

CRANFIELD UNIVERSITY



SCHOOL OF ENGINEERING

PhD THESIS

Christos Isseris

**Distortion Management Effects on
Jet Engine Performance**

Supervisor: P. Pilidis

November 2005

ProQuest Number: 10820947

All rights reserved

INFORMATION TO ALL USERS

The quality of this reproduction is dependent upon the quality of the copy submitted.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if material had to be removed, a note will indicate the deletion.



ProQuest 10820947

Published by ProQuest LLC (2018). Copyright of the Dissertation is held by Cranfield University.

All rights reserved.

This work is protected against unauthorized copying under Title 17, United States Code
Microform Edition © ProQuest LLC.

ProQuest LLC.
789 East Eisenhower Parkway
P.O. Box 1346
Ann Arbor, MI 48106 – 1346

Executive Summary

Boundary layer separation studies have been going on for many years. The concept of flow control is not new, with boundary layer blowing or suction to delay separation known since Prandtl. Many flow control methods are considered nowadays to introduce improvements on flow characteristics both for internal and external applications.

This work is concerned with the application of flow control methods for improving the flow characteristics within intake ducts. Flow control applications are presented with a challenge to improve the flow characteristics of intake ducts with complex geometries that are used to deliver airflow to buried propulsion systems, common for military engine applications. This work is analysing the flow control capabilities for improving s-shaped ducts flow characteristics and how these improvements translate onto engine performance. For the purpose of this study a methodology was created in order to simulate an intake/compressor/engine interaction and analyse the characteristics of all three components. Three-dimensional flow simulations have been carried out for the intake and compressor models and compressor performance results have been transferred to an engine performance tool to investigate performance parameters under different inflow conditions.

The case studies involved axial inflow simulations and cases with inflow of different pitch and yaw angles. The influence of these conditions has been transferred from the intake model to the compressor model and the performance characteristics used to analyse the engine operation under these conditions. Flow control methods have been applied to the system in order to minimise the negative effects of the inflow and improve the flow characteristics at the intake duct. Furthermore, air or power off-takes requested from the engine in order to operate the flow control mechanisms have also been considered. The results showed that the application of flow control introduces improvements throughout the system components. These improvements are sufficient in order to compensate for the air and power off-takes requested by the engine for the use of the flow control mechanisms. Although the results support such a conclusion it is also important to point out that some case studies especially for low distorted flows showed a very small improvement in engine performance which can conclude that flow control may not be as effective when the cost and complexity introduced to the system is considered.

Table of Contents

Executive Summary.....	i
List of Figures.....	v
List of Tables	viii
Nomenclature.....	ix
Acknowledgements.....	xi
1. Introduction.....	1
1.1 Introduction.....	2
1.2 Review of previous work.....	3
1.2.1 Boundary layer separation control.....	3
1.2.2 Flow control methods assessment.....	8
1.3 Project methodology.....	11
1.4 Overview of dissertation.....	13
2. Intake modelling and flow simulation.....	17
2.1 Introduction.....	18
2.2 Review of previous work.....	19
2.2.1 Intake design and flow mechanics.....	19
2.2.2 Computational and experimental intake work.....	20
2.3 Diffusing s-duct flows.....	22
2.3.1 Terminology.....	22
2.3.2 Mass flow and pressure recovery definitions.....	22
2.3.3 Fluid mechanics of diffusing s-duct flows.....	24
2.3.4 Distortion.....	25
2.4 Numerical formulation.....	28
2.4.1 Grid generation.....	28
2.4.2 Engine face boundary conditions.....	30
2.4.3 Turbulence modelling.....	30
2.5 S-shape intake model validation.....	31
2.5.1 Grid assessment.....	31
2.5.2 Turbulence models assessment.....	34
2.6 CFD results.....	38
2.6.1 Distortion simulation cases.....	38
2.6.1.1 Pitch angles.....	39
2.6.2 Flow control simulation cases.....	42
2.6.2.1 FC design configuration.....	42

2.6.2.2 Injection mass flow rate.....	45
2.6.2.3 Flow control methods.....	46
2.6.2.4 Flow control cases.....	48
3. Compressor modelling and flow simulation.....	52
3.1 Introduction.....	53
3.1.1 CFD approach.....	57
3.2 Fan preliminary design.....	58
3.2.1 Design objectives and specifications.....	59
3.2.2 Design guidelines and limitations.....	59
3.2.3 Design variables.....	61
3.2.4 Final design specifications.....	62
3.3 CFD fan modelling.....	64
3.3.1 Geometry and grid generation.....	65
3.3.2 CFD simulation.....	67
3.3.2.1 Grid generation.....	68
3.3.2.2 Boundary conditions.....	68
3.3.2.3 Solver parameters.....	68
3.3.3 Method limitations.....	69
3.4 CFD simulation analysis.....	70
3.4.1 Compressor grid analysis.....	71
3.4.2 Turbulence model analysis.....	73
3.4.3 Frozen rotor simulation.....	75
3.5 Clean compressor cases.....	76
3.5.1 Flow analysis.....	78
3.5.2 Choke analysis.....	83
4. Engine performance simulation.....	86
4.1 Introduction.....	87
4.2 Low bypass turbofan engine modelling.....	89
4.2.1 Engine performance simulation.....	89
4.2.1.1 Effect of flight velocity.....	90
4.2.1.2 Effect of altitude.....	92
4.2.1.3 Effect of shaft rotational speed.....	93
4.2.2 Intake pressure losses.....	95
4.2.2.1 Effect of flight velocity.....	95
4.2.2.2 Effect of altitude.....	96
4.2.2.3 Effect of shaft rotational speed.....	97
4.2.3 Bleed off-takes.....	99

4.2.3.1 Effect of flight velocity.....	100
4.2.3.2 Effect of altitude.....	101
4.2.4 Power off-takes.....	102
4.2.4.1 Effect of altitude.....	103
4.3 Engine modelling at distorted conditions.....	105
4.3.1 Engine performance simulation.....	105
4.3.1.1 Effect of shaft rotational speed.....	106
4.3.1.2 Effect of altitude.....	107
4.4 Engine modelling at flow control conditions.....	108
4.4.1 Engine performance simulation.....	108
4.4.1.1 Effect of altitude.....	109
5. Results and discussion.....	112
5.1 Introduction.....	113
5.2 Axial flow case studies.....	114
5.2.1 Intake flow simulation.....	114
5.2.2 Compressor flow simulation.....	119
5.2.3 Engine performance.....	122
5.3 Yaw flow case studies.....	124
5.3.1 10 degrees yaw angle cases.....	125
5.3.1.1 Intake flow simulation.....	125
5.3.1.2 Compressor flow simulation.....	131
5.3.1.3 Engine performance.....	134
5.3.2 30 degrees yaw angle cases.....	135
5.3.2.1 Intake flow simulation.....	135
5.3.2.2 Compressor flow simulation.....	142
5.3.2.3 Engine performance.....	144
5.4 Angle of attack case studies.....	146
5.4.1 Intake flow simulation.....	147
5.4.2 Compressor flow simulation.....	151
5.4.3 Engine performance.....	156
6. Conclusions and future work.....	159
6.1 Conclusions.....	160
6.2 Future work.....	162
References.....	165

List of Figures

Figure 1.1 Massless Jets.....	11
Figure 1.2 Methodology Flow Diagram.....	12
Figure 1.3 RAE Intake Model M2129.....	14
Figure 1.4 Rotor/Stator Passage Grid.....	15
Figure 1.5 Two spool low bypass turbofan engine (Courtesy of RR).....	16
Figure 2.1 Input Coordinates.....	27
Figure 2.2 RAE Intake Model M2129.....	28
Figure 2.3 Model Blocks.....	29
Figure 2.4. Wall treatment.....	32
Figure 2.5 Grid comparison in DC60.....	33
Figure 2.6 Grid comparison in pressure losses.....	33
Figure 2.7 Turbulence models comparison in DC60.....	35
Figure 2.8 Turbulence models comparison in PR.....	36
Figure 2.9 Starboard wall pressure recovery.....	36
Figure 2.10 Flow recirculation region.....	37
Figure 2.11 Flow recirculation region.....	37
Figure 2.12 Port wall pressure recovery.....	38
Figure 2.13 DC60 at angle of attack.....	39
Figure 2.14 Pressure recovery at angle of attack	40
Figure 2.15 Engine face velocity vectors.....	41
Figure 2.16 Pathlines at 40 degrees.....	41
Figure 2.17 Total pressure profiles.....	42
Figure 2.18 Flow separation region.....	43
Figure 2.19 Flow control design configuration.....	44
Figure 2.20 Flow control configuration at 40 degrees pitch.....	45
Figure 2.21 Air injection mass flow rates.....	46
Figure 2.22 Flow control methods.....	47
Figure 2.23 Flow control methods.....	47
Figure 2.24 FC angle of attack cases.....	48
Figure 2.25 FC angle of attack cases.....	49
Figure 2.26 FC yaw cases (DC60).....	49
Figure 2.27 FC yaw cases (PR).....	50
Figure 2.28 Yaw angle case studies.....	50
Figure 3.1 Axial compressor configuration.....	53
Figure 3.2 Three-dimensional phenomena in compressors.....	54
Figure 3.3 Mid-radius De Haller number distribution.....	64
Figure 3.4 Mid-radius diffusion factor distribution.....	64
Figure 3.5 Visualisation of hub, shroud and blade profiles.....	65
Figure 3.6 Modelling of rotor, stator and whole passage.....	66
Figure 3.7 Grid view for hub, mid and tip and interface plane.....	66
Figure 3.8 Fan 3D model.....	67
Figure 3.9 Grid comparison Pressure ratio.....	72
Figure 3.10 Grid comparison Efficiency.....	72
Figure 3.11 Turbulence model comparison Pressure ratio (fine grid).....	74
Figure 3.12 Turbulence model comparison Efficiency (fine grid).....	74
Figure 3.13 Frozen rotor and mixing plane comparison	75

Figure 3.14 Frozen rotor and mixing plane comparison.....	76
Figure 3.15 Clean map Pressure ratio.....	77
Figure 3.16 Clean map Efficiency.....	77
Figure 3.17 Clean design speed line.....	78
Figure 3.18 Static pressure distribution at 50% of the rotor and stator blade... 79	
Figure 3.19 Relative Mach number contours at the hub, mid height and shroud at DP.....	80
Figure 3.20 Relative Mach distribution at DP along the first stage.....	81
Figure 3.21 Shock wave structure at DP.....	81
Figure 3.22 Flow speed at DP.....	82
Figure 3.23 Shock waves structure at 50 % blade span for two different outlet conditions.....	83
Figure 3.24 Shock waves structure from hub to tip, choke analysis, fine grid..	84
Figure 3.25 Flow recirculation at the stator leading edge.....	85
Figure 3.26 Flow recirculation at the stator trailing edge.....	85
Figure 4.1 Compressor map.....	88
Figure 4.2 Clean running line.....	90
Figure 4.3 Effect of flight velocity on engine thrust.....	90
Figure 4.4 Effect of flight velocity on engine specific fuel consumption.....	91
Figure 4.5 Effect of altitude on engine thrust.....	92
Figure 4.6 Effect of altitude on engine specific fuel consumption.....	93
Figure 4.7 Effect of shaft rotational speed on engine thrust.....	94
Figure 4.8 Effect of shaft rotational speed on engine sfc	94
Figure 4.9 Effect of Mach number on PR penalties.....	95
Figure 4.10 Effect of Mach number on PR penalties.....	96
Figure 4.11 Effect of altitude on PR penalties.....	97
Figure 4.12 Effect of altitude on PR penalties.....	97
Figure 4.13 Effect of shaft rotational speed on PR penalties.....	98
Figure 4.14 Effect of shaft rotational speed on PR penalties.....	98
Figure 4.15 Effect of Mach number on Bleed Air Penalties.....	100
Figure 4.16 Effect of Mach number on Bleed Air Penalties.....	101
Figure 4.17 Effect of Altitude on Bleed Air Penalties.....	101
Figure 4.18 Effect of Altitude on Bleed Air Penalties.....	102
Figure 4.19 Compressor characteristics map.....	103
Figure 4.20 Power off-takes thrust penalty.....	104
Figure 4.21 Power off-takes sfc penalty.....	104
Figure 4.22 Distortion running line.....	105
Figure 4.23 Effect of TET on engine thrust.....	106
Figure 4.24 Effect of TET on engine specific fuel consumption.....	107
Figure 4.25 Effect of altitude on engine thrust.....	107
Figure 4.26 Effect of altitude on engine specific fuel consumption.....	108
Figure 4.27 Flow control running line.....	109
Figure 4.28 Effect of altitude on engine thrust.....	110
Figure 4.29 Effect of altitude on engine specific fuel consumption.....	110
Figure 5.1 Intake duct total pressure distribution.....	115
Figure 5.2 Total pressure distortion profiles.....	115
Figure 5.3 Secondary flow injection angle.....	116
Figure 5.4 Secondary flow pathlines.....	117

Figure 5.5 Axial cases pressure recovery.....	117
Figure 5.6 Engine face total pressure profiles at different injection angles....	118
Figure 5.7 Axial cases engine face distortion.....	119
Figure 5.8 Compressor passages shock waves formation.....	120
Figure 5.9 Compressor pressure ratio map.....	120
Figure 5.10 Compressor efficiency map.....	121
Figure 5.11 Axial cases pcn 1.0 pressure ratio results.....	121
Figure 5.12 Percentage thrust with altitude variation.....	123
Figure 5.13 Percentage thrust with altitude variation.....	124
Figure 5.14 Yaw angle inflow.....	125
Figure 5.15 Velocity vectors flow visualisation.....	126
Figure 5.16 Recirculating area comparison.....	127
Figure 5.17 Total pressure contours along the intake.....	127
Figure 5.18 Injection flow visualisation.....	128
Figure 5.19 Total pressure profiles at engine face.....	129
Figure 5.20 Recirculating area comparison.....	129
Figure 5.21 DC60 for 10 degrees yaw.....	130
Figure 5.22 Pressure recovery for 10 degrees yaw.....	131
Figure 5.23 Compressor flow simulation results.....	132
Figure 5.24 Compressor pressure ratio 10 degrees yaw.....	133
Figure 5.25 Compressor efficiency 10 degrees yaw.....	133
Figure 5.26 Engine performance at 10 degrees yaw.....	134
Figure 5.27 Thrust vs sfc for 10 degrees yaw.....	135
Figure 5.28 Flow characteristics at 30 degrees yaw inflow.....	136
Figure 5.29 Flow visualisation throughout the intake duct.....	137
Figure 5.30 Flow optimisation for intake pressure recovery.....	138
Figure 5.31 Flow optimisation for engine face distortion.....	139
Figure 5.32 Engine face distortion for different angles of injection.....	139
Figure 5.33 Flow visualisation with and without flow control.....	140
Figure 5.34 Engine face profile with and without flow control.....	140
Figure 5.35 Engine face distortion for 30 degrees yaw.....	141
Figure 5.36 Intake pressure recovery at 30 degrees yaw.....	141
Figure 5.37 Flow analysis along the blade span.....	143
Figure 5.38 Pressure ratio map characteristics.....	143
Figure 5.39 Efficiency map characteristics.....	144
Figure 5.40 30 degrees yaw angle engine performance.....	145
Figure 5.41 30 degrees yaw engine performance.....	146
Figure 5.42 Intake flow visualisation at angle of attack.....	147
Figure 5.43 Engine face profiles for angle of attack cases.....	148
Figure 5.44 Injection and suction at flow control plane.....	149
Figure 5.45 Engine face profiles with and without flow control.....	150
Figure 5.46 40 degrees angle of attack pressure recovery.....	150
Figure 5.47 40 degrees angle of attack DC60.....	151
Figure 5.48 Relative Mach number across the blade span.....	152
Figure 5.49 Compressor total pressure profiles for suction.....	153
Figure 5.50 Inlet axial velocity profiles for 40 deg angle of attack cases.....	154
Figure 5.51 40 degrees compressor pressure ratio maps.....	155
Figure 5.52 40 degrees compressor efficiency maps.....	156

<i>Figure 5.53 Thrust vs TET for angle of attack.....</i>	<i>157</i>
<i>Figure 5.54 Thrust vs sfc for angle of attack.....</i>	<i>157</i>

List of Tables

<i>Table 1.1 Flow Control Methods Matrix.....</i>	<i>9</i>
<i>Table 2.1 Turbulence Models Comparison.....</i>	<i>34</i>
<i>Table 3.1 Fan design definition.....</i>	<i>59</i>
<i>Table 3.2 Compressor mid radius properties.....</i>	<i>63</i>
<i>Table 3.3 Nodes distribution</i>	<i>65</i>
<i>Table 3.4 Percentage variation at grid comparison.....</i>	<i>72</i>
<i>Table 3.5 Turbulence model comparison</i>	<i>74</i>
<i>Table 4.1 Rolls-Royce EJ200 general specifications.....</i>	<i>89</i>
<i>Table 5.1 Axial case studies comparison.....</i>	<i>122</i>
<i>Table 5.2 10 degrees yaw engine performance.....</i>	<i>134</i>
<i>Table 5.3 30 degrees yaw results.....</i>	<i>145</i>
<i>Table 5.4 40 degrees angle of attack results.....</i>	<i>156</i>

Nomenclature

A	Cross sectional area
AIP	Aerodynamic Interface Plane
C _p	Specific Heat (at constant pressure)
DNS	Direct Numerical Simulation
DP	Design Point
HP	High pressure
IGV	Inlet Guide Vanes
LE	Leading edge
LP	Low pressure
M	Mach number
N	Rotational speed
NDMF	Non-Dimensional Mass Flow
P	Total pressure
PCN	Non-dimensional rotational speed
Pr	Pressure recovery
PR	Pressure ratio
RPM	Revolution Per Minute (rotational speed)
FC	Flow Control
SLS	Sea Level Static
T	Total temperature
TE	Trailing edge
TET	Turbine Entry Temperature
W	Work
W, $\dot{m}$	Mass flow
amb	Ambient conditions
h	Enthalpy
k	Turbulent Kinetic Energy
l	Length
p	Static pressure
q	Dynamic pressure (or Dynamic head)
s.f.c.	Specific Fuel Consumption
t	Static temperature
u	Velocity

Greek symbols

ρ	Density
η	Efficiency

γ	Specific heat ratio
ψ	Generic fluid dynamic quantity
ε	Turbulent Dissipation Rate
ω	Specific Dissipation Rate

Subscripts and Superscripts

i	Ideal
o	Total
1	Inlet
2	Outlet
60	Over 60 degrees
360	Over 360 degrees

Acknowledgements

The author would like to express his gratitude to a number of people that helped to complete this thesis. In particular I wish to thank my supervisor Proffesor Pericles Pilidis for his guidance, constructive criticism and his encouragement and patience throughout this work.

I would also like to thank BAE SYSTEMS for giving me this opportunity to work for this project, and in particular Nick Sellars and Peter Curtis for the support and useful comments.

A special thanks to all my friends for their encouragement and support.

Finally I will be forever indebted to my family Nikos, Anna and Stamatis Isseris for their love support and understanding.

CHAPTER ONE

Introduction

1.1 Introduction

The trend of modern aircraft has been towards designs that develop higher thrust without sacrificing manoeuvrability or increasing overall weight. As engines become smaller and more powerful, a persistent problem in intake design and engine/intake compatibility becomes more significant, the adverse effect of the inlet pressure distortion on engine performance.

The compatibility between the engine and the intake forms an integral part of propulsion system design and development. Intake and engine must remain aerodynamically compatible throughout an aircraft flight envelope. This is to say that situations which lead to compressor stall, engine surge or other malfunctioning of the propulsion system must be avoided or at least reduced to a tolerably low frequency of occurrence. Such situations are produced by departures of the airflow, as delivered by the intake to the engine, from the ideal of a flow uniform in pressure, temperature, gaseous and other content and uniformly axial in direction. No real flow ever achieves this ideal, but in the pursuit of high intake pressure recovery on design, flows are usually sufficiently close to it as to pose no compatibility problem. In the outer region of the flight envelope, however, or in non-standard atmospheric conditions, serious deterioration in flow quality can and do occur.

Buried propulsion system installations are common for military engines to reduce radar cross-section (RCS), and also for some configurations of the Blended Wing Body (BWB) design. The S-duct inlet for such cases may be undesirably long to avoid significant losses, or pressure distortions that could cause the engine to stall. Such flow distortion promotes perturbations in the axial, circumferential and radial velocity components and in flow angles at entry to the fan or low pressure compressor of the engine. These perturbations represent the result of the interaction between the flow fields of the intake and the fan and are very complex indeed as they vary across the engine face and with time, when the intake flow is turbulent. Moreover, the perturbations are not characteristic of the intake flow alone and could not be derived without the presence of the engine simulation.

As intakes tend to become smaller and more complex, to achieve compatibility between the engine and the intake, the problem of distortion needs to be addressed. The way to do that is with the use of flow control systems. The definition of flow control that has been adopted in this work is the use of a small

modification (e.g. fluid injection) to change the behavior of a much larger flow. The key emphasis is on the ability to modify the behavior of the larger flow without having to act on the scale of that flow.

Flow control could be used to enable a more aggressive inlet duct without the associated performance penalties from separation; indeed vortex generators are already in use to reduce inlet distortion in some aircraft such as the F-18A/B. While inlet length is a significant propulsion/airframe integration challenge for manned vehicles, the inlet could become a limiting factor for the entire vehicle size for some unmanned configurations.

1.2 Review of Previous Work

The concept of flow control is not new, with boundary layer blowing or suction to delay separation known since Prandtl[1], and in use today in areas such as supersonic inlets. In addition to mitigating separation, flow control has the potential to enhance or modify mixing, create a controllable “virtual” shape, manage wakes, control boundary layer drag, etc.; an excellent review of flow control is given by Gad-el-Hak[2].

1.2.1 Boundary Layer Separation Control

This section gives a review to some of the studies that have been carried out throughout the years on control of boundary layer separation and the methods used in order to improve flow characteristics either actively or passively.

Wallis [3] first showed that a pitched, skewed blowing jet generates a streamwise vortex as it adds high momentum fluid to the flow. This generation of vortices introduces a cross-stream mixing that can be used to eliminate separation zones.

Johnston and Nishi [4] looked at eliminating stalled regions in other words zones of detached or separated flow, by using vortex generator jet (VGJ) method. The method employs a flat plate with spanwise arrays of small, skewed and pitched jets from holes in the plate surface. The experiments were carried out at low-speed ($U = 15\text{m/s}$). The results showed that this particular method creates vortices in the boundary layer downstream of the jet holes, like the vortices created behind solid vortex generators. It is also shown that these vortices introduce a mixing of the boundary layer stream and the freestream that can reduce and even eliminate separation.

Selby et al. [5] performed a parametric investigation to find the optimum combination of vortex generator jet geometric and flow properties to control separation on an aft-facing ramp in low speed flow (40m/s). The study showed that VGJs had significant separation-control when placed 40 boundary layer thicknesses upstream of the separation point but were even more effective when placed just upstream of the separation point. This study also showed that an array of jets that are placed in order to produce co-rotating vortices was more effective than jets that produced counter-rotating vortices.

Studies have been also carried out in order to compare the effectiveness between passive and active methods. Lin et al. [6] performed a study comparing the performance of passive and active methods on an aft-facing ramp and found VGJs to be one of the most effective methods.

Ball [7] carried out a study of seven different blowing schemes, showing that a combination of discrete blowing holes and solid vortex generators produced the best results, providing a total pressure recovery increase of approximately 1% with a blowing mass flow of only 0.4% of diffuser inlet mass flow, suggesting that some combination of high-momentum fluid injection and vortex generation may provide a relatively efficient means for improving diffuser performance.

The effect of blowing vortex generator jets on the performance of a highly-offset diffuser was investigated experimentally by Senseney, Buter and Bowersox [8]. The study showed that without blowing, the flow on the lower surface of the diffuser was massively separated. Blowing at 0.5% mass flow ratio through three lower surface vortex generator jets, reduced the size of the separated-flow region and the exit plane boundary layer thickness, increased pressure recovery and decreased turbulence intensity. However, the exit plane total and static pressure fields were more distorted with blowing than without.

Raghunathan et al. [9] carried out an experimental investigation in order to assess passive vortex generator jets in controlling boundary layer separation. The PVGJs consisted of an array of jets upstream of the diffuser and were arranged symmetrically on both diverging walls of the diffuser. This study also showed that with the use of co-rotating vortex generator jets there was an improvement in both pressure recovery and stall angle.

Vakili et al. [10] used pulsed injection to change the effective throat area of a nozzle. Qualitative experimentation was used to understand the relevant mechanisms involved. Tests were run in a water tunnel, comparing steady injection to pulsed injection. Flow visualization showed that jet penetration is increased with the pulsation of the jet. Each pulse produces a vertical mass of flow having a vortex ring structure. Results for different exit geometry jets indicate improvements in the effective blockage for the pulsed injection compared to the steady injection.

