0.0577
Design I: Optimal PR (L = 10 m)	0.9884	0.0064	0.102
Design II (L = 11 m)	0.9876	0.0045	0.0659
Design III (L = 12 m)	0.9865	0.0038	0.0540
Design IV (L = 12 m)	0.9853	0.0029	0.0470
Design V: Optimal Distortion (L = 12 m)	0.9833	0.0025	0.0429

The differences between the optimal distortion and PR ducts obtained using the new distortion metric are shown in Figure 4.21 and 4.23. Figure 4.21 illustrates how the total pressure distortion at the outlet is “flattened” and smeared circumferentially around the bottom of the outlet. As shown by the Mach number contours, the maximum velocity occurs over the second bend of the outer curvature for the current optimized duct as compared to the DC(60) optimized duct where it is shifted towards the outlet; the current design has a higher velocity at the mid-section of the duct and a gradual acceleration as the flow progresses towards the outlet which contributes to the “flattening” of the low total pressure region. As shown in Table 4.8, it is interesting to note how the current optimized design has a lower DC(60) value compared to the original optimized design but also a lower PR. This is a consequence of the flow acceleration which evidently has increased the dynamic pressure at the outlet as well as redistributing the distortion and the increased velocity has resulted in the reduced PR. Analyzing Figure 4.12, the PR of the current optimal distortion duct falls within the region which has not been fully explored by the optimizer. However, it is difficult to assess if an increase in iterations would be sufficient to discover the current design using the DC(60) metric since the change in distortion coefficient also changes the design space.

Figure 4.21: Mach number contours (symmetry plane) and normalized total pressure distribution (engine inlet; A-A) of optimal distortion ducts [Top = new distortion coefficient (D); Bottom = DC(60)]

The comparison of cross sectional curvature of the optimal distortion ducts with respect to the undeformed cross sections is presented in Figure 4.22. The length of the current duct has been extended to the maximum allowable limit of 12 meters just as for the DC(60) distortion metric. In contrast to the DC(60) duct, the cross sections following the inflection point have a narrower area at the bottom. This allows the low total pressure region to expand, redistributing itself circumferentially along the bottom of the outlet. It also aids in the acceleration of the flow through the second half of the duct leading to the reduced height of the low total pressure region. Also, towards the inlet the bottom of the cross section have been “pulled up” which in combination with the subsequent cross section being “pulled down” provides a higher acceleration over the first inner bend; this increases the momentum of the boundary layer flow and therefore is less affected by the adverse pressure gradient.

Figure 4.22: Comparison of the cross sectional curvature of the optimal distortion ducts [Left = new distortion coefficient (D); Right = DC(60)]

The current optimal PR duct has a length of ten meters compared to the nine meter length of the duct obtained from the DC(60) optimization, however, both ducts have the same PR value of 98.8%. The Mach number contours shown in Figure 4.23 illustrate how the lengthening of the duct has reduced the maximum velocity within the duct and thus, the pressure losses. However, it seems that this is balanced by the increased viscous dissipation throughout the duct resulting in the identical PR. As such, it is interesting that the current design was not found during the optimization using the DC(60) distortion metric. This is most likely explained by comparing the DC(60) values of both duct found in Table 4.8. The DC(60) value of the current design is approximately 43% worse than the original optimized duct since the low total pressure region has shifted up towards the center leading to a decreased area-averaged total pressure. It is conjectured that since the DC(60) value of the current design is far outside the Pareto front of the initial optimization it is unlikely to be found by the optimizer without a substantial amount of iterations.

Figure 4.23: Mach number contours (symmetry plane) and normalized total pressure distribution (engine inlet) of optimal PR ducts [Top = new distortion coefficient (D); Bottom = DC(60)]

It is interesting to see how the change in the distortion coefficient has also changed the optimal PR duct shape. The PR values are the same for both ducts, but they have different cross sectional shapes and length. The inlet cross section of the current optimal PR duct is similar to that of the DC(60) optimal distortion duct as shown in Figure 4.24. The inlet is shaped as an upside-down water droplet resulting in a narrow area at the bottom; the subsequent widening of the area leads to flow diffusion. However, the current design has a much more aggressive change in inlet shape as compared to the DC(60) duct. The inlet area is less than the intermediate cross section leading to a diffusing S-duct and thus reduced pressure losses.

