Fluid flow studies of the F-5E and F-16 inlet ducts

Imran Halimi Ibrahim

2008

Imran, H. I. (2008). Fluid flow studies of the F-5E and F-16 inlet ducts. Master's thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/14581>

<https://doi.org/10.32657/10356/14581>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**FLUID FLOW STUDIES OF THE
F-5E AND F-16 INLET DUCTS**

IMRAN HALIMI BIN IBRAHIM

2008

**FLUID FLOW STUDIES OF THE F-5E AND F-16
INLET DUCTS**

**IMRAN HALIMI BIN IBRAHIM
SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING
2008**

FLUID FLOW STUDIES OF THE F-5E AND F-16 INLET DUCTS

IMRAN HALIMI BIN IBRAHIM

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in fulfillment of the requirement for the degree of
Master of Engineering

2008

TABLE OF CONTENTS

ABSTRACT	I
ACKNOWLEDGMENTS.....	III
LIST OF TABLES.....	IV
LIST OF FIGURES	VI
NOMENCLATURE	XIII
Chapter 1 INTRODUCTION	1
1.1 Overview of the Northrop F-5 Tiger.....	1
1.1.1 F-5E Propulsion System	3
1.2 Overview of the General Dynamics F-16 Falcon	4
1.2.1 Brief History and Evolution.....	4
1.2.2 Procurement by the Republic of Singapore Air Force (RSAF) ...	7
1.2.3 The F-16 Inlet	9
1.3 Objectives	10
1.4 Scope.....	10
1.5 Outline of Report	11
Chapter 2 LITERATURE REVIEW	12
2.1 Types of Intakes.....	12
2.1.1 Subsonic Intakes	13
2.1.2 Supersonic Intakes	14
2.1.2.1 Internal Compression Intakes	14
2.1.2.2 External Compression Intakes	16
2.1.2.3 Mixed Compression Intakes	16
2.2 Intake/Engine Compatibility	17
2.3 Technology Implementation in Current Aircrafts.....	18

2.4	Flow in Curved Pipes, Bends and Ducts.....	28
2.5	F-16 Intake Studies	31
2.6	Taguchi and ANOVA analysis	32
Chapter 3	NUMERICAL METHODOLOGY	36
3.1	The Aerodynamic Duct Concept	36
3.2	COMSOL PDE Calculator.....	39
3.2.1	Governing Equations	40
3.2.2	Solution Scheme	44
3.2.2.1	Using the Segregated Solving Approach	45
3.3	FLUENT Solver Scheme	45
3.4	Performance Parameters	48
3.4.1	Calculations for Total Pressure Coefficient, Distortion Coefficient (DC) and Pressure Recovery.....	48
Chapter 4	VALIDATION / VERIFICATION ANALYSIS	52
4.1	Viewgraph Norm	52
4.2	Grid Convergence Index.....	53
4.3	Validation and Verification.....	55
4.3.1	Circular S-duct.....	55
4.3.1.1	Comparison with Experimental Results.....	58
4.3.1.2	GCI Study of the Circular S-duct.....	61
4.3.2	F-5E Intake.....	62
4.3.2.1	Comparison with FLUENT Simulations	63
4.3.2.2	GCI Study of F-5E Duct	66
4.3.3	F-16 Intake	68
4.3.3.1	Comparison of airflow through the F-5E and F-16 intake at different orientations.....	70
4.3.3.2	GCI Study of the F-16 Duct.....	78
4.3.3.3	Specifications of Intake a Pre-Entry region	79
Chapter 5	F-16 Maneuverability Simulations	82

5.1	Effects of Pre-entry Separation	82
5.2	Effects of Angles of Attack (F-16)	83
5.2.1	Total Pressure and Velocity Vector Diagrams.....	86
5.3	Effects of Angle of Sideslip (F-16).....	95
5.3.1	Total Pressure and Velocity Vector Diagrams.....	96
5.4	Correlation to F-16 Flight Placard	102
Chapter 6	TAGUCHI AND ANOVA ANALYSIS	105
6.1	F-5E Manoeuvrability Design.....	105
6.1.1	Orthogonal Array	105
6.1.2	Analysis of Variance (ANOVA or Yates' method).....	109
6.2	F-16 Maneuverability Design	112
6.2.1	Orthogonal Array	112
6.2.2	Analysis of Variance (ANOVA or Yates' method).....	116
Chapter 7	Conclusion and Future Work.....	120
7.1	Conclusion	120
7.2	Future Recommendations	122
REFERENCES		124
APPENDIX A		130
APPENDIX B		136

I. ABSTRACT

The intake represents the front of a complex aerodynamic system. The shape of the intake diffuser follows the contours of the aircraft for external aerodynamic reasons. However the meandering shape can result in unwanted secondary flow or swirl build up at the Aerodynamic Interface Plane (AIP), resulting in engine flutter and stall. Using the Finite Element Method (FEM) Partial Differential Equations (PDE) calculator COMSOL, the compressible flow equations were obtained by the coupling of the continuity and momentum equations in the k - ϵ turbulence model and the energy equation conduction and convections model. The standard validation and verification techniques employed in the computational analysis were conducted in the study of F-5E and F-16 aerodynamic intakes. The results were validated using experimental results of the circular S-duct. Static pressure contours of the S-duct were compared at the azimuthal angles 0° , 90° and 180° with reasonable agreement. The Grid Convergence Index (GCI) study done on the circular S-duct, F-5E and F-16 intakes using the pressure recovery as variable. Results showed that fine grid solution error had a maximum error of 2.18%, 4.48% and 3.17% respectively. Comparisons were then made on the flow through the F-5E and F-16 intake only showing that the doubly offset F-5E intake had a more adverse effect on secondary flow formation. The addition of the pre-entry domain resembling that of the underbelly of the fuselage in addition to the intake itself was then implemented on the F-16 intake. Different maneuverability conditions (Mach number, AOA and AOS) were conducted on this F-16 geometry. The results were investigated via total pressure and swirl flow diagrams. Air quality was defined as the pressure recovery and the distortion coefficient that exists at the AIP. Addition of the pre-entry separation area resulted in more pressure recovery losses compared to studying the intake alone. However with increasing AOA, the underbelly of the fuselage acts as a flow straightener. Pressure losses were essentially small up to 20° AOA. The pressure recovery

and distortion characteristics of the intake with relation to changing AOS shows symmetry in the z-axis, the axis parallel to the transverse of the intake, as the intake was fundamentally a singly offset entity. The results were also correlated on a typical F-16 flight placard. Finally, the Taguchi's Method (TM) and Analysis of Variance (ANOVA) were conducted on the F-5E intake and the F-16 intake with pre-entry domain. They were used to analysis the effects of the factors and their interactions on the performance parameters of the intake. The results showed that for the F-5E intake, Mach number had the most effect on pressure recovery while AOA affected distortion most considerably. The F-16 intake, shielded at the underbelly of the fuselage, showed that the factor that resulted in pressure recovery change most considerably is the AOS, while AOA affected distortion most considerably. The two results were further reinforced with results obtained by ANOVA. The differences in results between the F-5E and F-16 studies are attributed to the boundary layer growth at the front of the F-16 intake (in the pre-entry region) and centerline curvatures of the two intakes (doubly offset F-5E intake versus the singly offset F-16 intake).

II. ACKNOWLEDGEMENTS

The author would like to acknowledge the Almighty for assisting by inspiring patience and consistency during times of hardship.

Special thanks to:

- Supervisor, A/P Ng Yin Kwee, for his knowledge and guidance in complex aerodynamic concepts.
- RSAF Personnel, Mr R. Gunasekaran and Major Kenneth Wong, for providing equipment and relevant documents in the use of the project.
- Companion, Wu Yashi, for constant encouragement.
- Family members, for love and support.

III. LIST OF TABLES

Table 3-1 The k- ϵ modeling constants used.....	42
Table 4-1 Calculation of discretization error for the circular S-duct. The superscripts of the results correspond to the subscripts of the numerical mesh cases in the first row. Variables are defined in flowchart by Celik [58]	62
Table 4-2 Calculation of discretization error for the F-5E duct. The superscripts of the results correspond to the subscripts of the numerical mesh cases in the first row.....	67
Table 4-3 Calculation of discretization error for the F-16 intake. The superscripts of the results correspond to the subscripts of the numerical mesh cases in the first row.	78
Table 6-1 Factors and levels Mach number, angles of attack and sideslip used in the F-5E simulations. Mach numbers of 0.4, 0.5 and 0.6 were used. For angles of attack and sideslip α and β of 20° and -20° were used.....	106
Table 6-2 Layout of F-5E aerodynamic maneuverability orthogonal array.	107
Table 6-3 The off-line TM results of pressure recoveries and distortion coefficients of the various factors for the F-5E intake.....	108
Table 6-4 The ranking of the relative significance of the factors for the F-5E intake.....	108
Table 6-5 The summary table denoting Factors A, B and C describing pressure recovery for the F-5E intake.	109
Table 6-6 Summary table for interaction AB used in the F-5E simulations....	110

Table 6-7 Summary table for interaction AC used in the F-5E simulations....	110
Table 6-8 Summary table for interaction BC used in the F-5E simulations.	110
Table 6-9 ANOVA table for pressure recovery used in the F-5E simulations.	111
Table 6-10 Distortion Coefficient ANOVA results used in the F-5E simulations.	112
Table 6-11 Factors and levels Mach number, angles of attack and sideslip used in the F-16 simulations. Mach numbers of 0.4, 0.6 and 0.8 were used. For angles of attack and sideslip α and β of 0° , 10° and 20° were used.....	113
Table 6-12 Layout of F-16 aerodynamic maneuverability orthogonal array...	114
Table 6-13 The off-line TM results of pressure recoveries and distortion coefficients of the various factors for the F-16 intake.. ..	115
Table 6-14 The ranking of the relative significance of the factors used in the F- 16 simulations.	116
Table 6-15 The summary table denoting Factors A, B and C describing pressure recovery used in the F-16 simulations.. ..	116
Table 6-16 Summary table for interaction AB used in the F-16 simulations. .	117
Table 6-17 Summary table for interaction AC used in the F-16 simulations. .	117
Table 6-18 Summary table for interaction BC used in the F-16 simulations. .	117
Table 6-19 ANOVA table for pressure recovery used in the F-16 simulations.	118
Table 6-20 Distortion Coefficient ANOVA results used in the F-16 simulations.	118

IV. LIST OF FIGURES

Figure 1.1 A USAF F-5E with afterburners executing a maneuver and sketch plans of the F-5E Knaack [1].	2
Figure 1.2 The primary and secondary propulsion system of the F-5E illustrates the complexity of the overall intake aerodynamic system. At all flight Mach numbers, the main duct is exposed to freestream conditions from Wolf and Farrell [4].	3
Figure 1.3 A US Airforce YF-16 in base [5].	4
Figure 1.4 An F-16D Block 52 at Luke AFB Dewitte and Vanhaste [6].	5
Figure 1.5 Israeli F-16 A/B double seater variants. These versions are normally used for training purposes.	8
Figure 2.1 The A340 with podded subsonic installations [5].	13
Figure 2.2 The British Aerospace Harrier in vertical landing mode from Hirschberg [12].	14
Figure 2.3 Three operations of the internal compression inlet shown in Oates [3].	15
Figure 2.4 An example of an external compression inlet with the presence of oblique and normal shocks near the cowl lip shown in Oates [3].	16
Figure 2.5 The mixed compression inlet shown in Oates [3].	17
Figure 2.6 The proportional correlation between complexity and Mach number tabulated in Goldsmith and Seddon [13].	19

Figure 2.7 F-111A intake and engine locations shown in Goldsmith and Seddon [13].	20
Figure 2.8 The F-111A Aardvark from General Dynamics with wings partially swept back [5].	20
Figure 2.9 Close-up of the inlet portion of the intake taken by Ibrahim [14].	21
Figure 2.10 The Mirage fighter with conical intake at take-off as shown in Su [15].	21
Figure 2.11 The F-18 air induction system from Goldsmith and Seddon [13].	22
Figure 2.12 The F-18 Hornet at sonic boom [16].	22
Figure 2.13 EAP aircraft with hinged cowl lip [5].	23
Figure 2.14 The Rafale intake and fuselage shaping [5].	24
Figure 2.15 Planview of the Rafale aircraft [5].	24
Figure 2.16 The YF-22A Raptor at flight from Hayles [17].	25
Figure 2.17 Planform view of the Lockheed YF-22A from Goldsmith and Seddon [13].	25
Figure 2.18 Front view of the YF-23A Black Widow 2 [18].	26
Figure 2.19 Planform view of the Lockheed YF-23A from Goldsmith and Seddon [13].	26
Figure 2.20 The F-15 with afterburners turned on [5].	27
Figure 2.21 F-15 intake/engine integration [5].	27
Figure 2.22 Comparison of the A-12 and F-14 in planform view from Goldsmith and Seddon [13].	28

Figure 3.1 Axial locations of flow stations for a complete engine nacelle. The symbols ∞ , c , f and e refer to upstream infinity, duct entrance, engine face and exit respectively.	36
Figure 3.2 Flow through an intake in a supersonic stream (a) $CFR < 1$; (b) $CFR = 1$; (c) $CFR > 1$ by Goldsmith and Seddon [13].	38
Figure 3.3 Overview of the uses of COMSOL.	39
Figure 3.4 Measurement of the distortion coefficient (DC) factor. The circular AIP boundary is investigated with θ taken as 60°	49
Figure 4.1 Viewgraph norm qualitative and quantitatively compares experiment and computational results.	52
Figure 4.2 The outlet of the second bend of the circular S-duct by Vakili <i>et al.</i> [22].	55
Figure 4.3 Normalised distance along duct centerline, S/D	56
Figure 4.4 The circular duct is meshed with tetrahedral elements, resulting in 5848 elements and 63948 degrees of freedom.	57
Figure 4.5 δ_w^+ values for the circular S-duct.	57
Figure 4.6 Comparison of total pressure coefficient and velocity vector diagrams of Station 6 between experimental results done by Vakili <i>et al.</i> [22] and COMSOL results.	58
Figure 4.7 Comparison of static pressure coefficient vs. normalized centerline distance between experimental done by Harloff <i>et al.</i> [24] and COMSOL results along the 0° azimuth angle (top) of the duct.	59

Figure 4.8 Comparison of static pressure coefficient vs. normalized centerline distance between experimental done by Harloff <i>et al.</i> [24] and COMSOL results along the 90° azimuth angle (side) of the duct.	59
Figure 4.9 Comparison of static pressure coefficient vs. normalized centerline distance between experimental done by Harloff <i>et al.</i> [24] and COMSOL results along the 180° azimuth angle (bottom) of the duct.....	60
Figure 4.10 Monotonic Convergence of pressure recoveries with increasing mesh elements.....	61
Figure 4.11 The SOLIDWORKS and COMSOL F-5E geometry. Note to export geometry to COMSOL, adequate guide curves are required in the lofting process.....	63
Figure 4.12 Five stations on the port side F-5E intake.	64
Figure 4.13 Total pressure isobars analysis of the F-5 port side duct. The outlet boundary shows non axis-symmetry character, an undesirable compressor intake feature.....	66
Figure 4.14 Convergence of pressure recovery increase with mesh elements. The trend shows with increasing mesh, pressure recovery converges to a value of about 0.978.	68
Figure 4.15 Duct area distribution of the F-16 inlet used in COMSOL compared to the NSI by Hawkins [32] and MCID-713 and MCID-771 by Hagseth [33].	69
Figure 4.16 The F-16 inlet with a single offset y-axis.....	69
Figure 4.17 The doubly offset x- and y-axis of the F-5E inlet.	69

Figure 4.18 Total pressure coefficient contours for F-16 at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 0^\circ$	72
Figure 4.19 Total pressure coefficient contours for F-5E at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 0^\circ$	73
Figure 4.20 Total pressure coefficient contours for F-16 at Mach 0.6, $\alpha = 20^\circ$ and $\beta = 0^\circ$	74
Figure 4.21 Total pressure coefficient contours for F-5E at Mach 0.6, $\alpha = 20^\circ$ and $\beta = 0^\circ$	75
Figure 4.22 Total pressure coefficient contours for F-16 at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 20^\circ$	76
Figure 4.23 Total pressure coefficient contours for F-5E at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 20^\circ$	77
Figure 4.24 Monotonic Convergence of the simulations conducted on the F-16 intake.....	78
Figure 4.25 A 1/48 SOLIDWORKS model of F-16. The boxed portion highlights the simulation geometry. The fuselage in the geometry is taken to be a flat plate.....	79
Figure 4.26 The geometry of the simulations showing the two subdomains involving airflow through the underside of the fuselage and the internal airflow through the intake. The fuselage is offset a distance of 10 inches showing the diverter height.	80
Figure 4.27 The geometry of the underbelly fuselage together with the F-16 intake used in the simulations. The underbelly is assumed to be a flat plate. ...	80

Figure 4.28 Mesh setting for the simulation of fuselage and intake with a coarser setting at the fuselage portion.....	81
Figure 4.29 Viscous wall units of the fuselage underbelly as well as the walls of the intake satisfy the limits imposed in the circular S-duct.	81
Figure 5.1 The addition of the fuselage results in pre-entry separation of the flow. This results in lower total pressure recoveries in the Mach range of 0.1 to 0.9. Results are obtained by simulations in COMSOL.....	82
Figure 5.2 The F-16 Falcon duct losses compared other fighter aircrafts by Seddon and Goldsmith [2]. The numerical results of the F-16 Falcon, obtained from COMSOL simulations, show similarity to the Hawk duct losses.....	83
Figure 5.3 A comparison of inlet pressure recovery for the NSI, MCID-713, MCID- 771 and COMSOL F-16 inlet.....	84
Figure 5.4 Axial total pressure plots for all the AOA used in the simulations. The high and low total pressure starts at the lower and higher portions of the geometry initially and diffuses to the opposite direction as the AOA rises.....	85
Figure 5.5 Distortion effects are minimized at AOA 5°, and increase with changing AOA due to the effects of pre-entry separation.	86
Figure 5.6 Total pressure plot and velocity vector diagram of AOA -10°.	88
Figure 5.7 Total pressure plot and velocity vector diagram of AOA -5°.	89
Figure 5.8 Total pressure plot and velocity vector diagram of AOA 0°.....	90
Figure 5.9 Total pressure plot and velocity vector diagram of AOA 10°.....	91
Figure 5.10 Total pressure plot and velocity vector diagram of AOA 20°.....	92
Figure 5.11 Total pressure plot and velocity vector diagram of AOA 30°.....	93

Figure 5.12 Total pressure plot and velocity vector diagram of AOA 40°.....	94
Figure 5.13 The effect of pressure recovery on AOS shows a decreasing recovery value as AOS increases.....	95
Figure 5.14 The effect of distortion on AOS shows a steadily increasing distortion coefficient value with increasing AOS.....	95
Figure 5.15 Total pressure plot and velocity vector diagram of AOS 5°.....	97
Figure 5.16 Total pressure plot and velocity vector diagram of AOS 10°.....	98
Figure 5.17 Total pressure plot and velocity vector diagram of AOS 15°.....	99
Figure 5.18 Total pressure plot and velocity vector diagram of AOS 20°.....	100
Figure 5.19 Total pressure plot and velocity vector diagram of AOS 25°.....	101
Figure 5.20 A typical flight placard of a military fighter.....	103
Figure 5.21 F-16 flight placard from Hagseth [33]. The Normal Shock Inlet (NSI) in the F-16 shows similar geometric properties as that used in the COMSOL simulation as shown in Figure 5.3.....	103

V. NOMENCLATURE

$\dot{m}$	mass flow rate	(kg/s)
ρ	density	(kg/m ³)
V	average velocity	(m/s ²)
A	cross-sectional area	(m ²)
p	static pressure	(Pa)
$\mathbf{u}$	velocity vector	(m/s)
a	speed of sound	(m/s)
M	Mach number	(= u/a)
η	dynamic viscosity	(kg/m/s)
η_T	turbulent viscosity	(kg/m/s)
F	force per unit volume	(N/m ²)
k	turbulent energy	(m ² /s ²)
ε	turbulent dissipation	(m ² /s ³)
$\mathbf{I}$	identity matrix	
I_s	turbulence intensity	
l	characteristic length	(m)
D	hydraulic diameter	(m)
$C_{\varepsilon 1}, C_{\varepsilon 2}, C_{\mu}, \sigma_k, \sigma_{\varepsilon}$	k- ε modeling constants in Chapter 3	
T	Temperature	(K)
C_p	specific heat of capacity at constant pressure	(J/kg.K)
Q	heat source	(W/ m ³)
k	thermal conductivity	(W/m.K)
k_T	turbulent thermal conductivity respectively	(W/m.K)
δ_w	normal wall distance	(m)
δ_w^+	normal wall distance in viscous units	
Re	Reynolds number	
p_0	total pressure	(Pa)
q	dynamic pressure	(Pa)
$DC(\theta)$	distortion coefficient	
θ	angle of distortion domain (60°)	
ψ	pressure recovery	
α	angle of attack used in Chapter 6	(°)
β	angle of sideslip used in Chapter 6	(°)
Subscripts		Superscripts
∞	upstream infinity	—
value		averaged
c	duct entrance	
f	engine face	
e	exit	

Chapter 1 INTRODUCTION

As of all air-breathing vehicles, the supply of air to the engine of the vehicles has to be at optimum conditions. The role of the intake or inlet of an aircraft is to supply atmospheric air to the engine. English semantics will not distort the real message being conveyed and hence both terms, intake and inlet, will be used interchangeably. The definitions of 'optimal conditions' have been widely discussed as having maximum total pressure recovery and minimum distortion.