Sedor [11] also used pulsed jet nozzle blowing in a low-speed wind tunnel to investigate directional control at moderate to high angles of attack. Pulsing jets were tested to determine if mass flow could be reduced without compromising yaw control effectiveness. In this study it was shown that frequency modulation of the blowing jet has an effect on yawing moment. As blowing coefficient increases, the frequency for improving effectiveness over continuous blowing decreases. Results indicated that pulsed blowing is slightly more effective than steady blowing at certain angles of attack.

Another study of pulsed vortex generator jets for flow separation was carried out by McManus et al [12]. The experiments were performed in a small scale wind tunnel, configured with a divergent lower wall and an array of three jet orifices were positioned upstream of the divergent section. The pulsed jet amplitude and frequency were variable. Results showed that pulsed VGJs enhance cross-stream mixing in the boundary layer and as a result forestall separation. Flow visualization results indicate that the mechanism leading to mixing enhancement is related to the formation of large turbulent structures caused by the pulsed injection. A comparison of pulsed and steady VGJs showed that the mass flow requirements for separation control using pulsed VGJs are greatly reduced over those for steady jets.

The interaction of pulsed vortex generator jets with a zero pressure gradient turbulent boundary layer has also been studied by Johari and McManus [13]. In this study the improved performance of the pulsed vortex generator jets is attributed to the formation, stretching and bending of the starting vortex ring. The shear in the boundary layer amplifies the vorticity of the starting ring and re-directs it into the streamwise direction. Flow visualization results showed that pulsed VGJs create vortex ring-like structures that are inclined with respect to the freestream.

Amitay et al. [14] used arrays of fluidic actuators based on synthetic jets (zero mass flux), in order to study the reattachment of flow within a separated flow region. In this study the jets were placed downstream of the separation point and the results showed that the jets even though they are placed downstream of the separation point, lead to flow reattachment. This reattachment of the flow showed a reduction in losses and an increase in volume flow rate.

A novel test facility has been built by Techburg and Virginia Tech (VT) and research has been carried out by Rabe et al. [15] in serpentine inlets, analyzing the inlet fan-face distortion in terms of total pressure distribution. Surface-microphone measurements made on different sections of the inlet show that pressure fluctuation measurements can be used to distinguish differences between the separated and the attached flow regimes throughout the inlet. In this case 0.8% and 1% of the overall mass flow has been injected just before the separated region to control the separated flow and improve fan pressure distortion. The face distortion was of circumferential nature and both cases showed a distortion migration that resulted in an improved fan-face pressure distribution.

Experimental investigations on boundary layer control have been carried out by McLean and Herring [16] using multiple discrete wall jets for delaying separation. The effectiveness of a spanwise array of small discrete blowing nozzles has been used in order to analyze the overall performance of a combination of spacing, axial location and momentum flux of the nozzles. The study showed that over a wide range of axial locations the performance of the jets was not strongly sensitive to axial location. Performance continued to improve as the nozzles moved upstream but the rate of improvement was not large. When the nozzles were moved downstream into the immediate vicinity of the unblown separation point, performance deteriorated noticeably. This was based on the insufficient distance between the nozzle and the separation point, which stopped the merge of the jets. The spacing must be more than two nozzle spacing from the separation point. The study also showed that very high momentum flux corresponds to a disimprovement of performance because of an increase in skin friction losses.

Hernandez et al. [17] investigated numerical active control of two unsteady separated flows by introducing periodic disturbances near the point of separation. Using suction and blowing at the wall it was shown that a privileged

frequency of disturbances can greatly reduce pressure fluctuations up to eight times and the mean reattachment length has been reduced by 20%.

Studies have also been made for controlling boundary layer separation using low profile, sub-boundary-layer scale vortex generators known as micro-vortex generators. Lin [18] from NASA's Langley research centre showed that embedded streamwise vortices provide the most effective and efficient means of mitigating flow separation in turbulent boundary layers. The study was made on a high-lift aerofoil and separation alleviation on the flap significantly increased the lift on the order of 10%, reduced the drag on the order of 50% and increased lift-to-drag ratio on the order of 100% for a high lift aerofoil that was optimized for maximum lift. The main advantage of micro-vortex generator devices is their size, which makes it possible to use without extracting any penalties.

McKinzie [19] studied the control of boundary layer separation over a rearward facing ramp by using mechanical excitation. The study showed that by using an oscillating vane for controlling the separation, there are two variables that affect the control performance. These variables are the vane's oscillating frequency and the displacement amplitude of the vane. A parametric study of the maximum values of the static pressure recovery as a function on excitation frequency for several inlet velocities at constant displacement amplitude showed that the optimum excitation frequency increases in proportion with inlet velocity. At the cases of changing displacement amplitude at constant oscillating frequency it can be seen that pressure recovery increases with increasing displacement amplitude until a maximum is reached.

Haas et al. [20] review the progress in the development of airflow control for aero engine combustors using the phenomenon of boundary layer separation. Three concepts have been conceived and anal used based on controlling separation. The first was an upstream flow injection that employed a two-dimensional jet blowing in a cross flow. The second concept employs boundary layer suction and the third configuration employs control flow downstream. The upstream flow injection concept has provided the highest performance. The boundary layer control configuration did not provide sufficient flow separation in order to draw any final conclusions. The downstream blowing concept required very large amounts of control flow to achieve a significant improvement.

Mccormick [21] studied a new concept for boundary layer separation control, which converts acoustic oscillations into mean fluid motions. This concept has an acoustically excited neck that is curved in the downstream tangential direction. In this manner the boundary layer flowing over the neck is energized via suction removal of the approaching low momentum fluid on the in-stroke and tangential blowing on the out-stroke thereby making it in the time average more resistant to separation. It has been showed that the concept is energy efficient and can completely suppress boundary layer separation.

Erbsloh et al. [22] investigated the application of flow control to a civil turbofan intake for the purpose of relaxing the compressor face distortion due to crosswind constraint at take-off. Flow control was implemented using both vane and air jet vortex generators. Experimental results showed that compressor face total pressure distortion compared to the unactuated baseline could be reduced by a maximum of 52% using the vane vortex generators and 34% using the air vortex generator jets. The air bleed required for the air jet vortex generators was approximately 0.1% of the engine mass flow. The study also showed that flow control efficiency and effectiveness are affected by jet velocity and orifice spacing.

While the vortex generator jet concept used for controlling boundary layer separation has been studied for a variety of flows, investigations that consider the effect of intake flow improvement to compressor performance and overall engine performance are scarce. Hence, the primary objective of the current study was to evaluate the effect of flow control jets implementation on the total pressure recovery of the intake and the effect that this has in the overall performance of an aero engine.

1.2.2 Flow Control Methods Assessment

Based on the literature review presented in the previous section of this chapter and the control strategies and classification schemes review, a matrix was created where various techniques of flow control have been considered and an assessment was carried out on the applicability of these methods for the purpose of this work.

The purpose of this study is to introduce sophisticated control systems that can be used to alter the flow field of an intake and improve engine design and performance. Based on this consideration some selecting parameters can be

introduced that could help to assess the applicability of flow control techniques into this study.

These parameters involve the cases of wall bounded flows, active methods and systems, reactive feed forward or feedback loops, drag reduction, mixing augmentation, separation prevention, etc. Based on these parameters the assessment of various flow control methods has been carried out.

The matrix that is presented below, in table 1.1, has been produced based on a review into theoretical, computational and experimental studies that have been carried out on the basis of flow control techniques and their benefits. Many discussions with people involved in this area also helped in the production of this table. For the creation of table 1.1, Reference 2 has been used.

Methods	Strategies	Classify	Goals	Disadvantages
Suction	Transpiration	Passive and Active	Stability modifying Wave Cancellation Velocity Profile	Backflow Maintainability Reliability
Shaping	Pressure Gradient	Passive	Stability modifying Velocity Profile	Passive nature
Wall Cooling and Heating	Viscosity Gradient	Active	Stability modifying Wave Cancellation Velocity Profile	Applicable mainly for hypersonic cases
Plate Vibration	Wall Motion	Active	Wave Cancellation	Impractical, only used in experiments
Compliant Coating	Wall Motion	Passive	Wave Cancellation	Passive nature
Injection	Transpiration	Active	Momentum Addition Mixing Augmentation	Weight of system Bleed-air penalty Turbulence Increase
Fixed Surfaces	Pressure Gradient	Passive	Momentum Addition Mixing Augmentation	Passive nature
Wall Roughness	Pressure Gradient	Passive	Mixing Augmentation	Passive nature
Massless Jets	Transpiration	Active	Momentum Addition Mixing Augmentation	Early stages of research

Table 1.1 Flow Control Methods Matrix

The purpose of this study is to introduce sophisticated control systems that can be controlled in an active manner in order to achieve the best quality flow to

introduce into the engine. For this reason some of the techniques above are introduced with a disadvantage due to their passive nature. Of course passive nature control methods have a great advantage which is their simplicity, but because of the variation of flight conditions that this study is concerned, the focus is towards methods that can be controlled at any given condition.

From the active methods reviewed above, two of them are of impractical nature. The plate vibration due to the impracticality that introduces to the structure, and the wall cooling and heating due to the reason that it can only be used in hypersonic applications where cryogenically-fueled aircraft are concerned and requisite heat sink can be provided.

Therefore, the methods that are left to consider are all based on a transpiration strategy and involve suction, injection and a combination of two with the use of massless jets.

Suction is a well used method that has been studied and tested in many occasions. The disadvantages though are quite considerable because of the effect that have in economical areas. Because of the nature of the structure where many holes are used in order to achieve a desirable change, the maintainability and reliability of these surfaces are a major issue. The optimization of suction rate and distribution is also a major problem. In a more aerodynamically study the application of suction shows a case of backflow which has a negative effect to the flow field.

Injection is another active method that can be used for the purpose of this study. This method is based on the idea of introducing bleed-air from the engine to the intake flow field in order to increase near wall momentum and mixing of the flow. The disadvantages of this method are the weight of the system due to the use of pumps that are used to circulate the bleed-air and the penalty that this bleed-air introduces into the performance of the engine. There is another disadvantage of disturbances introduced into the flow field due to injection but experimental studies showed that these can be controlled with the use of pulsed injection jets into the main flow.

The final active method of this review is the use of massless jets. This is method that is based in both suction and injection but eliminating most of the disadvantages of both methods explained above. The idea of this method is based on the use of a type of membrane below the main surface that can be

actively driven and using the main mass flow introduce a suction/injection system without the use of bleed-air from the engine. This can be seen in Figure 1.1.

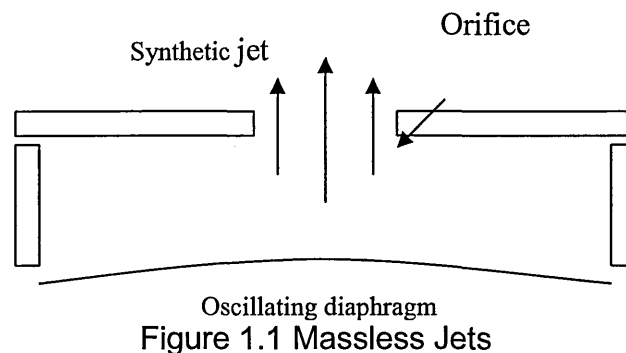


Figure 1.1 Massless Jets

With such a system main disadvantages of suction and injection methods can be overcome. For example there is no penalty to the engine performance due to bleed-air. The weight of the system is not a main consideration anymore. Maintainability and reliability of suction surfaces due to blocking is not a problem because of the jet injection that follows. Finally there can be a balance between momentum addition near the wall from injection and backflow due to suction. One of the main concerns of this method at this stage is the amplitude of these massless jets and how can they bring about the desired changes to the flow.

1.3 Project Methodology

From the literature review it is apparent that there has been considerable investigation into analyzing various methods of controlling boundary layer separation both for internal and external flows. There has been further research on internal flow control separation for serpentine inlets. Most of this research is based on the use of experimental test facilities and some on computation fluid dynamics modeling and simulation. These studies have been carried out taking into consideration the intake component individually raising further questions when considering intake/engine compatibility and overall engine performance.

For this reason this study has been concentrated in developing a methodology that takes into consideration the modeling, simulation and analysis of the intake, compressor and overall engine performance and the effects of flow distortion and control. One of the advantages of this overall analysis can be seen for example in the amount of mass flow extracted from the engine and used for the secondary flow injection. Many investigations have been carried out on injection

flow control methods that use air mass flow extracted from the engine. Although these studies show an improvement in fan-face distortion, it is impossible to analyze if this improvement is sufficient to overcome the penalty that the engine suffers from the air extracted.

Based on this objective a methodology has been developed that takes into consideration the modeling and simulation of intake/compressor/engine integration. The main areas that are involved in this study are:

- Engine simulation and performance
- Intake modelling
- Intake/engine compatibility
- Flow distortion modelling
- Intake distortion management
- Control systems integration

Figure 1.2 gives a simple representation of the flow diagram, based on the methodology used for this study.

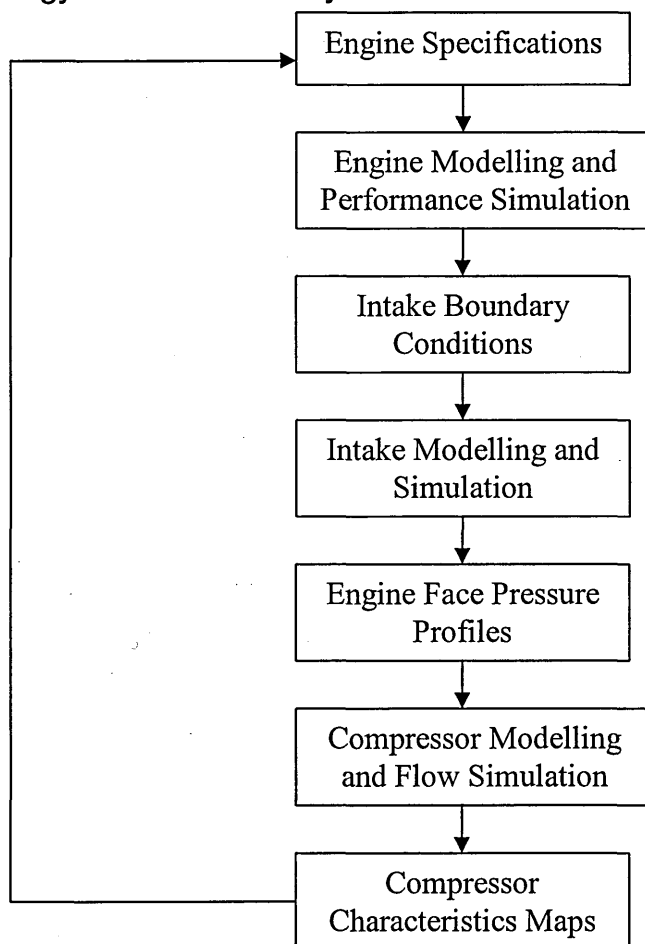


Figure 1.2 Methodology Flow Diagram

Following the flow diagram in Figure 1.2, the procedure starts by gathering the engine specifications for the study. Having obtained the parameters needed to model the design point performance of the engine, for example mass flow, pressure ratio, bypass ratio, turbine entry temperature etc, the next step is to run the performance simulation of the engine at off-design conditions in order to obtain the mass flow rate that the engine requests at different shaft rotational speeds.

Once the mass flow of the engine is given the next step is to run compressible flow calculations in order to obtain the intake boundary conditions in terms of pressures and temperatures for specified intake geometry. Having calculated the boundary conditions the flow simulation of the intake can be achieved with the use of CFD packages. The outlet results include the pressure profile that needs to be extracted and used as inlet conditions for the compressor flow simulation.

A FORTRAN program is used in order to transfer the outlet conditions of the intake and translate them into inlet conditions for the compressor model. This provides the conditions needed to run the compressor model and simulate the flow characteristics. The outlet results of the compressor model are analyzed in terms of mass flow, pressure ratio and efficiency. These results give the final compressor characteristic maps.

Once the complete maps are provided from the CFD simulations, the data are transferred to the engine performance simulation code and introduced as non-dimensional compressor characteristics. The engine simulation is carried out for different off-design conditions and the overall engine performance is analyzed in terms of thrust, specific fuel consumption etc.

1.4 Overview of Dissertation

The chapters that follow are based on the three components integration and an analysis is being carried out on modeling simulation and overall performance of these components.

Chapter 2 presents an overview of previous experimental and computational studies on intakes and engine face distortion analysis. Then the modeling and simulation of the M2129 s-shaped duct is being carried out followed by a discussion on grid generation and formulation of boundary conditions. CFD

Fluent commercial package has been used to simulate the intake flow conditions. An investigation into grid generation and turbulence models has been carried out before the final validation of the flow conditions with previous experimental results. A representation of the RAE intake model M2129 can be seen in Figure 1.3.

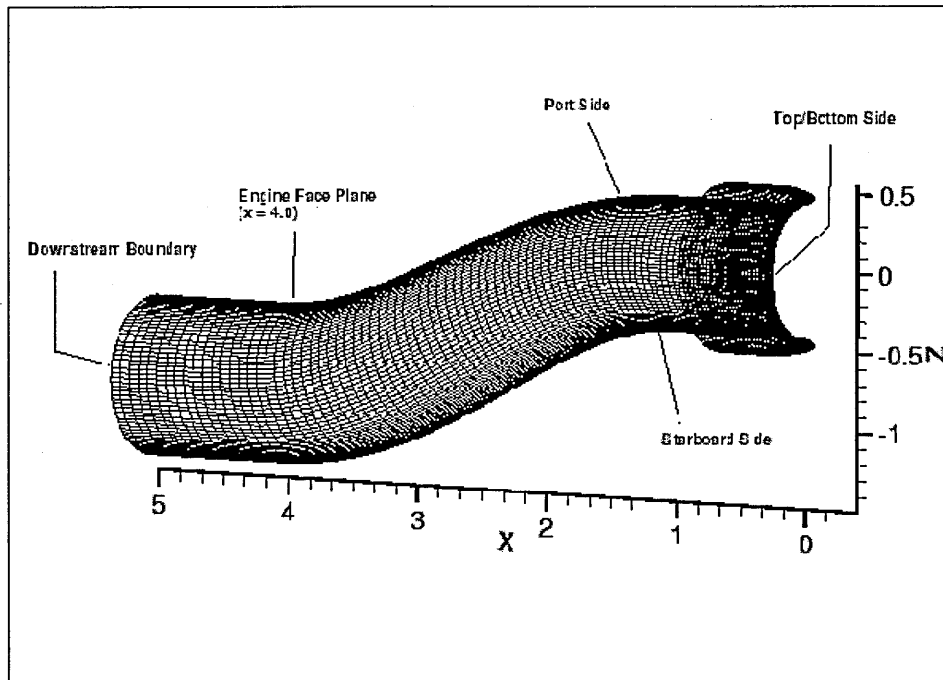


Figure 1.3 RAE Intake Model M2129

Chapter 2 continues by examining the operation of an air intake at normal conditions (computationally steady) for various mass flow rates. Further steady intake study is being carried by examining the problem of intakes at incidence. Various angles of yaw and pitch are studied, focusing on the effects on pressure recovery and distortion. Various mass flow rates are again examined. Finally, the second chapter ends with the results of flow control cases for different mass flows and incidences using both injection and suction flow control methods.

Chapter 3 presents an introduction on axial compressors and the computational fluid dynamics approach. Following the introduction is the fan design of a low bypass two spool turbofan engine and the consideration into design guidelines, limitations, variables and processes. The fan design is carried out and the final design specifications are presented. The fan preliminary design is followed by the generation of the three-dimensional geometry of the blade and the definition

of the camber line, blade thickness and final blade shape. Once the 3d-geometry is produced the next section covers the grid generation and an introduction into CFX-Turbogrid that is used for this purpose. This section ends with the grid size comparison and the final grid generation. The final grid of the rotor/stator passage can be seen in Figure 1.4. The sections that follow in this chapter analyze the simulation of the clean, distorted and flow control cases with further discussions in numerical methods used, model performance analysis, model performance improvements and different simulation techniques for greater accuracy.

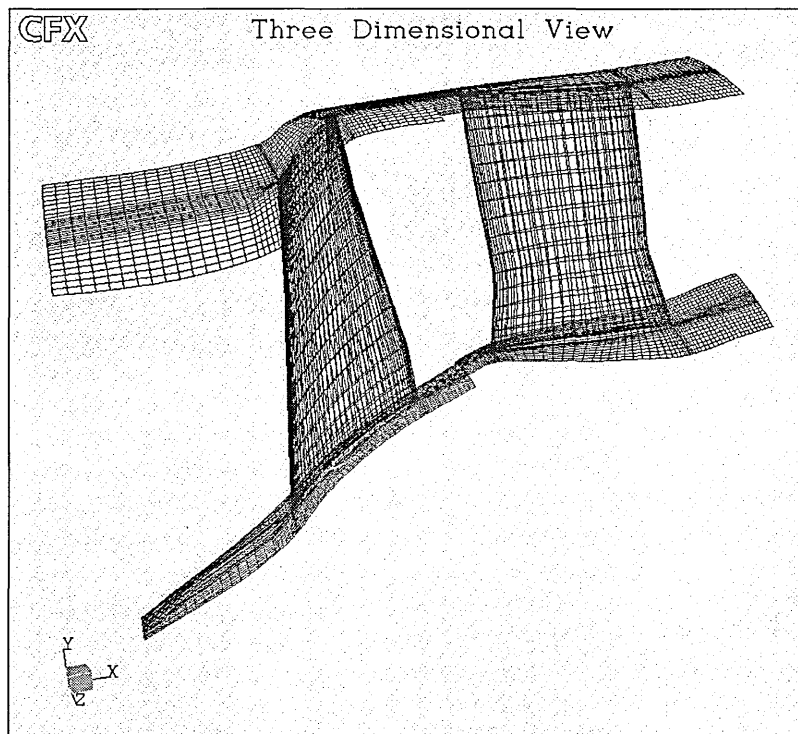


Figure 1.4 Rotor/Stator Passage Grid

Chapter 4 covers the work on engine modeling and performance simulation and the analysis of distortion effects on overall engine performance. The chapter starts with an introduction on engine modeling and continues with the modeling and simulation of the low bypass, two spool turbofan engine. Figure 1.5 shows a representation of a two spool low bypass turbofan engine.

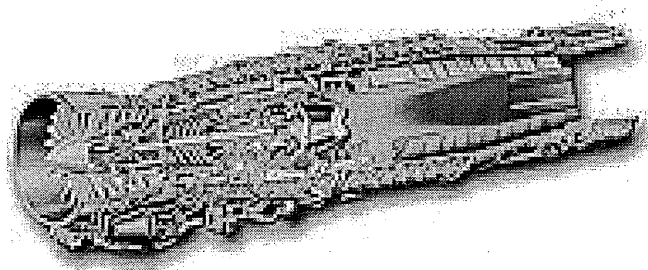


Figure 1.5 Two spool low bypass turbofan engine (Courtesy of Rolls Royce)

Results of the engine performance parameters such as variation of thrust and specific fuel consumption with flight Mach number, altitude and intake pressure recovery are presented and trend lines analyzed. A study into the effects of bleed-off takes is being carried out and different amounts of air bleed have been extracted from different stages of both the low and high compressors to investigate the penalties suffered by the engine. Finally an analysis is being carried out on engine pseudo-transient performance using a steady state code, hence the term pseudo-transient modeling and simulation. This final study helps to investigate the effect of improved engine face distortion into compressor surge margins at transient conditions by analyzing the movement of the operating line of the engine on the compressor characteristics map.

The three chapters mentioned earlier include the modeling, simulation and validation of the main components of this work methodology. Chapter 5 presents various cases following the complete methodology steps and results. The cases considered include suction and injection flow control methods, pitch and yaw angles, and comparisons between various performance results.

The work is completed with the sixth and final chapter, which contains the overall discussion and conclusion of this work and suggestions on future improvements of the methodology and further work on flow control investigations.

CHAPTER TWO

Intake modelling and Flow Simulation

2.1 Introduction

This chapter covers the intake component used for this study. A review of previous computational and experimental studies has been carried out and issues on intake modelling and flow simulation are discussed. Finally the validation of the RAE M2129 intake duct and the analysis of CFD results are presented.

2.1.1 The Design and Role of an Air Intake

Intake and engine compatibility is a crucial issue for the overall performance and handling attributes of an aircraft. The efficiency of the intake can have major impact to the performance of the engine both on effectiveness and safety terms.

Intake design parameters are set, based on the idea of air being supplied to the engine at various engine operating conditions by minimising pressure losses and provide a uniform stream of air to the engine face. This uniform stream of air is defined as the internal flow and can be described as one in which pressure, temperature and density are uniform in the radial direction (a direction normal to the centreline of the intake). External flow does not enter the air intake but is affected by the presence of the intake and so is still of vital importance as factors such as aerodynamic drag will be influenced.

There are a number of operational requirements that can affect the design of an intake and introduce further complexity to the component. Placement of store bays, and location of undercarriage wells can influence the design. Nowadays an interest is shown in developing low observability aircrafts in order to reduce the radar cross section. Those type of aircraft need intake designs that are able to hide the compressor face since this is a strong source of radar deflection. Ways that this can be achieved include the use of radar absorbing materials or use of more complex highly offset design intakes in order to eliminate a line of sight view from the intake to the compressor face. External surfaces can be shaped so that all radar reflections get diverted away from the direction of the threat.