Figure 4.24: Comparison of the cross sectional curvature of the optimal PR ducts [Left = DC(60); Right = new distortion coefficient (D)]

It is evident from the preceding analysis that the objective functions have a significant impact on the optimized shape of the ducts. Although the two alternate objective functions both measured the distortion at the outlet, they each led to different optimized designs. This is expected since the objective functions are crucial to defining the design space and thus, the relationship between the two objectives will significantly alter the optimal designs. After analyzing the method in which both distortion coefficients attempt to minimize the total pressure distortion at the duct outlet, it seems the alternate distortion coefficient (D) is the better performance metric. This is because the alternate distortion attempts to redistribute the low total pressure region circumferentially around the annulus of the duct outlet providing a more circumferentially uniform total pressure distribution which may be more beneficial to the fan since this reduces the height of the distortion.

Chapter 5

Conclusions and Recommendations

A fully autonomous optimization strategy has been developed using commercially available software and applied to the design of BLI intakes, for a “straight-through” duct as well as a S-duct. The objective of the optimization was to maximize the pressure recovery (PR) of the intakes as well as minimizing the total pressure distortion present at the engine inlet. In this work, the geometry was created using B-spline surfaces and the shape optimization of the intakes was performed using a free form deformation technique coupled with a multi-objective genetic algorithm. The performance of the intakes was evaluated using ANSYS CFX’s steady-state reynolds averaged navier-stokes (RANS) equations with the shear stress transport (SST) turbulence model.

The optimization of the “straight-through” duct was able to provide significant improvements in both PR and distortion. The reduction in distortion was accomplished by accelerating the flow through the duct, reducing the size of the low total pressure region and increasing the dynamic pressure at the outlet, resulting in a 56.7% decrease in distortion. In contrast, the PR was increased by widening the cross sections as well as shortening the duct length, thus reducing the velocity within the duct and the surface area resulting in reduced pressure losses and a 0.7% increase in PR.

The S-duct optimization was able to produce designs with improved PR and distortion. Similar to the “straight-through” duct, the reduction in distortion was

accomplished by accelerating the flow and shifting the region of maximum velocity towards the duct outlet, increasing the dynamic pressure at the engine inlet. This is beneficial for the propulsor as a higher inlet velocity would reduce the likelihood of flow separation over the blades and thus the susceptibility to stall. Also, the duct was extended to the maximum allowable length in order to reduce the adverse pressure gradient thus, reducing the deceleration of the boundary layer flow and the susceptibility to separation. This was also confirmed by increasing the length limits which led to the optimizer again increasing the length to the maximum allowable limit. The PR was increased by shortening the duct as much as possible before causing the flow to separate and widening the cross sections to reduce the velocity within the duct as well as shifting the maximum velocity region to the outlet.

The sensitivity of the optimal designs to the centerline curvature was also investigated, which indicated a significant coupling between the centerline and the duct performance, as expected. This study revealed that for increased PR, the centerline is no longer “S-shaped” and is representative of an elongated arc, almost producing a straight path from the inlet to the outlet. This was done in order to reduce the flow acceleration through the duct, which reduces the pressure losses due to friction. In contrast, the optimal distortion duct had an aggressive S-shape with the inflection point moved upstream to allow the counter-rotating vortices to shift the low total pressure region towards the center of the outlet, providing a more uniform total pressure distribution. Finally, the effects of distortion objective function on the optimal designs were investigated by using an alternate distortion coefficient. This distortion coefficient was able to improve distortion by distributing the low total pressure region around the bottom surface of the outlet. Interestingly, the change in distortion coefficient also affects the shape of the optimal PR duct. The cross sectional shapes were similar, but were more aggressive in the deformation.

5.1 Recommendations

There are various paths that can further the research and refine the optimized designs presented in this thesis. The following considerations may benefit the future researchers who wish to continue this work:

1. To build on this research, it would be beneficial to consider exploring the optimal designs throughout the full flight envelope including takeoff, landing, climb, and in the presence of strong crosswinds.
2. In addition, it would be interesting to see how the optimal shape changes for different S-duct offsets since there is no established BLI aircraft, this research can help designers in their development of the airframe.
3. Also, a different inlet shape, such as that shown in Figure [2.4](#), would be worth exploring to assess how the initial inlet shape effects the optimal duct shape.
4. Finally, in terms of optimization, using a gradient-based optimizer coupled with a gradient-free optimizer can help utilize the benefits of both types of algorithms.