Intakes are diverging ducts in nature. Usually due to the short length and aerodynamic considerations of fighter planes, it is inevitable that the shape of the intake has to be specially designed and curved according to the fuselage or the position at which it is housed. However, the resultant curves of the intakes will cause the formation of secondary flows in the intake. These secondary flows may cause separated flow, swirls and pressure distortion at the engine face which is undesirable for optimum engine performance. Maximizing pressure recovery and space constraint represents one of the many conflicting aerodynamic design goals that will be further discussed in a later section.

Two types of the fixed wing combat aircrafts that are currently being studied are the Northrop F-5 Tiger and the General Dynamic F-16 Falcon. The history and evolution of the both fighter aircrafts will be shown in the next section.

1.1 Overview of the Northrop F-5 Tiger

The F-5E/F Tiger II are part of a family of widely used light supersonic fighter aircraft, designed and built by Northrop in the United States, beginning in 1960s. They were a replacement of the F-5A with lengthened and enlarged, with increased wing area and more sophisticated avionics and called F-5E and eventually received the official popular name Tiger II. Overtime several

variants of the F-5 emerged at different countries, such as the single-seat F-5S and the two-seat F-5T versions. Powered by General Electric J85-GE-21 Engines, the F-5 can reach a maximum speed of 917 knots (1,060 mph, 1,700 km/h) or Mach 1.6 at 36000 feet as shown in Knaack [1]. Figure 1.1 shows the F-5E in flight.

Figure 1.1 A USAF F-5E with afterburners executing a maneuver and sketch plans of the F-5E Knaack [1].

Fighter aircraft, such as the F-5E, are generally designed for multi-mission capability. A mission scenario could involve a high-speed/ low-altitude penetration, transonic maneuvering and supersonic dash to fulfill the vehicle's intended objectives. The task of the inlet (used by the Americans) or intake (used by the British) is to provide high quality airflow to the engine as explained in Seddon and Goldsmith [2]. These two terms, inlet and intake, will be used interchangeably in the author's study. Other potential sources of airflow demand are: engine periphery and/or exhaust nozzle cooling, electronic equipment cooling and boundary/layer bleed from the inlet ramp or spike and internal cowl surfaces.

In the twin engine F-5E fighter, each intake is side mounted on the aircraft as explained in Oates [3]. Figure 1.1 also shows the top, front and port side of the F-5E. The design of the intakes follows the contours of the main body of a fighter plane for external aerodynamic reasons. It is because of these reasons that a meandering shaped diffuser results. The curvature of an intake

could result in unwanted secondary flows and formations of vortices which would inadvertently affect the engine performance and could result in flutter and stall. Flow separation in the inlet can affect air quality leading to the engine and degrade the aircraft's performance either through reduced inlet pressure recovery and/or increased flow distortion.

1.1.1 F-5E Propulsion System

A block diagram is shown in Figure 1.2 which identifies both the primary and secondary air flow paths to show the complexity of an overall aerodynamic intake system shown in Wolf and Farrell [4]. The F-5E basically consists of two induction systems powered by J85-GE-21 engines used as the primary propulsion units. Separate secondary airflow systems are used for the engine bay cooling. There are also two airframe mounted ejectors and a common keel. In each induction system, there is the inlet ramp bleed system and auxiliary takeoff door, plenum and duct slot. The secondary system for engine bay cooling includes takeoff door, plenum, bay and keel cooling airflow and airframe-mounted ejector.

Figure 1.2 The primary and secondary propulsion system of the F-5E illustrates the complexity of the overall intake aerodynamic system. At all flight Mach numbers, the main duct is exposed to freestream conditions from Wolf and Farrell [4].

1.2 Overview of the General Dynamics F-16 Falcon

1.2.1 Brief History and Evolution

The F-16 was designed as a light-weight fighter with a very high power weight ratio for good performance, a fly-by wire control system, and a semi-reclining pilot's seat for a good field of vision. Having a maximum short-endurance speed of Mach 2.05 (1353 mph) and the maximum sustained speed is Mach 1.89 (1247 mph) at 40,000 feet, the F-16, known as the Fighting Falcon, can serve as an air combat fighter and has also developed into an exceptional multi-purpose fighter. Its maximum level of speed at 40,000 feet is more than Mach 2 or 1,320 miles per hour. Its service ceiling is more than 50,000 feet. The F-16 has excellent turning abilities and capabilities which has made it an outstanding and highly utilized multi-purpose fighter plane.

The evolution of the F-16 Falcon began with the initiation of its prototype, YF-16, in late 1973 (Figure 1.3). Since then, modifications have been made to strike a balance between improving capabilities and maintaining the performance levels of the original design. Detailed information of these modifications can be obtained from aircraft magazines and websites. However, this discussion is focused on the engine improvements and how they are related to the overall characteristics of the aircraft.

Figure 1.3 A US Airforce YF-16 in base [5].

Generally, the modifications were made in groups, or blocks, to track things on the production line. A block was an important term in tracing the F-16's evolution and it is also a numerical milestone. The block number increases whenever a new production configuration for the F-16 was established. The first production aircrafts that followed the two YF prototypes and the eight full-scale development F-16s, were the Blocks 1 and 5. Today, the latest F-16s are designated Block 50/52. Block 50 has a General Electric (GE) engine while Block 52 has a Pratt and Whitney (P&W) engine shown in Figure 1.4. (From Block 30/32 onwards, a major block designation ending in 0 signifies a GE engine; one ending in 2 signifies a P&W engine.) Block 55 and Block 60 are often used to describe concepts for advanced versions of the F-16, but these terms do not indicate engine selection.

Figure 1.4 An F-16D Block 52 at Luke AFB Dewitte and Vanhaste [6].

Capability improvements come through upgrading weaponry systems and positional modifications of aircraft structures such as pods and fuel tanks. These structures can be placed tucked under the wings, attached to the centerline, and affixed to the intake. The airframe itself has been reinforced to handle some of these additions. The good news is that these increases in

capability are packed into an airframe which is capable of sustaining aircraft maneuverability that could reach up to nine Gs, or nine times the gravitational pull exerted by the earth. The bad news is that these new capabilities also cause the aircraft to experience additional weight and drag.

This is also why even though the F-16 may have been designed as a lightweight fighter; it has not been completely immune to the "pound a day" problem that applies to all fighters. The empty weight of the YF-16 with two AIM-9 missiles was 14,023 pounds. The prototypes, however, did not have any fire control radar or the avionics characteristic of an operational airplane. When the first batch of Block 10 F-16A arrived, the average weight of each was 15,600 pounds. The latest F-16C/D Block 50 today weighs an average of 19,200 pounds. The increase equates to less than half a pound a day, but it is additional weight nonetheless.

Fortunately, the engine makers, P&W and GE, have stepped up to the scale and offset much of the weight and drag gains with improvements in engine performance. The original P&W engine on the YF-16 developed over 23,000 pounds of thrust. The engines on the Block 50/52 aircraft developed nearly 30,000 pounds of thrust. The next round of engine improvements is expected to boost this figure to as high as 37,000 pounds. As such, even though weight has increased, the thrust-to-weight ratio of the F-16 has actually improved over time.

Other than improving maximum thrust, idle to afterburner characteristics have also improved. Since the evolution from hydromechanical to electrical systems, the engines could go from idle to full afterburner in just two seconds. Previous engines took about six to eight seconds to spool up. This responsiveness creates a huge payoff in performance and aircraft handling. Engine reliability and ease of maintenance have also been improved significantly. Today's F-16 engines can be expected to deliver eight to ten years of operational service between depot inspections.

In addition, the digital engine controls, first introduced on P&W powered F-16s in 1986, have also improved in performance. Older hydromechanical controls had to be trimmed to operate at the most challenging point within the F-16's flight envelope while digital engine controls automatically adjust to the operating environment so that they optimize engine performance at all points within the flight envelope. This optimization has increased thrust by more than ten percent in some areas of the F-16 flight envelope. All engines that are being built today for the F-16 have second-generation digital engine controls.

1.2.2 Procurement by the Republic of Singapore Air Force (RSAF)

The procurement of the F-16 by RSAF followed an interesting evolution as explained in Dewitte and Vanhaste [6]. The RSAF obtained the F-16 aircrafts in 5 batches and at present operates 70 F-16 Fighting Falcons, 62 of which are versions F-16C/D block 52 aircraft. These aircraft are all equipped with state-of-the-art armament. In January 1985, the RSAF initially ordered the J79 variant. J79 variant is a cost reduced unit powered by the GE J79 turbojet.

In the mid-1985s, the F100 P&W powered engines became available and the RSAF changed the initial J79 orders to the F-16 A/B versions (4 single and 4 double-seater versions) shown in Figure 1.5. The planes would be equipped with the P&W engines. This purchase was under the Peace Carvin I Foreign Military Sales program, and was intended to replace the ageing Hawker Hunters which were still serving with the RSAF.

Figure 1.5 Israeli F-16 A/B double seater variants. These versions are normally used for training purposes.

Singapore received its first two-seater P&W F100-PW-220 powered F-16 on February 20th, 1988. The Machines were initially delivered to Luke Air Force Base, where the RSAF trains its F-16 pilots. At that time, Singapore had also leased nine F-16A's previously used by the 'Thunderbirds' flight demonstration team from 1993 to 1996, for training at Luke. At the end of the F-16A training at Luke, a deal was struck with Lockheed Martin to buy a dozen newly-built F-16C/D Block 52's (4 Charlie-models and 8 Delta-models). 2004, donated F-16 A/B to the Republic of Thailand Airforce (RTAF).

In 1993, Dr. Yeo Ning Hong, the Defense Minister then, announced plans for the purchase of 5 F-16C's and 6 F-16D's. A year later, the RSAF increased the order to 18 Block 52 Machines (8 C's and 10 D's). The 18 block 52 Machines would all be powered by the F100-PW-229 engine. The F100-PW-229 is lighter and more powerful than earlier F100s, and had been flying at Edwards Air Force Base since mid-1990s in test ships. However, both engines are rated at 17,000 lbs and 29,000lbs of thrust (129kN) with afterburners on. On October 29th, 1997, the government of the Republic of Singapore announced it would buy another 12 F-16C/D Block 52 aircraft, 10 C-models and 2 D-models. Singapore revealed that it was ordering another 20 F-16s on July 21st, 2000. At first, it was not clear whether it would be a mix of C-models and D-models, but

eventually the Singapore government decided to make the order an all D-model lot.

1.2.3 The F-16 Inlet

The modifications of the F-16 engines call for the need to improve the intakes leading up to the engines as shown in Bergsmans [7]. There are essentially two forms of intake ducts used in the variants of the F-16: the 'small' or popularly Normal Shock Inlet (NSI) and the 'big-mouth' or Modular Common Inlet Duct (MCID). Block 30/32 added two new engines-the P&W F100-PW-220 and the GE F110-GE-100-to the F-16 line. Block 30 designates a GE engine, and Block 32 designates a Pratt & Whitney engine. A larger inlet was introduced at Block 30D for the GE-powered F-16s, which are often called "big-mouth" F-16s. The larger inlet, formally called the modular common inlet duct, allows the GE engine to produce its full thrust potential at lower airspeeds.

The 0.30m larger air intake enables the General Electric F110-GE-100 to offer 128.9 kN of thrust, while F-16C Blocks 32 and 42 Falcons introduced a 106.05-kN thrust Pratt & Whitney F100-PW-200s. USAF F-16C/D delivery totals slightly favor the GE engine.

The geometry of the F-16 intake resembles an S-duct, dominated by a curvature in the streamwise direction. The intake transcends from an elliptical inlet to a fully circular outlet at the compressor face. Curvature at only one axis allows an analysis of only one half of the intake. The curves are lofted in SOLIDWORKS and exported in the finite element program, COMSOL. The curved are initially part of a scaled model of the F-16 as shown in Bourke [8]. The characteristic length of the elliptical inlet is measured at its semimajor axis, resulting in a length of 1.05m. This value is crucial in determining the inlet conditions, such as turbulent kinetic energy and dissipation, derived the Reynolds number. The total length of the F-16 inlet is 5.08m, similar to the F-5E inlet.

1.3 Objectives

The objectives of this study are as follows:

1. To compare and discuss results obtained with the previous numerical analysis conducted on the intakes of the F-5E and the F-16.
2. To improve the investigations of the F-16 by adding on a pre-separation domain.
3. To investigate the maneuverability of the F-16 intake on performance parameters such as pressure recovery (ψ) and distortion coefficient DC (θ). Visualizations of the total pressure plots and swirl flows of transverse stations of the intake will also be shown.
4. To compare the results of Taguchi and ANOVA on the maneuverability of the F-16 intake together with the presence of the front fuselage with the previous results obtained on the F-5E intake only.

1.4 Scope

The study of intake aerodynamics covers a multitude of aerodynamic concepts that goes to the overall designing factors of the intake. Generally, the study can be divided into two categories: separation before and after the entry of the intake. Pre-entry separation can be resulted by the presence of the external geometries such as fuselage, splitter plate and entry lip. The airflow can also be disrupted within the intake by the bleed systems, flow diverters and landing gear geometries. Variable geometry at the external and internal flows can affect the capture flow ratio (CFR), as the capture area is allowed to change in magnitude to allow the optimum airflow to enter the intake.

To simulate flow within the subsonic and transonic Mach regions, the pre-entry separation is simulated by the presence of the fuselage only. The role of the splitter plate is only significant at supersonic regions. Also, due to the

bleed systems, flow diverters and landing gear geometries will be neglected within the intake.

In addition, only steady-state analysis simulations are conducted. This allows the measurement of total pressure distortion via the distortion coefficient $DC(\theta)$ method as used by Rolls-Royce as stated in Seddon and Goldsmith [2]. The other performance parameters used will be total pressure recovery ψ of the intake and swirl.

1.5 Outline of Report

Chapter 1 discusses the basic designs of the F-5E and F-16. The F-5E propulsion design and F-16 inlet types are mentioned. Chapter 2 lists previous literatures in the field of duct flow. The chapter mentions the study of the tradeoffs in jet inlet design. This study is then developed into the evolution of intake airflows. Implementation into the F-16 intake is then investigated. Finally an overview of the statistical analysis that will be conducted in this study is discussed. Chapter 3 discusses the methodology of our study. Essentially a numerical analysis, the governing equations and boundary conditions are noted. The program that will be used is a PDE calculator, by COMSOL [9] [10]. Comparison of the F-5E intake flow will also be done on the solution scheme used in FLUENT as conducted by Wu *et al.* [11]. The specifications of the geometry studied are also shown. As of all numerical analysis, qualitative and quantitative validations have to be made to reinforce credibility of the results made. This will be done in Chapter 4. Chapter 5 discusses the experiments conducted in the three aspects of maneuverability of the F-5E and F-16: increasing Mach number, changing Angles of Attack (AOA), and changing Angles of Sideslip (AOS). Chapter 6 compiles the previous works as well as additional combinations of the three factors to facilitate the analysis using Taguchi and ANOVA. Finally Chapter 7 discusses the conclusion and the future works.

Chapter 2 LITERATURE REVIEW

2.1 *Types of Intakes*

As mentioned previously, the primary purpose of the inlet is to bring the air required by the engine from freestream conditions to the conditions required at the entrance of the fan or compressor with minimum total pressure loss and flow distortion. The fan or compressor work best with a uniform flow of air at Mach number of about 0.5. Also, since the installed engine performance depends on the inlet's installation losses (additive drag, external forebody or cowl drag, bypass air, boundary layer bleed air, etc.) the design of the inlet should minimize these losses. The performance of an inlet is related to the following characteristics: high total pressure ratio, controllable flow matching, good uniformity of flow, low installation drag, good starting and stability, low signatures (acoustic, radar and infrared) and minimum weight and cost while meeting life and reliability goals. An inlet's overall performance must be determined by simultaneously evaluating all of these characteristics since an improvement in one is always accompanied with a compromise of another.

The design and operation of subsonic and supersonic inlet differ considerably due the characteristics of the flow. For subsonic inlets, near-isentropic internal diffusion can be easily achieved and the inlet flow rate adjusts to demand. The internal aerodynamic performance of a supersonic inlet is a major design challenge since efficient and stable flow diffusion is difficult to achieve over a wide range of Mach numbers is difficult to achieve. In addition, the supersonic inlet must be able to capture the required mass flow rate, which may require variable geometry to minimize inlet loss and drag and provide stable operation. This mass flow rate, associated to the Capture Flow Ratio (CFR) termed earlier, will be substantiated in the next section

Although capable of supersonic speeds, the F-5E intake has more subsonic than supersonic characteristics. For low supersonic speed (Maximum

Ma=1.5~1.7) aircraft, a subsonic intake is always used because it has a stable flow characteristic with good flow distribution at different flight speeds and different engine working conditions shown in Seddon and Goldsmith Seddon and Goldsmith [2]. The F-5E does not possess features such as compression surfaces and variable geometry for complex supersonic flow control.

2.1.1 Subsonic Intakes

Intakes on subsonic aircrafts divide broadly into two categories; ‘podded’ and ‘integrated’ installations as explained in Seddon and Goldsmith [2]. These normally relate to transport aircraft (civil or military) and combat aircraft respectively. With a podded installation, for example the Airbus 340 in Figure 2.1, the flow inside the intake has the shortest and most direct route possible to the engine and its pressure recovery is almost 100%. A problem may exist in the form of shock waves induction from the internal shaping of the cowl. Another significant problem is the effect on external aerodynamics on the aircraft wing and fuselage.

Figure 2.1 The A340 with podded subsonic installations [5].

With an integrated intake, such as the British Aerospace Harrier in Figure 2.2, internal flow problems are more significant. These are due to a) the duct being longer, usually containing bends and shape changes and b) the presence of aircraft surface ahead of the intake, wetted by internal flow.

Figure 2.2 The British Aerospace Harrier in vertical landing mode from Hirschberg [12].

2.1.2 Supersonic Intakes

Supersonic inlets may be classified into one of three types – external, mixed or internal compression – according to whether the supersonic diffusion occurs external to the inlet duct, partly external and partly internal to the duct, or entirely internal to the duct as shown in Oates [3]. The mixed compression inlet provides higher recovery at supersonic conditions however its increased weight, control complexity and bleed requirements must be balanced against the improved supersonic recovery. For a multimission aircraft, this is seldom a favourable trade.

2.1.2.1 Internal Compression Intakes

The internal compression inlet achieves compression through a series of internal oblique shock waves followed by a terminal normal shock wave positioned downstream of the throat (its stable location). This type of inlet

required variable throat area to allow the inlet to swallow the normal shock during starting. Fast reaction bypass doors are also required downstream of the throat to permit proper positioning of the terminal normal shock under different conditions.

Figure 2.3 Three operations of the internal compression inlet shown in Oates [3].

There are three conditions related to the operation of the internal compression inlet. At normal or ideal condition as shown in Figure 2.3a, the terminal normal shock is positioned downstream of the throat for stable operation. This occurs for an adequate area contraction ratio at a given Mach number. Movement of the terminal shock to the throat (which is an unstable location) will cause the total internal flow pattern to be completely disrupted as shown in the unstarted inlet condition in Figure 2.3b. This occurs when the throat area is not sufficiently changed when the free stream Mach number is brought up from subsonic to supersonic speeds. Starting of the inlet (Figure 2.3c) can be achieved when the area of the throat is made large enough for the normal shock to move back and touch the inlet lip. This is also known as the critical operation point.

The demise of the internal compression inlet is associated to the large area variation, problem of inlet unstart, poor performance at angles of attacks

and many other issues. Its analysis is of purely academic reasons. It is also included to bridge the gap between practical and academic worlds.

2.1.2.2 External Compression Intakes

The external compression type achieves supersonic flow diffusion through either one or a series of oblique shocks followed by a normal shock or simply through one normal shock. This is shown in Figure 2.4. The external compression inlet that achieves compression through only a single normal shock is called a 'pitot inlet' or 'normal shock inlet'. The simulations studied in this report use this notion of a single normal shock existing at the entrance of the intake.