Another important parameter in intake design as mentioned earlier, is the total pressure losses. Engine intakes should be designed in order to avoid major pressure losses. In general intake performance can be characterised by high total pressure ratio, good uniformity of flow across the engine plane, low

installation drag, low radar signatures, and low weight. These factors should all be considered and yet the final intake design must still meet longevity and reliability targets.

Subsonic and supersonic intakes tend to vary considerably. Subsonic intakes usually have fixed geometries. Due to the low speeds encountered it is possible for subsonic intakes to draw in air from a greater area than the highlight area. Thus, variable intake geometry is not required. The diffusing part of the intake tends to be shorter in length due to the lower speeds. However, longer diffuser parts are sometimes needed as on the RAE intake model 2129. This, for example, may be because of the need to bend the intake round an undercarriage well or weapons bay or for stealth reasons. Splitter plates are sometimes also used to help obscure the engine face for stealth reasons. Some well-known examples of aircraft that utilise s-shaped intakes include the F16 and Eurofighter Typhoon. The proposal by Boeing Aircraft Corporation for a Sonic cruiser has the engines at the rear of a diamond/delta shaped wing with s-shaped intake ducts supplying the compressor with air.

In the case of supersonic flow the design needs to be able to accommodate more complex flow characteristics and high compressibility effects. A major issue for supersonic intakes is the formation of shock patterns that form as a result of slowing the freestream to subsonic speeds for entry to the compressor. These shock patterns are designed to compress the incoming flow. Moving ramps are required to alter the position of the shock depending on the speed of the aircraft.

The final aspect of an aircraft intake is the downstream compressor face. The primary purpose of the compressor is to draw and compress air into the engine core. Pressure rise is in the direction of flow for a compressor (an adverse pressure gradient) and hence this increases the likelihood of boundary layer separation. Compressor stall leads to a rapid drop in the performance of the compressor and the possibility of engine surge or rotating stall.

2.2 Review of Previous Work

2.2.1 Intake Design and Flow Mechanics

Useful information on intake design and flow mechanics can be obtained from the three references that follow.

Goldsmith and Seddon [23] provided a detailed introduction to intake aerodynamics. This book covers intakes at subsonic and supersonic speeds. Distortion, pressure recovery, lip separation, and incidence are discussed and there are also sections on wind tunnel testing, computational techniques, and various designs.

The mechanics of flow in an enclosed curved geometry have been studied by Miller [24] and a good description has been provided. This includes the forces that act on the fluid and the resultant effect on the fluid flow downstream of the offset. This has obvious relevance to s-shaped intakes. The book also looks at the effect of diffusing geometries.

Useful information on experimental techniques and processes for determining intake pressure distortion data between test methods and facilities have been developed by Dardis and Mayhew [25].

2.2.2 Computational and Experimental Intake Work

A number of studies have been carried out on computational and experimental intake work that covers a number of important issues in modelling, simulation and experimental test procedures.

An investigation into the effect of turbulence models used for simulating the flow characteristics of a highly serpentine duct were investigated by Kral [26]. By comparing the results with experimental data the conclusion was that the two-equation models better predict the flow than the algebraic and one-equation models when compared with experimental data.

Comparison of the three-dimensional Navier-Stokes computational results with new experimental measurements was conducted by Harloff et al [27]. Two studies were carried out with the first study indicating some inadequacies in either the grid resolution or algebraic turbulence model used. This study used a finer grid and the $k-\epsilon$ turbulence model. The results are in reasonable agreement but both turbulence models under-predict the length and angular extent of boundary layer separation and in both cases initiated further downstream than witnessed in experiment. They concluded that neither turbulence model adequately accounts for strong secondary flows with separation.

A study into the s-shaped intake duct M2129 has been carried out by Abrahamsen et al [28] using experiments and computations and the results are

compared with available experimental data. The study showed that both low cost experimental and computational methods are viable. The improved computational method consists of a non-linear numerical modelling approach to improve the predictive capability of CFD. The results obtained appear good although the paper only examines the relatively simple low mass flow case.

May [29] describes the first of a series of Aircraft Research Association (ARA) reports on s-duct flows. This particular report investigates the flow in the M2129 duct for high and low mass flow rates using several two-equation turbulence models. It was found that wall functions are inappropriate for modelling the secondary flow and separation, whereas a two-equation model, which is integrated through the sub-layer, provides a qualitative prediction in the separated region. It is then postulated that a further improvement may be obtained by including non-linear eddy viscosity terms and modifications to sensitise the model to adverse pressure gradients. The report found that, for the low mass flow case, all the turbulence models used failed to predict any secondary flow.

May [30] again examined the M2129 geometry for low and high mass flow demand. It was found that the two-equation model results for the low mass flow rate case are very similar to the results obtained using the algebraic turbulence model.

The $k-\epsilon$ turbulence model has been used by Zhang and Assanis [31] in order to evaluate the performance and accuracy of a three-dimensional Navier-Stokes flow code. The benchmark used is an S-duct of circular cross-section. They concluded that, given the limitations of the turbulence model in use, the numerical method yields satisfactory results giving a good qualitative description of the pressure field and quantitative prediction of the velocities.

Finally, Menzies [32] carried out an investigation into intake aerodynamics for s-shaped ducts, using computational fluid dynamics. His work involved the modelling of the RAE M2129 intake and the simulation of the flow field of this particular duct. He investigated the use of Euler and Navier-Stokes modelling. An analysis of the duct at low and high mass flow rates was carried out investigating the modelling methods. Euler results compared favourably with experiment and previous calculations for a low mass flow case. The high mass flow case Euler calculations proved more challenging. A converged steady solution could not be found. The N-S results utilising a $k-\omega$ turbulence model were much more realistic.

2.3 Diffusing S-Duct Flows

2.3.1 Terminology

When analysing intake flow field characteristics there are some important parameters that need to be taken into consideration. These parameters include the definition of the aerodynamic interface plane, the capture ratio, the contraction ratio, the pressure recovery, the mass flow parameter and mass flow capture ratio.

When conducting experimental cases it is generally difficult to obtain measurements at the compressor face while the engine is running. The flow field that is used as the aerodynamic interface plane must be forward of the compressor face but sufficiently close to the compressor face to have a very similar flow field. Based on this aerodynamic interface you can obtain engine face distortion values and intake pressure recovery values.

The capture ratio CFR helps describe the extent of the engine demand and can be defined as:

$$CFR = \frac{A_{\infty}}{A_{hl}}$$

Where A_{∞} is the area enclosed between the dividing streamlines (the envelope of the streamlines), i.e. it is the freestream air that actually gets drawn into the engine

And A_{hl} is the area of the disc that is created from the furthest protruding point of the cowl into the freestream (the leading edge).

The Contraction Ratio (CR) is defined as

$$CR = \frac{A_{hl}}{A_{ef}}$$

where A_{ef} is the engine face area.

This is an important geometrical definition used when considering engine demand and relates directly to intake highlight area and intake engine face area.

2.3.2 Mass Flow and Pressure Recovery Definitions

The Mass Flow Parameter (MFP) is a convenient term and can be defined, noting that the equation is a unique function of M (the local Mach number) only in a calorifically perfect gas, as

$$MFP = M \sqrt{\frac{\gamma}{R} \left(1 + \frac{\gamma-1}{2} M^2 \right)^{\frac{1}{2\gamma-1}}}$$

The value of the MFP peaks at around 0.0485 for a Mach number of 1.0 and falls thereafter in the supersonic regime. It is also useful to note that the formula for the MFP can be manipulated to give the Mass Flow Rate, MFR that can be defined as

$$MFR = \frac{A_{\infty}}{A_{hl}}$$

The MFR is also known as the Capture Flow Ratio, CFR. This parameter can be used to quantify engine demand. Based on the consideration of flow speed, the value of MFR changes from being greater than one for subsonic speed as the intake can 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), to a value that is less than one for high speed supersonic intakes where the intake draws in air from an area less than the highlight area (i.e. the intake is being supplied with more air than it requires).

Another important parameter in duct flow is the pressure recovery (PR). This can be defined as the ratio of the total pressure at the engine face to freestream total pressure (as described later in this section). The total pressure at the compressor face is actually taken at the AIP during wind tunnel testing. PR is defined as

$$PR = \frac{P_{t_{ef}}}{P_{t_{in}}}$$

In high speed flight the intake slows the airflow down for entry into the compressor and produces a corresponding increase in pressure. This is a form of ram compression. The pressure recovery factor is a measure of the efficiency of the intake and is a significant design parameter as a loss in total pressure can be directly related to a loss in engine thrust. Clearly a value of pressure recovery as close to unity as possible is desirable and would indicate an efficient intake with a low distortion across the compressor face and hence low susceptibility to engine surge.

At low Mach numbers air is generally being 'sucked' into the intake and so static pressure tends to suffer an overall drop. Therefore, for practical reasons, the

total pressure is used in the definition even though it only drops in relation to the freestream value. Also total pressure falls when there are losses in the flow that could occur as a result of boundary layer build up, shock waves, and separation. These losses can also be responsible for distortional effects across the compressor face and surge generation. Hence by quantifying these losses we have an effective way of describing the flow.

2.3.3 The Fluid Mechanics of Diffusing S-Duct Flows

The fluid mechanics of the airflow within an intake vary with the geometry of the intake. The RAE intake model 2129 is a diffusing s-duct where the cross-sectional area increases as you travel through the duct. There is also an offset in the y-plane between the highlight plane of the intake and the plane on which the compressor face lies creating an s-shaped type centreline. As a result there are a number of interesting characteristics that these flows exhibit.

After the first bend in the diffuser there is an interaction between the centrifugal pressure gradient and low energy region (such as that found in a boundary layer or separation region). If the air is to travel in a curved trajectory this requires a centrifugal force. As the air is turned, static pressure and velocity distributions change. The centrifugal pressure P_{cent} can be written as

$$P_{cent} \propto \frac{\rho V^2}{R}$$

where V is the mainstream velocity and R relates to the curvature of the bend in question. Due to the fact that the outer wall has a greater radius than the inner wall, the inner wall has a greater centrifugal pressure. For ideal fluid with a uniform energy distribution the static pressure increases with radius to balance the centrifugal force. The sum of the velocity and static pressures is the same everywhere. Hence the velocities decrease from the inside to the outside of the bend.

Real flows involve non-uniform energy distributions. Velocity distributions change from zero at the duct walls to a maximum in the core flow. Centrifugal and pressure forces acting on the faster moving core flow cause it to move towards the outside of the bend. However, there is an adverse pressure gradient created on the outside of the bend (region of increasing pressure). Near wall fluid that is energy deficient and approaches this adverse pressure gradient cannot pass through it. Instead, the flow moves round the walls towards the low static pressure on the inside of the bends. This movement of the low energy region towards the inside of the bend combined with the

movement of the core flow towards the outside of the bend sets up two cells of swirling secondary flows.

For the second bend the low energy flow is largely on the outside wall as a result of the first bend and is not driven back circumferentially by the method described previously. Hence the swirl pattern experienced at the engine face in a double bended intake is in the direction from the first bend and not the second bend.

This swirling flow can change the flow angle of attack on the compressor blades which can then lead to stall. Intake guide vanes (IGV) are fitted to some engines to combat this problem. Swirl is particularly susceptible in offset diffusers such as the one being considered in this thesis. Here the airflow is being delivered to the engine through a double bend (s-bend) as found on aircraft such as the F16.

Compressor blade stalling can act like a solid wall at the compressor face and can result in an engine surge. A resulting shock wave, often referred to as a 'hammershock', travels forwards out of the engine. Some common causes of compressor stalling are high distortion, cowl lip separation, and general flow unsteadiness. The distortion was described as the maldistribution of flow in terms of total pressure at the compressor face. Historically this has generally been a sufficient description of distortion although there have been a few notable exceptions. In one such case the failure to fully understand the nature of the intake flow field and the sensitivity of the engine to it led to some major problems. Swirling flow aggravated the effect of total pressure distortion and surge was encountered when the swirl was contra-rotational to the direction of the fan rotor blades.

2.3.4 Distortion

Engine/intake compatibility is purely concerned with the quality of the airflow that is delivered by the intake to the engine and how the engine is affected. This process should ideally be accomplished with the minimum total pressure loss and the flow distribution should be as uniform as possible. Distortion is the term given to the variation of total pressure across the engine face. Many aircraft have experienced intake compatibility problems due to the effects of distortion. It has been shown that a high degree of distortion can induce engine surge.

As previously mentioned air intakes must limit the possibility of compressor surge and stall. This can be done by eliminating non-uniformities in pressure across the engine face, although total elimination is not possible in real flows. Sources of distortion include wall separation due to high diffusion rates, shock/boundary layer interaction and inadequate boundary layer bleeds. Any non-uniform loss in total pressure across the intake entrance results in a degree of distortion of the flow and this will progress to the compressor face although a degree of attenuation is likely. Local degradation in total pressure leads to changes in the velocity vector orientation at the compressor face, which can cause compressor stall and possible surge.

Distortion can be quantified by a number of parameters and equations. The most popular descriptor used in the United Kingdom and introduced by Rolls Royce is the coefficient

$$DC = \frac{Pt_{ef} - Pt_{\theta}}{qt_{ef}}$$

Here, Pt_{θ} corresponds to the mean total pressure in the sector θ . The sector is chosen relating to the area with the worst distortion. The dynamic head h represented in the equation as qt_{ef} .

For the purpose of extracting data from the CFD cases and analysing engine face distortion in DC60 and pressure recovery form, two programs were created.

AWProfile Program

This program produces the input coordinates that can be read into Fluent CFD package in order to extract the data needed to run DC60 and Pressure recovery (PR) calculations.

Given an engine face radius and bullet radius, and by specifying the number of arms (NA) and number of points (NP), points are being positioned along the engine face to be equally area-weighted. An input coordinates example can be seen in Figure 2.1.

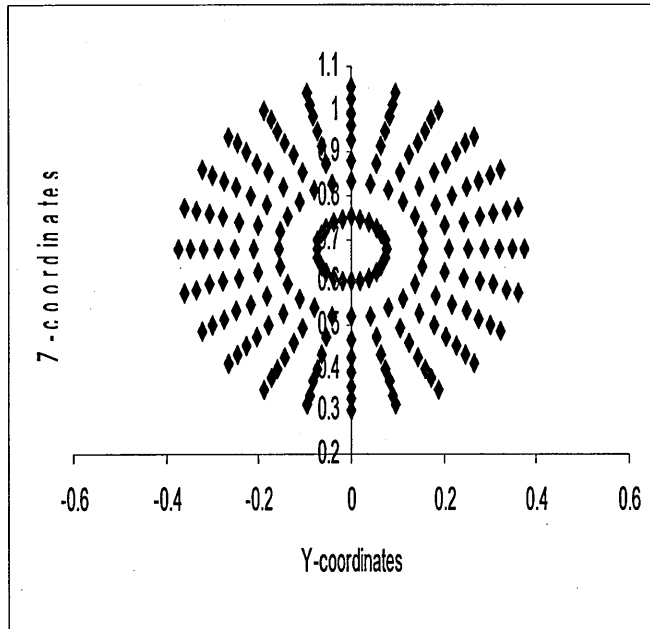


Figure 2.1 Input Coordinates

DCCalc Program

This program is used to analyze the data received from Fluent and run DC60 and pressure recovery calculations.

Given the pressure values obtained from Fluent for each point DC60 can be calculated.

As annuluses are equally area-weighted:

Engine face mean total pressure is calculated by

$$Pt_{ef} = \frac{SPt}{NA * NP}$$

SPt is the SUM of total pressures obtained by Fluent

NA is the number of arms

NP is the number of points at each arm

The mean total pressure for each arm can also be calculated

$$Pt_{arm} = \frac{SPt}{NP}$$

Next step is the calculation of the mean total pressure at 60° sector

$$Pt_{60} = (1/4) * (0.5 * Pt_{arm23} + Pt_{arm24} + Pt_{arm1} + Pt_{arm2} + 0.5 * Pt_{arm3})$$

Where the arm numbers are the arms around the circumference shown in Figure 2.1

Finally DC60 and PR can be calculated as

$$DC = \frac{Pt_{ef} - Pt_{60}}{qt_{ef}} \quad PR = \frac{Pt_{ef}}{Pt_{in}}$$

2.4 Numerical Formulation

This section describes the computational model used for the study. The code is introduced in the form of a summary of the features and techniques employed. Following a description of the code grid generation is considered. Finally there is a full description of the boundary conditions used, particularly those at the engine face.

2.4.1 Grid Generation

Grid generation for the s-duct is challenging due to the nature of the duct. The geometry examined in this study is the RAE intake model 2129, shown in Figure 2.2 and was used as test case 3 by AGARD Working Group 13 [33].

The duct geometry consists of a circular intake section, one engine face diameter long and of constant cross section, joined smoothly into an s-bend diffuser of circular cross section. There follows a further constant area axisymmetric cross sectional piece of one engine face diameter in length that terminates at the downstream boundary. The duct is circular in cross section throughout its length.

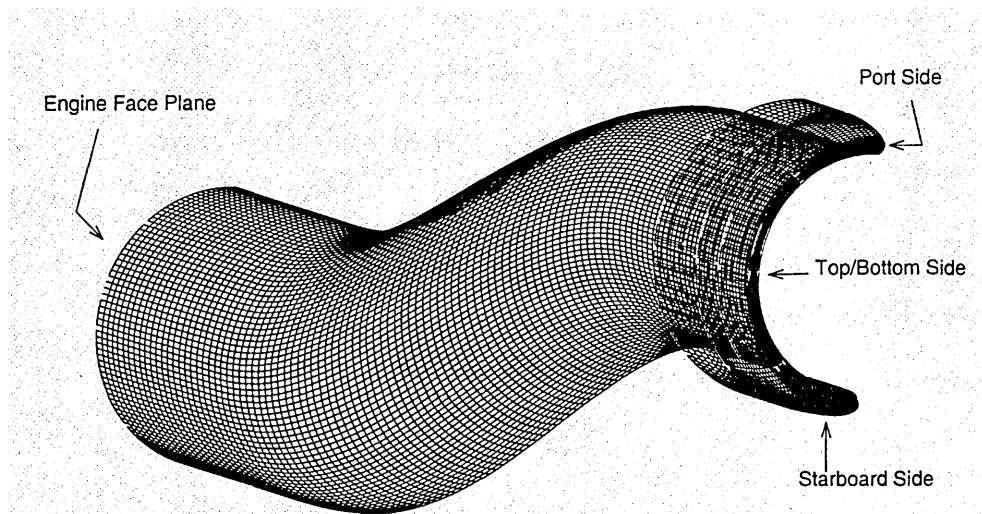


Figure 2.2 RAE Intake Model M2129

The offset of the centreline and the variation of the radius in the diffusing part of the intake can be defined by the following two equations:

$$\Delta Z = \Delta Z_{cl} \left[1 - \cos \left(\pi \frac{X_{cl}}{L} \right) \right]$$

$$\left(\frac{R - R_i}{R_{ef} - R_i} \right) = 3 \left(1 - \frac{X_{cl}}{L} \right)^4 - 4 \left(1 - \frac{X_{cl}}{L} \right)^3 + 1$$

The dimensions of the model used for this study were:

Duct length, $L = 4$ meters

Duct intake throat radius, $R_i = 0.338$ meters

Duct engine face radius, $R_{ef} = 0.375$ meters

Duct centreline offset, $D_z = 0.675$ meters

The Gambit commercial package was used to generate the geometry and grid. Gambit is the package supplied with Fluent that allows for all aspects of surface and solid modelling. The geometry of the intake model has been created in different blocks. Figure 2.3 represents the model and shows the intersections between the various blocks.

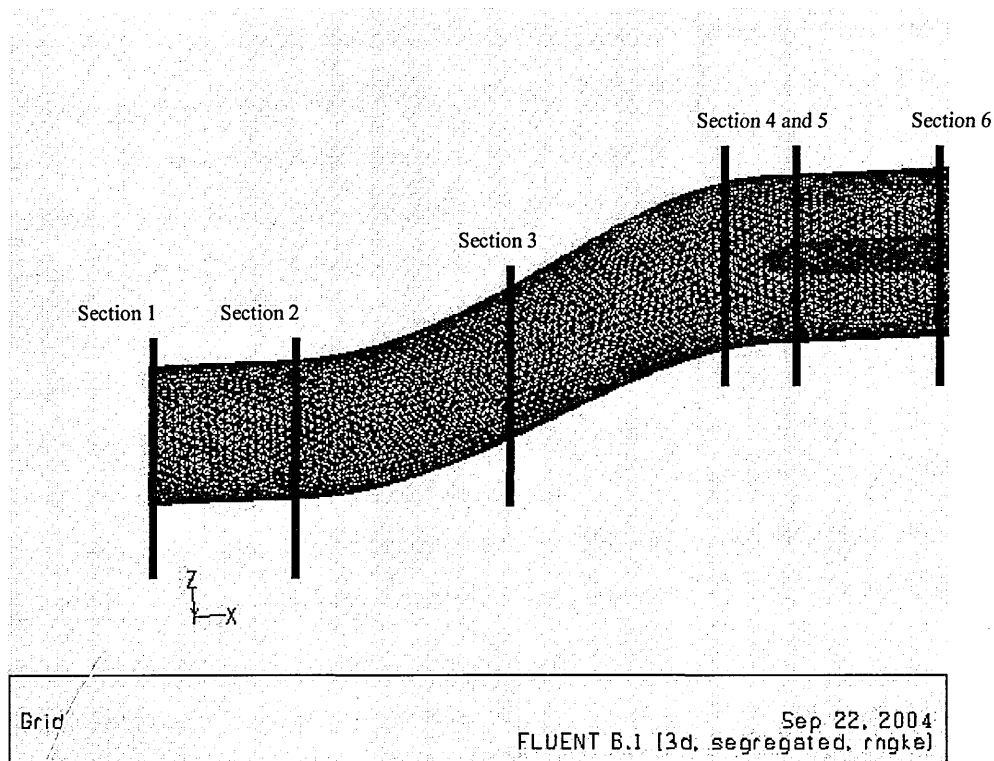


Figure 2.3 Model Blocks

Section 1 – Intake throat

Section 2 – Diffuser inlet

Section 3 – Diffuser mid

Section 4 – Diffuser outlet

Section 5 – Engine face

Section 6 – Intake outlet

Further case results will be analyzed based on these stations format.

2.4.2 Engine Face Boundary Conditions

It appears that applying a constant static pressure at the compressor face is a commonly used approach for the boundary condition and is the most straightforward method for simulating the engine face for strictly subsonic outflow. Having the areas of the intake the outlet static pressure can be calculated by specifying a mass flow rate. This mass flow rate is obtained from the engine performance model at specific atmospheric conditions. The downstream boundary is usually placed sufficiently far downstream from the aerodynamic interface plane, at least one engine face diameter length. A constant value of static pressure is not valid through the s-duct nor is it true for the total pressure at any location. Static pressure is only assumed constant at the engine face plane.

2.4.3 Turbulence Modelling

One of the aims of this work is to assess the performance of various turbulence closures in modelling complex internal flows. The flow is challenging with complex secondary flows and strong pressure gradients generated by localised acceleration and deceleration, placing high demands on turbulence models. Turbulence is an eddying motion that exists at high Reynolds numbers. Turbulence has a wide spectrum of eddy sizes with a corresponding spectrum of fluctuation frequencies. Turbulence has prevailing rotational motion that can be thought of as a tangle of vortex elements with highly unsteady vorticity vectors that are aligned in all directions. The largest eddies have sizes on the same order of magnitude as the flow domain, have low frequencies, and are effected by the boundaries and the mean flow. The smallest eddies, on the other hand, are determined by the viscosity of the fluid and have high frequency fluctuations. As the Reynolds number of a given flow increases, the width of the spectrum, or the difference between the largest and smallest eddies, increases. The large eddies extract kinetic energy from the mean motion and feed it to the large scale turbulent motion. The eddies may be considered as vortex elements that stretch each other. Due to this vortex stretching, energy is passed down the

cascade to smaller and smaller eddies until viscosity causes the dissipation of the eddies. Turbulence modelling remains a challenge in CFD.

For calculations employing turbulence model a variety of turbulence closure techniques are available. The two most popular are algebraic and the two-equation model (e.g. $k-\epsilon$ and $k-\omega$). This work has used the $k-\epsilon$ standard model, the $k-\epsilon$ RNG model, the $k-\omega$ model and a hybrid of the $k-\omega$ model called the SST model. The $k-\omega$ model has been widely used and has been successful, particularly for two-dimensional flows with adverse and favourable pressure gradients. It has also been found that the model appears to match measured properties of recirculating flows with no changes to the basic model and its closure coefficients. Problems found with the model include an unreliability when used in flows with boundary layer separation induced by an interaction with a shock wave. There have also been reported inaccuracies with flows over curved surfaces in some circumstances. The shear stress transport (SST) turbulence closure is a two-equation model that is a hybrid of the $k-\omega$ model. The SST model was devised in order to improve the prediction of the extent of separation in flows dominated by adverse pressure gradients. To this end it was expected that SST predictions would show improvements over standard $k-\omega$ predictions for flows in diffusing offset intakes. The $k-\epsilon$ on the other hand is a more stable model giving a more reliable prediction of turbulent flow characteristics, but under predicting highly recirculating flows like the M2129 flow field. For this reason, the $k-\epsilon$ RNG model can be used where the effect of swirl on turbulence is included, hence enhancing accuracy for swirling flows and improving flow simulation for this particular intake.