References

- [1] —. Iata forecast predicts 8.2 billion air travelers in 2037. <https://www.iata.org/en/pressroom/pr/2018-10-24-02/> [Accessed on 3-11-2020].
- [2] —. Air transport, passengers carried. <https://data.worldbank.org/indicator/IS.AIR.PSGR> [Accessed on 3-11-2020]. International Civil Aviation Organization, Civil Aviation Statistics of the World and ICAO staff estimates.
- [3] S. W. Ashcraft, A. S. Padron, K. A. Pascioni, G. W. Stout Jr, and D. L. Huff. “Review of Propulsion Technologies for N+3 Subsonic Vehicle Concepts”. Technical report, NASA/TM-2011-217239, 2011.
- [4] A. Plas. “*Performance of a Boundary Layer Ingesting Propulsion System*”. Master’s thesis, Massachusetts Institute of Technology, 2006.
- [5] E. Van Dam. “*Inlet Distortion Characterization of the Boundary Layer Ingesting D8 Aircraft*”. Master’s thesis, Delft University of Technology, Netherlands, 2015.
- [6] A. M. Ferrar. “*Measurements of Flow in Boundary Layer Ingesting Serpentine Inlets*”. Master’s thesis, Virginia Polytechnic Institute and State University, 2011.
- [7] A.M.O. Smith and Howard Roberts. “The Jet Airplane Utilizing Boundary Layer Air for Propulsion”. *Journal of Aeronautical Sciences*, pages 97–128, 1947.
- [8] L. H. Smith Jr. “Wake Ingestion Propulsion Benefit”. *Journal of Propulsion and Power*, 1993.

- [9] J. R. Lucas. “*Effect of BLI-Type Inlet Distortion on Turbofan Engine Performance*”. Master’s thesis, Virginia Polytechnic Institute, Virginia, 2013.
- [10] J. C. Gladin, J. S. Sands, B. K. Kestner, and D. N. Mavris. “Effects of Boundary Layer Ingesting (BLI) Propulsion Systems on Engine Cycle Selection and HWB Vehicle Sizing”. In *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, Nashville, Tennessee, 2012. AIAA 2012-0837.
- [11] D. K. Hall. “*Analysis of Civil Aircraft Propulsors with Boundary Layer Ingestion*”. PhD thesis, Massachusetts Institute of Technology, 2015.
- [12] A. M. Ferrar and W. F. O’Brien. “Progress in Boundary Layer Ingesting Embedded Engine Research”. In *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Atlanta, Georgia, 2012. American Institute of Aeronautics and Astronautics.
- [13] M. Drela. “Power Balance in Aerodynamic Flows”. *AIAA Journal*, 47:1761–1770, 2009.
- [14] S. Sato. “*The Power Balance Methods For Aerodynamic Performance Assessment*”. PhD thesis, Massachusetts Institute of Technology, June 2012.
- [15] A. Uranga, M. Drela, E. M. Greitzer, D. K. Hall, N. A. Titchener, M. K. Lieu, N. M. Siu, C. Casses, A. C. Huang, G. M. Gatlin, and J. A. Hannon. “Boundary Layer Ingestion Benefit of the D8 Transport Aircraft”. *AIAA Journal*, 55:3693–3708, 2017.
- [16] S. Pandya, A. Uranga, and A. Huang. “Computational Assessment of the Boundary Layer Ingesting Nacelle Design of the D8 Aircraft”. In *52nd Aerospace Sciences Meeting*, National Harbor, Maryland, 2014. AIAA SciTech.
- [17] A. Uranga, M. Drela, E. M. Greitzer, N. A. Titchener, M. K. Lieu, N. M. Siu, A. C. Huang, G. M. Gatlin, J. A. Hannon. “Preliminary Experimental Assessment