Figure 2.4 An example of an external compression inlet with the presence of oblique and normal shocks near the cowl lip shown in Oates [3].

2.1.2.3 Mixed Compression Intakes

At flight Mach numbers above 2.2, the mixed compression inlet is used to achieve compression in the inlet. The mixed compression inlet is more complex, heavier and costlier than the external compression inlet. It achieves compression through the external oblique shocks, internal reflected oblique shocks and the terminal normal shock. The ideal location of the normal shock is just downstream of the inlet throat to minimize total pressure loss while maintaining a stable operating location of this shock. Similar to the internal

compression inlet, the mixed compression inlet requires both fast reacting bypass doors to maintain the stability of the normal shock and variable throat area to allow the normal shock to be swallowed in the inlet. Due to the external oblique shock system, the complexity of the inlet variation throat area is considerably less than the system present in the internal compression inlet. Figure 2.5 shows the mixed compression inlet.

Figure 2.5 The mixed compression inlet shown in Oates [3].

2.2 Intake/Engine Compatibility

The definition of the quality of air that enters the engine has evolved together with the complexities of aircraft engine design as explained in Goldsmith and Seddon [13]. Engine installations in the early 1950s, such as the F-100, Sabre and F-101 aircraft in the USA and the Meteor, Javelin, Hunter and V-bombers in the UK, were treated in a largely qualitative way. For example, a General Electric Company Installation Handbook for Turbojet Engines circa 1952, which dealt quite extensively with installation performance corrections, stated by way of introducing inductions systems:

“This process must be accomplished with the least possible loss in total-pressure or head, with the best attainable flow distribution, and with the least amount of drag.”

It can be seen that total pressure loss has been a recognized factor in defining airflow quality. Drag is known to be a function of the external flow of

the intake, and not used as a performance parameter in this study. The evolution for the definition of flow distribution, or distortion, differed for the USA and UK. The abundance of numerical distortion descriptors, some very complex, were developed by engine manufacturers and used to correlate the effects of total pressure distortions on the stability of the engine. In the USA, these included the K_{DA} , K_{D2} , K_{θ} , K_{C2} , K_{ra2} , K_{A2} (for the Pratt and Whitney Aircraft), and the Method D index system comprising of IDC, IDR, ID (General Electric Company). These distortion measurement methods use the radial distance as the key factor in the calculations. In the UK, the main parameter is the DC (60) as used by Rolls-Royce. Circumference analysis is the underlying factor in the measurement of distortion. In keeping with the metric system of units, the DC (60) system was chosen as the parameter to study air quality at the compressor face.

2.3 Technology Implementation in Current Aircrafts

The need for increased speed and maneuverability pushed intake designers to come out with more complex intakes. The trend of increasing complexity with increase of maximum Mach number is shown in Figure 2.6. This correspondence is due, in part, to the fact that US aircraft are being compared. Generally, European and Soviet designers placed greater emphasis on simplicity and/or weight than high-pressure recovery and drag.

Fighter	Semi flush	BL diverter	Side plate	Compression surface	BL bleed	Variable geometry	Mixed compression	Max Mach
F-80	x							Subsonic
F-89		x						
F-5A		x	x					Mach No 1.2–1.9 Class
F-101		x	x					
F-102		x	x					
F-16		x	x					
F-18		x	x	x				Mach No 2.0–2.2 Class
F-104		x		x	x			
F-105		x	x	x	x	x		
F-106		x	x	x	x	x		
F-4		x	x	x	x	x		MACH 2.5
F-14		x		x	x	x		
F-15		x		x	x	x		
F-111		x	x	x	x	x		MACH 3+
YF-12				x	x	x	x	

Figure 2.6 The proportional correlation between complexity and Mach number tabulated in Goldsmith and Seddon [13].

One of the branches of the evolution of intake study was initiated by lessons learnt from the evaluation of the F-111 wing root intake (Figure 2.7). By tucking the intake under the wing in an armpit location, the intake was shortened (for lighter weight) and the wing was used to shield the intake and thus provide high pressure recovery and low distortion in supersonic maneuvering flight. An F-111A is shown in flight in Figure 2.8.

Figure 2.7 F-111A intake and engine locations shown in Goldsmith and Seddon [13].

Figure 2.8 The F-111A Aardvark from General Dynamics with wings partially swept back [5].

The Northrop F-5E had its intake side-mounted and reduced greater emphasis on simplicity a maneuvering by requiring only a diverter and side plate to prevent boundary layer ingestion and shock-boundary layer interaction. This reduced the maximum Mach number to 1.4.-1.6. Figure 2.9 shows the channel diverter, sideplate and bleed systems used at the entry of the F-5E.

Figure 2.9 Close-up of the inlet portion of the intake taken by Ibrahim [14].

Another series of aircrafts that was using side-mounted intakes was the France's Mirage series (Figure 2.10). The intakes were shaped semi-conical and this indicates a highly maneuverable fighter aircraft with high distortion levels, but the Mirage and Kfir were powered with turbojet engines to balance this factor. The approach by the designers of Mirage fighter series emphasizes factors of light weight, low supersonic drag and simplicity.

Figure 2.10 The Mirage fighter with conical intake at take-off as shown in Su [15].

The F-18 used the wing shielding method similar to the F-111, but provided slots in the root of the wing leading edge extension which allowed for the escape of fuselage boundary layer. These slots however, caused wing drag.

The supersonic intake in the F-18 positioned movable ramp and bleed exit doors near the inlet to reduce secondary flow formation at transonic and supersonic speeds (Figure 2.11). Figure 2.12 shows the F-18 breaking the sound barrier.

Figure 2.11 The F-18 air induction system from Goldsmith and Seddon [13].

Figure 2.12 The F-18 Hornet at sonic boom [16].

The F-16 used fuselage shielding and avoided the problem of fuselage boundary layer ingestion while maintaining the positive aspects of shielding. This was similar to the European Fighter Aircraft (EFA) underbody shielding, with a minor difference of hinged lower cowl lip to improve performance at high incidence as shown in Figure 2.13.

Figure 2.13 EAP aircraft with hinged cowl lip [5].

A unique approach implemented by Rafale designers seems to fuse the concepts used by the F-16 and F-18 together. The aircraft retains the significant advantages of fuselage shielding but at the same time provides a considerable amount of boundary layer relief at high angles of incidence, allowing low energy air from the underbody to flow through the diverter region and spilled above the intake. This is shown in Figure 2.14. A planform view is shown in Figure 2.15.

Figure 2.14 The Rafale intake and fuselage shaping [5].

Figure 2.15 Planview of the Rafale aircraft [5].

The US Air Force Advanced Tactical Fighter (ATF) prototype (the YF22 and YF 23) shows even more integration of intake and airframe than was achieved with the previous generation aircrafts. Several design themes are accommodated; those of shielding and lip stagger (to enhance performance at different angles of attack) as well as other objectives such as low radar observability and structural efficiency. No two surfaces of the intake leading edge are at the same angle and stagger and sweepback are combined so that all

lips are swept with respect to the airstream. Ducts for both aircrafts are long and mostly lined with radar absorbent materials. Also, radar observability is limited by duct planform curvature for the YF-22 (Figure 2.16 and Figure 2.17) and probably eliminated by both planform and side elevation curvature (duct offset) for the YF-23 (Figure 2.18 and Figure 2.19).

Figure 2.16 The YF-22A Raptor at flight from Hayles [17].

Figure 2.17 Planform view of the Lockheed YF-22A from Goldsmith and Seddon [13].

Figure 2.18 Front view of the YF-23A Black Widow 2 [18].

Figure 2.19 Planform view of the Lockheed YF-23A from Goldsmith and Seddon [13].

The influence of stealth consideration and the resulting blending of engine and airframe is emphasized by contrasting the YF-22A and YF-23A designs with a typical 1970s design, the F-15 aircraft (Figure 2.20) and its planform view in Figure 2.21. Although not strictly a strike fighter, the A-12, the US Navy's Advanced Tactical System (ATS) aircraft, carries the theme of engine/aircraft integration to its ultimate development. The almost completely

featureless thick delta flying wing is only interrupted by the top of a canopy that is not quite buried in the wing upper surface and by two nostril intakes situated just on the underside of the wing but very close to the wing leading edge. Interaction between engine airflow variation and wing performance could be prime importance of the aircraft's stability and control. Again, the change in design philosophy from that of the 1970s is emphasized by comparing the A-12 with the F-14 (Figure 2.22).

Figure 2.20 The F-15 with afterburners turned on [5].

Figure 2.21 F-15 intake/engine integration [5].

Figure 2.22 Comparison of the A-12 and F-14 in planform view from Goldsmith and Seddon [13].

2.4 Flow in Curved Pipes, Bends and Ducts

A chronological account on the subject of intake aerodynamics have been provided in Seddon and Goldsmith [2]. Sóbester [19] explains that intake aerodynamics is a multidisciplinary subject in which designers frequently find conflicting goals. Although our F-16 diffuser investigations represent only a part of the studies, the accounts provided in Sóbester [19] provide readers a comprehensive understanding on intake design and discuss the evolution of changing central themes.

Before the Second World War, Weske [20] and Henry [21] conducted experiments on a circular and elliptical duct with a single bend respectively. They concluded that the degree of curvature had an effect on the pressure recoveries of a diffuser. The degree of curvature had an effect on the pressure recoveries of the duct. The radius ratio (defined as the ratio of the radius of curvature of the centerline to the hydraulic diameter) is the most important design variable for the compound elbows affecting the pressure coefficient of the bend. The investigation of two factors; skin friction and flow separation; was conducted for the effects of pressure loss across a duct. Flow separation was a result of forces arising in airstream in the opposite direction of flow.

Although mentioned qualitatively, numerical data of the curvature was not presented.

The study of the circular S-ducts (or ducts having two bends) experimentally showed that adverse pressure gradients and non-uniform centrifugal force acting at the bends contributed to the formation of secondary flows from Vakili *et al.* [22]. Pressure probes placed at the end of the second bend indicated the formation of a low total pressure region resulting from the adverse static pressure gradients, as well as the higher centrifugal forces acting on the boundary layers of the fluid present at the bends. The circular duct geometry will be reproduced in the investigation as part of a validation analysis in a later section.

Consequently, the circular S-duct geometry became an excellent platform for validation of a turbulence code. Harloff *et al.* [23][24] and Wellborn *et al.* [25] performed a full 3-D Navier-Stokes equation numerical analysis solved with algebraic and two-equation turbulence models. They compared experimental and simulated results of a diffusing S-duct. The total and static pressure contours at selected transverse stations around the S-duct yielded good agreement. Contra-rotating vortices were found embedded in the lower part on the exit and expelled low-velocity fluid toward the center of the cross section. However, the simulated models under predicted the length and angular extent of the boundary layer separation near the second bend.

The use of computers initiated novel techniques to reduce flow separation. addition of external geometries was modified to the ducts to increase airflow quality. Lefantzi and Knight [26] added a bump near the end of the first bend to capture the secondary flow formation, resulting in lowered distortion coefficients conducted simulations on both coarse and fine grids simulations. However, the coarse grid simulations did not result in the formation of counter rotating vortices, as seen by previous researchers. The study of the effects of adding vane effectors on the M2129 circular S-duct was conducted by Mohler [27]. Again, the locations of the formations of secondary

flows enabled the modified circular S-duct as a testing bed for turbulence modeling and prediction.

The intakes of a modern fighter aircraft did not have the shape uniformity as that of a circular S-duct. As shown in the previous sections, an intake normally has an elliptical or rectangular inlet that transcends to a circular outlet. Also, the intake could possess single or multiple offset bends. The issues of changing cross-sectional shape as well as centerline curvature were studied in an F-16-like intake by Towne and Anderson [28] and Povinelli and Towne [29]. Towne and Anderson [28] performed simulations on an intake that had a single bend and an elliptical inlet that transcends to a circular outlet by. Povinelli and Towne [29] used another geometry that was rectangular at the inlet but also transcending to a circular exit but had multiple bends as done by. Also, in both studies, two other geometries to isolate the factors of changing cross-sectional area and centerline curvature were set up. The results show that the core flow in both geometries responded to the centrifugal forces governed by the meandering bends present. Vortex formation is formed in the intake because the low boundary layer flow is not able to dissipate quickly according to the static pressure gradients. In the Mach 0.9 flow, twin swirl flows are recorded at the outlet because of the symmetry of the F-16 intake. The factor of an intake centerline curvature was concluded to have more effect on the flow the transitioning cross section shape.

Ibrahim *et al.* [30] conducted similar analysis on the F-5E with two additional geometries, a fully elliptical and circular intake that had similar centerline curvature with the baseline F-5E intakes. Results showed that fully elliptical inlet produced the best pressure recovery.

A general study of comparing different inlet shapes with similar S-duct curvature was also conducted by Saha *et al.* [31]. They concluded that the Renormalized Group (RNG) k- ϵ model gave better prediction than the standard k- ϵ model. This was due to the fact that the superior capabilities of the RNG k- ϵ model of picking up transverse pressure gradients in ducts with streamline

curvature. The semi-circle intake was seen to be the best duct in terms of pressure recovery and elliptical inlet showed the least distortion at the exit.

The focus of the report will be towards the maneuverability of the intakes of the F-5E and F-16. It will be shown the flow features within the intakes are related to the effects of cross sectional transitioning and centerline designs of the intakes.

2.5 F-16 Intake Studies

Various studies pertaining to the development of the F-16 have been done. The prototype version YF-16 used a Normal Shock Inlet (NSI) because of its good airplane performance and low cost as conducted by Hawkins [32]. A simple normal-shock inlet was seen with the optimum inlet for the YF-16. The placement of the NSI at the underbelly of the fuselage provided shielding to enhance maneuverability. The aircraft forebody is a powerful straightener which is significant in maintaining high inlet pressure recovery and low distortion at high angles of attack (AOA or α). No degradation in pressure recovery was found up to $30^\circ \alpha$ and stable operation existed to $40^\circ \alpha$ subsonically. It would be seen in the simulations conducted in our studies that this conclusion was also confirmed.

In the face of changing engine airflow requirements, a study was conducted with the motive of providing a 'common' engine inlet that would provide optimum performance by Hagseth [33]. The study encompassed a complete inlet design, analysis and performance achieving a cost effective minimum design impact on the F-16 production sequence. The Modular Common Inlet Concept Design (MCID) intake would allow flexibility in the selection of future engines for the F-16 by efficiently providing performance over the large range of airflows Hagseth [33].

Other improvements of the F-16 intake aerodynamics included the implementation of a fixed double-ramp inlet with a throat slot bleed system was implemented on the F-16/J79, a derivative of the F-16 A/B conducted by

Hunter and Cawthon [34]. The improvements were seen in the supersonic region (Mach 2.0), with spill drag decrease over 60% and performance increase of about 20%. Also, at Mach 0.6 a bypass valve was activated to direct air into the compressor face.

The development of a practical variable-geometry inlet design had also been in the development for the F-16 through a long term program that began at the outset of the YF-16 prototype conducted by Hunter and Hawkins [35]. The concept was using conformal shaping combined with conventional external compression inlet design technique to enhance airplane integration and performance. This modification was well matched with the Pratt and Whitney (P&W) F100 engine. Results had shown a significant increase in performance above Mach 1.4, increased accelerations, higher sustained load factors, turn rates persistence as well as Mach 2.2 plus capability.

The importance of adding a pre-entry separation has also been widely discussed in Seddon and Goldsmith [2]. Stroud [36], Stitt and Wise [37] and Saha *et al.* [38]. The fuselage in this study was simulated as a flat plate with geometries similar to that used by Hawkins [32]. Details of the fuselage and intake geometry of the F-16 will be discussed in a later section.

2.6 Taguchi and ANOVA analysis

In the final part of this study, the Taguchi Method is used to evaluate the need to understand airflow quality through the intake via fighter aircraft maneuvers; three factors associated directly with aircraft maneuverability, that is, the Mach number (M), angles of incident (α) and yaw (β). Desirable air quality is defined as having high pressure recoveries as well as low distortion

The use of Advanced Statistical Techniques (AST) such as Design of Experiment (DoE) and Taguchi Method (TM) has been fairly common in the industrial engineering fraternity. DoE is a powerful tool for process optimization and studying the product and process behaviour. It is used for discovering the set of process variables that are influential on the process output

and the levels at which these variables should be kept to optimize process performance. TM, on the other hand, are used for maximizing product and process robustness by reducing variation due to undesirable external disturbances which cannot be controlled during actual production conditions in Montgomery [39].

Amongst the applications of AST in manufacturing organizations are the injection molding, welding and chemical processes. In these processes, the main purpose of employing these techniques is to tackle process quality problems in real-life situations. However, not all AST implications are used in the industrial engineering field.

The helicopter experiments conducted by Antony and Antony [40][41] identified factors which affect the time of flight and to determine the optimal factor levels which will maximize the time of flight. A two-level fractional factorial study with 16 experimental runs is selected. The advantage of a factorial design is that it allows the independent estimate of the main factor effects without the need of going through the maximum number of two-level factorial runs. Effects of the interactions of factors are also studied. Analysis of Variance (ANOVA) is carried out to identify the significant and main interaction effects in Antony and Antony [40]. TM was also carried out on the paper helicopter experiment; time including uncontrollable factors called noise in Antony and Antony [41]. Signal-to-noise (SNR) calculations are conducted using the larger-the-better quality characteristics, as the yield measured is actually the flight time.

Other recent studies include the applications of the statistical analysis in identifying tumours by Ng and Ng [42]. ANOVA and TM are conducted on these 3 two-level factorial design. As with the previous studies, the aim is to maximise the surface potential from the tumour size and location relative to the tumour conductivity. Again this uses the larger-the-better concept that was implemented in the previous study.

Perhaps a more interesting design experiment involved the optimizing of the CFD solver itself in Anderson *et al.* [43]. As of the previous two works, numerical methods were used to obtain the results needed. This study involved TM to identify trends in the solver's performance. The relative effects of turbulent model, Reynolds number, angle of attack and wall spacing (y^+) (factors) determines the level of error in the lift and drag coefficients (yield/result). Again, two-level factors are used in this study. A response table of the results was also set up. The results gathered help to pinpoint flow regions where non-linear interactive effects may be most important. In the presence of noise, TM can highlight solver parameters and flow regimes that may be insensitive to changes in the calculation environment, making them good candidates for off-design calculations. Other group of factors in CFD experiments may yield further conclusions about improving solver performance or behaviour over wide regions.

The study of the role of DoE in managing inlet air flow had also been conducted in Anderson *et al.* [43]. The goal of the study was to maintain optimal inlet performance over a range of mission variables and to explore the use of AST in understanding the management of inlet air flow. The M2129 inlet S-duct was chosen as the geometry of study. The factor variables were the number of vane effector units, height, length, inlet throat Mach number and angle of incident. Optimum flow is defined in three mission strategies, namely (1) Maximum Performance, (2) Maximum Engine Stability and (3) Maximum High Cycle Fatigue (HCF) Life Expectancy. The Maximum Performance mission minimized the inlet total pressure losses; the Maximum Engine Stability mission minimized the engine face distortion while the Maximum (HCF) Life Expectancy mission minimized the mean of the first five Fourier harmonic amplitudes.

Numerical parametric studies at the compressor region using TM have also been conducted by Wu *et al.* [11] and Ng *et al.* [44]. In both studies, 3 three-level factors were used to measure flow with the least distortion at the

compressor face. This results in an $L_{27} (3^3)$ orthogonal array design. Using the Taguchi off-line quality control method, the parameters; namely inlet x-axis distorted velocity coefficient, incidence angle and drag-to-lift coefficient; are ranked according to the degree of influence in distortion. Further case studies were also conducted by varying only one factor with multiple levels.

The study to maintain optimal inlet performance over a range of mission variables and to explore the use of AST in understanding the management of inlet air flow was conducted on the M2129 inlet S-duct. The factor variables were the number of vane effector units, height, length, inlet throat Mach number and angle of incident. Optimum flow is defined in three mission strategies, namely (1) Maximum Performance, (2) Maximum Engine Stability and (3) Maximum High Cycle Fatigue (HCF) Life Expectancy. The Maximum Performance mission minimized the inlet total pressure losses; the Maximum Engine Stability mission minimized the engine face distortion while the Maximum (HCF) Life Expectancy mission minimized the mean of the first five Fourier harmonic amplitudes.

The investigation aims to improve the previous statistical analysis done on the F-5E intake as conducted in Ibrahim *et al.* [30], as well as add on to the detailed analysis done on the maneuverability of the F-16 intake by as conducted in Ibrahim *et al.* [45][46]. The use of CFD in this study is motivated by the guidelines stated in Towne [47]. Detailed explanations of the guidelines are added in Appendix A.