2.5 S-Shaped Intake Model Validation

A study has been carried out to validate the intake duct used for this study with data from previous studies on the M2129 intake model. The main parameters that affected the results were the grid and the turbulence models.

2.5.1 Grid assessment

Grid generation is a complicated process and with major effects to the overall flow performance simulation. One of the main parameters of grid generation is the wall treatment and the creation of cells close to the wall. To start with, a coarse grid was produced. The mesh consisted of 148000 cells with no special wall treatment. This coarse grid could not give any accurate results for engine face distortion or duct pressure recovery. Maximum y^+ for this grid was 850.

Fluent CFD package suggest a y^+ value of 60 to 70. So it was necessary to introduce a finer grid that would give a reasonable y^+ value. The final grid produced for this work has a mesh consisted of approximately 487000 cells. This grid has been produced with the introduction of a boundary layer cell generation in order to increase the accuracy of the flow simulation near the wall. The effect of swirl on turbulence, as mentioned above, is included in the RNG model, enhancing accuracy for swirling flows. Effective use of this feature does, however, depend on an appropriate treatment of the near-wall region. The wall treatment introduced in this model can be seen in Figure 2.4 where the outlet of the model is presented.

This grid generation method reduced the y^+ value to 68. Figures 2.5 and 2.6 present the results of both the coarse and fine grid in terms of engine face distortion DC60 and pressure recovery with respect to throat Mach number.

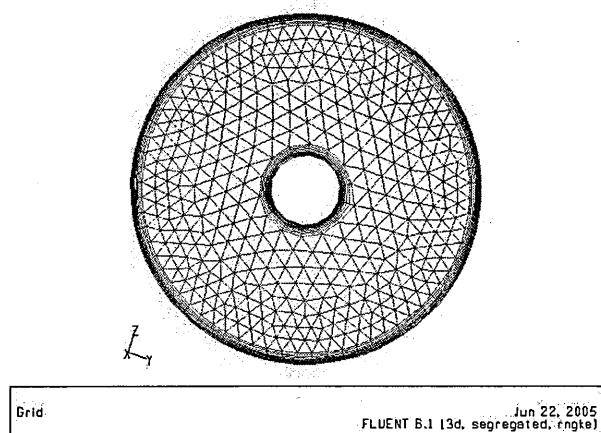


Figure 2.4. Wall treatment

Figure 2.5 shows results for both experimental and computational results in terms of DC60 parameter with respect to throat Mach number. The results show that the coarse grid under predicts engine face distortion and this is due to the limited accuracy of the flow simulation near the wall. This grid cannot obtain any flow separation characteristics of the M2129. The finer grid shows a great improvement in engine face distortion due to the improved grid analysis close to the wall. This gives the ability to the model to predict accurately enough, and even over predict as the results show, the flow separation region of the duct.

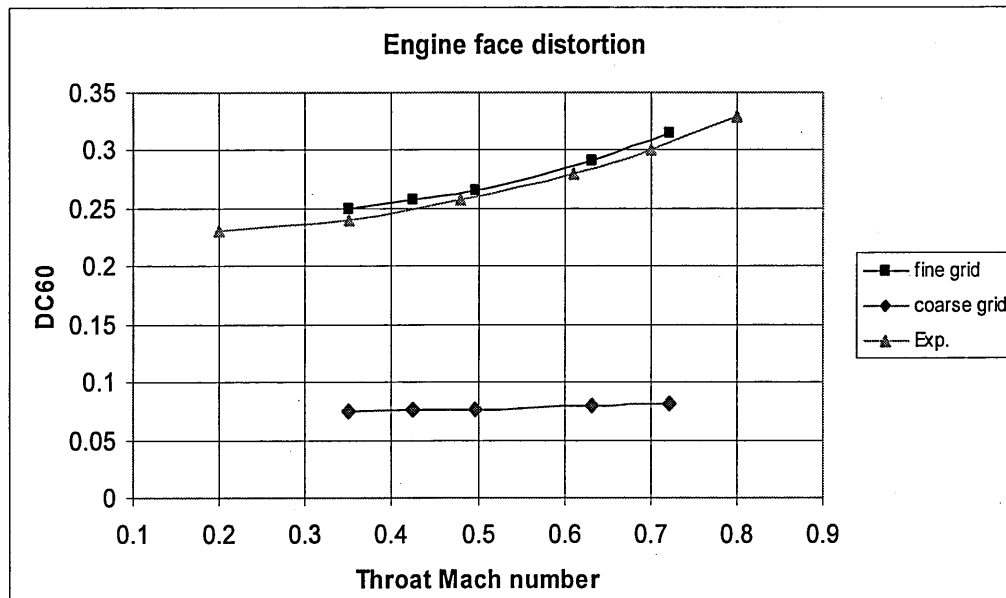


Figure 2.5 Grid comparison in DC60

In terms of intake pressure recovery, again the coarse grid cannot predict the pressure losses that the experimental results showed. This is because the flow simulated with the use of the coarse grid seems to be rather steady, hence introducing very small pressure losses to the flow. The finer grid again seems to over predict pressure recovery especially at higher mass flow rates.

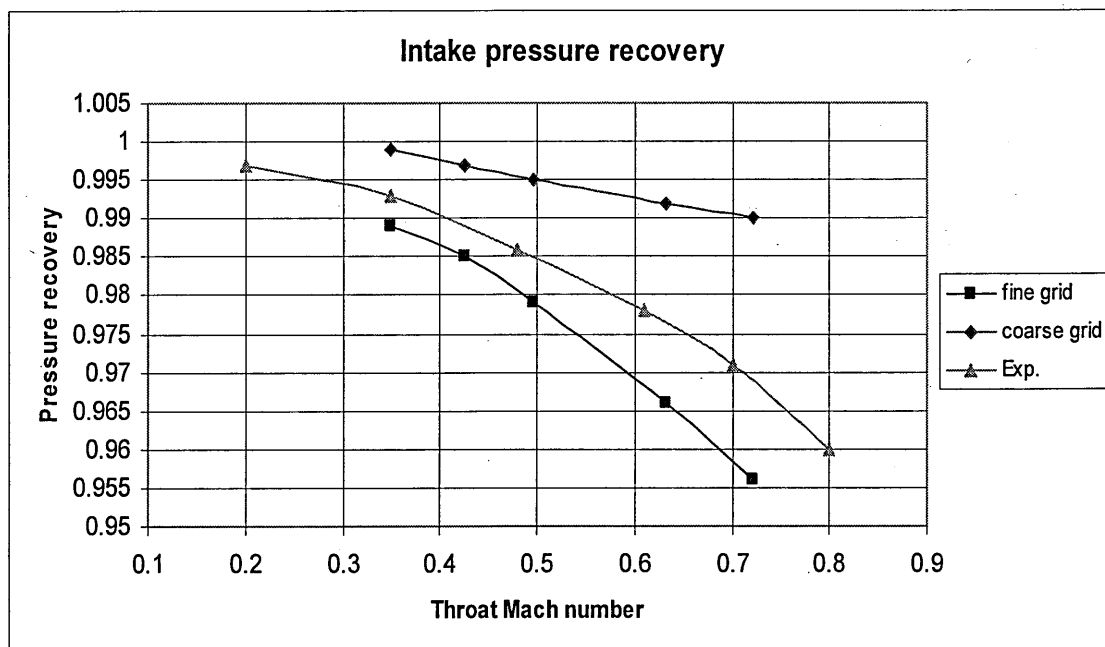


Figure 2.6 Grid comparison in pressure losses

2.5.2 Turbulence models assessment

Turbulence models assessment is an important part of computational fluid dynamics. Many studies have shown that different turbulence models can be accurate for different flow simulations. Studies mentioned earlier in this chapter, have shown that the use of the three-dimensional Navier-Stokes equations give reasonable results for this type of study. K- ϵ standard turbulence model has been shown to under-predict results for boundary layer separation and the intensity of any flow recirculation regions.

This section presents the results obtained from Fluent CFD code with the use of different turbulence models in order to simulate the highly complex flow of the M2129. The models used are the k- ϵ standard model, the k- ϵ RNG model, the k- ω standard model and the k- ω SST model. These results have been compared with experimental results and results obtained from a different CFD code known as FALCON. One of the models that are used the most is the k- ϵ standard model. Table 2.1 summarise the results obtained with the use of these turbulence models. It can be seen that all models give a lower prediction than the experimental data, whereas the RNG model slightly over predicts the engine face distortion and under predicts the pressure recovery.

	EXP	FALCON	k- ϵ	RNG	k- ω	SST
DC60	0.261	0.272	0.246	0.266	0.115	0.143
PR	0.985	0.986	0.98	0.981	0.9786	0.976

Table 2.1 Turbulence Models Comparison

A better understanding to the results that the turbulence models predicted can be seen in Figures 2.7 and 2.8, where engine face distortion and pressure ratio is plotted with respect to throat Mach number for every turbulence model.

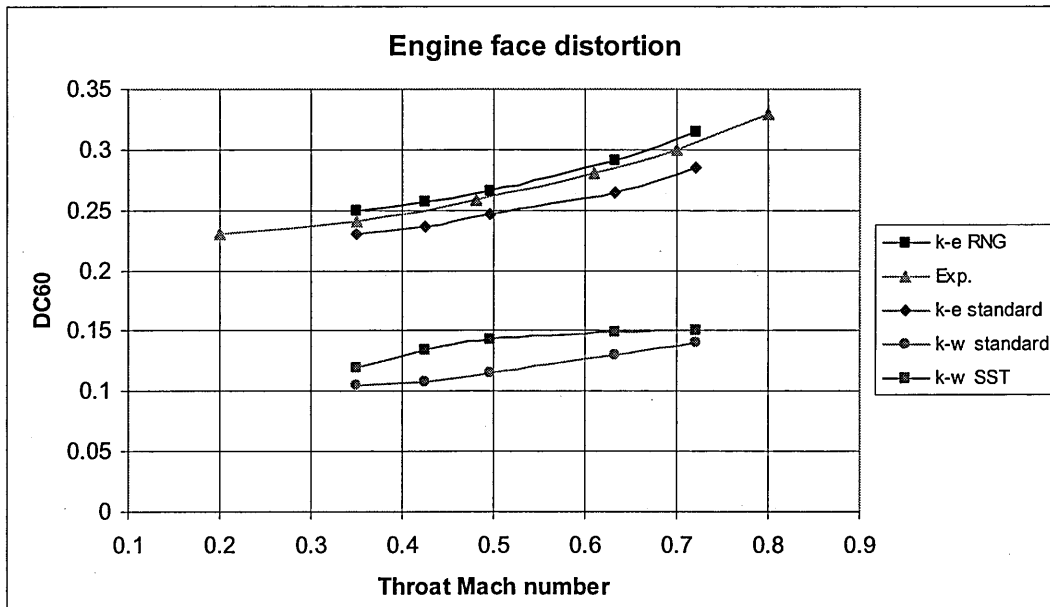


Figure 2.7 Turbulence models comparison in DC60

In Figure 2.7 it can be seen that all models give a lower prediction to engine face distortion except the k-ε RNG model. The reason for this is the ability that the particular model has in predicting more accurately highly flow recirculation regions. The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows therefore improving flow simulation for this particular intake.

The pressure recovery plotted with respect to throat Mach number can be seen in Figure 2.8. Here all models seem to over predict any pressure losses within the intake, with the prediction of the RNG model being closer to the experimental results. The main disadvantage of the RNG model, which can be seen from the graph, is that the higher the flow Mach number, hence the mass flow rate, the higher the error of prediction becomes.

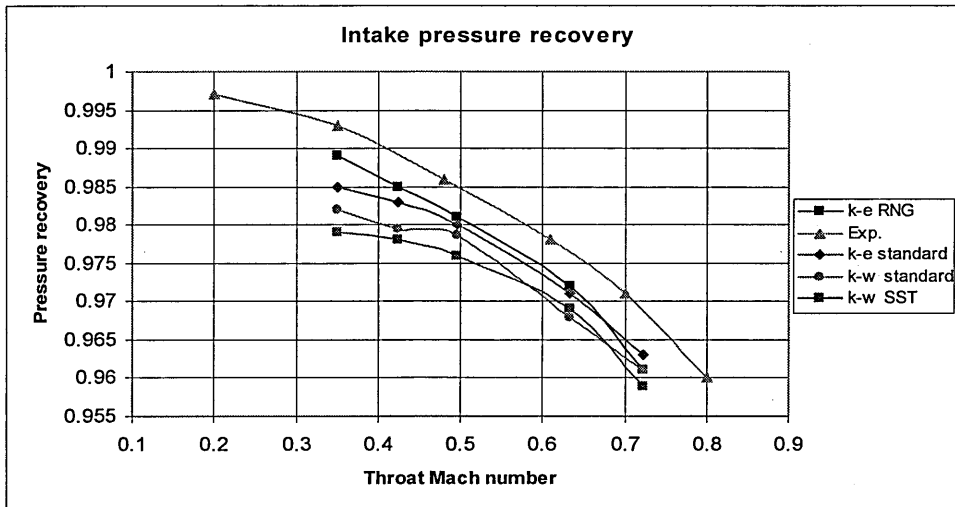


Figure 2.8 Turbulence models comparison in PR

The final analysis for the validation of the M2129 flow characteristics takes into consideration the pressure recovery to the port and starboard walls. Figures 2.9 and 2.12 present the results of this analysis in comparison to experimental results and previous studies.

The results in Figure 2.9 correspond to the starboard wall which is the region where the highly flow recirculation occurs. It can be seen that previous studies, carried out with the use of the $k-\omega$ SST model, could not predict the pressure drop shown in the region from 1.8 to 3 meters length of the duct. The $k-\epsilon$ RNG model was able to predict the swirl at that particular region.

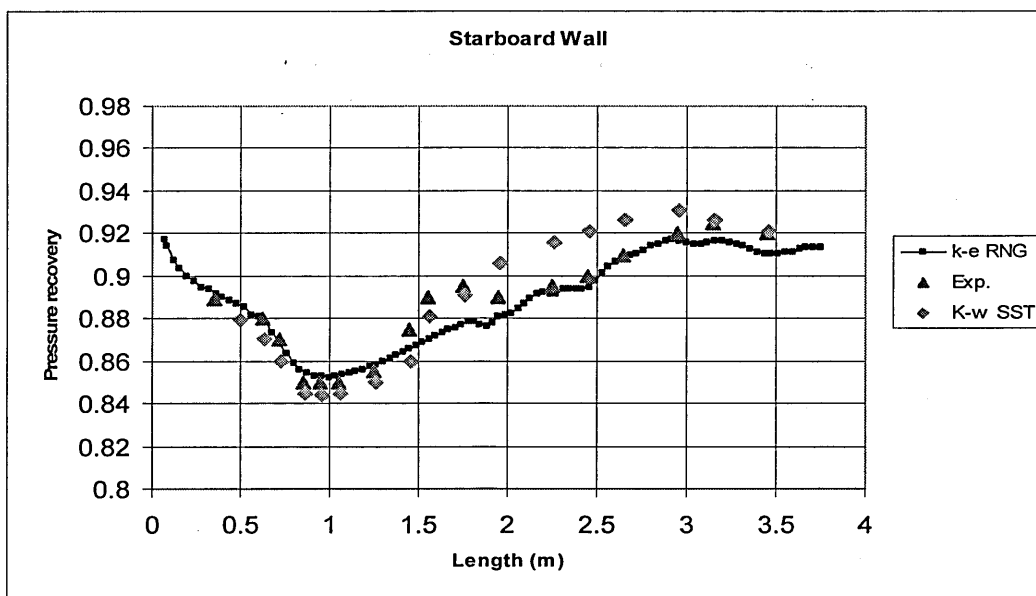


Figure 2.9 Starboard wall pressure recovery

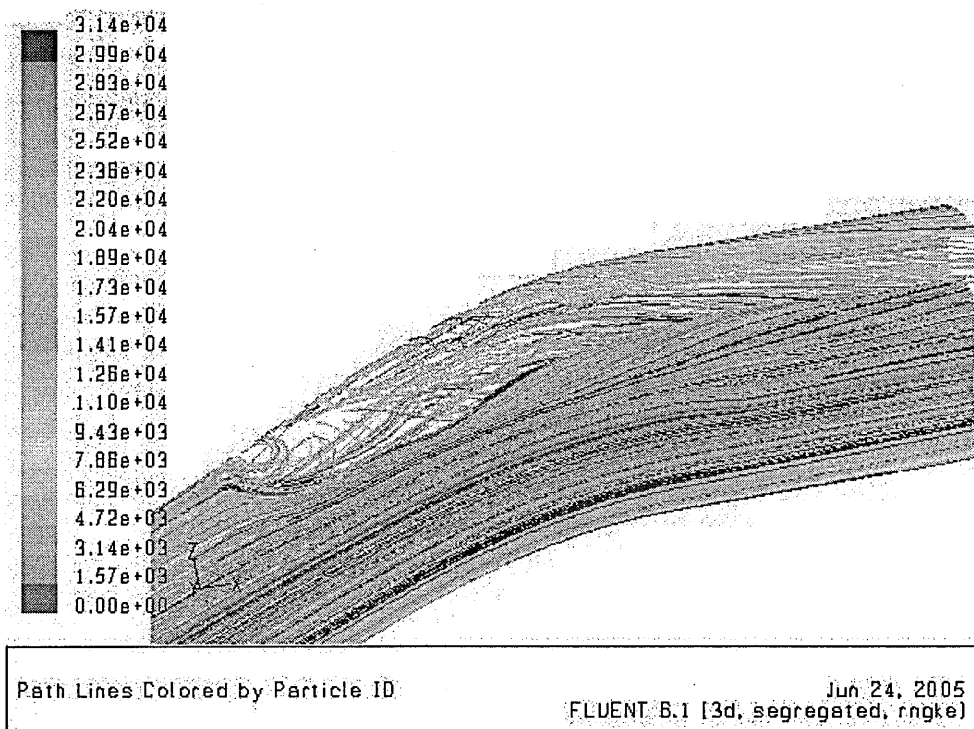


Figure 2.10 Flow recirculation region

The flow swirl presented in Figure 2.10 is translated into engine face distortion with a total pressure distribution that can be seen in Figure 2.11.

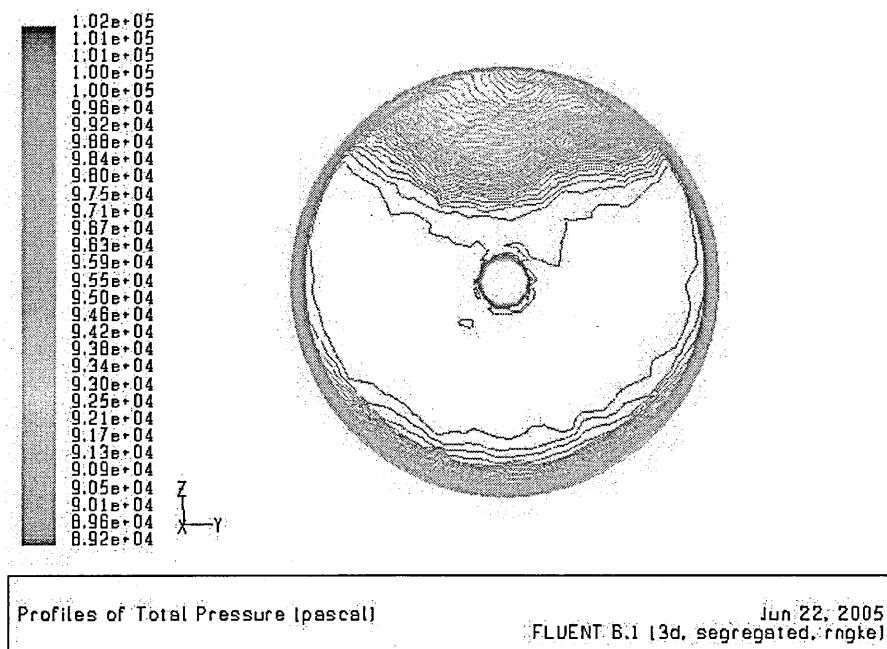


Figure 2.11 Flow recirculation region

Figure 2.12 produced for the validation of the CFD results is the pressure recovery results for the wall at port. Here again the $k-\varepsilon$ RNG model shows an over prediction of pressure losses but the results follow the experimental data in a better accuracy than the previous study, using the $k-\omega$ SST model.

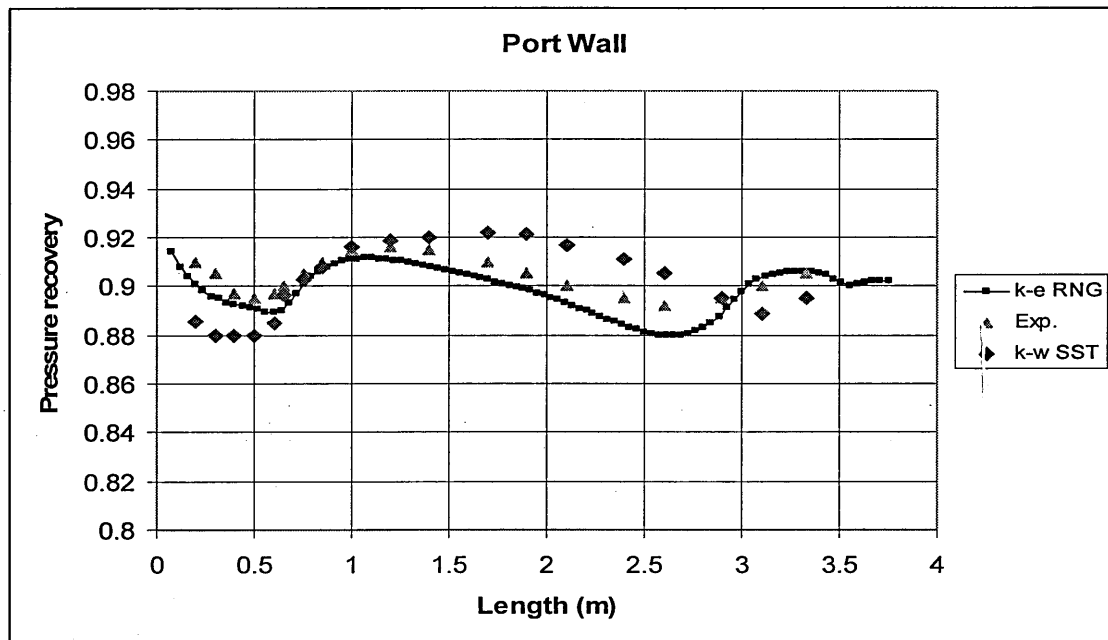


Figure 2.12 Port wall pressure recovery

2.6 CFD Results

As mentioned in chapter one of this dissertation, part of this work is to investigate the flow characteristics of the M2129 intake model at pitch and yaw angles and introduce flow control methods that can improve both engine face distortion and pressure recovery.

This section analyses the results obtained from CFD simulation for axial flow cases and different pitch and yaw angles and the improvement that can be obtained with the use of flow control techniques.

2.6.1 Distortion simulation cases

Flow simulation has been carried out for the M2129 intake model with the use of Fluent computational fluid dynamics software. The cases consider sea level inlet conditions and the mass flow rate for each case has been obtained from the performance simulation of a two-spool high bypass turbofan engine.

The results have been post-processed in terms of engine face total pressure distortion and intake pressure recovery. Flow visualisation post-processing has also been carried out.

2.6.1.1 Pitch angles

The Figures that follow present the post-processing of the various angle of attack cases carried out for this investigation. Figures 2.13 and 2.14 present the results of engine face total pressure distortion in terms of DC60 parameter and pressure recovery PR with respect to intake throat Mach number for all cases.