- of the Boundary Layer Ingestion Benefit for the D8 Aircraft”. In *52nd Aerospace Sciences Meeting*, National Harbor, Maryland, 2014. AIAA SciTech.
- [18] A. Uranga, M. Drela, E. M. Greitzer, D. K. Hall. “Analysis of the Aerodynamic Benefit from Boundary Layer Ingestion for Transport Aircraft”. *AIAA Journal*, 56:4271–4281, 2018.
 - [19] B. T. Blumenthal, M. D. Maughmer, S. Schmitz. “Computational Investigation of a Boundary-Layer-Ingestion Propulsion System”. *Journal of Aircraft*, 55:1141–1153, 2018.
 - [20] J. S. Gray. “*Design Optimization of a Boundary Layer Ingestion Propulsor Using a Coupled Aeropropulsive Model*”. PhD thesis, University of Michigan, 2018.
 - [21] J. R. Welstead and J. L. Feldery. “Conceptual Design of a Single-Aisle Turboelectric Commercial Transport with Fuselage Boundary Layer Ingestion”. In *54th AIAA Aerospace Sciences Meeting*, San Diego, California, 2016. AIAA SciTech Forum.
 - [22] J. P. Longley and E. M. Greitzer. “inlet distortion effects in aircraft propulsion system integration”. In *AGARD, Steady and Transient Performance Prediction of Gas Turbine Engines*, 1992.
 - [23] J. C. Gladin. “*A Sizing and Vehicle Matching Methodology for Boundary Layer Ingesting Propulsion Systems*”. PhD thesis, Georgia Institute of Technology, 2015.
 - [24] E. J. Gunn and C. A. Hall. “Aerodynamics of Boundary Layer Ingesting Fans”. In *ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*, Düsseldorf, Germany, 2014.
 - [25] S. Boller and W. F. O’Brien. “One-Dimensional Dynamic Wake Response in an Isolated Rotor Due to Inlet Pressure Distortion”. In *35th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Los Angeles, California, 1999. AIAA.

- [26] J. F. Ryman. “*Predication of Inlet Distortion Transfer Through the Blade Rows in a Transonic Axial Compressor*”. Master’s thesis, Virginia Polytechnic Institute and State University, June 2003.
- [27] L. W. Hardin, G. Tillman, O. P. Sharma, J. Berton, and D. J. Arend. “Aircraft System Study of Boundary Layer Ingesting Propulsion”. In *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Atlanta, Georgia, 2012. AIAA.
- [28] H. Kim and J. Felder. “Control Volume Analysis of Boundary Layer Ingesting Propulsion Systems With or Without Shock Wave Ahead of the Inlet”. In *49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, Orlando, Florida, 2011. AIAA 2011-222.
- [29] R. T. Kawai, D. M. Friedman, and L. Serrano. “Blended Wing Body (BWB) Boundary Layer Ingestion (BLI) Inlet Configuration and System Studies”. Technical report, NASA/CR-2006-214534, 2006.
- [30] J. Giuliani, J. Chen, T. Beach, and M. Bakhle. “Numerical Simulation of Boundary Layer Ingesting (BLI) Inlet/Fan Interaction”. In *50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference*, Cleveland, Ohio, 2014. AIAA Propulsion and Energy Forum.
- [31] D. J. Frohnepfel, W. F. O’Brien, and K. T. Lowe. “Fan Response to Inlet Swirl Distortions Produced by Boundary Layer Ingesting Aircraft Configurations”. In *51st AIAA/SAE/ASEE Joint Propulsion Conference*, Orlando, Florida, 2015. AIAA Propulsion and Energy Forum.
- [32] D. Perovic, C. Hall, and E. Gunn. “Stall Inception in a Boundary Layer Ingesting Fan”. *Journal of Turbomachinery-Transactions of the ASME*, 141, 9, 2019.
- [33] N. Chiereghin, L. Guglielmi, M. Savill, T. Kipouros, E. Manca, A. Rigobello, M. Barison, and E. Benini. “Shape Optimization of a Curved Duct with Free

- Form Deformations”. In *23rd AIAA Computational Fluid Dynamics Conference*, Denver, Colorado, 2017.
- [34] A. D’Ambros, T. Kipouros, P. Zachos, M. Savill, E. Benini. “Computational Design Optimization for S-ducts”. *Designs*, 2,36, 2018.
- [35] N. A. Harrison, J. Anderson, J. L. Fleming, and W. F. Ng. “Active Flow Control of a Boundary Layer-Ingesting Serpentine Inlet Diffuser”. *Journal of Aircraft*, 50, 1:pp. 262 – 271, 2013.
- [36] A. M. Ferrar, W. F. O’Brien, W. F. Ng, R. Florea, and D. J. Arend. “Active Control of Flow in Serpentine Inlets for Blended Wing-Body Aircraft”. In *45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Denver, Colorado, 2009.
- [37] D. Koo, D. W. Zingg, W. A. Chishty, H. M. Abo El Ella. “Optimizing Intakes for Embedded Engines”. In *ISABE-2019-24290*, Canberra, Australia, 2019. International Society of Air Breathing Engines.
- [38] S. Boyd and L. Vandenberghe. “*Convex Optimization*”. Cambridge University Press, 2004.
- [39] A. Rigobello. “*A Multi-objective shape optimization of an S-Duct intake through NSGA-II genetic algorithm*”. Master’s thesis, University of Padova, December 2016.
- [40] MathWorks. Pareto front for two objectives. <https://in.mathworks.com/help/gads/pareto-front-for-two-objectives.html> [Accessed on 28-07-2020].
- [41] J. Nocedal and S. Wright. “*Numerical Optimization*”. Springer, 2nd edition, 2006.
- [42] Gaurav Jain. “*A beautiful Analogy: Understanding gradient descent algorithm for solving linear regression problems*”, 2018 (ac-