Chapter 3 NUMERICAL METHODOLOGY

3.1 The Aerodynamic Duct Concept

A duct ‘captures’ a certain streamtube of air and divides the airstream into an internal flow and an external flow, as indicated in Figure 3.1 by Seddon and Goldsmith [2]. The external flow has the task of preserving the good aerodynamics of the airframe, as is normally a function of the drag force associated with the aircraft. The internal flow has the task of feeding air into the engine. The basic shape of the duct is important as an engine requires air at moderate subsonic speeds of about Mach 0.4 to 0.5. The front of the duct is essentially the diffuser, which is the focus of the study. The rear part of the duct is then convergent, simulating in essence the engine nozzle system.

Figure 3.1 Axial locations of flow stations for a complete engine nacelle. The symbols ∞ , c , f and e refer to upstream infinity, duct entrance, engine face and exit respectively.

Mass flow rate is a fundamental variable required in the understanding of internal flow. It is defined as:

$$\dot{m} = \rho VA \quad (3-1)$$

where ρ is the air density, V the assumed uniform velocity and A the cross-sectional area at a given station of the internal flow. Assuming no offtakes or additional inputs such as bleeds or bypass doors, by the principle of continuity, mass flow rate is constant throughout the internal flow streamtube, i.e.:

$$\rho_{\infty} V_{\infty} A_{\infty} = \rho_c V_c A_c = \rho_f V_f A_f \quad (3-2)$$

The term full flow is applied to a condition in which the stream tube arrives undisturbed at the entry station c, that is:

$$\text{full flow} = \rho_c V_c A_c = \rho_{\infty} V_{\infty} A_c \quad (3-3)$$

The capture flow ratio (CFR), sometimes called the capture area ratio, is the actual flow, defined by the streamtube area at upstream infinity, related to full flow, thus:

$$\text{CFR} = \text{flow ratio} = \frac{\rho_{\infty} V_{\infty} A_{\infty}}{\rho_{\infty} V_{\infty} A_c} = \frac{A_{\infty}}{A_c} \quad (3-4)$$

The CFR is a parameter that can be used to quantify engine demand. When the inlet can accept the mass flow rate of air required to position the terminal shock just inside the cowl lip, the inlet is said to be running at ‘critical’ condition (Figure 3.2a). CFR is equal to unity at this condition. When the inlet is not matched, the normal shock moves upstream. This is shown in Figure 3.2b and is called the ‘subcritical’ operation. The fraction of air spillage also increases and is associated with ‘spillage drag’, due to the change in momentum of the spilled air and the pressure forces on its streamtube. This results in a CFR value of greater than 1.

For subsonic speeds it is less meaningful but is retained, usually as a cowl highlight area. When considering low speed subsonic cases the CFR is greater than one as the intake needs to draw air in from an area greater than the highlight area (i.e. the intake is not being supplied with sufficient air to meet demand). High speed supersonic intakes tend to have a CFR less than one as the intake can draw in air from an area less than the highlight area (i.e. the intake is being supplied with more air than it requires).

For an intake in the supersonic stream, when CFR is less than one, which means that the intake is oversupplied with air, a shock wave stands out

ahead of the intake, as shown in Figure 3.2a. On the centre line the shock is normal, with subsonic Mach number immediately behind it. Some of the air spills outside of the intake, and the mass flow actually draw inside of the intake. The shock continuously approaches the intake entry as CFR is increased. When CFR is equal to one (as shown in Figure 3.2b), the intake is being supplied with as much air as it requires and the normal shock lies across the entry plane with the subsonic Mach number after the shock. If the air requirement is increased further, i.e. CFR greater than one (Figure 3.2c), there is not sufficient air to meet demand and the shock is sucked inside. Flow ratio remains unaltered and governed by the entry stream-tube. As the shock travels along the diffuser its Mach number increases, therefore the loss of total pressure also increases. Equilibrium is reached when the total pressure downstream of the shock has fallen sufficiently to compensate for the increase in area and leave the mass flow unchanged

Figure 3.2 Flow through an intake in a supersonic stream (a) $CFR < 1$; (b) $CFR = 1$; (c) $CFR > 1$ by Goldsmith and Seddon [13].

The condition of maximum pressure recovery at maximum flow (Figure 3.2b) is known as the ‘critical’ point or full flow. Operation at lower flow ratio (Figure 3.2a) is termed ‘sub-critical’ operation, while operation at maximum flow but lower pressure recovery (Figure 3.2c) is termed ‘supercritical’ operation.

3.2 COMSOL PDE Calculator

An overview of the uses of COMSOL [9] [10].is explained and shown in Figure 3.3. It has to be stressed, however, that solving the physics problem does not follow the steps in sequence. Rather, an iterative approach has to be taken. Definition of the problem is first needed on starting the program. The physics setting enables users to set up governing equations as well as boundary conditions. Partial differential governing equations are categorized into three forms: General, Coefficient, and Weak forms. A user also has the option to use the pre-defined physics settings available.

Figure 3.3 Overview of the uses of COMSOL.

Next, the geometry is drawn. Again, for this case, a 3-Dimensional drawing of the duct is needed. COMSOL enables a live connection with SOLIDWORKS so that changes can be done to the drawing in real time. The duct is lofted from curves of given coordinates at several transverse locations. The boundary conditions available are divided into two forms: Generalized Neumann and Dirichlet. Again there are several pre-defined boundary conditions in the provided physics settings.

In 3-D mode, COMSOL allows a user to generate either tetrahedral or prismatic meshes. Both these forms are unstructured. The choice of the meshing types depends on the shape of the geometry drawn.

Once the meshes are done, the problem can then be solved. A direct or iterative solver can be chosen, depending on the complexity of the problem. The size of the problem is related to the resultant degrees of freedom (DOF) generated. Generally, in 3-D mode, an iterative solver is chosen.

A turbulence model selected in COMSOL was similar to that of the example of turbulent flow through a 90° pipe bend by Idelchik [48]. The example compares results obtained using the k - ϵ turbulence model with experimental data and contains detailed descriptions of how to make the modeling as accurate as possible. The difference between the present studies and the 90° pipe bend was the non-dimensionality with respect to the inlet diameter of the 90° pipe bend. The 90° pipe bend was also simulated at low speeds, enabling the incompressibility character to exist. The present studies focuses on the fully compressible form of the Navier-Stokes equations.

3.2.1 Governing Equations

The present simulations have been performed using COMSOL [9] [10] based on the finite elements approach. The fully compressible model is expressed by coupling the continuity and momentum equations in the k - ϵ turbulence mode together with the energy equation obtained in the conduction and convection modes, as suggested by Smith [49]. The model is used on the circular S-duct, F-5E and F-16 intake. The k - ϵ physics settings were modeled using the General form. The modeling constants are included in Table 3.1.

Steady-state condition is assumed to be existing in the system, similar to the CFD works conducted by Saha *et al.* [38], Patel *et al.* [50] and Bharani *et al.* [51] where the effects of maneuver are studied on the bifurcated Y-duct. Subsonic inlet design are also studied using the steady-state assumption by

Zhang *et al.* [52]. Represented in tensor form in COMSOL, the governing equations are shown below:

Continuity equation:

$$\nabla \cdot (\rho \mathbf{u}) = 0 \quad (3-5)$$

Momentum equation

$$(\rho \mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \nabla \cdot \left[\boldsymbol{\tau} - \left(\frac{2\rho k}{3} \right) \mathbf{I} \right] + \mathbf{F} \quad (3-6)$$

Energy equation:

$$C_p (\rho \mathbf{u} \cdot \nabla) T = (\nabla \cdot (k + k_T) \nabla T) - \frac{T}{\rho} \frac{\partial \rho}{\partial T} (U \cdot \nabla p) + \boldsymbol{\tau} : \mathbf{S} + Q \quad (3-7)$$

Turbulence energy equation:

$$(\rho \mathbf{u} \cdot \nabla) k = \nabla \cdot \left[\left(\eta + \frac{\eta_T}{\sigma_k} \right) \nabla k \right] + \left[\eta_T P(\mathbf{u}) - \frac{2}{3} \rho k \nabla \cdot \mathbf{u} \right] - \rho \varepsilon \quad (3-8)$$

Dissipation equation:

$$(\rho \mathbf{u} \cdot \nabla) \varepsilon = \nabla \cdot \left[\left(\eta + \frac{\eta_T}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \left(C_{\varepsilon 1} \frac{\varepsilon}{k} \right) \left[\eta_T P(\mathbf{u}) - \frac{2}{3} \rho k \nabla \cdot \mathbf{u} \right] - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad (3-9)$$

where ρ is the density, $\mathbf{u}$ represents the velocity vector, p is pressure, η is dynamic viscosity, η_T is turbulent viscosity, $P(\mathbf{u})$ represents the rate of turbulence production, $\mathbf{I}$ represents the identity matrix, $\mathbf{F}$ is the volume force vector, C_p is the specific heat of capacity at constant pressure, T is the absolute temperature, Q is the heat source $(= -\frac{T}{\rho} \frac{\partial \rho}{\partial T} \left(\frac{\partial p}{\partial t} + U \cdot \nabla p \right))$, k and k_T are the thermal and turbulent thermal conductivity respectively.

The density term ρ appears in the governing equations and is related to the ideal gas equation.

$$\rho = \frac{p M_w}{RT} \quad (3-10)$$

where M_w and R are the molecular weight and universal gas constant of air.

The rate of turbulence production is represented as

$$P(\mathbf{u}) = \nabla \mathbf{u} : (\nabla \mathbf{u} + \nabla \mathbf{u}^T) - \frac{2}{3} \nabla \cdot \mathbf{u} (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \quad (3-11)$$

The viscous stress tensor τ is known as

$$\tau = 2(\eta + \eta_T) \mathbf{S} - \frac{2}{3} (\eta + \eta_T) (\nabla \cdot \mathbf{u}) \mathbf{I} \quad (3-12)$$

The modulus of mean rate of shear stress tensor $\mathbf{S}$ is defined as

$$\mathbf{S} = \frac{1}{2} (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3-13)$$

The operation ‘:’ denotes a contraction between tensors as defined by

$$\mathbf{a} : \mathbf{b} = \sum_n \sum_m a_{nm} b_{nm} \quad (3-14)$$

Hence the second term in the right side of the energy equation can be expressed as

$$\tau : \mathbf{S} = (\eta + \eta_T) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \frac{\partial u_i}{\partial x_j} \quad (3-15)$$

The turbulent thermal conductivity term k_T is defined as

$$k_T = \frac{C_p \eta_T}{Pr_T} \quad (3-16)$$

where C_p and Pr_T is the turbulent Prandtl number with a value of 0.85.

The turbulent viscosity term η_T can be empirically expressed as

$$\eta_T = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3-17)$$

Table 3-1 The k- ε modeling constants used.

Model	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	C_μ	σ_k	σ_ε
k- ε	1.44	1.92	0.09	1.0	1.3

The system was assumed to initiate at sea level pressure conditions with total pressure of 101325 Pa and temperature of 298K. The CFD simulations for

the F-5E and F-16 intake were conducted inlets having differing velocity quantities in the x, y and z-axis, resulting in changing Mach number, AOA and AOS. An integration coupling variable was also imposed on the inlet and outlet boundaries as used by Smith [49] to conserve the mass flux into the duct. The inflow boundary also defined the turbulent kinetic energy as

$$k = \frac{3}{2} (VI_s)^2 \quad (3-18)$$

where V is the resultant flow velocity.

The turbulent intensity scale, I_s , is defined as

$$I_s = 0.16 (Re)^{-1/8} \quad (3-19)$$

where Re is the Reynolds number based on the hydraulic diameter, which is defined as the semi-major axis of the inlets for the F-5E and F-16 and the diameter for the circular S-duct.

And the turbulent dissipation rate is expressed as

$$\varepsilon = C_\mu^{3/4} \frac{k^{3/2}}{l} \quad (3-20)$$

$$\text{where } l = 0.07D \quad (3-21)$$

and D is hydraulic diameter.

A open boundary condition was specified as the outflow condition. The equation at this outflow boundary is as follows:

$$n \cdot \nabla k = 0 \quad (3-22)$$

$$n \cdot \nabla \varepsilon = 0 \quad (3-23)$$

The wall boundaries were defined as that of the 90° pipe bend by Idelchik [48] which uses the behaviour of the log wall function. The velocity parallel to the wall, U , is described as

$$U^+ = \frac{U}{u_T} = \frac{1}{\kappa} \ln \left(\frac{\delta_w}{l^*} \right) + C^+ \quad (3-24)$$

The friction velocity, u_T , is defined by

$$u_T = \sqrt{\frac{\tau_w}{\rho}} \quad (3-25)$$

The value κ , known as the Von Karman's constant, is set at 0.42. The constant C^+ is a smooth wall constant and set as 5.5. The viscous length l^* is defined by

$$l^* = \frac{\eta}{\rho u_T} \quad (3-26)$$

The logarithmic wall functions are valid for values of δ_w^+ between 30 and 300.

The viscous units δ_w^+ is related to δ_w by

$$\delta_w^+ = \frac{\delta_w \rho C_\mu^{0.25} k^{0.5}}{\eta} \quad (3-27)$$

A universal wall temperature distribution is valid at high Reynold's numbers:

$$T^+ = \frac{\text{Pr}_T}{\kappa} \ln \left(\delta_w^+ \right) + \beta \quad (3-28)$$

where the β is a model constant set to 3.27. The boundary conditions of the different duct geometries used in the study are shown in the next chapter, where the validation of the ducts is examined.

3.2.2 Solution Scheme

The built-in solvers schemes in COMSOL [9] [10] are generally divided into two groups: Direct or Iterative. The direct solver solves a linear system by Gaussian elimination. This stable and reliable process is well suited for ill-conditioned systems. This solver requires less tuning and is often faster than the iterative solvers. With the present simulations conducted on a Pentium 4 Core 2 duo PC with 2GB RAM running on a 32-bit platform, a DOF limit of

approximately 20,000 can only be solved. Iterative solver requires proper preconditioning and can solve with a DOF limitation of approximately 100,000.

The direct solver chosen for our simulations is the PARDISO solver, while the iterative solver is the Generalized Mean Residual (GMRES) solver coupled with the VANKA preconditioner by Saad and Schultz [53].

3.2.2.1 Using the Segregated Solving Approach

This method solves for only a subset of the dependent variables at a time and hence requires less memory than a fully coupled approach. Smaller equation systems also have the advantage since they are better conditioned than large equation systems. A segregated approach therefore stabilizes the solving technique more than a fully coupled approach.

A group has to contain variables that, loosely speaking, “belong together.” In this case, modeling the mass transport of three species requires the segregating of the velocities and the pressure in one group and the concentration variables in another group. Similar solving methodology have been done by Ignat *et al.* [54] on 2-D NACA airfoils and Utne [55] on the 1-D steady state channel flow as well as the 2-D backward facing step with success.

3.3 FLUENT Solver Scheme

In the investigations of F-5E intake flows, the results of FLUENT 6.2 is used to compare with that obtained in COMSOL as conducted in Wu *et al.* [56]. Linear low-Re two-equation models seem to offer the best balance between accuracy and computational cost, but since they apply the Boussinesq approximation for the Reynolds stress tensor, they are not able to capture effects arising from normal-stress anisotropy. Second-moment closures offer a more exact representation of the Reynolds stresses but require longer computing times and careful numerical implementation for obtaining stable numerical solutions. Reynolds-stress models have been used in the past to

investigate shock/boundary layer interaction. These studies showed that in certain cases second-moment closures may provide better results than linear models, but in other cases the results are inconclusive.

A point implicit (Gauss-Seidel) linear equation solver is used in conjunction with an Algebraic Multi-Grid (AMG) method to solve the resultant scalar system of equations for the flow variable. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. The iteration consists of the following steps:

3.4 Performance Parameters

3.4.1 Calculations for Total Pressure Coefficient, Distortion Coefficient (DC) and Pressure Recovery

The total pressure coefficient used in the intake stations is measured by:

Total pressure coefficient

$$= (P_{total, local} - P_{static, inlet boundary}) / (P_{dynamic}) \quad (3-29)$$

The performance parameters used to measure the duct efficiency are the distortion coefficients and also pressure recovery. The measurements are taken at a plane near the outlet of the duct, also known as the Aerodynamic Interface Plane (AIP).

Total pressure distortion, also known as the distortion coefficient (DC), is measured by:

$$DC(\phi) = \max \frac{[\bar{p}_0 - \bar{p}_0(\theta)]}{\bar{q}} \quad (3-30)$$

where

$$\bar{p}_0 = \frac{\int_A p_0 dA}{\int_A dA},$$

$$\bar{p}_0(\theta) = \frac{\int_\theta p_0 dA}{\int_\theta dA},$$

$$\bar{q} = \frac{\int_A q dA}{\int_A dA}$$

and $\int_\theta$ indicates an integration over a pie-shape domain of angular size θ and $\int_A$ indicates an integration over the entire cross-sectional area A. $\bar{p}_0$ and q refers to the total and dynamic pressures respectively.

For this present paper, the θ value of 60° is taken, corresponding to the well known DC (60) parameter. The rationale for using the DC (60) parameter is its extensive usage in the British aerospace engineering field, as explained in

Chapter 2. The maximum is taken over all possible locations of the pie-shaped domain. An illustration of this measurement is shown in Figure 3.4 Measurement of the distortion coefficient (DC) factor. The circular AIP boundary is investigated with θ taken as 60o..

Pressure recovery, ψ , is defined as:

$$\psi = \frac{\bar{p}_{0_{AIP}}}{\bar{p}_{0_{inlet}}} \quad (3-31)$$

Figure 3.4 Measurement of the distortion coefficient (DC) factor. The circular AIP boundary is investigated with θ taken as 60°.

3.5 Numerical accuracy

The issue of numerical accuracy of the result encompasses the validation and verification of the solution. Validation and verification procedures are the primary means of assessing accuracy and are tools with which we build confidence and credibility in computational simulations as defined in Oberkamp [57].

Validation is defined as the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model. Validation deals with the physics of the problem.

It shows evidence on how a mathematical model is solved. Verification, on the other hand, is defined as the process of determining that a model implementation accurately represents the developer's conceptual description of the model and the solution to the model. Verification deals with the mathematics of the code or the solution. It provides a numerical gauge of how accurate a computer model is solved.

The motives of conducting validation tests are to characterize and minimize uncertainties and errors in the computational model and experimental data and increase confidence in the quantitative predictive capability of the computational model. In modelling validation experiments, an experiment is designed and executed to quantitatively estimate a mathematical model's ability to simulate a physical system or process. A validation experiment is designed to capture the relevant physics, all initial and boundary conditions, and auxiliary data. All important modeling input data are measured in the experiment and key modeling assumptions understood. Characteristics and imperfections of the experimental facility is also be included in the model.

Two groups form a subset of code verification: numerical algorithm verification and software quality assurances practices. In solution verification, three aspects are important: the verification of the input data, numerical error of the solution and verification of the output data. In verification of input data, proper input files, grids, physical and material data are used. For output, the steps for post-processing are checked.

The mathematics involved in this section comes mostly from measurement of the numerical error of the solution. Discretization error is a form of the numerical problem that arises due to the mapping of PDEs to discretized equations. An example of a discretized error is the truncation error that is present when used in solving linear PDEs using that Taylor series expansion. For non-linear PDEs, the relation between the discretized and truncation error is not that obvious. Discretization of the boundary conditions

can dominate the numerical accuracy if the order of accuracy is less than the interior conditions.

Approach in estimation of the discretization error is via two methods: a priori and posteriori error estimation. Error estimation is done before the numerical solution is computed in the a priori method. This method is used in the truncation error analysis in the finite volume and finite difference schemes. In the finite element scheme, the interpolation theory can be estimated using this method. This method is not practical though because the magnitude is only known within an unknown constant.

Chapter 4 **VALIDATION / VERIFICATION ANALYSIS**

4.1 Viewgraph Norm

The validation and verification analysis were implemented on two geometries: the circular S-duct as well as the F-16 intake. The methods used for measuring the accuracy of computational results have been either qualitative or semi-quantitative defined in Oberkamt [57]. The viewgraph norm formed, as shown in Figure 4.1 Viewgraph norm qualitative and quantitatively compares experiment and computational results. and used in this study, is semi-quantitative as it allows the user to visualize both the plots and the numerical results within the total pressure isobars.

Figure 4.1 Viewgraph norm qualitative and quantitatively compares experiment and computational results.

The a-posterior error estimation is done after a numerical solution was obtained. It could be used in a finite element scheme as an extrapolated error estimation method. Examples of these methods are the Richardson's extrapolation (h-extrapolation), order extrapolation (p-extrapolation) and the Grid Convergence Index (GCI) by Celik [58] which is conducted in this study. It is shown that the error estimation is within normal engineering practices of $\pm 5\%$.