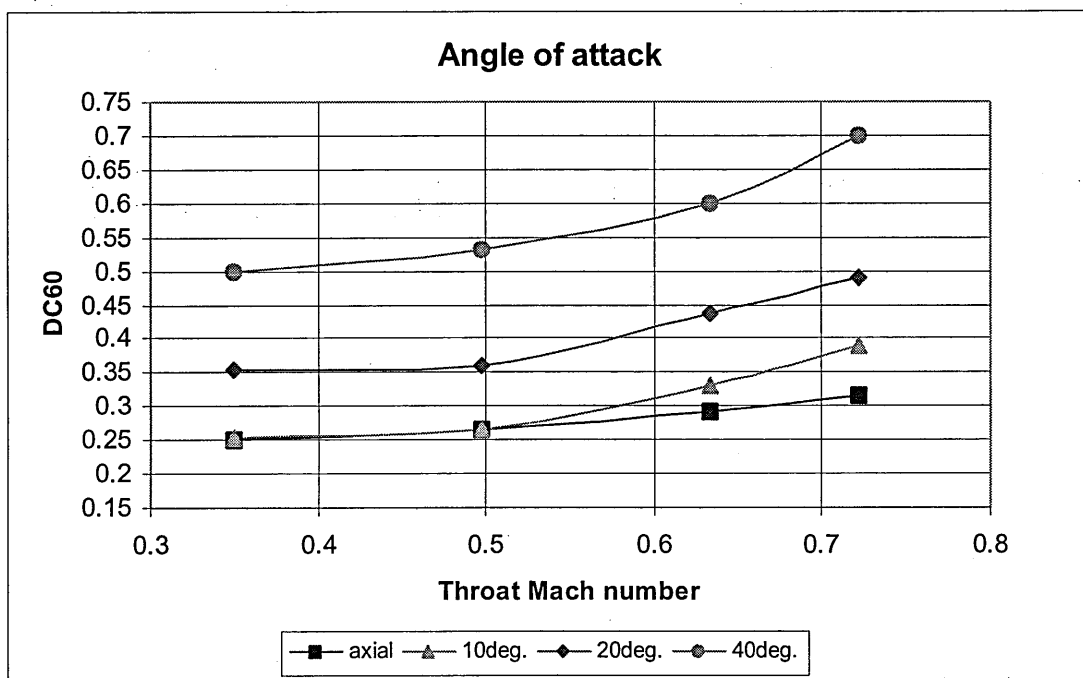


Figure 2.13 DC60 vs angle of attack and Mn

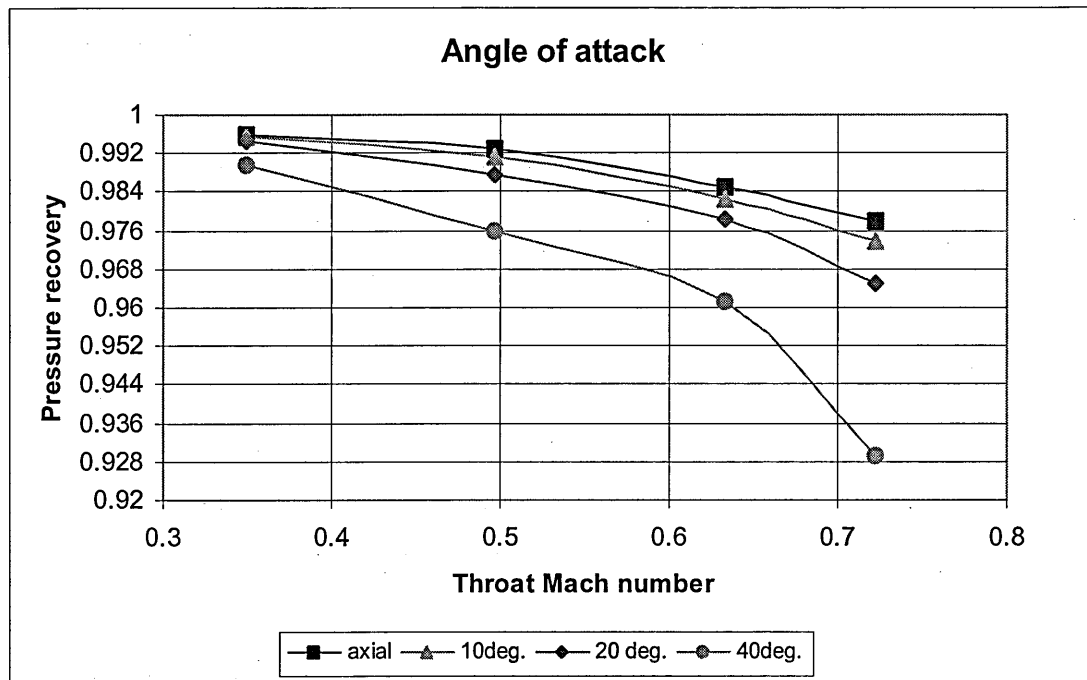


Figure 2.14 Pressure recovery vs angle of attack and Mn

In both graphs it can be seen that at an angle of attack of 10 degrees there is a decrease in DC60 and a minor increase in PR. When the angle of attack exceeds 20 degrees there is a major change in flow characteristics, which translate in a major change in both parameters. Especially for the 40 degrees angle of attack and the high mass flow rate cases where DC60 reach a value of 0.7 and the pressure recovery drops down to 0.935.

Flow visualisation post-processing has also been carried out and the Figures that follow present the flow characteristics for the 40 degrees angle of attack. This is the most extreme case investigated in this work and the flow visualisation shows a highly disturbed engine face profile both in terms of total pressure and velocity magnitude.

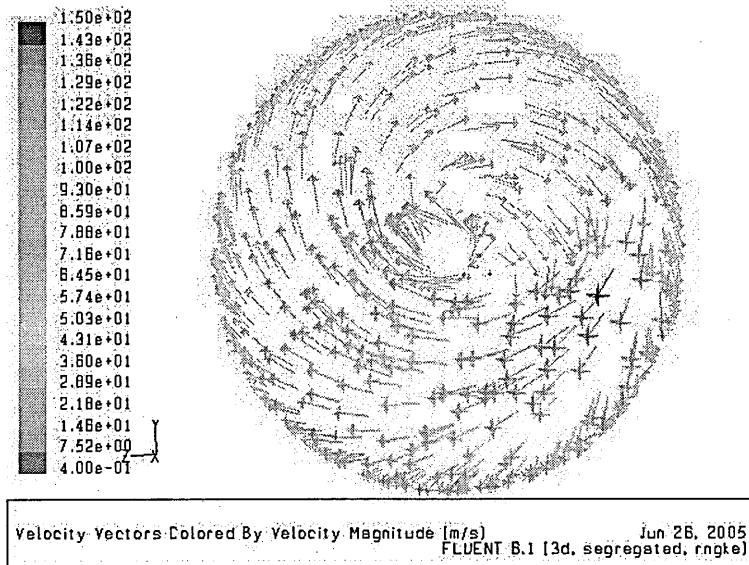


Figure 2.15 Engine face velocity vectors

The engine face visualisation shows an extreme recirculation for the 40 degrees angle of attack case.

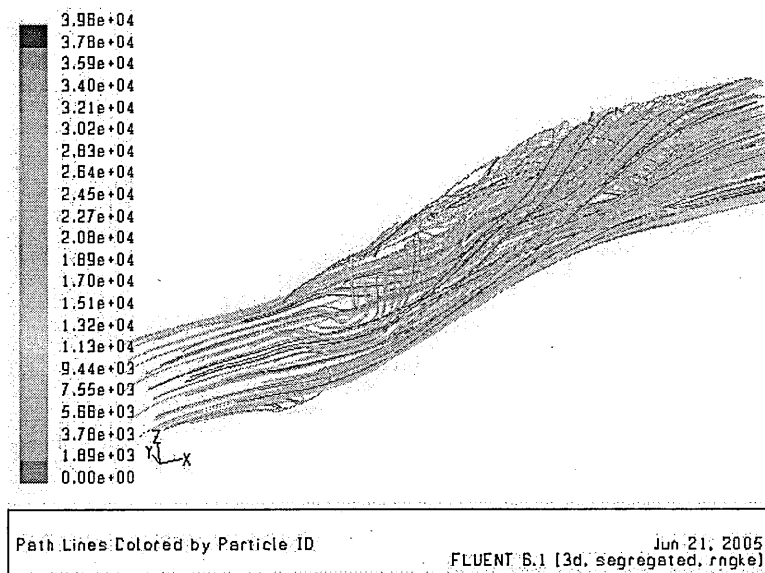


Figure 2.16 Pathlines at 40 degrees

The flow follows the circular wall of the duct introducing the engine face profile shown above. The characteristics of the flow within the duct can be seen in Figure 2.16. The pathlines presented, follow the wall of the duct introducing a recirculation effect to the flow characteristics throughout the duct and creating the engine face profile shown earlier. An analysis for the total pressure differences throughout the duct is presented in Figure 2.17. The results show a

movement of the lower total pressure regions from one profile to the other. The first profile shows the lower total pressure region at the bottom half of the duct. At the second profile the same region has moved to the side of the duct and the third region shows a low total pressure region at the top part of the duct. So the flow has a circular movement within the duct following the wall.

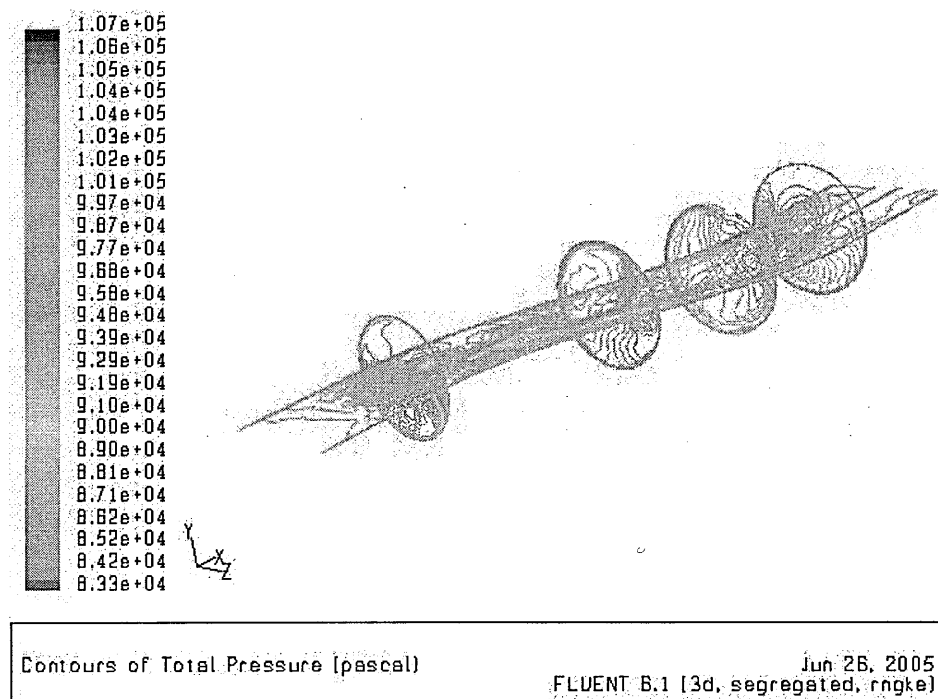


Figure 2.17 Total pressure profiles

2.6.2 Flow control simulation cases

Flow control methods have been used in this work in order to improve the flow characteristics of the M2129 intake model and introduce an improved flow profile to the engine face in order to have better compressor and overall engine performance.

This section presents the analysis of injection and suction flow control methods and an optimisation of these methods in terms of secondary mass flow rate and flow control design configurations. In the axial cases a study has been carried out considering different flow control design configurations.

2.6.2.1 Flow control design configuration

For the axial case the flow region that needs to be controlled is already known. This flow region is at the starboard wall of the intake. In Figure 2.18 a close up of this region is shown in terms of total pressure difference to the main flow.

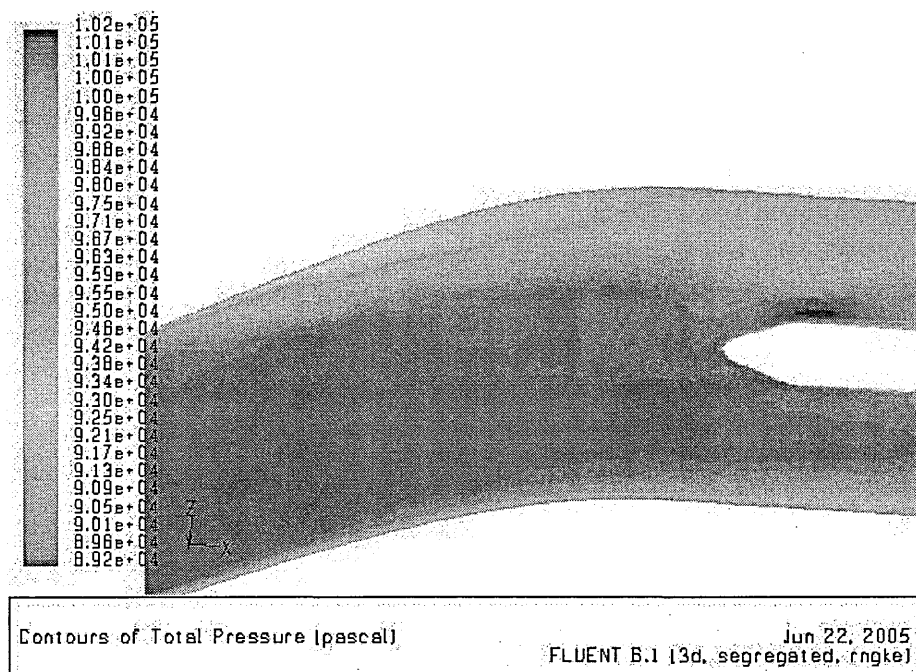


Figure 2.18 Flow separation region

The two design configurations have been analysed here. The first configuration involves two holes placed just upstream of the flow recirculation region where the major separation occurs. Secondary flow is injected into the duct main flow in order to reenergise the boundary layer and delay separation.

The second configuration is a series of eight holes placed all around the duct, far upstream of the flow recirculation zone where the diffusing part of the duct starts. This configuration reenergises the boundary layer in order to avoid any separation that occurs at the first bend of the duct hence solving the problem in an early stage.

Both configurations have been used for a number of flow simulations at various throat Mach number cases and the results can be seen in the graph that follows. Figure 2.19 presents the results of both configurations and the cases without flow control in terms of engine face distortion.

The results show that there is an improvement in engine face distortion when using the eight holes configuration. In order to be able to appreciate the above results an extra parameter needs to be considered. This parameter is the mass flow rate of the secondary airflow injection. In the results presented above the two holes configuration inject 0.6% of the overall mass flow, whereas the eight

holes configuration uses almost 1.0% of the engine overall mass flow. The next section presents an investigation into the effect of mass flow rate of the injected secondary flow.

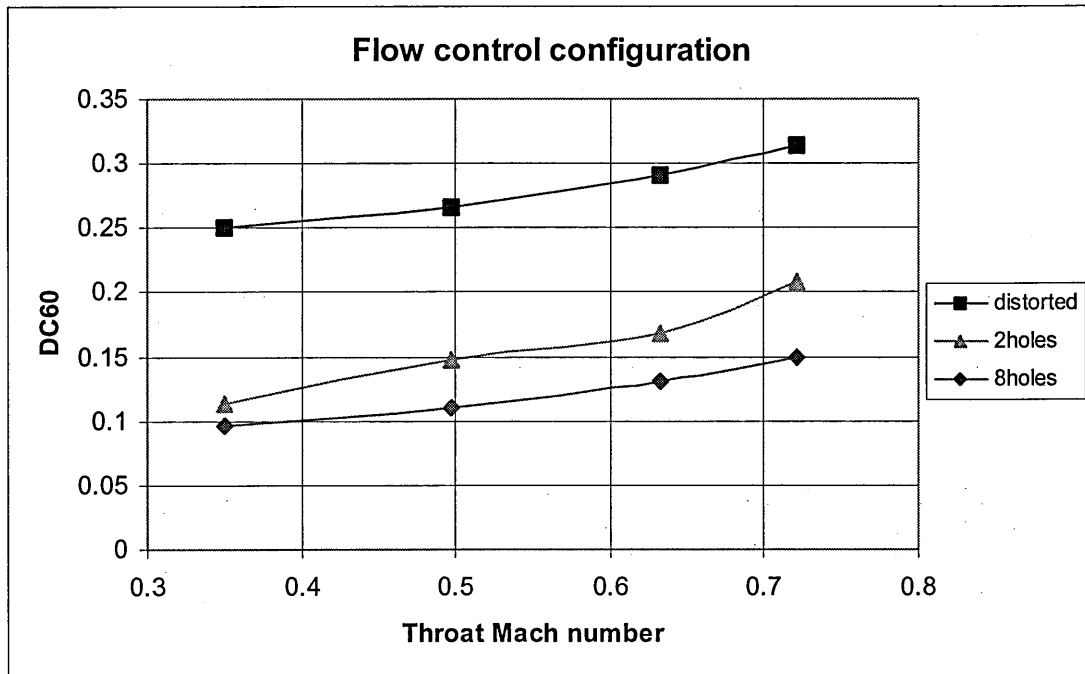


Figure 2.19 Flow control design configuration

Configuration of the flow control technique was also an issue when considering flow control for improving the flow at different pitch and yaw angles. In these cases it is difficult to isolate the problem into one region. Instead you need a flow control configuration that can attack the problem at an early stage and try to improve the flow throughout the duct. Therefore, for the cases of different pitch and yaw angles the eight holes configuration has been used. In the case of the 40 degrees angle of attack, because of the way that the flow enters the duct, when injecting air from all holes then the flow tends to worsen rather than improve. In this case air was injected in just the four holes where the extreme flow conditions were and the rest of the holes were treated as a wall. Results of these cases can be seen in Figure 2.20.

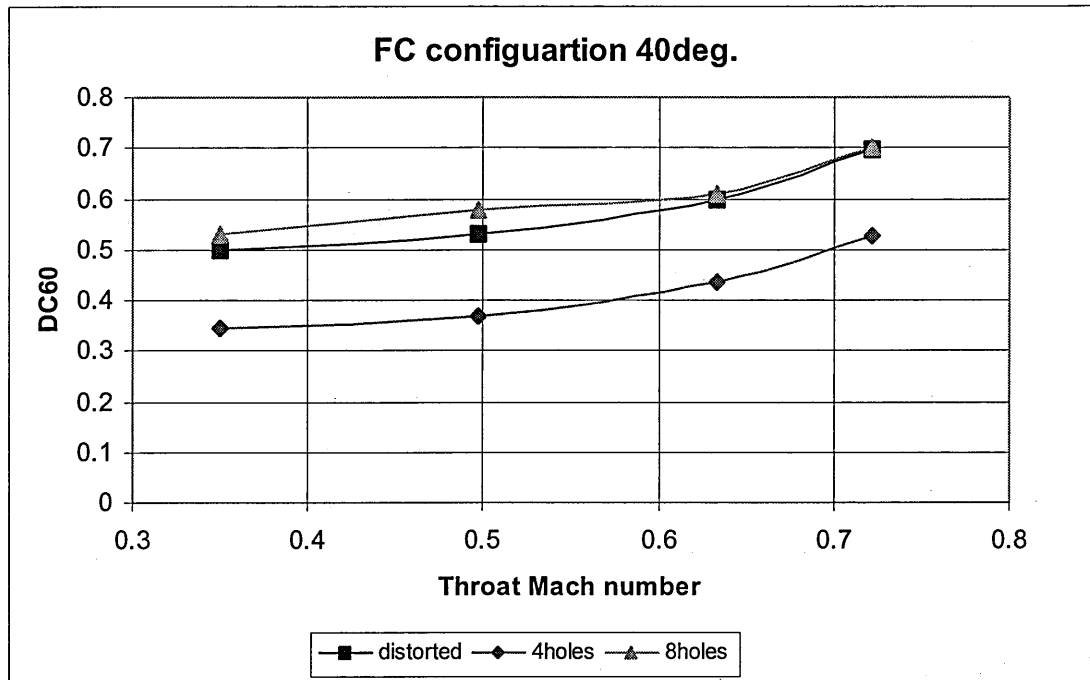


Figure 2.20 Flow control configuration at 40 degrees pitch

The results show that with the use of the eight holes, engine face distortion becomes worse, whereas the four holes configuration shows a clear improvement.

2.6.2.2 Injection mass flow rate

For the purpose of this study, three injection mass flow rates have been considered and the results are again presented in terms of engine face total pressure distortion DC60.

Figure 2.21 shows the trend lines of DC60 at various throat Mach number cases and using three variations of air injection mass flow rates.

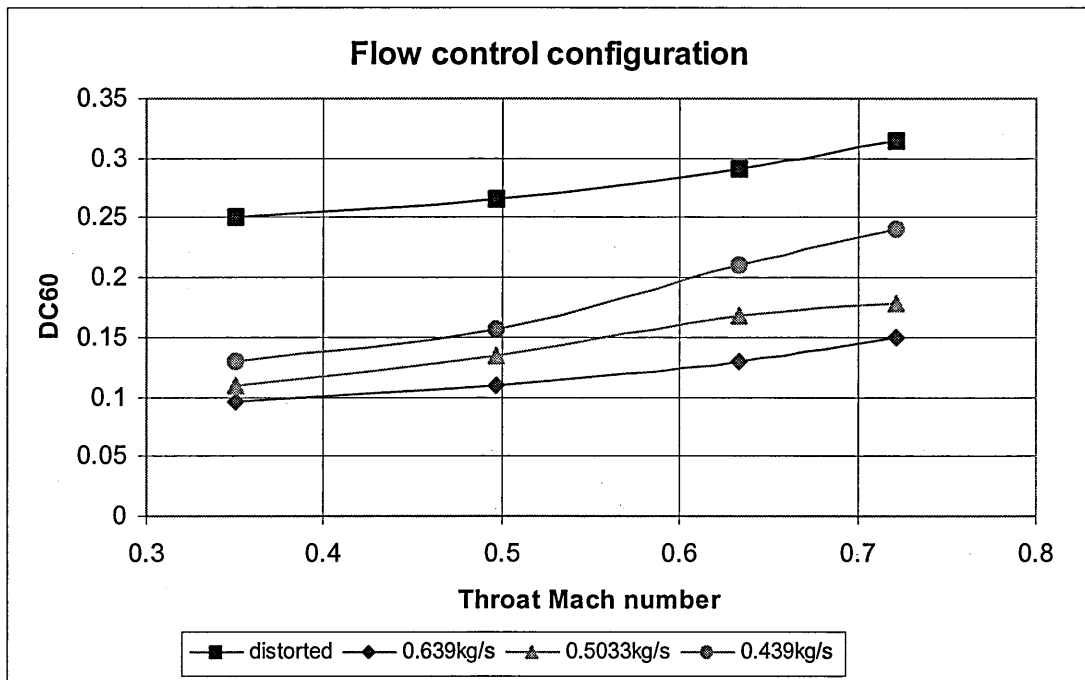


Figure 2.21 Air injection mass flow rates

The results show that an increase in the mass flow rate of the secondary flow, introduce a further improvement to the engine face distortion.

The case used earlier for the comparison between the two design configurations and the results presented here conclude that the eight holes configuration can give a better total pressure profile to the engine but the penalty that the engine will suffer from a higher mass flow bleed needs to be considered.

2.6.2.3 Flow control methods

The final investigation that has been carried out for the flow control study is the simulation and analysis of two active flow control methods, injection and suction. These methods have been tested in improving the flow characteristics at the 40 degrees angle of attack.

Figure 2.22 show a comparison between suction and injection DC60 results at different flow velocities and also with respect to the distorted case of 40 degrees. The results show that both methods work as well at low mass flow rate cases but at higher mass flow rate cases injection gives a better improvement in DC60.

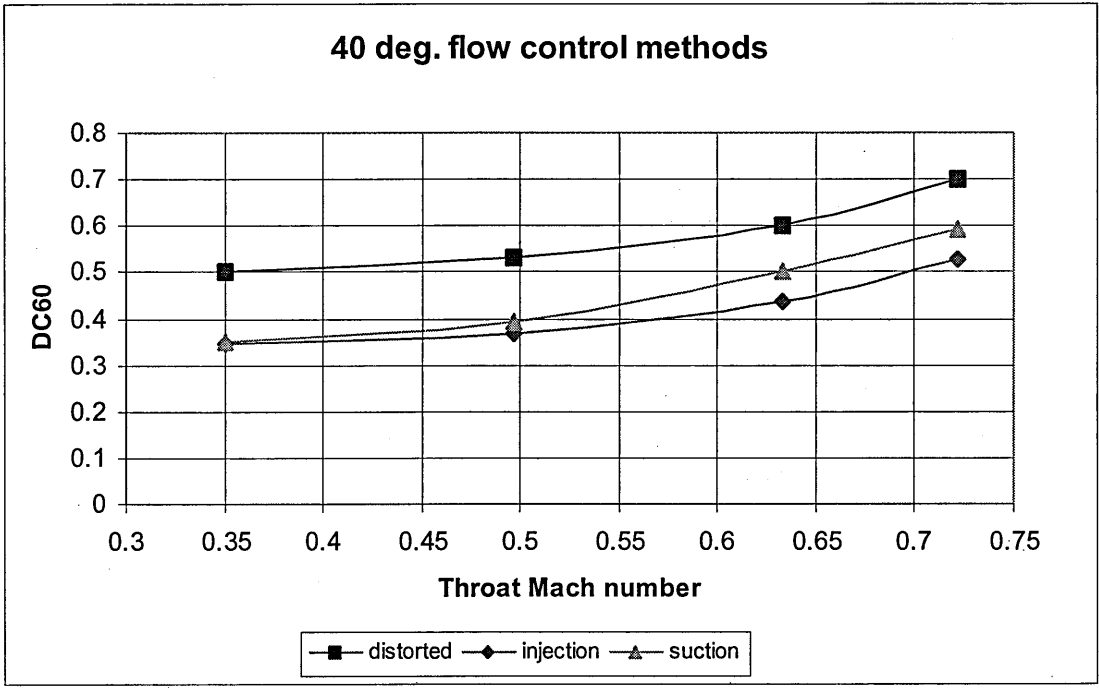


Figure 2.22 Flow control methods

In terms of pressure recovery the results, presented in Figure 2.23, are very similar. The reason for this is the pressure losses that injection or suction introduce into the flow.