- cessed May 20, 2020). <https://www.linkedin.com/pulse/beautiful-analogy-understanding-gradient-descent-algorithm-jain/>.
- [43] J. E. Hicken. “*Efficient Algorithms for Future Aircraft Design: Contributions to Aerodynamic Shape Optimization*”. PhD thesis, University of Toronto, 2009.
 - [44] Jason Hicken. Personal Communication.
 - [45] J. Larson, M. Menickelly, and S. M. Wild. “Derivative-Free Optimization Methods”. *Acta Numerica*, 28:287–404, 2019.
 - [46] W. Gordon and R. Riesenfeld. “B-Spline Curves and Surfaces”. *Computer Aided Geometric Design*, pages 95–126, 1974.
 - [47] L. Piegl and W. Tiller. “*The NURBS book*”. Springer, 2nd edition, 1995.
 - [48] T. Sederberg and S. Parry. “Free-form deformation of solid geometric models”. In *ACM Siggraph Computer Graphics*, volume 20, pages 151–160, 1986.
 - [49] S. R. Wellborn, B. A. Reichert, and T. H. Okiishi. “An Experimental Investigation of the Flow in a Diffusing S-Duct”. In *28th Joint Propulsion Conference and Exhibit*, Nashville, TN, 1992.
 - [50] Hans-Jürgen Fischer. 727-30-drawing-fischer-89. <https://www.flickr.com/photos/49681893@N05/40196225151/> [Accessed on 10-10-2020].
 - [51] The MathWorks. “*Global Optimization Toolbox: User’s Guide*”, 2020.
 - [52] M. Miller. “*The Multi-Objective Design of Flatback Wind Turbine Airfoils*”. Master’s thesis, Carleton University, 2016.
 - [53] J. H. Holland. “Adaptation in Natural and Artificial Systems: An introductory analysis with applications to biology, control, and artificial intelligence”. *University of Michigan Press*, 1975.
 - [54] MathWorks. gamultiobj algorithm. <https://www.mathworks.com/help/gads/gamultiobj-algorithm.html> [Accessed on 25-05-2020].

- [55] B. Liu, G. An, and X. Yu. “Assessment of curvature correction and reattachment modification into the shear stress transport model within the subsonic axial compressor simulations”. *Power and Energy*, 229:910–927, 2015.
- [56] J. Banke. Reduce fuel burn with a dose of bli. <https://www.nasa.gov/aero/reduce-fuel-burn-with-a-dose-of-bli> [Accessed on 23-02-2020].
- [57] J. Seddon and E. L. Goldsmith. *“Intake Aerodynamics: An account of the mechanics of flow in and around the air intakes of turbine-engined and ramjet aircraft and missiles”*. Collins Professional and Technical Books, 1985.
- [58] E. W. Conrad and A. E. Sobolewski. “Investigation of Effects of Inlet-Air Velocity Distortion on Performance of Turbojet Engine”. Technical report, NACA RM-E50G11, 1950.

Appendix A

Genetic Algorithm: Sensitivity Analysis

Genetic algorithms are sensitive to the population size and the number of generations, therefore a substantially high value for both parameters would allow sufficient exploration of the design space and produce a well developed Pareto front. However, this comes at the expense of significantly increasing the computational cost. Thus, a sensitivity study was performed, for the “straight-through” duct, by varying the population size and number of generations to determine how the Pareto front was affected as shown in Figures [A.1](#) to [A.5](#).

The number of generations were varied at a fixed population size of 50, starting from 20 generations. The initial population size was chosen based on a literature review of similar optimization studies, the authors of [39] and [33] performed the optimization of a S-duct with 36 and 18 design variables using population sizes of 50 and 40, respectively. It should be noted that [39] and [33] used 9 and 5 generations, respectively. However, since the above references do not have well defined Pareto fronts, the number of generations used in this thesis is significantly increased based on the present sensitivity analysis which shows well established Pareto fronts.