4.2 Grid Convergence Index

A 32-bit platform that is only capable of running a limited number of grid points. The complexity of a simulation is determined by the degrees of freedom (DOFs) that the system possesses. The DOF is related to the sparsity of the Jacobian matrix. The coupling of the Navier-Stokes equations to form the turbulence models in COMSOL presents high sparsity in the Jacobian Matrix. A VANKA presmoothing is needed together with the FGMRES iterative solver to suit this condition.

The Grid Convergence Study (GCI) presented in Celik [58] is used as verification tool in the current CFD research. The steps are presented as follows:

Define a representative cell, mesh or grid size h . For a 3-D case,

$$h = \frac{1}{N} \left[\sum_{i=1}^N (\Delta V_i) \right]^{1/3} \quad (4-1)$$

where ΔV_i is the volume of the i^{th} cell, and N is the total number of elements used in the simulations.

Select three significant sets of grids and run simulations to determine the values of the key variables, such as pressure, important to the objective of the simulation study. The calculation of pressure recovery is used in this study. It is desirable for a grid refinement factor, $r = h_{coarse}/h_{fine}$ be greater than 1.3. This grid refinement study has to be done systematically and could be done even if the grid is unstructured. The use of geometrically similar cells is also preferred.

Let $h_1 < h_2 < h_3$, $r_{21} = h_2 / h_1$ and $r_{32} = h_3 / h_2$ and calculate the apparent order, p of the method using the expression

$$p = \frac{1}{\ln(r_{21})} |\ln|\varepsilon_{32}/\varepsilon_{21}| + q(p)|, \quad (4-2)$$

$$q(p) = \ln\left(\frac{r_{21}^p - s}{r_{32}^p - s}\right), \quad (4-3)$$

$$s = 1 \times (\varepsilon_{32}/\varepsilon_{21}), \quad (4-4)$$

where $\varepsilon_{32} = \phi_3 - \phi_2$, $\varepsilon_{21} = \phi_2 - \phi_1$ denoting the solution on the k^{th} grid. Φ is taken as pressure recovery. Note that $q(p) = 0$ for a constant r . Negative values of $\varepsilon_{32} / \varepsilon_{21}$ indicates an oscillatory convergence. This shows that if the occurrence of the oscillatory solutions happens at a region where the solution are observed at the grids being in the asymptotic range, the occurrence should be taken as a sign of unsatisfactory calculation. If $\varepsilon_{32} = \phi_3 - \phi_2$ or $\varepsilon_{21} = \phi_2 - \phi_1$ is close to zero, additional grid refinement may be performed; if not the ‘exact’ solution may be reported as being obtained.

Calculate the extrapolated values from

$$\phi_{ext}^{21} = (r_{21}^p \phi_1 - \phi_2) / (r_{21}^p - 1), \quad (4-5)$$

Similarly, calculate ϕ_{ext}^{32}

Calculate and report the following error estimates, along with the apparent order p :

Approximate relative error:

$$e_a^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right|, \quad (4-6)$$

Extrapolated relative error;

$$e_{ext}^{21} = \left| \frac{\phi_{ext}^{12} - \phi_2}{\phi_{ext}^{12}} \right|, \quad (4-7)$$

The fine grid convergence index is then defined as:

$$GCI_{fine}^{21} = \frac{1.25 e_a^{21}}{r_{21}^p - 1} \quad (4.8)$$

4.3 Validation and Verification

4.3.1 Circular S-duct

The geometry used in Vakili *et al.* [22] to determine secondary flow separation was reproduced in COMSOL to determine the accuracy of the k- ϵ turbulence model implemented. The dimensions and boundary conditions are shown in Figure 4.2. The 30°-30° circular cross section S-duct is made from two symmetric sections. The duct inside diameter D is 0.1651m (6.5 inches) with the mean radius of curvature of $R = 0.8382\text{m}$ (33.0 inches). The entrance of the S-duct is connected to a pipe length which is 76.2cm or about 4.6 times the diameter. This section is required for the development of a turbulent boundary layer to the desired thickness. At the exit, the straight pipe section of 152.4 cm, about 10 times the diameter, is installed as shown in Figure 4.2.

Validation analysis included the qualitative comparison of the total pressure and velocity diagrams of the final station (station 6). This station represents the end of the second bend of the circular S-duct as indicated in Figure 4.2

Figure 4.2 The outlet of the second bend of the circular S-duct by Vakili *et al.* [22].

Figure 4.3 Normalised distance along duct centerline, S/D.

The experiment is conducted in a wind tunnel by Vakili *et al.* [22]. The inlet speed of the duct is chosen at Mach 0.6, resulting in a Reynolds number of 1.76×10^6 with respect to the diameter. The experimental measurements for flow angularity were done using a 5-port cone probe. The cone probe determines the local flow conditions, Mach number and total and static pressures. The cone probe was traversed along radial directions of the duct. Six stations were selected were to study localised total and static pressures. These six stations are shown in Figure 4.3. Ten azimuthal angles 20° apart were traversed at each station. Pressure was measured using differential transducers, calibrated to measure more data near the boundaries than at the centre. The total pressure in the tunnel fluctuated around 0.5psi during the runs, since this does not significantly influence the flow angularity, no corrections have been applied for this.

The free, tetrahedral mesh technique was used in COMSOL. The normal mode was selected and this resulted in the base mesh having 1662 mesh points and 5848 elements. The number of triangular boundary, edge and vertex elements were 1924, 280 and 20 respectively. This resulted in about 63948 degrees of freedom. The meshed generated is shown in Figure 4.4. The minimum meshing quality was 0.3235, higher than the required 0.3 that could affect the solution [10]. The element volume ratio was 0.0475.

A wall distance, δ_w , of 0.0007 was chosen, resulting in the δ_w^+ or y^+ value of between 30 to 300 to exist as the wall boundary condition. This was shown in Figure 4.5.

The problem was solved using a Direct PARDISO solver because of the segregation of dependent variables. This enabled a smaller number of DOFs to

be solved initially. The total time taken to solve was 1169 seconds or about 20 minutes for the k- ϵ model. The simulations were done on a Pentium 4 Core 2 duo PC with 2GB RAM running on a 32-bit platform. The order of iteration was set at 10^5 , with convergence defined at a value of 10^{-6} .

Figure 4.4 The circular duct is meshed with tetrahedral elements, resulting in 5848 elements and 63948 degrees of freedom.

Figure 4.5 δ_w^+ values for the circular S-duct.

4.3.1.1 Comparison with Experimental Results

Figure 4.6 shows a reasonable agreement of secondary flow separation between the experimental and numerical results. The correlating factors of adverse pressure gradient formation, boundary layer development and larger exposure of core flow to centrifugal forces contribute to the cause of secondary flow. The development of the boundary layer along the duct is directly influenced by the local pressure gradient. The adverse pressure gradient thickens the boundary layers. At the bends, the high kinetic energy flow, namely the inviscid core, is exposed to the larger centrifugal forces than the retarding viscous boundary layer. Hence, a pressure gradient in the transverse plane is resulted which is the main cause of the secondary flow.

Figure 4.6 Comparison of total pressure coefficient and velocity vector diagrams of Station 6 between experimental results done by Vakili *et al.* [22] and COMSOL results.

Figure 4.7, Figure 4.8 and Figure 4.9 show the static pressure coefficients along the normalized centerline distance for azimuthal angles of 0° , 90° and 180° respectively obtained from Harloff *et al.* [24]. The results show reasonable qualitative agreement between experimental and computational results. Separation occurs about 2.5m from the inlet of the duct. Adverse static pressure gradients are situated along the 2.5m distance, as shown in Figure 4.7

and Figure 4.9. The over and under-prediction of the results in Figure 4.7 and Figure 4.9 are mainly due to the coarse mesh being used in the simulation.

Figure 4.7 Comparison of static pressure coefficient vs. normalized centerline distance between experimental done by Harloff *et al.* [24] and COMSOL results along the 0° azimuth angle (top) of the duct.

Figure 4.8 Comparison of static pressure coefficient vs. normalized centerline distance between experimental done by Harloff *et al.* [24] and COMSOL results along the 90° azimuth angle (side) of the duct.

Figure 4.9 Comparison of static pressure coefficient vs. normalized centerline distance between experimental done by Harloff *et al.* [24] and COMSOL results along the 180° azimuth angle (bottom) of the duct.

The nature of the flow is further explained. A fully developed turbulent flow enters the first bend. As it enters, the inviscid core moves towards the inner bend and then towards the outer bend at the junction of the first and second bends. The flow enters the second bend with the inviscid core deflected towards the outer bend. In the second bend, the inviscid core again moved towards the inner bend, forming a low total-pressure flow at the outer portion of the bend. This formation is also attributed to the difference in static pressure gradients that existed in both the first and second bends, pushing the low velocity fluid towards the core.

Static pressure distributions identify the increase in static pressure toward the outer wall in the region of transition from the straight section to the first bend. This is the formation of the static pressure gradients which continues in the first half of the first bend. The pattern reverses at the start of the second bend; the static pressure increases towards the inner wall in the first bend and the outer wall in the second bend. Nearing the end of the second bend, the static pressure gradient is reduced in the flow downstream into the straight section pipe.

4.3.1.2 GCI Study of the Circular S-duct

Figure 4.10 Monotonic Convergence of pressure recoveries with increasing mesh elements.

Figure 4.10 shows the monotonic convergence of the pressure recoveries at the outlet boundary for the circular S-duct with increasing mesh elements. Increasing the mesh elements result in a converged total pressure value of about 0.968. A limited range of grid refinements were only done due to the 32-bit platform available as mentioned. The grid refinement factor, r , was from 1.032 to 1.147. ϵ was also calculated as zero, which could indicate that an ‘exact’ solution has been obtained. Hence two more additional refinements are made for further investigations. The results are shown in Table 4-1.

The numerical uncertainty for the fine-grid for N_3 , N_4 and N_5 solution are reported as 2.09 %, 1.54 % and 0.760 %; note these do not account for modeling errors. Hence the three grid analysis showed reasonably low numerical errors.

Table 4-1 Calculation of discretization error for the circular S-duct. The superscripts of the results correspond to the subscripts of the numerical mesh cases in the first row. Variables are defined in flowchart by Celik [58]

	ϕ = pressure recovery at the Aerodynamic Interface Plane (AIP), with monotonic convergence
N_1, N_2, N_3, N_4, N_5	2132, 3219 3608, 4491, 5848
$r_{21}, r_{32}, r_{43}, r_{54}$	1.147, 1.039, 1.076, 1.092
$\phi_1, \phi_2, \phi_3, \phi_4, \phi_5$	0.990, 0.984, 0.982, 0.981, 0.981
p	2.46
Φ_{ext}^{21}	1
e_a^{21}	0.671
e_{ext}^{21}	1.64
GCI_{fine}^{21}	2.09%
Φ_{ext}^{32}	0.995
e_a^{32}	0.118
e_{ext}^{32}	1.185
GCI_{fine}^{32}	1.5%
Φ_{ext}^{43}	0.988
e_a^{43}	0.12
e_{ext}^{43}	0.606
GCI_{fine}^{43}	0.76%

4.3.2 F-5E Intake

Several coordinate points were obtained from the RSAF and were used to draw curves (stations) along the intake. The curves were then drawn and lofted in SOLIDWORKS to obtain the 3-D geometry of the intake. It should be noted that adequate guide lines were needed to obtain a proper lofted geometry for export to COMSOL. The SOLIDWORKS and imported COMSOL F-5E geometries with boundary conditions are shown in Figure 4.11. The tolerance

for import was at 10^{-7} m to enable fine surface transitions to be imported into COMSOL.

The F-5E duct has an elliptical inlet that transcends to a circular outlet. Curvatures along the x and y-axis are present to conform to the body of the aircraft. Using similar conditions as the circular S-duct, the F-5E intake geometry is studied.

Figure 4.11 The SOLIDWORKS and COMSOL F-5E geometry. Note to export geometry to COMSOL, adequate guide curves are required in the lofting process.

4.3.2.1 Comparison with FLUENT Simulations

Due to the lack of experimental data, results for the F-5E intakes are compared with the analysis done by FLUENT. The simulations conducted by FLUENT have 210980 hexahedral cells, compared to 8289 tetrahedral elements used in COMSOL. The large difference in mesh refinement is due to different computer specifications used. FLUENT was running on a supercomputer while COMSOL could only be run on 32-bit 2GB system. The FLUENT results required five hours of computing time for convergence. COMSOL convergence and iteration settings were kept similar to that used in the Circular S-duct and required one hour for results to be converged.

Five total pressure contour plots were taken at different stations along the duct. The stations are at 0.5m, 1.5m, 2.5m, 3.5m and 4.5m along the y-axis

of the intake and are termed stations 1,2,3,4 and 5 respectively. Figure 4.12 illustrates the five stations.

Similar vortex formation is not seen for the flow simulations done for the F-5 duct. The dimensional results for the Mach 0.6 inlet obtained for both FLUENT and COMSOL compared in Figure 4.13 suggests a good agreement with the validation analysis done in the prior section. Regions of low total pressure were present at the top left side of the duct, and this character remained unchanged throughout the duct. At the outlet boundary (4.5m, Figure 4.13(e)), it can be seen that there is a non-axis symmetry formation, a character which is undesirable for the intakes of engines.

The results from COMSOL compared to the FLUENT overestimate the size of the core flow in the F-5E duct. This can be accounted the difference in grid size used in the simulation. Finer grid cells used FLUENT are able to account for gentler total pressure gradients, hence the boundary layer separation and migration from stations 1 to 5 can be captured. The low total pressure region migrates from the top left side of the station 1 to the top of station 5. This migration is not captured in the results obtained by COMSOL.

Although the details of the results by the two programs are largely different at the boundaries at the transverse stations, the global physics of the flow simulated are the same. Adverse secondary flow phenomena is not captured by both FLUENT and COMSOL. The advantage in using COMSOL comes with the savings of computer costs and time.

Figure 4.12 Five stations on the port side F-5E intake.

a) 0.5m

b) 1.5m

Figure 4.13 Total pressure isobars analysis of the F-5 port side duct. The outlet boundary shows non axis-symmetry character, an undesirable compressor intake feature.

4.3.2.2 GCI Study of F-5E Duct

Similar GCI steps used for the circular S-duct were implemented on the F-5E. A limited range of grid refinements were only done due to computing cost imposed on the 32-bit system. The grid refinement factor, r , was from 1.083 to 1.259. The value, ε , was also calculated as zero, which could indicate

that an ‘exact’ solution has been obtained. Hence two more additional refinements are made for further investigations. The results are shown in Table 4-2. The numerical uncertainty for the fine-grid for N_3 , N_4 and N_5 solution are reported as 4.48%, 2.70% and 0.48%; note these do not account for modeling errors.

Table 4-2 Calculation of discretization error for the F-5E duct. The superscripts of the results correspond to the subscripts of the numerical mesh cases in the first row.

	ϕ = pressure recovery at the Aerodynamic Interface Plane (AIP), with monotonic convergence
N_1, N_2, N_3, N_4, N_5	1877, 3743, 4753, 6280, 8289
$r_{21}, r_{32}, r_{43}, r_{54}$	1.259, 1.083, 1.097, 1.097
$\phi_1, \phi_2, \phi_3, \phi_4, \phi_5,$	0.997, 0.972, 0.968, 0.967, 0.966
p	2.31
Φ_{ext}^{21}	1.03
e_a^{21}	2.51
e_{ext}^{21}	3.46
GCI_{fine}^{21}	4.48%
Φ_{ext}^{32}	0.99
e_a^{32}	0.44
e_{ext}^{32}	2.11
GCI_{fine}^{32}	2.7%
Φ_{ext}^{43}	0.971
e_a^{43}	0.091
e_{ext}^{43}	0.379
GCI_{fine}^{43}	0.48%

Figure 4.14 shows the convergence of the pressure recoveries at the AIP with increasing meshing elements. The pressure recovery of a Mach 0.6 inlet is conducted over a range of mesh element values of 1877 to 8289. The limited mesh convergence analysis is due to the computing cost imposed in refining meshes.

Figure 4.14 Convergence of pressure recovery increase with mesh elements. The trend shows with increasing mesh, pressure recovery converges to a value of about 0.978.

4.3.3 F-16 Intake

The geometries of the F-16 intake are obtained from curves lofted in *SolidWorks*. The geometry is very similar to that used in the Normal Shock Inlet (NSI) used in the YF-16 by Hawkins [32]. The area distribution of the simulated F-16 intake, together with other variants; the NSI, MCID-713 and 771 intakes are shown in Figure 4.15. The curved are initially part of a 1/20th scaled model of the F-16. The characteristic length of the elliptical inlet is measured at its semi-major axis, resulting in a length of 1.05m. This value is crucial in determining the inlet conditions, such as turbulent kinetic energy and dissipation, derived from the Reynolds number. The total length of the F-16 inlet is 5.08m (Figure 4.16), longer than the 4.38m of the F-5E inlet (Figure 4.17). The F-16 intake has an inlet and outlet area of 0.478 m² and 0.620 m² respectively. The F-5 has the areas 0.128 m² and 0.202 m² respectively.

Figure 4.15 Duct area distribution of the F-16 inlet used in COMSOL compared to the NSI by Hawkins [32] and MCID-713 and MCID-771 by Hagseth [33].

Figure 4.16 The F-16 inlet with a single offset y-axis.

Figure 4.17 The doubly offset x- and y-axis of the F-5E inlet.

4.3.3.1 Comparison of airflow through the F-5E and F-16 intake at different orientations.

Comparisons of airflow through increasing AOA and AOS are made for the F-5E in Ibrahim *et al.* [46] and F-16 intakes. As seen from Figure 4.16 and Figure 4.17, the singly offset intake of the F-16 affects the airflow through the intake less significantly than the doubly offset F-5E intake. The results of Mach 0.6 at three orientations ($\alpha = 0^\circ$ and $\beta = 0^\circ$, $\alpha = 0^\circ$ and $\beta = 20^\circ$ and $\alpha = 20^\circ$ and $\beta = 0^\circ$) are compared.

The plots in the intake illustrate of the diffusion of the airflow through the duct. The plots measure the total pressure contours dimensionalized with the inlet boundary. The F-16 intake at $\alpha = 0^\circ$ and $\beta = 0^\circ$ shown in Figure 4.18 illustrates a symmetrical total pressure contour plot throughout the transverse stations. This is due to the singly offset feature along the y-axis of the intake. For the F-5E intake, at $\alpha = 0^\circ$ and $\beta = 0^\circ$, (Figure 4.19) a region of high total pressure at the bottom of the intake persists throughout the intake. This is due to the presence of the z-axis curvature of the duct, resembling that of the S-duct. Low total pressure region initiates at the top and right side of the intake but remains relatively low throughout the duct. These two features conform to the intake geometry as the intake shape transcends from an elliptical inlet to a circular outlet.

Increasing the AOA or α results in the diffusion of the high and low total pressure regions to the opposite ends to each other for the F-16 as shown in Figure 4.20. This can also be seen clearly in the AIP. For the F-5E however, this effect is not clearly shown (Figure 4.21). A high total pressure region is formed at the top part of station 1, and diffuses slightly to the upper right side to conform to the shape of the intake in stations 2, 3, 4 and 5. The AIP illustrates the region of significant low total pressure formation which is not directly opposite to the high pressure region. It can also be seen that a negative total pressure value exists in the intake.

The increase in AOS or β results in similar total pressure coefficient plots for the F-16 and F-5E intake, shown in Figure 4.22 and Figure 4.23 respectively. The high and low total pressure regions are at the right and left side of the intake. However it can be seen that the low total pressure region for the F-5E intake is more significant than that of the F-16.

Figure 4.18 Total pressure coefficient contours for F-16 at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 0^\circ$.

Figure 4.19 Total pressure coefficient contours for F-5E at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 0^\circ$.

Figure 4.20 Total pressure coefficient contours for F-16 at Mach 0.6, $\alpha = 20^\circ$ and $\beta = 0^\circ$.

Figure 4.21 Total pressure coefficient contours for F-5E at Mach 0.6, $\alpha = 20^\circ$ and $\beta = 0^\circ$.

Figure 4.22 Total pressure coefficient contours for F-16 at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 20^\circ$.

Figure 4.23 Total pressure coefficient contours for F-5E at Mach 0.6, $\alpha = 0^\circ$ and $\beta = 20^\circ$.

4.3.3.2 GCI Study of the F-16 Duct

Figure 4.24 Monotonic Convergence of the simulations conducted on the F-16 intake.

The monotonic convergence of the F-16 intake is shown in Figure 4.24. The GCI study conducted on the F-16 with the pre-separation region with shows indexes of 2.33% and 3.17%, within the engineering norm of 5%; note these do not account for modeling errors. Hence the grid analysis showed reasonably low numerical errors (Table 4-3).