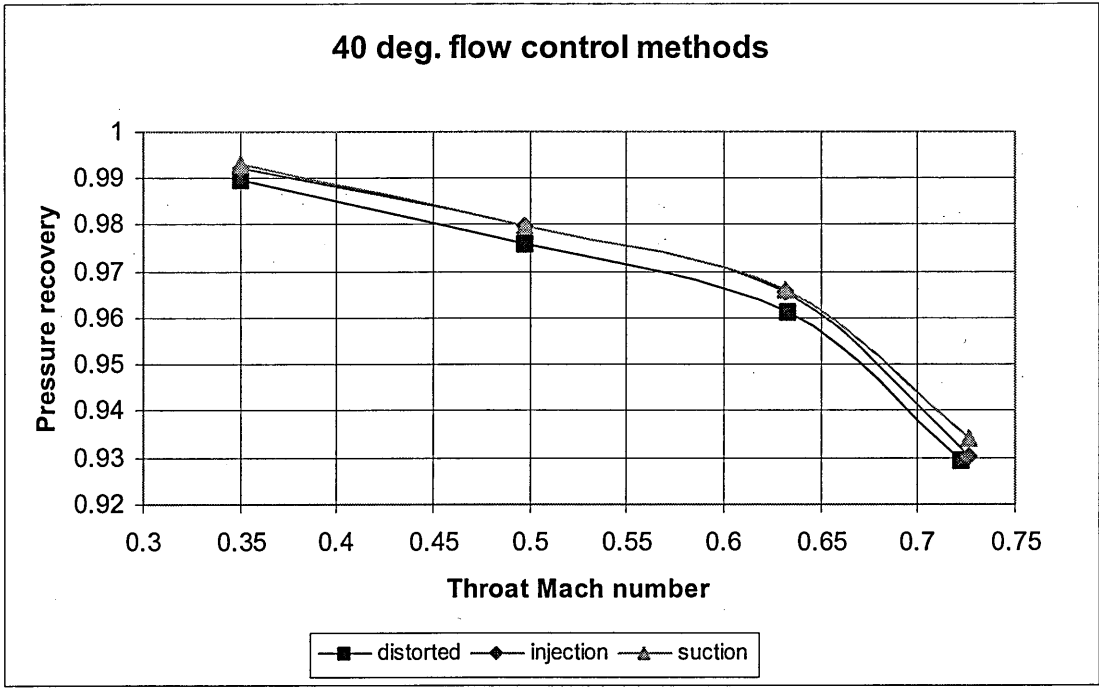


Figure 2.23 Flow control methods

Although there is an improvement into the flow characteristics as presented in the previous Figure there are also some pressure losses introduced from these methods. This is due to the high intensity of injection or suction, which is used in order to make the methods efficient in such an extreme flow field.

Again here one thing that needs to be taken into account is the penalty that the engine suffers from the bleed air used for controlling the flow. Although the improvement with injection is higher the engine performance penalty may prove that suction is overall a better method to consider. This will be analysed in the engine performance chapter.

2.6.2.4 Flow control cases

This final section, presents the results of both distorted and flow control cases, for all angles that have been analysed. All cases use injection for flow control.

The results for all the angle of attack cases are presented in terms of DC60 and intake pressure recovery with respect to throat Mach number in Figures 2.24 and 2.25 respectively.

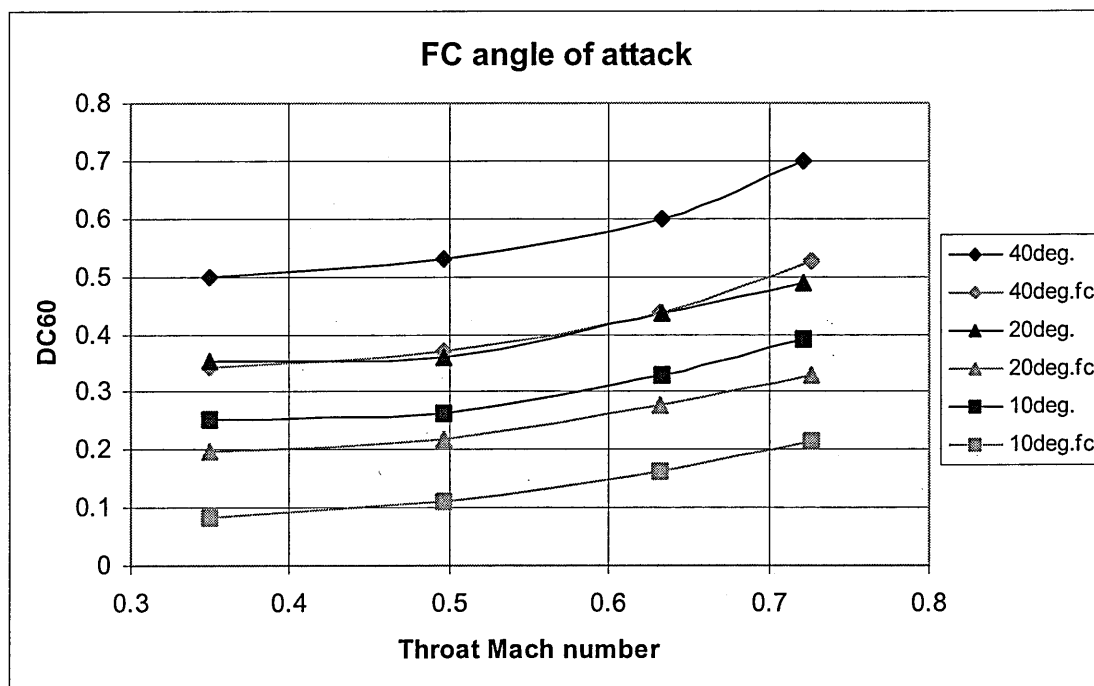


Figure 2.24 FC angle of attack cases

Both Figures presenting the angle of attack cases engine face distortion and pressure recovery results show an improvement of both parameters throughout

the intake throat mach number section. It can be seen that the improvements are better for the engine face distortion results rather than the pressure recovery. This can be explained by the pressure losses introduced to the system when secondary flow is injected into the intake duct. Although the flow characteristics are improved the pressure recovery improvement of the flowfield is minimised due to the secondary flow injection.

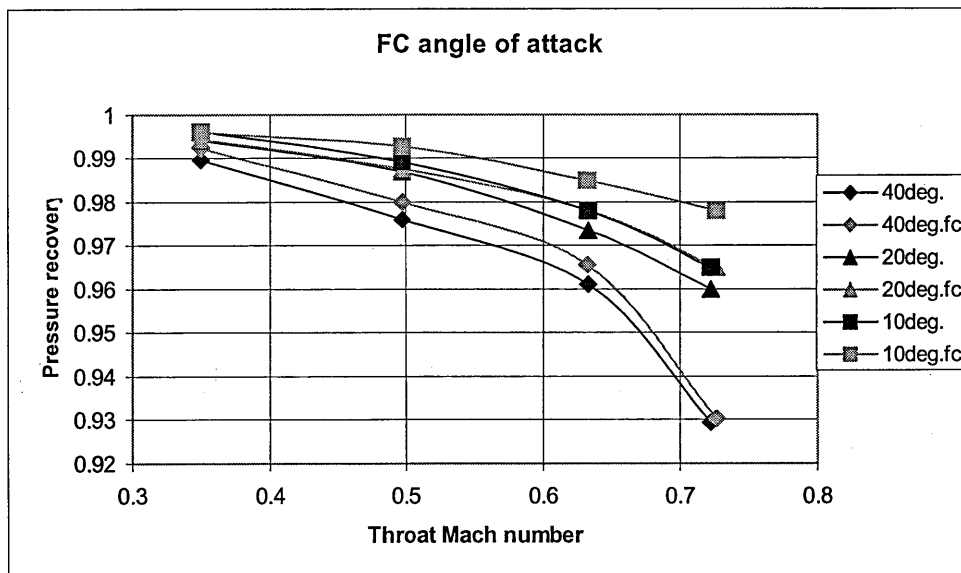


Figure 2.25 FC angle of attack cases

In a similar manner to the previous Figures, the Figures 2.26 and 2.27 present all the results obtained from the flow simulation for the yaw inflow cases.

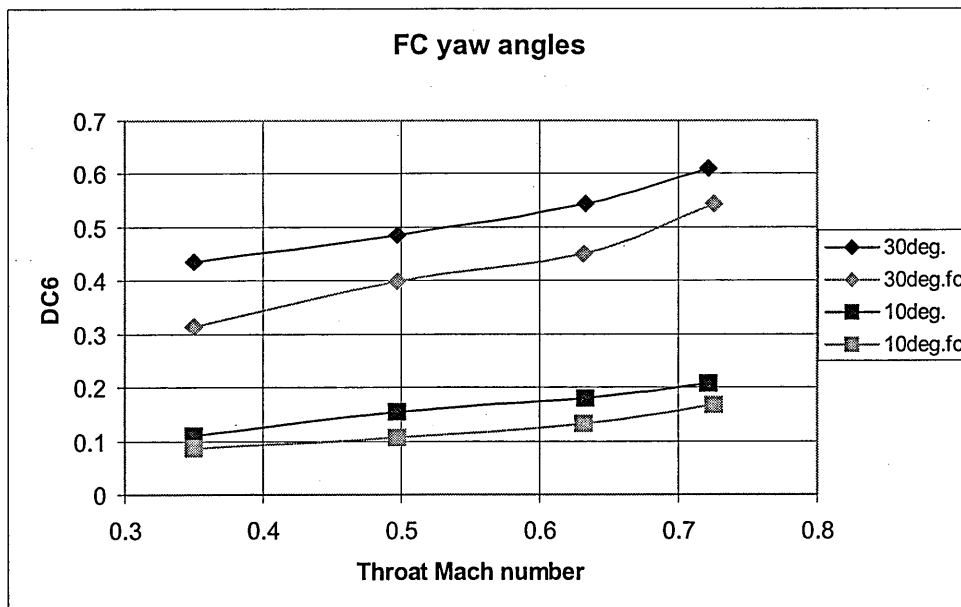


Figure 2.26 FC yaw cases (DC60)

Following the analysis that has been carried out earlier for the axial case study, the 10 degrees yaw angle seems to introduce an improved flowfield into the intake duct. This can be explained by the results shown in Figure 2.28 below.

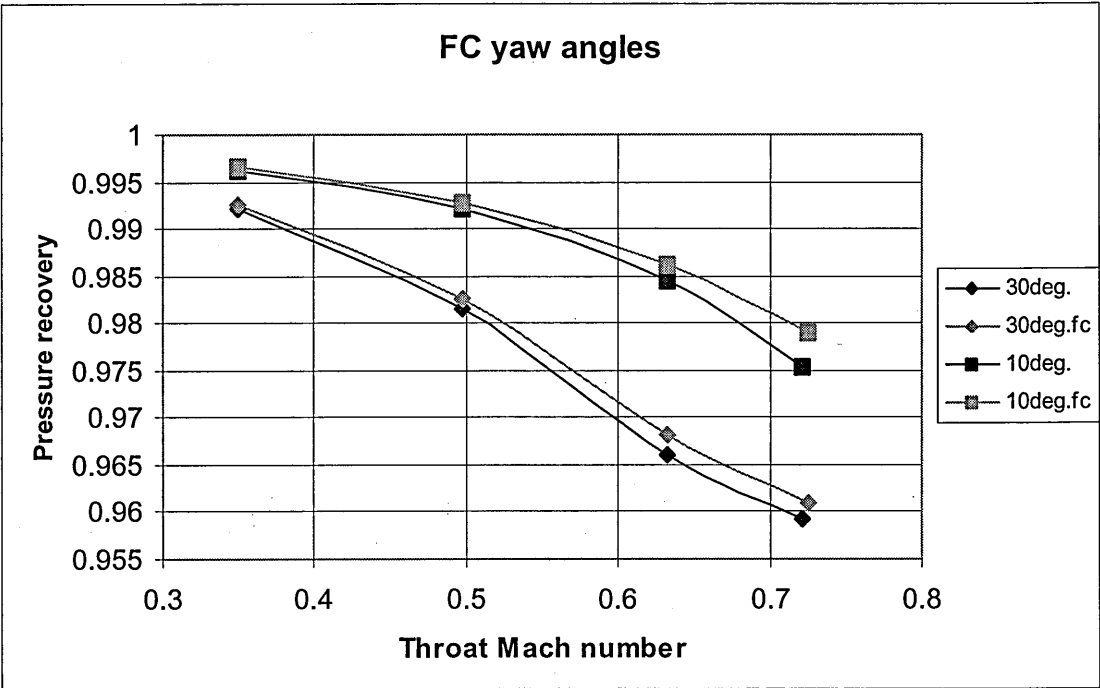


Figure 2.27 FC yaw cases (PR)

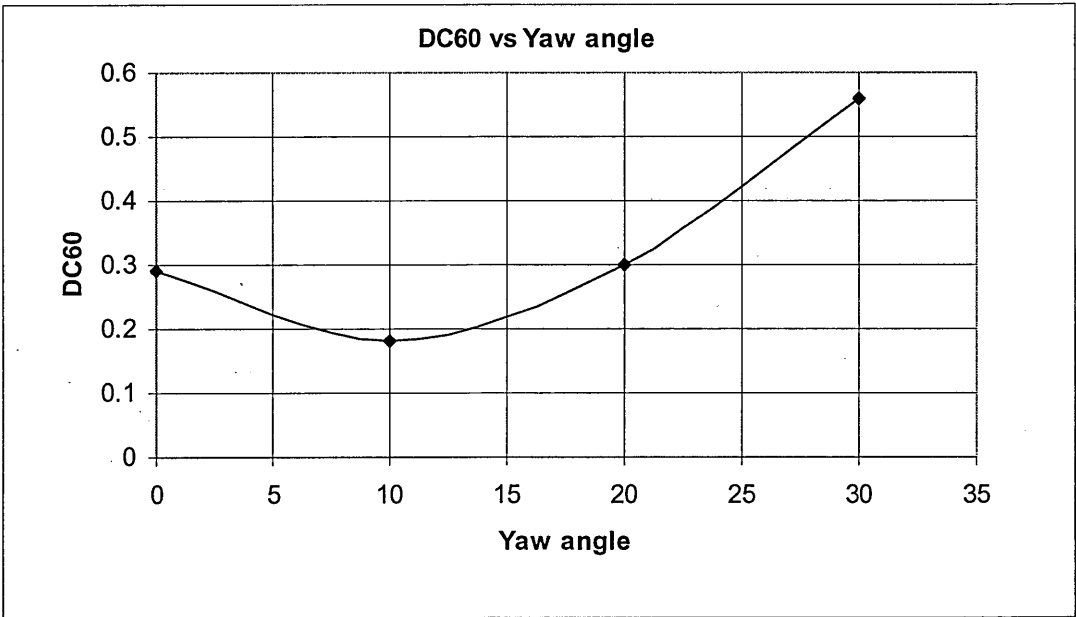


Figure 2.28 Yaw angle case studies

Figure 2.28 presents an investigation that has been carried out into the inflow yaw angle and how that affects the flowfield of the intake duct. The results show

that up to 10 degrees yaw angle inflow there is an improvement of the engine face distortion hence the intake flowfield. From then on the engine face distortion increases again. This can be explained due to the geometry of the s-shaped duct used for this study. The air coming into the intake at 10 degrees yaw angle flows parallel to the lower bend of the s-shaped duct avoiding any flow redirections and further distortion to the flowfield. For this reason a yaw angle of 10 degrees inflow is considered the optimum for engine face distortion results.

CHAPTER THREE

Compressor modelling and simulation

3.1 Introduction

Compressor design and analysis is a complex area within the gas turbine community. Many studies have been carried out through the years on the aerodynamics of compressor components. The compressor configurations that exist are separated in two types, the axial and centrifugal compressors. The history of gas turbine has shown that centrifugal compressors tend to have limitations in obtaining both high-pressure ratios and high efficiencies. Axial flow compressors can provide higher-pressure ratios and efficiencies compared to the centrifugal configuration and also greatly improve the flow capacity. Therefore nowadays the axial flow compressor configurations dominate the field for large powers and the centrifugal compressors are restricted to the lower end of the power spectrum where the flow is too small to be handled efficiently by axial blading.

The axial compressor system, functions in a way where shaft work is converted into an increase in stagnation pressure. This aim is achieved by passing the air through an alternative chain of rotating and stationary blades. Rotors add energy to the flow by increasing the tangential velocity of the air (and therefore its total pressure and temperature), while stators convert the velocity into static pressure (decreasing the losses due to friction which are proportional to the velocity squared) and prepare the air to the following rotor.

The process mentioned above tends to introduce several losses, resulting in an increase in the entropy of the flow. Compressor design considerations involve the minimisation of the sources that introduce these losses to the system.

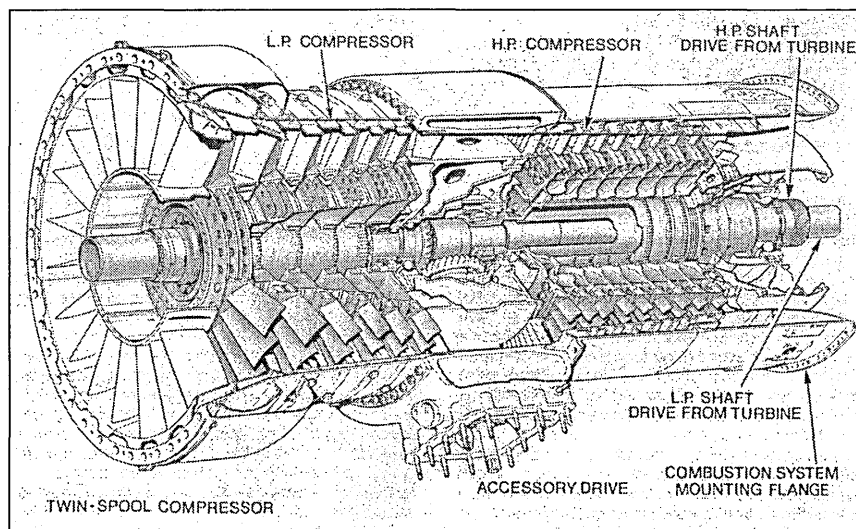


Figure 3.1 Axial compressor configuration [37]

As mentioned earlier, analysing the flow of an axial compressor is a complex three-dimensional phenomenon. Mattingly [34] suggested that in order to be able to understand the basic phenomena that occur in such a complex configuration, the complete flow field can be analysed as the sum of three less complex two-dimensional flows, which are:

- Throughflow field

This field takes into consideration the variation of the flow properties in the axial and radial coordinates whereas the flow properties for the tangential direction are considered to be constant.

- Cascade field

The cascade field covers the analysis of the variation of the flow along stream surfaces and the tangentially through blade rows

- Secondary flow field

The secondary flow field takes into consideration the effects of the lower velocity of the fluid inside the boundary layers that causes the fluid to flow from regions of higher pressure to regions of lower pressure.

Very important considerations for analysing the axial flow of the compressor are the flow unsteadiness, compressibility and viscosity. These considerations make the analysis of the overall system even more complex. Adding to that for this particular fan model used for this study is the high velocities concerned which give rise to flow characteristics such as shock waves and the introduction of even higher losses. Figure 3.2 gives a representation of the main phenomena, introduced due to the three dimensional nature of the problem.

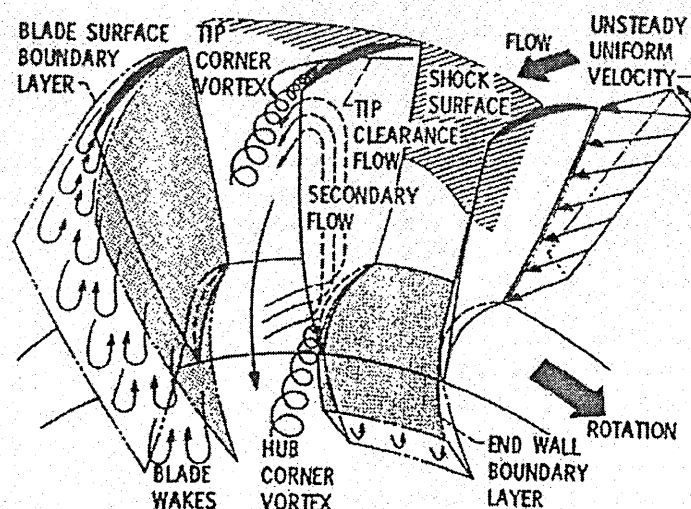


Figure 3.2 Three-dimensional phenomena in compressors [37]

In order to be able to carry out a preliminary design of such a complex system, some assumptions need to be made that will help to simplify the problem and create an initial design of the compressor components. Among these principal assumptions are for example flow steadiness, axial symmetry, blade-element flow and adiabatic flow of the overall system. The first two assumptions tackle the problems linked to the complex variations of the flow whereas the third assumes the flow about blade profiles formed by the intersection of a surface of revolution and the compressor blading (Johnsen and Bullock, [35]). In this study all the calculations are carried out at several radial locations from hub to shroud, defining a three-dimensional flow field. The radial locations chosen for the computations, in reality become theoretical meridian planes (planes containing the stream lines) with the further assumption of radial equilibrium (which is completely justified in case of free vortex design).

The initial step for the initiation of the design is to specify the general requirements of the compressor system. Once these requirements are in place the steps that follow are within a standard procedure that covers the principal phases below

- establish the dimensions for the overall annulus configuration
- produce the velocity triangles at design point operation
- select the shape and configuration of the blades
- determine the procedure for the off-design performance

Throughout the process of design and simulation of the compressor a number of assumptions are used and physical laws are stated. For example one of the important laws is surely the law of continuity of mass flow. In terms of turbomachinery aspects, Euler's turbomachinery equation, which links the work done by the rotating blade on the flow with the change in enthalpy of the flow itself across the rotor, is used.

$$H_e - H_{i_1} = \omega(r_e V_{we} - r_i V_{wi})$$

this equation can be further simplified if we assume the gas to be calorifically perfect, e.g. with constant specific heats.

$$c_p (T_e - T)_{i_1} = \omega(r_e V_{we} - r_i V_{wi})$$

This, together with a prescribed variation of swirl along the span of the blade, leads to the complete definition of the velocity triangles. The simplest and more diffuse way of variation of swirl is the free-vortex design, which allows having both constant work done in every radial positions and radial equilibrium (the centrifugal force exerted on the mass of fluid is balanced by the variation in pressure).

The overall feasibility study of the blade takes into consideration important parameters such as De Haller number, stage loading parameter and diffusion factor, which specify the limitations of the load that is physically possible to be carried by one blade row.

The passage from the air angles to the blade angles is done through the choice or the evaluation of two parameters: incidence and deviation. Both choices are driven by empirical laws: the first is usually selected in order to have low losses and a reasonable stall margin, the second is predicted from empirical relations and the blade angles are adjusted in order to obtain the desired velocity triangles.

The selections of the general blade geometry leads directly to the definition of the precise blade shape, through the choice of the type of blade that is more adapt for the specific case.

Nowadays the most sophisticated type of blades available is the PVD (prescribed velocity distribution) blades. Both camber and thickness distribution are specified arbitrarily in order to achieve the desired velocity and pressure distribution. This process requires however a complex CFD approach to the blade design that exceeds the aims of this work.

An easier way of designing the blade is the choice of the geometry among several blade shapes that have been tested during the years and are available in the literature (e.g. blades of families such as C series or NACA-65 series). However, the one that appears to have the biggest advantages in terms of performance and simplicity is without any doubts the double circular arc (DCA) or biconvex blade. This choice is principally due to the characteristics of the compressor (it is a transonic fan and the DCA blades show the biggest advantages at Mach numbers between 0.8 and 1.2) and to the simplicity of designing a double circular arc blade.

Once the complete geometry is defined, it is necessary to study the off design performance of the compressor, and assess the overall design procedure. If this does not give satisfactory results the geometry needs to be modified. This last task can be accomplished again by using some empirical relations developed through the years, but in this case the general specifications of the project suggested the direct use of CFD.

Several studies are available on the subject of aerodynamic compressor design and useful suggestions can be found in Ramsden [36], Mattingly [34], Cumpsty [37].

3.1.1 CFD approach

The use of Computational Fluid Dynamics (CFD) in turbomachinery applications has exponentially increased in last years due to the availability of powerful computers and software. Since the first applications in the aerospace industry (dated 1970s) big improvements have been accomplished in both potentiality and reliability of this tool, despite of the extreme complexity of the phenomena involved. The need of a precise and complete solution of the Navier-Stokes equations also for distorted flow has become very important because of both the lack of accuracy demonstrated by the analytical methods mentioned earlier and for the great costs involved in experimental tests. Another great advantage of CFD simulation is given by the possibilities of testing the particular component before its realisation.