It was found that increasing the number of generations from 40 to 50 provided a negligible increase in the optimal pressure recovery (PR) of 0.12% and only a 7.0%

reduction in the optimal distortion. As such, it was determined that a maximum of 50 generations is adequate and further increases in generations would significantly increase the computational time with negligible improvement in performance.

Next, the sensitivity to the population size is performed by varying the population size with the number of generations fixed at 40. The number of generations was reduced to 40 to see if the population size or number of generations had more impact on the optimization as well as reducing the number of iterations to reduce the computational time. The population size was changed to 60 and 70, which provided a 0.09% and 0.05% increase in the maximum PR, respectively. The reduction in the optimal distortion was 1.4% and 2.8% for the population size of 60 and 70, respectively. Therefore, it seemed that a population size of 50 and 50 generations provided the best combination with respect to computational time and a well defined Pareto front.

Number of Generations:

Figure A.1: Evaluation history and Pareto front using a population size of 50 and a maximum of 20 generations

Figure A.2: Evaluation history and Pareto front using a population size of 50 and a maximum of 40 generations

Figure A.3: Evaluation history and Pareto front using a population size of 50 and a maximum of 50 generations

Population Size:

Figure A.4: Evaluation history and Pareto front using a population size of 60 and a maximum of 40 generations

Figure A.5: Evaluation history and Pareto front using a population size of 70 and a maximum of 40 generations

Appendix B

Grid Convergence Study

A mesh convergence study was performed to ensure an adequate number of elements was used for the simulations. The mesh convergence is determined by analyzing the change in pressure recovery (PR) and distortion, DC(60), for different mesh densities since, these are the two parameters of interest. The convergence analysis for the “straight-through” duct and the S-duct are presented in Figures B.1 and B.2.

Figure B.1: Pressure recovery (PR) and distortion [DC(60)] for the “straight-through” duct (duct length = 6 m)

Figure B.2: Pressure recovery (PR) and distortion [DC(60)] for the S-duct (duct length = 6 m)

As shown in the above figures, the PR value and the DC(60) coefficient for the “straight-through” duct and the S-duct do not vary significantly after 300,000 and 370,000 elements, respectively. However, the slight variation is accompanied by significant increases in computational time as shown in Table B.1 and thus, it was determined that a grid of approximately 300,000 and 370,000 elements for the “straight-through” duct and the S-duct was adequate to assess the performance of the ducts.

Table B.1: CFD simulation times with respect to the mesh density

	”straight-through” duct	S-duct
Mesh #1	55 seconds	58 seconds
Mesh #2	1 minute 5 seconds	1 minute 11 seconds
Mesh #3	1 minute 40 seconds	1 minute 50 seconds
Mesh #4	2 minutes 42 seconds	2 minutes 16 seconds

Appendix C

Outlet Pressure Variation

Figure C.1 and C.2 show the variation in the outlet static pressure with respect to the change in inlet area for the “straight-through” duct and the S-duct, respectively. The trend line equations were implemented within the optimization in order to maintain the respective free stream Mach number.

Figure C.1: Outlet pressure variation with respect to inlet area for “straight-through” duct

Figure C.2: Outlet pressure variation with respect to inlet area for S-duct

Appendix D

Turbulent Profiles

The turbulent kinetic energy and turbulent eddy frequency profiles were also extracted from the 2D flat plate simulation, in addition to the total pressure profile, to ensure proper representation of the turbulent boundary layer profile. Figures D.1 and D.2 show the turbulent kinetic energy and turbulent eddy frequency profiles which are applied to the inlet boundary condition along with the total pressure profile.

Figure D.1: Turbulent kinetic energy profile from 2D flat plate simulation

Figure D.2: Turbulent eddy frequency profile from 2D flat plate simulation

Appendix E

Area Constraint

Although the control points are allowed to vary $\pm 20\%$ from their original location, the ratio of the inlet area to any of the following cross sectional areas (i.e. intermediate cross sections) must be less than or equal to 1.04 ($\frac{A_{inlet}}{A_{intermediate}} \leq 1.04$) for the “straight-through” duct and less than or equal to 1.1 for the S-duct. This is to ensure that the flow does not choke and was determined using a combination of the isentropic area ratio equation and CFD simulations. The “straight-through” duct has a lower ratio due to the higher free stream Mach number and thus, slight changes in area will cause the flow to choke.