Table 4-3 Calculation of discretization error for the F-16 intake. The superscripts of the results correspond to the subscripts of the numerical mesh cases in the first row.

	ϕ = pressure recovery at the Aerodynamic Interface Plane (AIP), with monotonic convergence
N_1, N_2, N_3, N_4	2215, 4788, 5912, 8479
r_{21}, r_{32}, r_{43}	1.293, 1.073, 1.128
$\phi_1, \phi_2, \phi_3, \phi_4$	0.993, 0.984, 0.981, 0.980
p	1.53
Φ_{ext}^{21}	1.011
e_a^{21}	0.90
e_{ext}^{21}	1.83
GCI_{fine}^{21}	2.33%
Φ_{ext}^{32}	1.008
e_a^{32}	0.29
e_{ext}^{32}	2.47
GCI_{fine}^{32}	3.17%

4.3.3.3 Specifications of Intake a Pre-Entry region

Figure 4.25 shows a SOLIDWORKS model of the F-16. The highlighted box indicates the simulation geometry. To simulate a possible pre-entry separation, the underside of the fuselage as well as a splitter plane of the intake is added as indicated in Figure 4.26. The dimensions of the geometries are assumed as F-16 form. The underside of the fuselage is assumed to be a flat plate having a length of 12 ft or about 4 meters. A curved surface is not used because when merged with the F-16 intake, error is prompted due to the failure of convergence. At other intake geometries investigated, the program also fails to recognize a merged curved domain. A diverter height of 3.3 inch or 10 centimeters is present so that boundary layer air from fuselage does not enter the inlet as conducted by Hawkins [32]. The splitter plate, gun port, inlet strut and nose wheel are not featured because of their indirect relations to the internal flow of the intake. The cowl blow-in doors are also not featured since they are only used during take-off and landing to allow additional air entering the intake. The boundary conditions used in the simulations are indicated in Figure 4.27.

Figure 4.25 A 1/48 SOLIDWORKS model of F-16. The boxed portion highlights the simulation geometry. The fuselage in the geometry is taken to be a flat plate.

Figure 4.26 The geometry of the simulations showing the two subdomains involving airflow through the underside of the fuselage and the internal airflow through the intake. The fuselage is offset a distance of 10 inches showing the diverter height.

Figure 4.27 The geometry of the underbelly fuselage together with the F-16 intake used in the simulations. The underbelly is assumed to be a flat plate.

Figure 4.28 shows the meshed figure of the F-16 intake together with the separation region.

Figure 4.28 Mesh setting for the simulation of fuselage and intake with a coarser setting at the fuselage portion.

Figure 4.29 Viscous wall units of the fuselage underbelly as well as the walls of the intake satisfy the limits imposed in the circular S-duct.

Figure 4.29 shows the viscous wall units illustration of the fuselage underbelly (for pre-entry separation) and the walls of the intake. They are obtained from equation 3.25 and satisfy the limits imposed on the circular S-duct by Vakili *et al.* [22]. However with changing AOA and AOS this criterion is not fulfilled due to the coarse mesh setting used. The viscous wall units could go up to a few thousands. As mentioned with the previous geometries, the author is confronted with an ‘out of memory’ error when finer mesh is produced. As a result, certain boundary layer growth could not be adequately captured, as shown with the F-5E comparison with FLUENT. However, the physics of the flow will not be truly affected.

Chapter 5 F-16 Maneuverability Simulations

In this section, the maneuverability of the F-16 intake together with the pre-entry region is investigated. Maneuverability is defined in terms of the speed (Mach number) as well as the orientation (Angle of attack and sideslip) of the F-16. The changes in maneuverability are obtained by varying the u , v and w terms in the inflow boundary

5.1 Effects of Pre-entry Separation

Figure 5.1 shows the effects of adding a pre-entry separation domain to the geometry. Two points can be noted; the trends show a decreasing pressure recovery with increasing Mach number and a larger decrease in pressure recovery for the F-16 duct with fuselage from Mach 0.3 above. The decrease in pressure recovery with increasing Mach points to the build up in secondary flow within the intake with increasing free-stream velocity. Significant deviation from Mach 0.3 above shows the effect of the pre-entry separation and its importance in the inclusion of the F-16 intake simulations.

Figure 5.1 The addition of the fuselage results in pre-entry separation of the flow. This results in lower total pressure recoveries in the Mach range of 0.1 to 0.9. Results are obtained by simulations in COMSOL.

Figure 5.2 shows the comparison of aircraft duct losses with increasing Mach number. The F-16 Falcon intake losses resemble that of the Hawk fighter aircraft. This could be due to the single engine similarity that both aircraft uses. The Harrier also uses a single engine but the intake is modified for Vertical and Short Takeoff and Lift (VSTOL) purposes. The F-18 Hornet uses a Y-bifurcated type intake.

Figure 5.2 The F-16 Falcon duct losses compared other fighter aircrafts by Seddon and Goldsmith [2]. The numerical results of the F-16 Falcon, obtained from COMSOL simulations, show similarity to the Hawk duct losses.

5.2 Effects of Angles of Attack (F-16)

The following section represents an extension to the study by Ibrahim *et al.* [59] which had been conducted in weakly compressible form. The F-16 simulations are compared with data of other F-16 intakes that have evolved in its history by Hawkins [32], Hagseth [33] and Hunter and Hawkins [35]. The comparison of inlet pressure recoveries in Figure 5.3 shows that the trend of the simulated results closely resembled the NSI values up to Angle of Attack (AOA) 10°. The pressure recovery from AOA -10° to 10 was about 0.98. Above 10°, the NSI intake only exhibited more significant losses beyond AOA 30°. The F-

16 geometry used in this study only exhibited significant losses from AOA 20°. This could be due to the absence of other geometries to guide the flow through the intake, such as the splitter plate at the lip inlet as well as landing gear house within the intake. Close to the entrance of the inlet, the presence of the wings coupled with the fuselage helps in guiding the air through. Also, the geometry simulated did not include bleeds and diverters which improved in the inlet pressure recoveries.

Figure 5.3 A comparison of inlet pressure recovery for the NSI, MCID-713, MCID- 771 and COMSOL F-16 inlet.

Figure 5.4 and shows the effects of total pressure diffusion within the geometry with increasing the AOA. The figures were to illustrate the movements of the higher and total pressure regions of the fuselage and hence did not include the actual contour legends. At AOA -10°, the high total pressure regions are at the low end of the fuselage, creating a low total pressure area at the underside (top) of the fuselage. At AOA 5°, it can be seen that the airflow that reaches the intake is at its optimum quality in terms of distortion.

Figure 5.4 Axial total pressure plots for all the AOA used in the simulations. The high and low total pressure starts at the lower and higher portions of the geometry initially and diffuses to the opposite direction as the AOA rises.

5.2.1 Total Pressure and Velocity Vector Diagrams

Figure 5.5 shows the effects of distortion with increasing AOA. Distortion decreases from -10° to 0° , but steadily increases from 0° onwards. At negative angles, the presence of the pre-entry domain plays a minimal part in affecting distortion. However with increasing positive angles, that factor couple with the singly offset character of the duct plays a more vital role in increasing distortion.

Figure 5.5 Distortion effects are minimized at AOA 5° , and increase with changing AOA due to the effects of pre-entry separation.

From Figure 5.6 to Figure 5.12, the total pressure coefficient plots and velocity vector diagrams of the different AOA configurations are shown. The orientation of the figures is looking aft wards in relation to the reader. It can be seen at the different AOAs, the total pressure plots and velocity vector diagram are virtually symmetrical, which is characterised by the singly offset feature of the F-16 intake. At AOA -10° , the high total pressure regions can be clearly seen to be split into two portions. The velocity vector diagram at AOA -10° indicate the movement of twin swirls at the top side. The direction of the movements of the twin swirls remain at AOA -5° , changes from AOA 0° onward. The pre-entry domain and the curvature of F-16 intake results in the growth the low total pressure region increases in at the bottom side of the intake,

resulting the low velocity air to build up. From AOA 5° to AOA 10° , the twin swirls seem to exhibit diminishing strength, indicating the function of the fuselage as a flow straightener. However from AOA 20° onwards, the strength of the swirls increases with increasing AOA.

Figure 5.6 Total pressure plot and velocity vector diagram of AOA -10° .

Figure 5.7 Total pressure plot and velocity vector diagram of AOA -5° .

Figure 5.8 Total pressure plot and velocity vector diagram of AOA 0°.

Figure 5.9 Total pressure plot and velocity vector diagram of AOA 10°.

Figure 5.10 Total pressure plot and velocity vector diagram of AOA 20°.

Figure 5.11 Total pressure plot and velocity vector diagram of AOA 30°.

Figure 5.12 Total pressure plot and velocity vector diagram of AOA 40°.

5.3 Effects of Angle of Sideslip (F-16)

Since the intake of the F-16 possessed only a single offset along the z-axis, perfect symmetry occurs along y-axis. The characteristics of pressure recovery and distortion coefficient illustrate this symmetry, shown in Figure 5.13. Without the presence of the fuselage as a flow straightener, varying the AOS affects the pressure recovery (Figure 5.13) of the intake more significantly compared to AOA (Figure 5.3). Also, the distortion coefficient of the F-16 decreases to a minimum of 0.1 with changing AOS. (Figure 5.14)

Figure 5.13 The effect of pressure recovery on AOS shows a decreasing recovery value as AOS increases.

Figure 5.14 The effect of distortion on AOS shows a steadily increasing distortion coefficient value with increasing AOS.

5.3.1 Total Pressure and Velocity Vector Diagrams

From Figure 5.15 to Figure 5.19, the total pressure and swirl flow diagrams shows increasing AOS orientations at the AIP. The views are looking aft wards.

The total pressure diagrams indicate a shift in core flow in the direction of increasing AOS. The lower total pressure segments in each orientation are positioned at the opposite direction to that of the core flow.

The swirl flow diagrams show that bulk swirl in the counterclockwise direction grow in strength with increasing AOS. This tendency is generally due to the transitioning cross sectional area shape of the F-16 intake. The singly offset feature of the F-16 intake also contributes to the formation of only a single a dominant bulk flow direction.

Figure 5.15 Total pressure plot and velocity vector diagram of AOS 5°.

Figure 5.16 Total pressure plot and velocity vector diagram of AOS 10°.

Figure 5.17 Total pressure plot and velocity vector diagram of AOS 15°.

Figure 5.18 Total pressure plot and velocity vector diagram of AOS 20°.

Figure 5.19 Total pressure plot and velocity vector diagram of AOS 25°.

5.4 Correlation to F-16 Flight Placard

The flight simulations results can be represented in a flight placard as done in Ibrahim *et al.* [60]. For a tactical fighter, a typical placard is reproduced in Figure 5.20 by Oates [3] provides a framework for discussion of the various demands that can be placed on the inlet/engine system. The placard provides information of the aircraft and engine envelope. The focus of the aircraft combat arena is denoted by box E, which in practice may be fairly extensive. It is at this area that the severe angles of attack and sideslip can confront the inlet.

The F-16 maneuverability correlated from engine/inlet design by Hagseth [33] is used to determine the end most points of the maneuverability of F-16. The correlation, shown in Figure 5.21, is described in the three factors that are of focus in this work: the Mach number and angles of attack and sideslip. The area produced in the placard decreases with increasing Mach number, implying a lower maneuverability capability with increasing speed. At supersonic speed (Mach 1.6), the area derived from the placard indicates a significant decrease from the two subsonic speeds (Mach 0.6 and Mach 0.9), due to interactions of the boundary layer and shock waves formed at the inlet portion of the intake.

Another characteristic that is prominent in Figure 5.21 is the symmetry of along $\text{AOS} = 0^\circ$. This is due to the singly offset feature of the F-16 intake. This understanding enables the simulation of only one side of the group of AOS, saving computing cost and time. From Figure 5.21, it can be seen that the positive ‘end points’ of Mach 0.6, marked as ‘X’, are defined at:

- 1) $\text{AOA } 30^\circ$ and $\text{AOS } 0^\circ$
- 2) $\text{AOA } 30^\circ$ and $\text{AOS } 5^\circ$
- 3) $\text{AOA } 25^\circ$ and $\text{AOS } 15^\circ$
- 4) $\text{AOA } 0^\circ$ and $\text{AOS } 15^\circ$
- 5) $\text{AOA } -13^\circ$ and $\text{AOS } 5^\circ$

6) AOA -13° and AOS 0°

Simulations are conducted at the positive ‘end points’. The pressure recoveries and distortions at the negative ‘end points’ are expected to be similar. The pressure recovery is at minimum when AOA 25° and AOS 15° , while distortion is maximum at AOA 30° and AOS 5° as shown in Table 5.1. At other combinations of AOA and AOS, however, it can be seen that the stability margins of the F-16 does not prove conclusive when compared to pressure recovery and distortion of airflow through the intake fluctuate at 0.850~0.979 and 0.135~0.420 respectively. This indicates that the stability margins of the F-16 are determined by other aerodynamic factors pertaining to the overall flight. Design of Experiment (DOE) analysis of Taguchi and ANOVA will be done on the results pertaining to the factors of Mach number, angles of attack and angles of sideslip in the next section

Figure 5.20 A typical flight placard of a military fighter.

Figure 5.21 F-16 flight placard from Hagseth [33]. The Normal Shock Inlet (NSI) in the F-16 shows similar geometric properties as that used in the COMSOL simulation as shown in Figure 5.3.

Table 5.1 The correlation of Pressure Recovery and Distortion with the 'end points' of the F-16 placard at Mach 0.6.

Mach Number	AOA	AOS	Total Pressure Recovery	Total Pressure Distortion
6	30°	0°	0.965	0.350
0.6	30°	5°	0.945	0.420
0.6	25°	15°	0.850	0.160
0.6	0°	15°	0.913	0.135
0.6	-13°	5°	0.976	0.310
0.6	-13°	0°	0.979	0.220

Chapter 6 TAGUCHI AND ANOVA ANALYSIS

6.1 *F-5E Manoeuvrability Design*

The use of Taguchi's Method (TM) and ANOVA was done to realize the effects of factors and their interactions on the performance parameters of the intake. For TM, the F-5E manoeuvrability design consists of three factors; Mach number, angles of attack (incidence, α) and sideslip (yaw, β); and represented as symbols A, B and C respectively. Mach numbers of 0.4, 0.5 and 0.6 are investigated. For angles of attack and sideslip, -20° , 0° and 20° are tested. The values chosen are shown in Table 6-1. Pressure recovery (ψ) and distortion (DC (60)) is measured at the AIP, an interface which is near to the outlet boundary of the intake.

6.1.1 Orthogonal Array

The three factors in Table 6-1 are arranged into an orthogonal array using three columns of $L_{27} (3^3)$, as shown on the 1st column of Table 6-2. The numerical experiments are carried out with 3 factors each containing 3 levels, resulting in 27 possible combinations. The numbers 1 to 27 on the 1st column of the table are called numerical experiment numbers. For the numerical experiment No.1 (row 1), factors A, B and C are at, using Table 6-3, Mach 0.4, $\alpha = 0^\circ$ and $\beta = 0^\circ$. Similarly, for numerical experiment No. 6 (row 6), calculation is done using factors in Table 6-2: Mach 0.4, $\alpha = 20^\circ$ and $\beta = -20^\circ$.

Table 6-1 Factors and levels Mach number, angles of attack and sideslip used in the F-5E simulations. Mach numbers of 0.4, 0.5 and 0.6 were used. For angles of attack and sideslip α and β of 20° and -20° were used.

	Levels		
Factors	1	2	3
A	M = 0.4	M = 0.5	M = 0.6
B	$\alpha = 0^\circ$	$\alpha = 20^\circ$	$\alpha = -20^\circ$
C	$\beta = 0^\circ$	$\beta = 20^\circ$	$\beta = -20^\circ$

The results produced for different values of Mach numbers are compared by the average values of ψ and DC (60) that used M = 0.4 (No. 1-9), M = 0.5 (No.10-18) and M = 0.6 (No. 19-27). The average value of ψ is denoted as $\overline{\psi}$ and they are:

$$\overline{\psi}_{M=0.4} = (0.990+0.979+0.952+0.973+0.967+0.945+0.959+ 0.953+0.936)/9 = 0.962$$

$$\text{and } \overline{\psi}_{M=0.5} = 0.921$$

$$\overline{\psi}_{M=0.6} = 0.906$$

$$\overline{DC(60)}_{M=0.4} = (0.128+0.293+0.488+0.447+0.394+0.462+ 0.422+0.459+0.383)/9 = 0.386$$

$$\text{and } \overline{DC(60)}_{M=0.5} = 0.429$$

$$\overline{DC(60)}_{M=0.6} = 0.453$$

The different values of α is also compared by the average values of ψ and DC (60) over the numerical experiments using $\alpha = 0^\circ$ (No. 1-3, 10-12, 19-21), $\alpha = 20^\circ$ (No. 4-6, 13-15, 22-24) and $\alpha = -20^\circ$ (No. 7-9, 16-18, 25-27). Likewise, the $\overline{DC(60)}$ values for different β values are compared in the same way: $\beta = 0^\circ$ (No. 1,4,7,10,13,16,19,22,25), $\beta = 20^\circ$ (No. 2,5,8,11,14,17,20,23,26) and $\beta = -20^\circ$ (No. 3,6,9,12,15,19,21,24,27). The results are summarised in Table 6-3.

Table 6-2 Layout of F-5E aerodynamic maneuverability orthogonal array.

No.	Factors			Parameters			Results	
	A	B	C	Mach number (M)	Angle of attack (α)	Angle of sideslip, yaw (β)	Pressure Recovery (ψ)	Distortion Coefficient (DC (60))
1	1	1	1	0.4	0°	0°	0.984	0.135
2	1	1	2	0.4	0°	20°	0.973	0.296
3	1	1	3	0.4	0°	-20°	0.944	0.496
4	1	2	1	0.4	20°	0°	0.966	0.449
5	1	2	2	0.4	20°	20°	0.966	0.398
6	1	2	3	0.4	20°	-20°	0.940	0.464
7	1	3	1	0.4	-20°	0°	0.955	0.429
8	1	3	2	0.4	-20°	20°	0.949	0.463
9	1	3	3	0.4	-20°	-20°	0.926	0.387
10	2	1	1	0.5	0°	0°	0.963	0.181
11	2	1	2	0.5	0°	20°	0.958	0.337
12	2	1	3	0.5	0°	-20°	0.891	0.557
13	2	2	1	0.5	20°	0°	0.933	0.486
14	2	2	2	0.5	20°	20°	0.916	0.419
15	2	2	3	0.5	20°	-20°	0.867	0.533
16	2	3	1	0.5	-20°	0°	0.893	0.495
17	2	3	2	0.5	-20°	20°	0.896	0.518
18	2	3	3	0.5	-20°	-20°	0.858	0.422
19	3	1	1	0.6	0°	0°	0.961	0.210
20	3	1	2	0.6	0°	20°	0.913	0.374
21	3	1	3	0.6	0°	-20°	0.869	0.610
22	3	2	1	0.6	20°	0°	0.905	0.514
23	3	2	2	0.6	20°	20°	0.906	0.455
24	3	2	3	0.6	20°	-20°	0.849	0.545
25	3	3	1	0.6	-20°	0°	0.869	0.539
26	3	3	2	0.6	-20°	20°	0.879	0.531
27	3	3	3	0.6	-20°	-20°	0.819	0.473

Table 6-3 The off-line TM results of pressure recoveries and distortion coefficients of the various factors for the F-5E intake.

Parameter	Value	$\bar{\psi}$	$\overline{DC(60)}$	Data Range for $\bar{\psi}$	Data Range for $\overline{DC(60)}$
M	0.956	0.391	0.070	0.082	0.956
	0.908	0.439	0.429		0.908
	0.886	0.472	0.453		0.886
α	0.940	0.355	0.046	0.119	0.940
	0.916	0.474	0.463		0.916
	0.894	0.473	0.464		0.894
β	0.937	0.382	0.052	0.117	0.937
	0.928	0.421	0.408		0.928
	0.885	0.499	0.487		0.885

From the values of relative varying ranges as shown in Table 6-3, it can be seen that the Mach number (M) plays the most important factor in determining the outcome of pressure recovery ($\bar{\psi}$), followed by the angle of sideslip (β) and the angle of attack (α). For the distortion coefficient ($\overline{DC(60)}$), the orientation of the aircraft, however, has a more important effect than its speed. The angle of attack (α) plays the most significant effect on distortion, followed by the angle of sideslip (β) and the Mach number (M).

Table 6-4 The ranking of the relative significance of the factors for the F-5E intake.

Parameter	$\bar{\psi}$	$\overline{DC(60)}$
M	1	3
A	3	1
B	2	2

The Taguchi off-line quality control method allows the ranking of the parameters according to their degree of influence of the pressure recovery and distortions tabulated in Table 6-4.