While many studies have been carried the past years on three dimensional CFD simulations for undistorted conditions, the accuracy and confidence of these simulations are still very limited and this is one of the reasons that experimental testing is a priority for such complex flow field analysis and even CFD results evaluation.

Since the first three dimensional simulation of an isolated rotor (NASA rotor 67) the use of CFD for complex turbomachinery flows studies is increased exponentially and nowadays examples of successful simulation with undistorted condition are quite common also for complex geometries such as transonic fans.

An example is given by Hirai, Kodama, Nozaki, Kikuchi, Tamura and Matsuo [38] who manage to perform a complete simulation of the all rotor under

distorted condition by linking together 162 processors for a total main memory of 41 GB. The strategy adopted by other studies has been to reduce the number of blade passages in the simulation, in order to make the study acceptable in relation to the computer capabilities. A further example is given by Davis, Hale and Beale [39] who studied the effects of steady and unsteady inlet flow distortion obtaining a good matching with the experimental data.

3.2 Fan preliminary design

The starting point of the fan design is the provision of the design specifications that are concerned with the objective of the overall system. The variables involved are parameters that can bring the desired performance of the fan by modifying the initial recommended values. This modification is an iterative process that can be carried out several times until the desired results are obtained. This analysis and assessment of the final design is based on parameters that define and limit the loads that can be carried by the compressor blade rows, referring to the existing geometries. These parameters include the De Haller number, diffusion factor and air deflection. A preliminary design process can be found in Ramsden [36] and a fan design study has been carried out by Ghisu [40], Charalambous [41] and Iurisci [42] based on the limitations and variables mentioned earlier.

The procedure that has been followed during the design of this fan starts with a complete definition of the design specifications of the fan. Then, an assessment of the parameters that influence the loading limitations is carried out. Design variables specifications follow and an explanation of the overall procedure is given. A complete summary of choices and changes made in the design process through the time is clearly too vast and complex to be explained within this section. A detailed analysis of all the final choices made on the design variables and the reasons for those will be however presented.

The various points analysed in this section have been stated below:

- Design specifications
- Design guidelines and limitations (loading parameters)
- Design variables
- Final design specifications

3.2.1 Design objectives and specifications

The objectives of the design include major parameters to the overall performance of the fan and have been specified with information from the public domain. Table 3.1 gives the design specifications of the fan.

Fan pressure ratio	4.33
Mass flow	39 kg/s
Inlet Mach number	0.6
Rotational speed	12000 rpm
Target isentropic efficiency(1)	0.88
Inlet diameter	0.74 m
Inlet annulus area(2)	0.376 m ²

Table 3.1 Fan design definition

The fan has been chosen to be designed with constant tip diameter, to use completely the rotational power of the spool. The limit to the rotational speed of the spool is in fact given by the blade tip speed and (if there aren't other constraints) the compression of the air is easier with a big blade speed, that is obtained raising as much as possible the tip diameter. There is however a limit of size that imposes the maximum diameter to not exceed 0.74 meters.

(1) An isentropic efficiency of 0.88 corresponds to a polytropic efficiency of 0.90

$$\eta_p = \frac{\frac{\gamma-1}{\gamma} \ln(R_c)}{\ln \left(\frac{R_c^{\frac{\gamma-1}{\gamma}} - 1}{\eta_{is}} + 1 \right)}$$

(2) The inlet annulus area has been calculated with the continuity of mass flow equation

$$A = \frac{W}{\rho V_a}$$

3.2.2. Design guidelines and limitations

In order to carry out the preliminary design of the fan, assumptions need to be made that will make the procedure less complex and give the basis for an acceptable and feasible design. These assumptions concern various parameters that take into consideration the flow speed, blade shape and loading of the blades. In this section, the guidelines and limitations for the design of the fan will be presented and explained.

Inlet Mach number is an important parameter to the design of the fan and based on the literature review, the inlet Mach number limit is shown to be fixed by the maximum relative Mach number at the blade tip. For the design of a transonic fan, the relative Mach number is suggested to be 1.6 by Hill [43]. Mattingly [34] suggests a maximum tip blade speed of 460 m/s. Seddon [23] suggests an inlet absolute Mach number of 0.6.

Based on these assumptions a number of parameters have been specified for this particular design.

- Rotational speed: based on the limit of the tip blade speed of 460 m/s the rotational speed has been chosen to be 12000 rpm.
- Air deflection: in order to avoid the introduction of big losses to the system, the air deflection is limited to 30 degrees to avoid any separation.
- Blade twist: this is the difference between the root and tip average flow angles. Hill [43] gives a limitation of 35 degrees to avoid any boundary layer stresses.
- Stage loading: this parameter is defined as the ratio between the work done in the stage and the blade velocity squared. The recommended maximum value for axial flow compressors is 0.35 – 0.4.

$$\psi = \frac{\Delta H}{U^2}$$

This parameter limits the work that can be done in the stage by relating it to the rotational speed (the bigger is the blade speed the bigger is the work that can be done).

- De Haller number: this is a very important parameter for the design of compressors, giving the ratio between the exit and inlet air velocity relative to the blade. This parameter limits the amount of deceleration that can be achieved across a blade row. The value for this parameter is assumed to be higher than 0.7.

$$DH = \frac{V_e}{V_i}$$

- Diffusion factor: the De Haller parameter mentioned earlier is used basically to define the mid radius air angles. For the choice of blade chord and number of blades the diffusion factor is used, where it relates

the amount of deceleration across the blade row to other parameters such as the ratio of blade chord and blade spacing, known as solidity σ and the deviation that is imposed to the air. A recommended value for this parameter is 0.45.

$$DF = 1 - \frac{V_e}{V_i} + \frac{|V_{ew} - V_{iw}|}{2\sigma V_i}$$

From the parameters mentioned above, the most important are the parameters that control the loading of the blades. These parameters, in a sense, measure the loading of each blade row and can specify if the blade is overloaded. If that is the case then the design is considered to be inadequate.

3.2.3 Design variables

In the previous section the guidelines and limitations of the design have been presented. In order to follow these limitations and achieve a design that is within the constraints of these parameters, some variables within the design procedure need to be adjusted in order to achieve final acceptable design.

The variables that the iterative procedure used are specified below.

- Temperature rise: although the design specifications of pressure ratio and isentropic efficiency give a fixed total temperature for this design, the variation of the temperature within each stage is not fixed and can vary.
- Velocity direction and modulus is another parameter that can vary at the inlet of each stage and the exit of the compressor.
- Annulus area between the rotor and stator can be specified and fixed once the two variables above have been decided.
- Blade aspect ratio is also a variable and defines the ratio between the span of the blade and its chord.

$$AR = \frac{h}{c_m}$$

In this case the blade aspect ratio is affected by the blade chord, which is defined by the velocity triangles. These are used to determine the annulus configuration, hence the blade height.

- Variation of the chord along the span of the blade
- Number of blades

A change in the number of blades or in the chord length allows increasing or decreasing the solidity of the blade itself at the specified

radial location. This has a direct influence in the value of diffusion factor and therefore changing these two parameters is a way of decreasing the loading on the blade.

Based on the design specifications, guidelines and variables mentioned earlier the design process has been carried out in order to specify the annulus configuration, the analysis of the velocity triangles, the hub to tip design and the blade shape. Once the process is complete the final design is produced and it can be seen in the section that follows.

3.2.4 Final design specifications

The previous sections of this chapter covered the specifications for the fan design and the parameters involved in order to achieve a satisfactory design. The process followed is an iterative procedure that takes into consideration the adjustment of different variables that can fulfil the limitations of the loading parameters. This iterative procedure has been carried out and the final adjustment of the variables is presented in this section. These adjustments represent the final design of the three-stage fan.

- **Stages temperature rise**

The basic idea for the design of a three-stage fan is to avoid overloading of the first and final stages. This overloading can cause many problems at off-design conditions. Especially in the case of the first stage where the hub to tip ratio is low, it is important to introduce a free vortex design of the blades. For this reason the temperature rise for the three stages has been chosen to be 53, 59 and 53 degrees respectively.

- **Stage inlet velocity**

The Mach number at the fan inlet is suggested to be 0.6 as specified earlier. The main factors that dictate the flow velocity of each stage are the pressure ratio and the pressure losses. For this design the decision was made to keep the velocities of all three stages at the same value in order to have a balance between the increased pressure ratio and increased pressure losses to the system.

- **Annulus configuration**

The annulus configuration has been suggested by using the average areas at inlet and outlet of the stage. Once this was specified then a small decrease has been introduced in order to increase the velocity at the rotor exit, which will in turn affect the loading of the rotor.

- Chord length and number of blades

The choices of the chord length and of the number of blades are strictly related. The same air deflection could be obtained either by increasing the chord of the blade or increasing the number of the blades. This is reflected in the diffusion factor, which decreases if one of these two variables or both of them is increased. Clearly a too low value of diffusion factor is not positive, because it means that the same deviation could be obtained with a lower chord or with less blades and that probably the losses and the size of the compressor are bigger than what is really needed. The choice of these two variables has then been done with the objective of a diffusion factor in the order of 0.45. The chord has been kept constant along the span of the blade, but sometimes it has been slightly increased at the blade hub where the deviation is bigger.

The final specifications for each stage are shown in Table 3.2. Only the mid radius data are shown for simplicity.

Stages	First		Second		Third	
$\Delta T(K)$	53.00		59.00		53,05	
α_4	22		27		27	
$V_4(m/s)$	204.92		202,02		202.02	
AR	2.00		1.75		1.50	
ψ	0.42		0.38		0.31	
V_{a2m}/U_{2m}	0.45		0.46		0.41	
degree of reaction	0.94		0.65		0.66	
	Rotor	Stator	Rotor	Stator	Rotor	Stator
Blades No.	19	23	29	31	41	47
Inlet abs Mach	1.14	0.74	1.00	0.67	0.93	0.63
Inlet rel. Mach	0.60	0.74	0.58	0.67	0.52	0.63
De Haller No.	0.71	0.93	0.71	0.70	0.71	0.73
Diffusion factor	0.39	0.20	0.44	0.43	0.39	0.43

Table 3.2 Compressor mid radius properties

The obtained values of De Haller number and diffusion factor are shown in Figures 3.3 and 3.4.

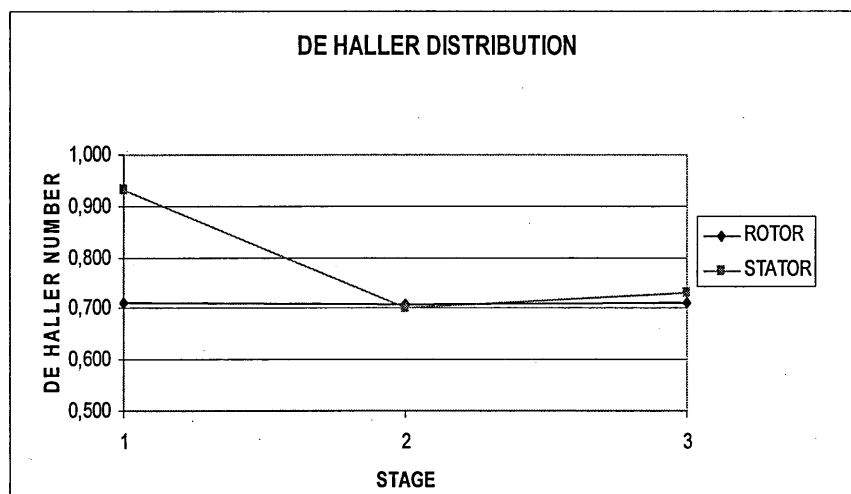


Figure 3.3 Mid-radius De Haller number distribution

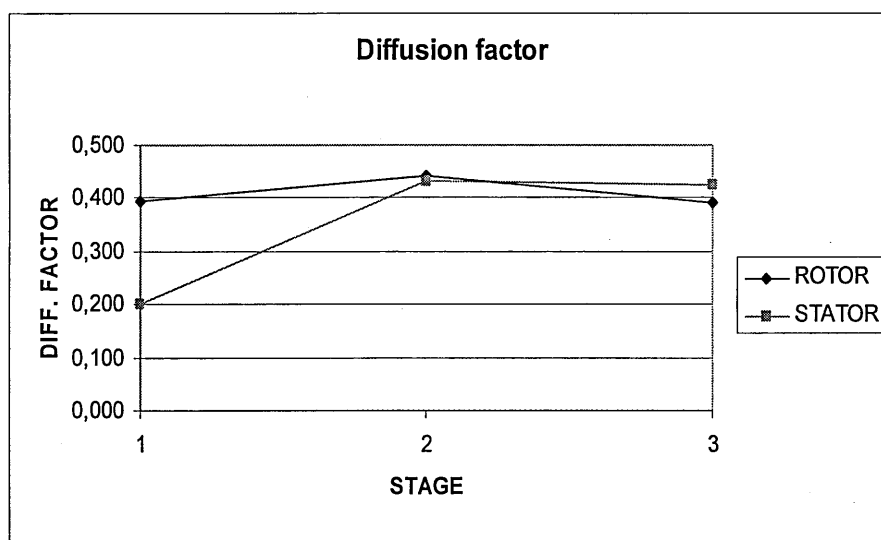


Figure 3.4 Mid-radius diffusion factor distribution

3.3 CFD Fan Modelling

This study is concerned with the effect of pressure distortion profiles to the performance of the first stage of a low bypass engine fan and the overall performance of the engine. The modelling and simulation of the first stage of this fan has been carried out with the use of CFX-Turbogrid and CFX-Tascflow respectively. The sections that follow analyse the procedure for the three-dimensional geometry and modelling of the fan stage.

3.3.1 Geometry and grid generation

Based on the results of the preliminary design of the fan blades the next step was to produce the three-dimensional geometry of both the rotor and the stator. For the purpose of this task the program used was CFX-Turbogrid. This program has been developed in such way as to be able to produce 3D blade geometries with a limited amount of information.

From the preliminary design, three files of Cartesian coordinates have been obtained that represent the hub, shroud and blade profiles. A visualisation of these curves can be seen in Figure 3.5.

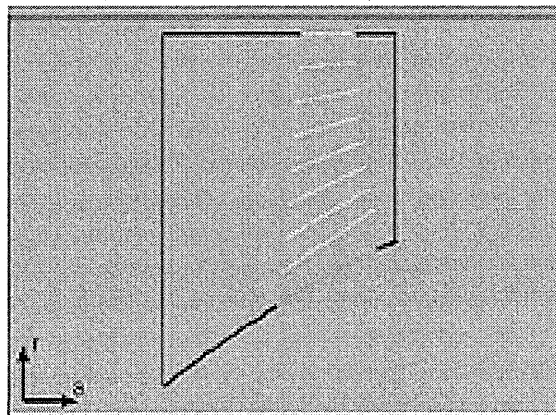


Figure 3.5 Visualisation of hub, shroud and blade profiles

In order to obtain the 3D geometry and grid of the model, CFX-Turbogrid provides pre-defined grid topology templates that can be used to optimise grid generation and minimise time.

Once the grid was created the next step was to carry out a grid independent solution analysis to make sure that the results obtained from the simulations are not affected by the grid generation. The optimum result have been used for this study. The final models and nodes distributions can be seen in Table 3.3.

NODES NUMBER	Meridional dir.	Tangential dir.	Spanwise dir.	Total size
ROTOR	70	34	23	54740
STATOR	42	56	23	58824

Table 3.3 Nodes distribution

Based on this node distribution the modelling of the rotor and stator has been carried out separately in CFX-Turbogrid and then both models have been connected in CFX_Tascflow to produce the final design passage. This model can be seen in Figure 3.6.

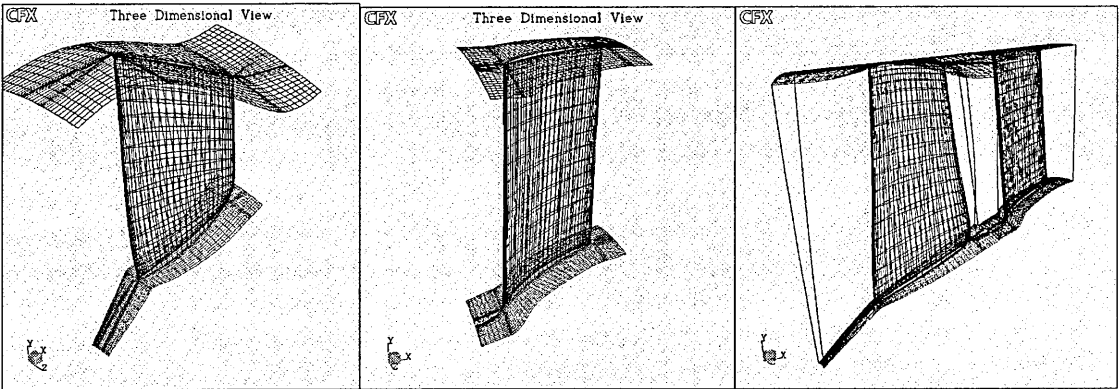


Figure 3.6 Modelling of rotor, stator and whole passage

More visualisation of the grid at the hub, mid, and tip region in the meridional direction, at the inflow and at the interface plane can be seen in Figure 3.7.

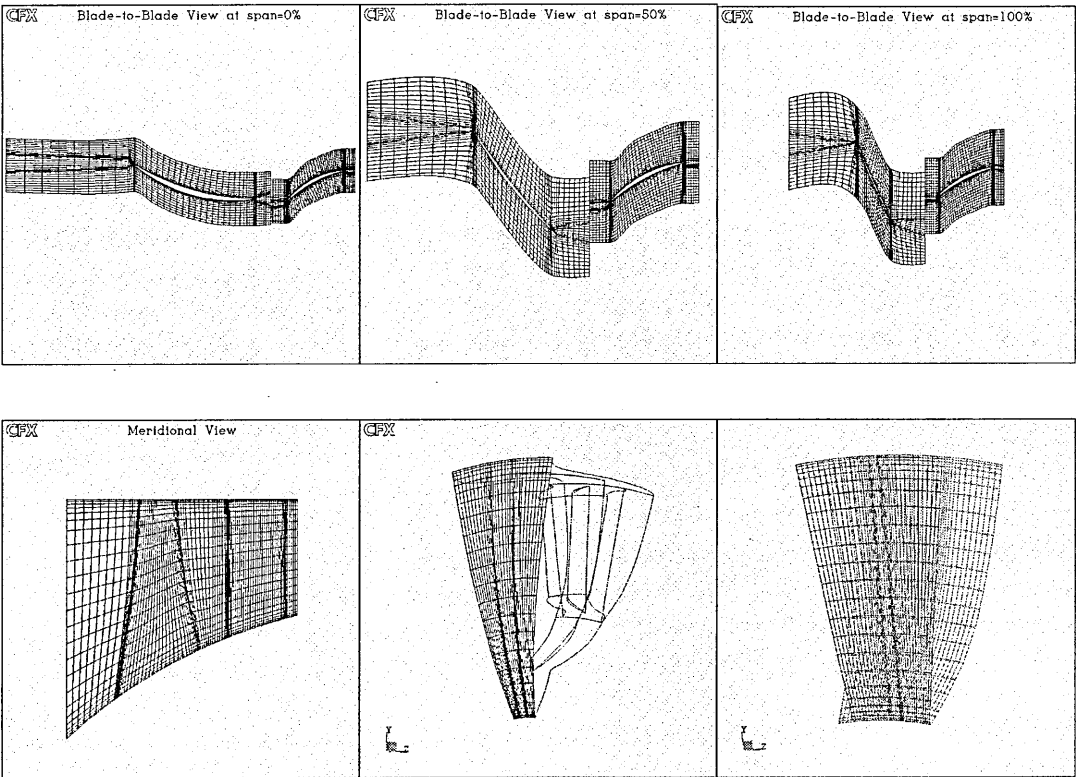


Figure 3.7 Grid view for hub, mid and tip and interface plane

Although as mentioned earlier this study is concerned with the effect of distortion to the first stage of the fan, Figure 3.8 presents the complete three stages fan model.

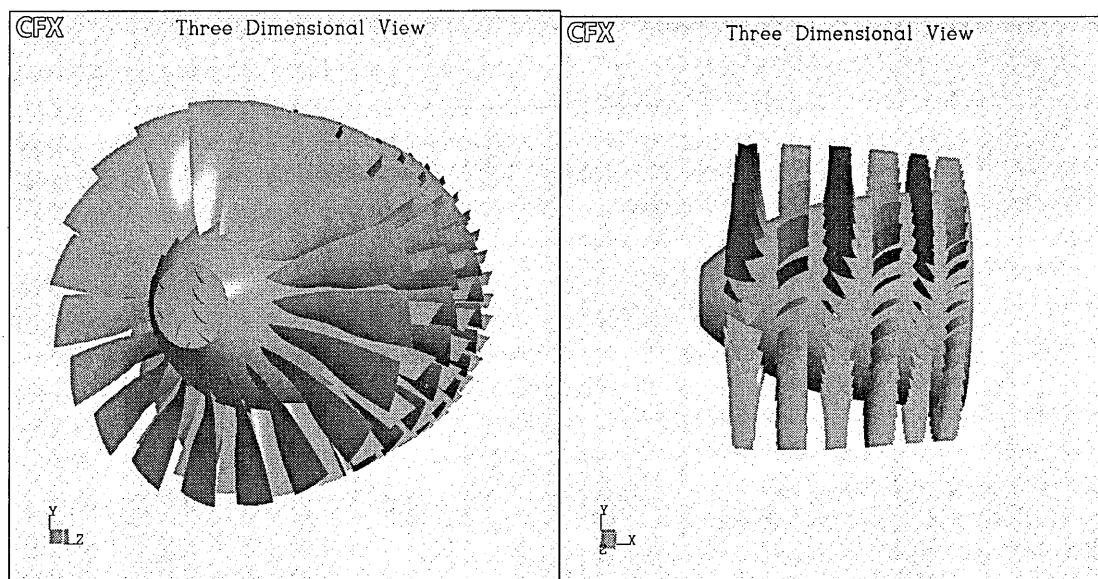


Figure 3.8 Fan 3D model

3.3.2 CFD Simulation

For the flow simulation of the fan stage, the commercial software CFX-Tascflow has been used. This software is suitable for turbomachinery applications due to the structure of the package and the ability to attach different blocks of grid and specify different interfaces and rotating and stationary components.

This commercial package, gives the possibility to the user to apply different types of boundary conditions both at inlet and exit, use various turbulence models and run the simulation either steady or unsteady. For this study the boundary conditions were specified as inlet total pressure and outlet static pressure.

The model run at steady simulation and the turbulence model used was the $k-\epsilon$. $k-\epsilon$ is the most prominent and used turbulence model for turbomachinery application and was used in this study because it has proven to be stable and numerically robust and has a well established regime of predictive capability. It is recommended by Tascflow for medium or coarse grid for offering a good compromise in terms of accuracy and robustness. Within Tascflow the $k-\epsilon$ turbulence model uses a scalable wall function approach to avoid inconsistency with very fine grid.

3.3.2.1 Grid attachment

As mentioned earlier in the geometry creation package the two components (rotor and stator) have been created separately. In order to bring them together and create the rotor-stator passage CFX-Tascflow is used. Here both grids are imported and then the number of rotors and stators has been chosen to give the overall stage. The way that the program transfers information from the rotor exit to the stator inlet is by collecting all aerothermodynamics information at the exit of the rotor and makes a massflow average and passes them to the inlet of the stator. The main reason for using this type of simulation is to avoid great computational time processes. By using this simulation type some local fluid phenomena, like for example the rotor wake influence on the stator, cannot be taken into consideration. For this reason a study has been carried out to analyse the effect that this type of simulation has on the results accuracy. This study will be presented in the sections that follow.

3.3.2.2 Boundary conditions

The flow is considered compressible and the boundary conditions specified are:

- the total pressure at the inlet;
- the static pressure at the rotor outflow as an average value;
- the walls are defined as adiabatic and for the rotor is given the rotational speed;
- the value of k and the turbulent viscosity ratio is also specified

The stator and rotor are specified to belong to different frame of reference: one stationary and the other rotating. The static pressure at the outflow and the total pressure at the inlet, for the design point, are got from the Excel program of the compressor design. Changing the average static pressure at the outflow, without changing the RPM, the total pressure and temperature at the inlet we can simulate different conditions at off design condition, in essence at the same non-dimensional speed line of the compressor map.

3.3.2.3 Solver parameter

For the simulation some parameters have to be set, and they are different for the steady state and the transient simulation. The convergence accuracy and time depends from their value.

Residuals: for the steady state simulation they are set to 10^{-6} , for the transient one they are set to 10^{-4} .

Time step: the time step (in Tascflow called "STIME") is set for both steady state and transient simulation. It is a powerful parameter that can control the

stability of the solution (the smaller STIME the more stable it is) and the velocity of convergence (the bigger STIME the faster the solution converge if it is enough stable). A compromise value must be found and after different try my final choice are:

- Steady state simulation $STIME = 10^{-4}$
- Transient simulation $STIME = T/60 = 8.3333138 \times 10^{-4}$ (where T is the period of the rotation)

The computational time was of the order of some hours (2 to 5) for the steady state simulation and of some days (2 to 3) for the transient.