6.1.2 Analysis of Variance (ANOVA or Yates' method)

To compare the results obtained using TM, a qualitative ANOVA approach on the analysis of intake performance is conducted. Similarly, the factors considered are Mach number and angles of attack and sideslip. The Mach number factor has 3 levels while the remaining factors of angle of attack and sideslip have 3 levels each. Again, this resulted in a total of 27 runs.

The performance parameters of the intake are measured by the efficiency of the pressure recovery and distortion coefficient. Mach number is varied at the transitional zone between subsonic and supersonic. The tests are done with Mach 0.4, 0.5 and 0.6. Both the angles of attack and sideslip are varied similarly; at -20° , 0° and 20° . The readings are conducted at the AIP; a hypothetical plane between the inlet of the compressor face and the exit of the diffuser. Firstly, for pressure recovery, the factors A (Mach number), B (Angle of Attack) and C (Angle of Sideslip) are compiled in the summary in Table 6-5.

Table 6-5 The summary table denoting Factors A, B and C describing pressure recovery for the F-5E intake.

Pressure Recovery (ABC Summary Table)									
	C1			C2			C3		
	B1	B2	B3	B1	B2	B3	B1	B2	B3
A1	0.984	0.966	0.955	0.973	0.966	0.949	0.944	0.940	0.926
A2	0.963	0.933	0.893	0.958	0.867	0.896	0.891	0.867	0.858
A3	0.961	0.905	0.869	0.913	0.849	0.879	0.869	0.849	0.819

The observation is again taken at the AIP. Although only three factors were studied, tabulating the ANOVA results proved to be a mathematically tedious because the differences in levels for Factors A with B and C. To summarize, individual factors are removed for the calculation of their respective means. This results in summary tables for interactions AB, AC and BC. The grand mean is reflected at the bottom right of the three tables. The tables are

shown in Table 6-6, Table 6-7 and Table 6-8 respectively. This method is further explained in Farmen [61].

The means are calculated by taking the average of either the rows or the columns. For example, in Table 6-6, the A-means column is obtained by calculating the average of the A row values. The B-means row is obtained by calculating the average of B-column values. In Table 6-6, Table 6-7 and Table 6-8, the Grand Mean value is obtained by averaging the calculated mean values.

Table 6-6 Summary table for interaction AB used in the F-5E simulations.

Pressure Recovery (AB Summary Table)				
	B1	B2	B3	A-means
A1	0.967	0.957	0.943	0.956
A2	0.937	0.889	0.882	0.903
A3	0.914	0.868	0.856	0.879
B-means	0.940	0.905	0.894	0.913

Table 6-7 Summary table for interaction AC used in the F-5E simulations.

Pressure Recovery (AC Summary Table)				
	C1	C2	C3	A-means
A1	0.968	0.963	0.937	0.956
A2	0.930	0.907	0.872	0.903
A3	0.912	0.880	0.846	0.879
C-means	0.937	0.917	0.885	0.913

Table 6-8 Summary table for interaction BC used in the F-5E simulations.

Pressure Recovery (BC Summary Table)				
	C1	C2	C3	B-means
B1	0.969	0.948	0.901	0.940
B2	0.935	0.894	0.885	0.905
B3	0.906	0.908	0.868	0.894
C-means	0.937	0.917	0.885	0.913

The ANOVA results are tabulated in Table 6-9 using Yates' method. The sum of square (SS) of the third order interactions (abc) are pooled to provide estimation of error variance. SS in the table denotes sum of square which is given by $Ct^2/n = Ct^2/8$, Ct being contrast and n being the number of data points; DOF is the degree of freedom; MS or mean square, which is given by SS/DOF, is also equal to the estimates of variance effect; error of the method is defined by variance error, which is given by (Ct_{abc}) ; and finally F, the variance effect divided by variance error, gives SS/error. From the F-table at 5% level of significance, we tabulated that $P = F_{0.05; 2, 8} = 4.46$. It can be seen that for Source A, which is the factor of Mach number, the value calculated from F_0 is higher than P, implying that Mach number plays a significant role in affecting the pressure recovery.

Table 6-9 ANOVA table for pressure recovery used in the F-5E simulations.

Pressure Recovery ANOVA Table					
Source	SS	DOF	MS	F	P
A	0.049	2	0.025	26.237	4.46
B	0.005	2	0.002	2.434	4.46
C	0.005	2	0.003	2.904	4.46
AB	0.001	4	0.000	0.306	3.84
AC	0.001	4	0.000	0.175	3.84
BC	0.001	4	0.000	0.393	3.84
ABC (taken as error)	0.054	8	0.007		
Total	0.117	26.000	0.037		

Subsequently, only the ANOVA result for distortion coefficients is shown in Table 6-10. Since a lower distortion is desired, the distortion coefficient values obtained in the Orthogonal Array are inversed and reflected in the ANOVA table. It shows that Factors A, B and C and Interactions AC and BC affect the distortion of airflow. However, Factor B or the angle of attack affects this distortion most significantly.

Table 6-10 Distortion Coefficient ANOVA results used in the F-5E simulations.

Distortion Coefficient ANOVA Table					
Source	SS	DOF	MS	F	P
A	0.017	2	0.009	9.229	4.46
B	0.023	2	0.011	12.196	4.46
C	0.005	2	0.003	5.078	4.46
AB	0.000	4	0.000	0.017	3.84
AC	0.002	4	0.000	4.485	3.84
BC	0.011	4	0.003	4.318	3.84
ABC (taken as error)	0.037	8	0.005		
Total	0.096	26	0.031		

The results obtained substantiate the previous observation of the Taguchi's Method. Also, they give a qualitative conclusion to the levels of factors and interactions on the effects of Mach number, angles of attack and sideslip to the pressure recovery and distortion of the F-5E intake.

6.2 F-16 Maneuverability Design

Similarly for the F-16 intake, the three factors: Mach number, angles of attack (incidence, α) and sideslip (yaw, β) are represented as symbols A, B and C respectively. Mach numbers of 0.4, 0.6 and 0.8 are investigated. Because of the intake symmetry, for angles of attack and sideslip, 0° , 10° and 20° are tested. The values chosen are shown in Table 6-11. Pressure recovery (ψ) and distortion (DC (60)) is measured at the Aerodynamic Interface Plane (AIP), an interface which is near to the outlet boundary of the intake.

6.2.1 Orthogonal Array

The three factors in Table 6-11 are arranged into an orthogonal array using three columns of $L_{27} (3^3)$, as shown on the 1st column of Table 6-12. The numerical experiments are carried out with 3 factors each containing 3 levels,

resulting in 27 possible combinations. The arrangement of the factors and levels are similar to that conducted on the F-5E. The numbers 1 to 27 on the 1st column of the table are called numerical experiment numbers. For the numerical experiment No.1 (row 1), factors A, B and C are at, using Table 6-12, Mach 0.4, $\alpha = 0^\circ$ and $\beta = 0^\circ$. Similarly, for numerical experiment No. 6 (row 6), calculation is done using factors in Table 6-12: Mach 0.4, $\alpha = 10^\circ$ and $\beta = 20^\circ$.

Table 6-11 Factors and levels Mach number, angles of attack and sideslip used in the F-16 simulations. Mach numbers of 0.4, 0.6 and 0.8 were used. For angles of attack and sideslip α and β of 0° , 10° and 20° were used.

Factors	Levels		
	1	2	3
A	M = 0.4	M = 0.6	M = 0.8
B	$\alpha = 0^\circ$	$\alpha = 10^\circ$	$\alpha = 20^\circ$
C	$\beta = 0^\circ$	$\beta = 10^\circ$	$\beta = 20^\circ$

Table 6-12 Layout of F-16 aerodynamic maneuverability orthogonal array.

No.	Factors			Parameters			Results	
	A	B	C	Mach number (M)	Angle of attack(α)	Angle of sideslip (β)	Pressure Recovery (ψ)	Distortion Coefficient (DC (60))
1	1	1	1	0.4	0°	0°	0.992	0.051
2	1	1	2	0.4	0°	10°	0.917	0.108
3	1	1	3	0.4	0°	20°	0.907	0.131
4	1	2	1	0.4	10°	0°	0.982	0.213
5	1	2	2	0.4	10°	10°	0.954	0.182
6	1	2	3	0.4	10°	20°	0.933	0.390
7	1	3	1	0.4	20°	0°	0.983	0.280
8	1	3	2	0.4	20°	10°	0.915	0.255
9	1	3	3	0.4	20°	20°	0.903	0.237
10	2	1	1	0.6	0°	0°	0.981	0.126
11	2	1	2	0.6	0°	10°	0.902	0.123
12	2	1	3	0.6	0°	20°	0.887	0.154
13	2	2	1	0.6	10°	0°	0.974	0.209
14	2	2	2	0.6	10°	10°	0.932	0.112
15	2	2	3	0.6	10°	20°	0.896	0.356
16	2	3	1	0.6	20°	0°	0.966	0.274
17	2	3	2	0.6	20°	10°	0.889	0.236
18	2	3	3	0.6	20°	20°	0.868	0.245
19	3	1	1	0.8	0°	0°	0.968	0.216
20	3	1	2	0.8	0°	10°	0.895	0.147
21	3	1	3	0.8	0°	20°	0.876	0.192
22	3	2	1	0.8	10°	0°	0.953	0.232
23	3	2	2	0.8	10°	10°	0.871	0.037
24	3	2	3	0.8	10°	20°	0.842	0.333
25	3	3	1	0.8	20°	0°	0.951	0.257
26	3	3	2	0.8	20°	10°	0.842	0.258
27	3	3	3	0.8	20°	20°	0.809	0.243

The results produced for different values of Mach numbers are compared by the average values of ψ and DC (60) that used M = 0.4 (No. 1-9), M = 0.6 (No.10-18) and M = 0.8 (No. 19-27). The average value of ψ is denoted as $\bar{\psi}$ and they are:

$$\bar{\psi}_{M=0.4} = (0.992+0.917+0.909+0.982+0.954+0.933+0.983+0.915+0.903)/9 = 0.943$$

and $\overline{\psi}_{M=0.6} = 0.922$

$\overline{\psi}_{M=0.8} = 0.890$

$\overline{DC(60)}_{M=0.4} = (0.051 + 0.108 + 0.131 + 0.213 + 0.182 + 0.390 + 0.280 + 0.255 + 0.237) / 9$
 $= 0.205$

and $\overline{DC(60)}_{M=0.6} = 0.193$

$\overline{DC(60)}_{M=0.8} = 0.189$

From the values of relative varying ranges as shown in Table 6-13, it can be seen that the angle of sideslip (β) plays the most important factor in determining the outcome of pressure recovery ($\overline{\psi}$), followed by the Mach number (M) and the angle of attack (α). For the distortion coefficient ($\overline{DC(60)}$), the orientation of the aircraft had a more important effect than its speed. The angle of attack (α) played the most significant effect on distortion, followed by the angle of sideslip (β) and the Mach number (M).

Table 6-13 The off-line TM results of pressure recoveries and distortion coefficients of the various factors for the F-16 intake..

Parameter	Value	$\overline{\psi}$	$\overline{DC(60)}$	Data Range for $\overline{\psi}$	Data Range for $\overline{DC(60)}$
M	0.4	0.943	0.205	0.053	0.009
	0.6	0.922	0.204		
	0.8	0.890	0.213		
α	0°	0.925	0.139	0.023	0.115
	10°	0.926	0.229		
	20°	0.903	0.254		
β	0°	0.972	0.206	0.092	0.091
	10°	0.902	0.162		
	20°	0.880	0.253		

Table 6-14 The ranking of the relative significance of the factors used in the F-16 simulations.

Parameter	$\bar{\psi}$	$\overline{DC(60)}$
M	2	3
α	3	1
β	1	2

The Taguchi off-line quality control method in this research method allows the ranking of the parameters according to their degree of influence of the pressure recovery and distortion. This ranking of the relative significance table is tabulated in Table 6-14.

6.2.2 Analysis of Variance (ANOVA or Yates' method)

ANOVA is also used to measure the performance parameters of the intake via the efficiency of the pressure recovery and distortion coefficient. Mach number is varied at the transitional zone between subsonic and supersonic. The tests are done are Mach 0.4, 0.6 and 0.8. Both the angles of attack and sideslip are varied similarly; at 0°, 10° and 20° degrees. Similar to the F-5E study, for pressure recovery, the factors A (Mach number), B (Angle of Attack) and C (Angle of Sideslip) are compiled in the summary table in Table 6-15.

Table 6-15 The summary table denoting Factors A, B and C describing pressure recovery used in the F-16 simulations..

Pressure Recovery (ABC Summary Table)									
	C1			C2			C3		
	B1	B2	B3	B1	B2	B3	B1	B2	B3
A1	0.992	0.982	0.983	0.917	0.954	0.915	0.907	0.933	0.903
A2	0.981	0.974	0.966	0.902	0.896	0.889	0.887	0.896	0.868
A3	0.968	0.953	0.951	0.895	0.842	0.842	0.876	0.842	0.809

This results in summary tables for interactions AB, AC and BC. The grand mean is reflected at the bottom right of the three tables. The tables are shown in Table 6-16, Table 6-17 and Table 6-18 respectively.

Table 6-16 Summary table for interaction AB used in the F-16 simulations.

Pressure Recovery (AB Summary Table)					
	B1	B2	B3	A-means	
A1	0.939	0.956	0.934	0.943	
A2	0.923	0.922	0.908	0.918	
A3	0.913	0.879	0.867	0.886	
B-means	0.925	0.919	0.903	0.916	Grand mean

Table 6-17 Summary table for interaction AC used in the F-16 simulations.

Pressure Recovery (AB Summary Table)					
	C1	C2	C3	A-means	
A1	0.986	0.928	0.914	0.943	
A2	0.974	0.896	0.884	0.918	
A3	0.957	0.860	0.842	0.886	
C-means	0.972	0.895	0.880	0.916	Grand mean

Table 6-18 Summary table for interaction BC used in the F-16 simulations.

Pressure Recovery (AB Summary Table)					
	C1	C2	C3	B-means	
B1	0.981	0.904	0.890	0.925	
B2	0.970	0.897	0.890	0.919	
B3	0.966	0.882	0.860	0.903	
C-means	0.972	0.895	0.880	0.916	Grand mean

From the F-table (Table 6-19) at 5% level of significance, we tabulated that $P = F_{0.05, 2, 8} = 4.46$. It can be seen that for Source A, which is the factor of Mach number, the value calculated from F_0 is higher than P, implying that Mach number plays a significant role in affecting pressure recovery.

Table 6-19 ANOVA table for pressure recovery used in the F-16 simulations.

Pressure Recovery ANOVA Table					
Source	SS	DOF	MS	F	P
A	0.025	2	0.013	13.561	4.46
B	0.001	2	0.001	0.562	4.46
C	0.020	2	0.010	10.429	4.46
AB	0.002	4	0.000	0.411	3.84
AC	0.001	4	0.000	0.294	3.84
BC	0.000	4	0.000	0.098	3.84
ABC (taken as error)	0.063	8	0.008		
Total	0.112	26	0.032		

Subsequently, only the ANOVA result for distortion coefficients is shown in Table 6-20. It shows that Factor B (AOA) and Interaction AC (Mach number and AOS) affect the distortion of airflow with higher F compared to P values.

Table 6-20 Distortion Coefficient ANOVA results used in the F-16 simulations.

Distortion Coefficient ANOVA Table					
Source	SS	DOF	MS	F	P
A	0.003	2	0.002	1.781	4.46
B	0.025	2	0.012	13.231	4.46
C	0.001	2	0.000	0.490	4.46
AB	0.001	4	0.000	0.138	3.84
AC	0.028	4	0.007	7.337	3.84
BC	0.002	4	0.000	0.410	3.84
ABC (taken as error)	0.017	8	0.002		
Total	0.076	26	0.024		

The results obtained substantiate the previous conclusions of the Taguchi's Method on the F-16 intake. Also, they give a qualitative conclusion to the levels of factors and interactions on the effects of Mach number and angles of attack and sideslip to the pressure recovery and distortion of the F-16 intake.

The F-5E Taguchi results revealed that Mach number had the most effect on pressure recovery, and AOA affected distortion most considerably. The F-16 intake, shielded at the underbelly of the fuselage, showed that the factor that resulted in pressure recovery change most considerably is the AOS, while AOA affected distortion most considerably. The two results were further reinforced with results obtained by ANOVA. The differences in results obtained are attributed to the boundary layer growth at the front of the F-16 intake (in the pre-entry region) and centerline curvatures of the two intakes (doubly offset F-5E intake versus singly offset F-16 intake)

Chapter 7 Conclusion and Future Work

7.1 Conclusion

The Navier-Stokes compressible equations coupled with the $k-\epsilon$ turbulence equations were used in COMSOL to solve fluid flow simulations in the F-5E and F-16 intakes. Standard validation and verification techniques were employed in the computational analysis. The semi-qualitative viewgraph norm method was used in validating experimental results of the circular S-duct with reasonable agreement. The Grid Convergence Index (GCI) study done on the circular S-duct, F-5E and F-16 intakes, using pressure recovery as the variable. Results showed that fine grid solution error had a maximum error of 2.18%, 4.48% and 3.17% respectively. The values were within the bandwidth engineering norm of $\pm 5\%$. The geometry of the F-5E comprised only on the intake itself, whereas the F-16 intake consisted of an additional pre-entry domain resembling that of the underbelly of the fuselage in addition to the intake. Both studies investigated the effects of airflow in the intake with changing Mach number, Angles of Attack (AOA) and Sideslip (AOS). Air quality was defined as the pressure recovery and the distortion coefficient that exists at the AIP. The results were studied via total pressure coefficient and swirl flow diagrams. The Taguchi method (TM) and ANOVA statistical analysis were also used to investigate the factors and their interactions on the performance parameters of the intakes.

Comparison of the F-5E and F-16 intakes showed that the obvious geometrical differences of the F-5E and F-16 intakes affected the secondary flow developments adversely within each intake. The F-5E intake is positioned above the wing, attached to the fuselage. It is a doubly offset intake, with a predominant offset along the z-axis, the axis parallel to the transverse of the intake, compared to the singly offset F-16 (also along the z-axis). The presence of a doubly centerline offset of the F-5E intake caused secondary flows to be

excessive within the intake of the F-5E. Increasing the Mach number resulted in an increase in the secondary flow strength, affecting pressure recovery most considerably. The predominant z-axis centerline offset also resulted in the AOA affecting the distortion most considerably.

The addition of the pre-entry separation region of the F-16 resulted in various numerical and visual differences in the flow field. Increasing Mach number increases the boundary layer growth within pre-entry separation region. The diverter present at the intake lip only offsets the boundary layer up to Mach 0.2. At higher Mach numbers, the pressure recoveries of the two geometries (with and without the pre-entry domain) begin to deviate. The presence of the pre-entry region also acts as a shield for increasing AOA up to 20° . This enabled the F-16 intake to maintain its high pressure recoveries with changing AOA. Bulk swirls can be seen forming at increasing values of AOS.

The distortion effects in the F-16 intake were almost opposite to that obtained in the pressure recovery results. The presence of the pre-entry region resulted in the factor of changing AOA to have the most distorted flow within the intake. This is also reinforced by the fact that the intake has a singly z-axis offset character.

The F-5E's Taguchi's Method results showed that Mach number had the most effect on pressure recovery, and AOA affected distortion most considerably. The F-16 intake, shielded at the underbelly of the fuselage, revealed that the factor which resulted in pressure recovery change most considerably is the AOS, while AOA affected distortion most considerably. ANOVA further reinforced the results of the factors obtained by TM as well as indicated the role of the interactions of the factors on both intake performance parameters. The differences in results obtained are attributed to the boundary layer growth at the front of the intake (in the pre-entry region) and centerline curvatures of the two intakes.

Both TM and ANOVA showed that the AOS affects the pressure recovery while the AOA affects distortion most predominantly in the F-16

intake. The presence of the forebody as a flow straightener helped in guiding the air up towards the end of the intake. However this effect also results in high distortion at the AIP of the F-16 intake because of the distorted nature of the incoming air at the inlet. The changing cross-sectional shape as well as offset of the intake also contributed to the nature of this distortion.

7.2 Future Recommendations

The author suggests that running the simulations on a computer with a better platform is essential to obtain a finer grid solution. Better experimental validations and a more detailed GCI verification study can be done. At present, the determining step to this stage is the availability of the use of a 64-bit system. The author believes that with such a system, more accurate simulations can be obtained.

In addition to the presence of the forebody, the presence of additional bleed or ramp systems to accommodate subsonic/ supersonic flow can also produce results similar to that in real-life situations. The inside of the F-16 intake also has compartments such as the landing wheel house that could affect airflow through the intake.

One of the factors that can be considered for practical applications in future work can incorporate unsteady effects at the inlet of the duct. These can be associated with an engine surge, dynamic maneuver, or a separation regions introduced passive or active flow separation devices.

Another parameter that can be added to the study of F-16 intakes is the amount of blockage that is initiated at the inlet boundary. The size and shape of the blockage can be correlated to the pressure recovery and distortion coefficient that is resulted at the AIP.

In addition to the use of variable geometry and secondary bypass, other novella ideas could also be implemented in future studies. In 2000, a modification of the F-16 calls for the use of the Diverterless Supersonic Intake (DSI) which essentially adds a 'bump' near the inlet mouth to counter pre-

separation Hehs [62]. The appealing nature to the modification is that it does not include any moving parts which add on to the overall weight of the aircraft. The positions and geometries of these features are not readily available at the moment for simulating purposes.

Finally, the efforts can build on existing studies of compressor interface distortion. Engine thrust ratio; the value of the average output divided by the maximum output thrust; can be measured by multiplying the pressure recovery with a constant of about 1.5 as stated in Seddon and Goldsmith [2]. Also, calculations of drag force and thrust measurements of the F-16 maneuvers can be used to demonstrate the actual flight envelope of the aircraft. As drag force is a function of the external flow intake, measurements from spillage drag can be estimated at different CFR values. With further studies of the two terms (drag and thrust) obtained, the flight envelope from technical data can be better investigated and studied with different flight conditions obtained from the simulations.

REFERENCES

1. Knaack, M.S., *Encyclopedia of US Air Force Aircraft and Missile Systems: Volume 1, Post-World War II Fighters, 1945-1973*. Office of Air Force History, Washington, DC.
2. Seddon, J. and E.L. Goldsmith, eds. *Intake Aerodynamics*. 2nd ed. AIAA Education Series. 1999, Blackwell Science Ltd.
3. Oates, G.C., *Aircraft Propulsion Systems Technology and Design*, ed. A.E. Series. 1989.
4. Wolf, J.T. and J.L. Farrell, *F-5E Air Vehicle And J85-GE-21 Engine Interface Control Document*, in *Northrop Corporation Aircraft Division*. 1972.
5. *Non-profit website for the aerospace industry*. [cited February 2008; Available from: www.aerospaceweb.org.
6. Dewitte, L. and S. Vanhaste. *The Ultimate F-16 Reference*. [cited 2007; Available from: <http://www.f-16.net>.
7. Bergsmans, W. *Fighter planes*. [cited 2007; Available from: <http://home.iae.nl/users/wbergmns/info/f16.htm>.
8. Bourke, J. *The ABCs of Radio Control - Aircraft, Boats and Cars*. [cited 2008; Available from: <http://www.rcgroups.com/forums/showthread.php?t=111585>.
9. *COMSOL Multiphysics 3.2*, in *Chemical Engineering Module User's Guide*. 2005.
10. *COMSOL Multiphysics 3.3*, in *Release Notes*. 2007. p. 55-57.
11. Wu, Y.L., E.Y.K. Ng, and K.Wong, *Numerical study of the swirl flows in F-5E intake under subsonic speeds*. *Mathematical and Computer Modelling*, 2008. **48**(3-4): p. 447-467.
12. Hirschberg, M.J. *V/STOL: The First Half-Century*. AIAA 1997 [cited 2007; Available from: <http://www.aiaa.org/tc/vstol/same.html>.

13. Goldsmith, E.L. and J. Seddon, *Practical Intake Aerodynamic Design*. AIAA Education Series. 1993: Blackwell Science Ltd.
14. Ibrahim, I.H., *The F-5E port side inlet*, in *Source from RSAF Open House 2007*. 2007: Singapore
15. Su, G. *Air Force World*. [cited 2008; Available from: www.AirForceWorld.com.
16. *Digital Seismograph Station- Interesting Images*. . [cited 2007; Available from: <http://www.geocities.com/CapeCanaveral/Hangar/7195/index6.html>.
17. Hayles, J. *Aeroflight*. [cited 2007; Available from: http://www.aeroflight.co.uk/types/usa/lockheed_martin/f-22/F-22_Raptor.htm.
18. *ATF Program YF-23A*. [cited Cited 2007; Available from: http://www.dreamlandresort.com/black_projects/yf23a.htm. .
19. Sóbester, A., *Tradeoffs in Jet Inlet Design: A Historical Perspective*. Journal of Aircraft, 2007. **44**(3): p. 13.
20. Weske, J.R., *Pressure Loss in Ducts with Compound Elbows*, in *NACA Wartime Report*. 1943.
21. Henry, J.R., *Design of Power-Plant Installations Pressure-Loss Characteristics of Duct Components*, in *NACA Wartime Rept*. 1943.
22. Vakili, A., J.M. Wu, W.R. Hingst, and C.E. Towne, *Comparison of Experimental and Computational Compressible Flow in a S-Duct*. AIAA Paper 84-0033, 1984
23. Harloff, G.J., C.F. Smith, J.E. Bruns, and J.R. DeBonis, *Three-Dimensional Compressible Turbulent Computations for a Diffusing S-Duct*, N. TM-81672, Editor. 1992.
24. Harloff, G.J., C.F. Smith, J.E. Bruns, and J.R. DeBonis, *Navier-Stokes Analysis of Three-Dimensional S-ducts*. Journal of Aircraft, 1993. **30**(4).

25. Wellborn, S.R., T.H. Okiishi, and B.A. Reichert, *A Study of the Compressible Flow Through A Diffusing S-Duct*, N.T.M. 106411, Editor. 1993.
26. Lefantzi, S. and D. Knight, *Automated Design Optimization of a Three-Dimensional S-shaped Subsonic Diffuser*. Journal of Propulsion and Power, 2002. **18**(4): p. 9.
27. Mohler, S.R.J., *Wind-US Flow Calculations for the M2129 S-duct using Structured and Unstructured Grids*, NASA/CR-2003-212736, Editor. 2004.
28. Towne, C.E. and B.H. Anderson, *Numerical Simulation of Flows in Curved Diffusers with Cross-Sectional Transitioning Using a Three Dimensional Viscous Analysis*, in *NASA TM-81672*. 1981.
29. Povinelli, L.A. and C.E. Towne, *Viscous analyses for flow through subsonic and supersonic intakes*, N.T.M. E-3209, Editor. 1986, Lewis Research Centre: Cleveland, Ohio
30. Ibrahim, I.H., E.Y.K. Ng, K. Wong, and R. Gunasekaran, *Effects of centerline curvature and cross-sectional shape transitioning in the subsonic diffuser of the F-5 Fighter Jet*. Journal of Mechanical Science and Technology (Selected paper from AICFM9-246, special issue), 2007. **22**: p. 1993-1997.
31. Saha, K., S.N. Singh, V. Seshadri, and S. Mukhopadhyay, *Computational Analysis on Flow Through Transition S-Diffusers: Effect of Inlet Shape*. Journal of Aircraft, 2007. **44**(1): p. 7.
32. Hawkins, J.E., *YF-Inlet Design and Performance*. Journal of Aircraft, 1974. **13**(6).
33. Hagseth, P.E., *F-16 modular common inlet design concept*, in *23rd Joint Propulsion Conference*. 1987: San Diego, CA.
34. Hunter, L.G.J. and J.A. Cawthon, *Improved Supersonic performance for the F-16 inlet Modified for the J79 Engine*. Journal of Propulsion, 1984. **1**(1).

35. Hunter, L.G.J. and J.E. Hawkins, *F-16 Variable-Geometry Inlet Design and Performance*, in *17th Joint Propulsion Conference*, S. AIAA-1981-1394, and ASME, Editor. 1981: Colorado Springs.
36. Stroud, J.F., *Experimental Investigation of the Effect of Forebody Bluntness on the Pressure Recovery and Drag of a Twin-Scoop Inlet-Body Combination at Mach Numbers Of 1.4 and 1.7*, in *NACA RM A51K14*. 1952.
37. Stitt, L.E. and G.A. Wise, *Investigation Of Several Double-Ramp Side Inlets*, in *NACA RM E54D20*. 1954.
38. Saha, K., S.N. Singh, and V.S. V., *Effect of Angle of Attack on Performance of Twin Intake Duct*, in *37th AIAA Fluid Dynamics Conference and Exhibition (AIAA-2007-4357)*. 2007: Miami, FL.
39. Montgomery, D.C., *Design and Analysis of Experiments*. 3rd ed. 1991, New York: Wiley.
40. Antony, J. and F.J. Antony, *Teaching Advanced Statistical Techniques to Industrial Engineers and Business Managers*. *Journal of Engineering Design*, 1998. **9**(1): p. 12.
41. Antony, J. and F.J. Antony, *Teaching the Taguchi method to industrial engineers*. *Journal of Work Study*, 2001. **50**(40): p. 9.
42. Ng, E.Y.K. and W.K. Ng, *Parametric study of the biopotential equation for breast tumour identification using ANOVA and Taguchi method*. *International Federation for Medical and Biological Engineering*, 2007(44): p. 9.
43. Anderson, B.H., D.N. Miller, M.C. Gridley, and J. Agrell, *The Role of Design-of-Experiments in Managing Flow in Compact Air Vehicle Inlets*, NASA/TM—2003-212601, Editor. 2003.
44. Ng, E.Y.K., N.Y. Liu, and J.Siak, *Parametric Study of Inlet Distortion Propagation in Compressor with Integral Approach and Taguchi Method*. *International Journal of Computational Methods in Engineering Science and Mechanics*, 2005(6): p. 9.

45. Ibrahim, I.H., E.Y.K. Ng, K. Wong, and R. Gunasekeran. *A Study of Secondary Flows in Curved Subsonic Diffusers*. in *COMSOL*. 2007. SEGI college, Kuala Lumpur, Malaysia: Also presented at Aerospace Technological Seminar 2007 (ATS 2007, RSAF).
46. Ibrahim, I.H., E.Y.K. Ng, K. Wong, and R. Gunasekeran, *Applying Taguchi's off-line quality control method and ANOVA on the maneuverability of the F-5E intake* International Association of Mathematical and Computer Modelling (submitted, IAMCM), 2008.
47. Towne, C.E., *Inlet, Duct, and Nozzle Flows*, in *Applied Computational Fluid Dynamics*. 1998, Marcel Dekker.
48. Idelchik, I.E., *Handbook of Hydraulic Resistance*. 1996: Begell House
49. Smith, D. *Using Coupling Variables to Solve Compressible flow, Multiphase flow and Plasma Processing Problems*. in *COMSOL Users Conference*. 2006.
50. Patel, T., S.N. Singh, and V. Seshadri, *Characteristics of Y-Shaped Rectangular Diffusing Duct at Different Inflow Conditions*. *Journal of Aircraft*, 2005. **42**(1): p. 113-119.
51. Bharani, S., S.N. Singh, V.Seshadri, and R.Chandramouli, *Effect of angle-of-turn on the performance of divided intake ducts* Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 2004. **218**(1): p. 22-32.
52. Zhang, W.L., D.D. Knight, and D. Smith, *Automated Design of a Three-Dimensional Subsonic Diffuser*. *Journal of Propulsion and Power*, 200. **16**(6): p. 1132-1140
53. Saad, Y. and M.H. Schultz, *GMRES: A generalized minimal residual algorithm for solving nonsymmetric linear systems*. *SIAM J. Sci. Statist. Comput.*, 1986(7): p. 9.
54. Ignat, L., D. Pelletier, and F. Illinca, *A Universal Formulation of Two Equation Models for Adaptive Computation of Turbulent Flows*.

- Computer Methods in Applied Mechanics and Engineering, 2000(18): p. 1119-1139.
55. Utnes, T., *Two - Equation (k,e) Turbulence Computations by the Use of a Finite Element Model*. International Journal For Numerical Methods In Fluids, 1988. **8**: p. 965-975.
 56. Wu, Y.L., E.Y.K. Ng, and K.Wong, *Numerical study of the swirl flows in F-5E intake under subsonic speeds*. Mathematical and Computer Modelling, August 2008. **48**(3-4): p. 447-467.
 57. Oberkamp, W.L. *Verification and Validation in Computational Simulation*. in *Transport Task Force Meeting*. 2004. Salt Lake City, Utah.
 58. Celik, I., *Procedure for estimation and reporting of discretization error in CFD applications*. ASME Journal of Fluids Engineering, 2005.
 59. Ibrahim, I.H., E.Y.K. Ng, K. Wong, and R. Gunasekeran, *Effect of angle of attack on the YF-16 inlet*. International Journal of Aerodynamics (IJAD), (submitted), 2008.
 60. Ibrahim, I.H., E.Y.K. Ng, K. Wong, and R. Gunasekeran, *The Flight Maneuverability Characteristics of the F-16 and Correlation with its Intake Total Pressure Recovery and Distortion*. Submitted to the Aerospace Science and Technology Journal, 2008.
 61. Farnen, B. *The three-way analysis of variance*. Department of Psychology, Anderson University [cited; Available from: <http://bill.psyc.anderson.edu/exdes/ex5.htm>, .
 62. Hehs, E. *JSF Diverterless Supersonic Inlet*. Code One An Airpower Projection Magazine 2000 [cited 2007; Available from: http://www.codeonemagazine.com/archives/2000/articles/july_00/divertless_1.html.
 63. Dick, S.J. *SP-3300 Flight Research at Ames, 1940-1997*. 2007 [cited 2008; Available from: <http://history.nasa.gov/SP-3300/ch3.htm>.

APPENDIX A

Tradeoffs in Jet Inlet Design: A Historical Perspective

A historical perspective on the chronological account of the modifications in the intake aerodynamics of fighter aircraft was conducted Sóbester [19] . In essence, the approach of an inlet design is a multi-objective and multidisciplinary nature. It is a compromise between numbers of conflicting goals. The emphasis of these goals has however shifted with time. A good inlet design normally has this set of goals; maximizing pressure recovery, while minimizing drag, distortion, weight, cost, complexity and other constraints.

The primary function of the intake is to adequately supply the aircraft engine with air under all conditions of aircraft operation, with no undue impairment on the aptitude of airframe. Similar to the goals of the inlet design, the definitions of ‘properly supplied’ and ‘airframe aptitude’ have changed since the early days of jet design. Most of the intake development problems have been attributed to the behavior of the boundary layer in the adverse pressure gradient environment accompanying the retardation of flow.

One of the most conflicting relationships is pressure recovery and drag. In a pitot type installation, clean freestream air can be captured. This results in good pressure recoveries. However, the increase in drag will be resulted from the space taken up from the fuselage.

Twin mounted intakes have a balance between good pressure recovery and low inlet drag. Two types of twin mounted intakes are the scoop and the flush or submerged intake. The scoop intake exhibits this balance, but the flush/submerged intake compromises pressure recovery with lower drag. Boundary layer ingestion was seen to be the cause of the undesirable pressure recoveries.

A subsonic study of the North American YF-93A aircraft investigated drag and ram pressure recoveries as performance parameters. For the drag parameter, scoop inlet marginally better up to Mach 0.89. Flush intake causing

lower drag at high end of subsonic range. For the pressure recovery parameter, at low mach numbers, submerged intake show better pressure recovery, with scoop intakes being more efficient above Mach 0.87. The results proved to be the cause of the demise of the NACA flush intake as a main source of air on fast jet aircraft (Figure A.1).

Figure A.1 NACA's North American YF-93A research aircraft, with scoop intakes on number one (left) and submerged (flush) intakes on number two Dick [63].

When the issue of 'space' got involved in the conflicting goals of drag and pressure recovery, jet intakes were installed at the wing leading-edge, as that of the F-16 Tiger. However, increasing speeds and sweep angles spelt the end of wing root intakes. This was due to the high spillage drag involved with the increasing sweep angles. Excess airflow tended to spill out at the most rearward part of the intake, instead of uniformly distributed along the lip. Pressure losses at low speeds were also attributed to the sweepback effect. The Nimrod MRA.4 uses desweep intake lips; an improvement from their initial Nimrod MR.2 design (Figure A.2)

Figure A.2 The intake of the Nimrod MR.2 showing two inlet openings mounted at the wing leading edge Sóbester [19].

For supersonic aircrafts, the issue of capture area was of main significance. A large capture area optimizes low speed characteristics, however neglects the spillage drag that is incurred. A small capture area obviously could result in insufficient air entering the engine. A variable capture area could compromise on both the polarizing effects. This is illustrated in the Harrier GR9 (Figure A.3) when the airplane is hovering, and also the MIG-29 during take-off and taxi. A variable geometry during flight conditions reaps benefits in which variation incurred during different angles of attack can be adapted by varying intake lip thickness as well as changing the positions of the shocks that could occur internally or externally

Figure A.3 Blow-in doors on the Harrier: fully open in the hover (Sea Harrier FA2, left) and partially closed (Harrier GR.9, right) Sóbester [19].

The types of intakes used in the supersonic aircrafts also showed a compromise between space and the overall efficiency of the intake. Annular intakes used by the MiG-21F and Mirage 2000 generated oblique shock waves directly in front of the fuselage to retard flow motion shown in Figure A.4. This type of intake is efficient a structural and weight standpoint, however the demise of the annular intake was mainly due to the limited geometry variation that enabled aircraft speed increase.

Figure A.4 Annular intakes and bullet centerbodies on the 1956 MiG-21F (left) and the 1978 Mirage 2000 (right) Sóbester [19].

Rectangular intakes used in the F-14 Tomcat (Figure A.5) and F-15 Eagle (Figure A.6) had better geometry variation possibility. This resulted in lower risk of surge in asymmetric flow conditions as well as less distortion at high angles of attack. Finally, the fixed normal inlet used in the F-16 Falcon, compared to the variable geometry intake, incurred acceleration time penalties

as well as decrease in turn rate. The evolution of the F-16 intakes is further analyzed in the next section.

Figure A.5 The F-14 Tomcat climbing with wings swept forward.

Figure A.6 The F-15 Eagle gaining altitude.

Generally, the supersonic inlets may be classified into one of three types – external, mixed or internal compression – according to whether the supersonic diffusion occurs external to the inlet duct, partly external and partly internal to the duct, or entirely internal to the duct, respectively. The mixed compression

inlet provides higher recovery at supersonic conditions however its increased weight, control complexity and bleed requirements must be balanced against the improved supersonic recovery. For a multimission aircraft, this is seldom a favorable trade.

APPENDIX B

Inlet, Duct, and Nozzle Flows, in Applied CFD

The age of computing set off the synergy of both experimental and numerical analysis in the study of fluid flow through diffusers. In the field of numerical analysis, the Computational Fluid Dynamics (CFD) modeler plays a crucial role in balancing the two. Towne [47] provided a general guide for a CFD modeler. For simplicity, the author will use this template in the analysis of the duct flow.

The study begins with the need to understand the flow problem. To understand the flow problem, gathering necessary information is needed. The amount of details required depends on the use of the results. Once the problem is understood, these steps will be determined: (1) the type of CFD analysis and flow modeling. (2) the particular code to be used. (3) the input parameters running the code.

Several questions concerning the flow are required to be answered to give a better picture of the flow.

- 1) Will the flow be compressible or incompressible?
- 2) Will the boundary layers be thin or thick?
- 3) Are regions of separated flow possible?
- 4) Will the flow be laminar, turbulent, or transitional?
- 5) Will shock waves be present?
- 6) Will three-dimensional effects be important?
- 7) Will real-gas effect be significant?
- 8) What are the appropriate reference conditions?
- 9) What are the appropriate boundary conditions?

A suitable Computer Aided Design (CAD) system is used to assist in the drawing of the geometry for the system. A question arises as to how detailed

should the drawing of the geometry compared to the actual system. Certain geometrical features could just be modeled, without be computed. For example, by specifying a surface roughness factor in the turbulence model, rivet heads impeding a flow could be modeled without actually including them.

The next issue to be discussed is the computational grid. Grid generation can be done by the computer or the user. For example, a customized program can be used to generate the grid algebraically, giving the user a control on specifying the grid size and shape. In most cases, this will be the most time-consuming step in the entire solution process.

There are several points to keep in mind when generating the grid for an application. Firstly, the grid points should be concentrated on the regions where large flow gradients are expected. These include boundary layers near solid surfaces or shear layers at shockwaves. In addition, a balance has to be struck between grid density and computer resource. If the grid is made too dense, the plus point is that a more accurate result solution will be obtained. However, computer resources will be used. Secondly, highly non-orthogonal grids should be avoided. This is because excessive grid skewness may adversely affect the solution. Thirdly, the grid should be smooth. Sharp changes between grid lines would again result in numerical difficulties.

Other inputs to the model are: the choice and magnitude of artificial viscosity model, solution algorithm and iteration control parameters, boundary conditions and type of turbulence model. Under these conditions, it may take several “iterations” – running a case, examining the results, changing the input/and or mesh, and re-running the case – to get the first good results for a new case. It is because of this that running a successful CFD code is sometimes known to be a combination of science and art.