3.3.3 Limitation of the method

The method used for the simulation has a number of limitations, some depends on the way the solver solves the equation, others depend on my particular grid and setting.

The solver limitations are basically due to the way the Navier stokes equations are solved (by Reynolds-average mode) and for the use of a turbulent model (to close the system of equations) that is not accurate. Besides, to solve the equations, numerical methods are used, so a numerical diffusion in the results must be considered. The error in the solution depends also on the “residual” set, that will be small but always a finite value.

The limitations of my particular case depend basically on:

- the grid used, even though it is a grid of 114000 points, a more precise one would give more accurate results;
- no tip clearance is left to simulate the tip leakages;
- the walls are set as adiabatic but in the reality there is a heat exchange between air flowing and compressor elements;
- the thermodynamic properties are transmitted from the rotor to the stator by a mass average, so some local physical phenomena are lose in the average process;
- the static pressure at the stator outlet is set to be constant (as average value) for all the circumferential extent, even for the distorted case, and we know that this is not true, because it decrease in the distorted region. See also next chapter.

These limitations set the level of trust of the solution. So we know that all the results are affected by errors, that the real behaviour of the same geometry will be a bit different from the one studied by CFD, in particular in those region where instabilities and unsteady phenomena are accentuated, like the stall

region. The instability of the numerical solution near the stall region, indeed, makes it very difficult to investigate the surge region. Anyway, CFD remain a very powerful and cheap way to study complex flow field like the one described in this thesis, and it can still give good results (in engineering sense) and a very good sensitivity of the flow behaviour.

3.4 CFD Simulation Analysis

In chapter one an explanation has been given to the procedure on which this study is based. The fan component modelling is used in order to simulate a three-dimensional flow through the fan blades and obtain results that can be used to produce the compressor characteristics maps for clean and distorted inlet conditions. The model that is used for this process is a 'heavy' model with almost two million elements. This introduces further restrictions to the amount of runs that can be produced. For this reason some assumptions had to be made and the simulation procedures used for this work had to be further analysed due to the simplifications that followed.

The three main areas concerned for this study are:

- Grid analysis
- Turbulence models study
- Frozen rotor simulation

Grid analysis has been carried out with a coarse and a finer grid in order to analyse the compromise on the results accuracy when using a coarse grid. The reason for using a coarse grid is to cut down on computing time.

Turbulence models are another important issue when dealing with CFD simulation. In this case the commercial package used has a limitation when using the $k-\omega$ model. In order to use that particular turbulence model the number of elements need to be increased to a specified number. This is another case of increasing computing time.

CFX-Tascflow used for the simulation of the fan component, offers two different simulation procedures for the interaction between the rotor and stator. The frozen rotor is the most time consuming and the procedure follows the simulation of the flow for a fixed rotor position. This is a more realistic simulation process but in order to have completed results all the possible positions of rotor and stator interactions need to be covered and this add more runs to the

procedure. In order to avoid this complexity the simulation has been carried out with the assumption that the flow is fully mixed after it leaves the rotor and this simulation process is known as the mixing plane process.

The three areas mentioned above correspond to the limitations that the methodology introduces to the accuracy of the results. So a study has been carried out for these three compressor simulation procedures in order to analyse the effect on the results accuracy.

3.4.1 Compressor grid analysis

For the purpose of this study the analysis of CFD results was used. The simulation of three dimensional flow characteristics and the use of reasonably accurate models with size limitations were analysed.

A grid study has been carried out in order to analysis the penalty in accuracy that comes with the use of a coarse grid. The use of finer grids tends to increase the computational time needed for a simulation procedure. For the purpose of this study computational time is limited due to the amount of flow simulations that need to be carried out in order to obtain a complete compressor characteristics map. Therefore this study is essential in terms of estimating any differences in the results given from these simulations.

The conditions used for the simulations are exactly the same in terms of both inlet total pressure and outlet static pressure. The conditions have been given from the performance simulation program and indicate flow analysis at sea level conditions.

Figure 3.9 and 3.10 show the trends of the speed lines for the coarse and fine grid and the differences that occur.

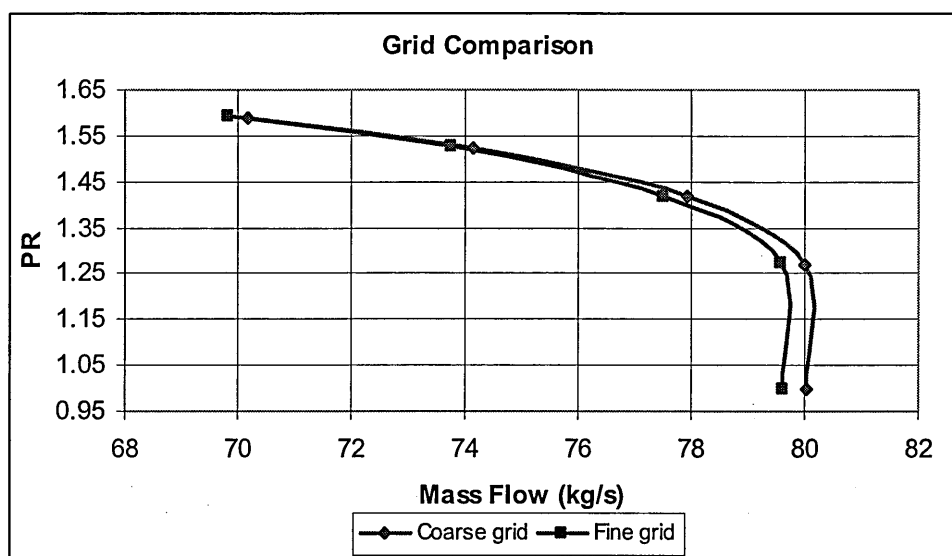


Figure 3.9 Grid comparison Pressure ratio

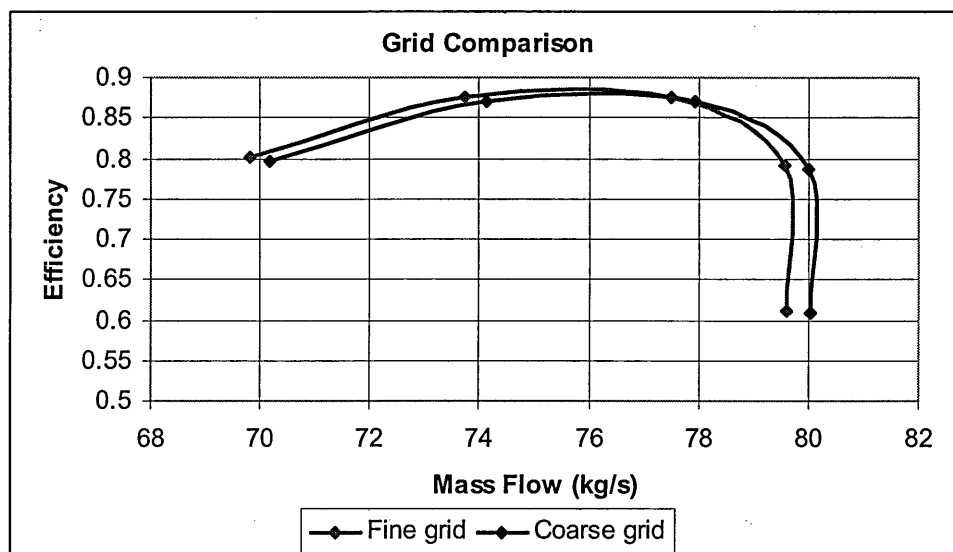


Figure 3.10 Grid comparison Efficiency

The variations in percentages from the coarse to the fine grid are worked out below in three different points of the constant speed line. The points used here are shown in Figure 3.17.

	$\Delta W \%$	$\Delta PR \%$	$\Delta \eta \%$
Point 1	-0.5267	0.1399352	0.68691
Point 2	-0.5267	0.2260398	0.68692
Point 3	-0.5267	0.2433519	0.68692

Table 3.4 Percentage variation at grid comparison

Analysing the differences of the two grid simulations it can be seen that in terms of mass flow the finer grid at all three points under-predicts the results. In terms of pressure ratio and efficiency there is an increase for the finer model.

Another interesting point to notice is that the variation of both mass flow and efficiency are the same throughout the constant speed line whereas pressure ratio tends to increase when moving towards the top of the speed line.

In general, the variation of the results for all three parameters throughout the constant speed line are very small and when compared to the time gained with the use of the coarse grid then it safe to assume that the best way forward is to use the coarse grid. The lack of experimental data in order to be able to validate the model is another limiting factor for this study.

In terms of computational analysis the fine grid permits to analyse flow behaviours very close to the wall than the coarse grid. The coarse grid does not capture any flow recirculation. Either way the coarse grid is considered reliable based on the above comparison. Taking into account its limitations, it is possible to use the coarse model to obtain reasonable results in terms of performance and flow behaviour.

3.4.2 Turbulence model analysis

Another important area that needs to be considered for these flow simulations is the turbulence model used. Turbulence models tend to effect the results of the flow simulations especially when analysing the three dimensional characteristics of complex flows.

For the purpose of this study the turbulence models used are the $k-\varepsilon$ and $k-\omega$ models. The simulation parameters are the same to the grid comparison study and the results obtained are analysed in terms of mass flow, pressure ratio and efficiency.

Figure 3.11 and 3.12 show the trends of pressure ratio and efficiency comparing the $k-\omega$ with $k-\varepsilon$ model results for the fine grid.

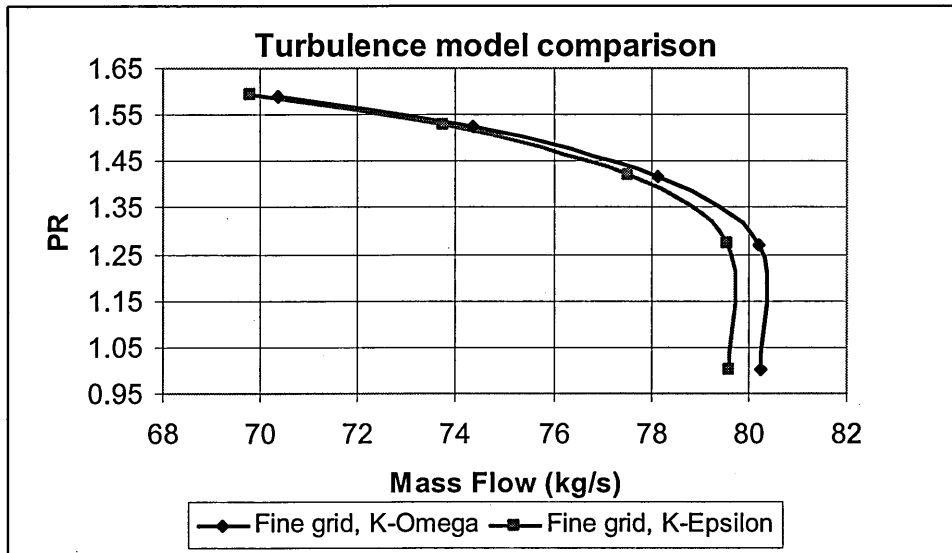


Figure 3.11 Turbulence model comparison Pressure ratio (fine grid)

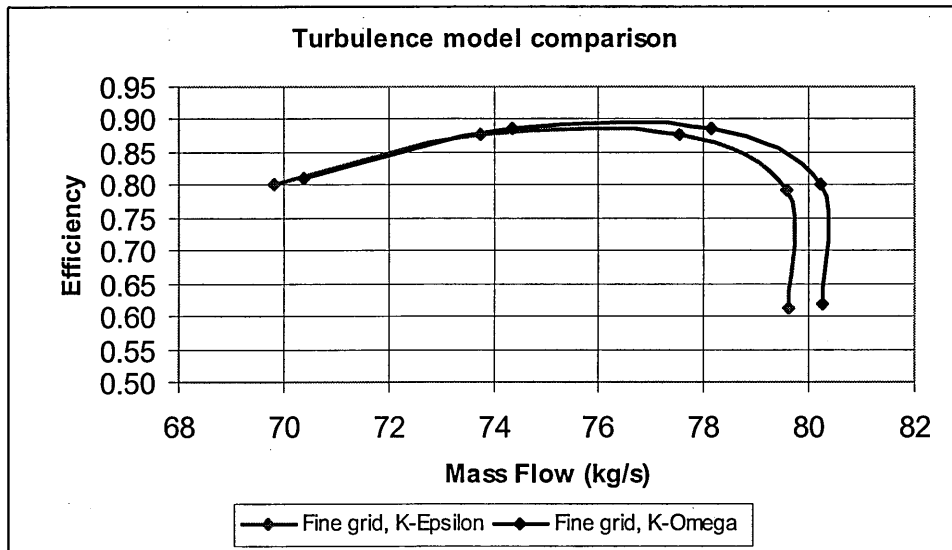


Figure 3.12 Turbulence model comparison Efficiency (fine grid)

The variation of mass flow, pressure ratio and efficiency throughout the constant speed line with the $k-\omega$ model and the $k-\epsilon$ model can be seen in Table 3.5 below.

Comparison with the fine grid			
	ΔW %	ΔPR %	$\Delta \eta$ %
Point 1	0.80624	-0.1443	1.15569
Point 2	0.80623	-0.233	1.15569
Point 3	0.80624	-0.2507	1.15569

Table 3.5 Turbulence model comparison

From the results shown in Table 3.5 it can be seen that the use of the $k-\omega$ model gives an increase in the compressor efficiency and an increase in mass flow. In terms of pressure ratio there is a reduction throughout the constant speed line.

These results are most likely due to the different fluid treatment approach close to the wall employed by the $k-\omega$ model. The flow is differently modelled around the blades, the hub and the shroud of rotor and stator.

Again the main limitation in order to draw any useful conclusions is the lack of experimental data for this model. For this reason it is not possible to specify which model gives the more accurate results. The differences given from the two turbulence models are highest for the efficiency with a more than 1% increase using the $k-\omega$ model. This study will use the $k-\varepsilon$ model only because it has been suggested that it is a more complete model and used previously for a number of applications with reasonable success.

3.4.3 Frozen rotor simulation

As mentioned earlier the flow simulation tool used for the analysis of the compressor stage three-dimensional flow has two procedures of modelling the interaction between the rotor and stator. These procedures are based on the idea either of a frozen rotor where an analysis is carried out at a particular position of the rotor stator interaction or the use of a mixing plane where it is assumed that the flow reaching the inlet to the stator is fully mixed.

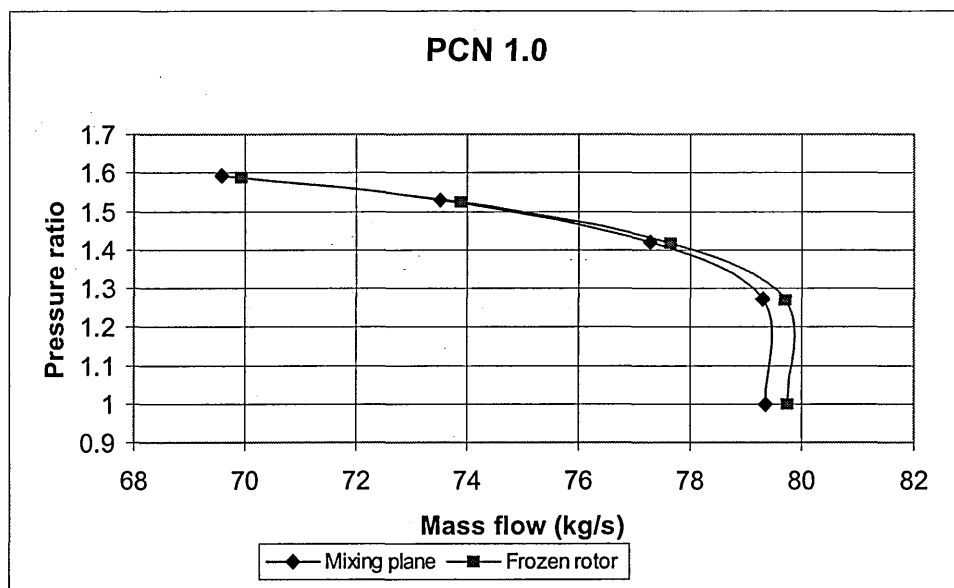


Figure 3.13 Frozen rotor and mixing plane comparison

The two Figures in this section present a study that has been carried out for this work, where the two methods are considered and compared in order to analyse any variations or results discrepancies between the two methods.

Figure 3.13 shows the compressor characteristics plotted as pressure ratio with mass flow at the design point rotational speed and Figure 3.14 shows the same simulation procedure results in terms of efficiency and mass flow.

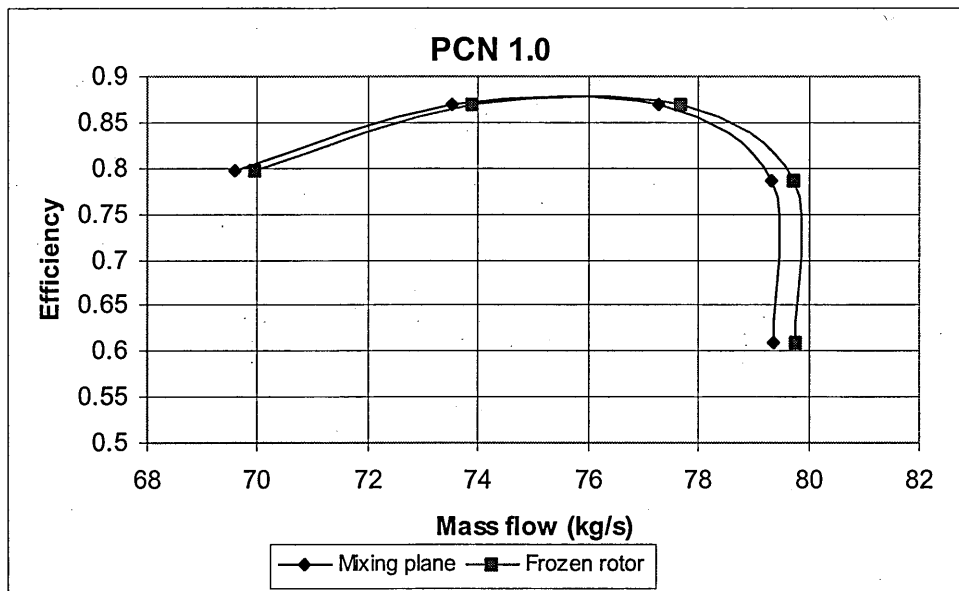


Figure 3.14 Frozen rotor and mixing plane comparison

Results from both Figures present a variation between the cases of frozen rotor application and the mixing plane. These variations amount to a maximum of 0.5% which when considering the time difference of the two modelling procedures it is more appropriate to follow the mixing plane application. Another point that needs to be made is that for the compressor stage model there are no validation data in order to draw conclusions on which method is more accurate. So the logical thing in this case is to go with the procedure with less computational time, considering the time restrictions for this study.

3.5 Clean compressor

The compressor simulation initiates with the flow simulation at clean conditions. Clean conditions are considered when the inlet compressor face is uniform both in terms of pressure and temperature. The compressor model used for this study has 19 rotors and 23 stators but for the purpose for the clean case simulation there is only a rotor/stator combination used. The reason for this is

that all passages will be introduced to exactly the same initial flow and it is safe to assume that the flow characteristics will be the same for every passage. This helps to minimise on computational time.

Figures 3.15 and 3.16 that follow present the pressure ratio, efficiency and mass flow results obtained from the flow simulation for the whole first stage.

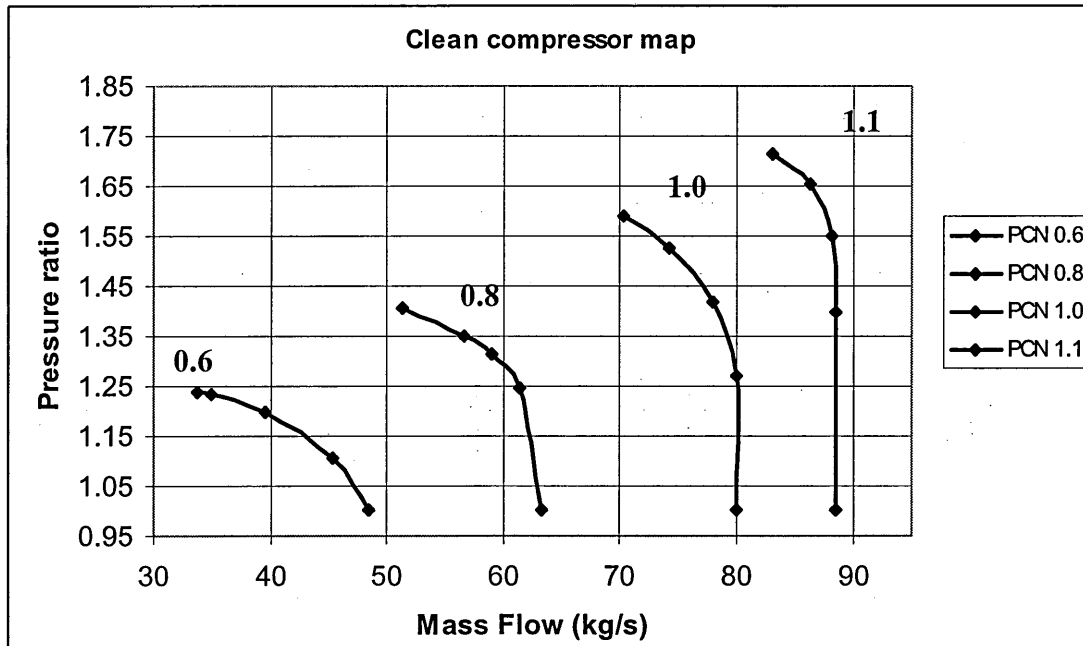


Figure 3.15 Clean map Pressure ratio

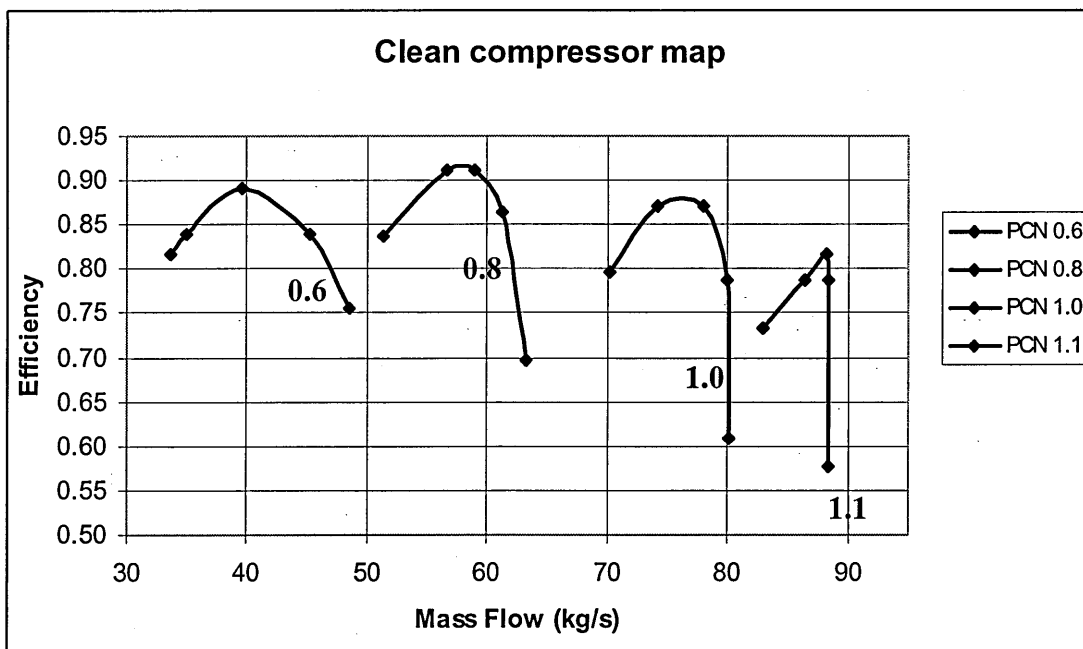


Figure 3.16 Clean map Efficiency

The runs have been carried out at four different rotational speeds, which are presented in terms of a decimal fraction of the non-dimensional rotational speed (PCN). PCN 1.0 is considered to be the design point rotational speed. The final point for each speed line, correspond to the last possible numerical simulation that was carried out. The simulation program tends to diverge for any further point. This last point has been obtained by setting the highest outlet static pressure possible at steady state simulation and is defined as the numerical surge.

The real surge is an unsteady phenomenon and it needs to be modelled with a transient simulation, while the simulations pursued are only steady. Although obtaining the surge point of the compressor is an important aspect in modelling, this work does not go into this area of study.

3.5.1 Flow analysis

This section presents the flow analysis for the simulation under clean conditions. The analysis has been carried out for the design point rotational speed and all the parameters have been analysed for three points on the speed line that correspond to the point closest to choke conditions (point1), the point closest to the design point (point2) and the pint closest to the numerical surge (point3). Figure 3.17 that follows present the three points mentioned above.

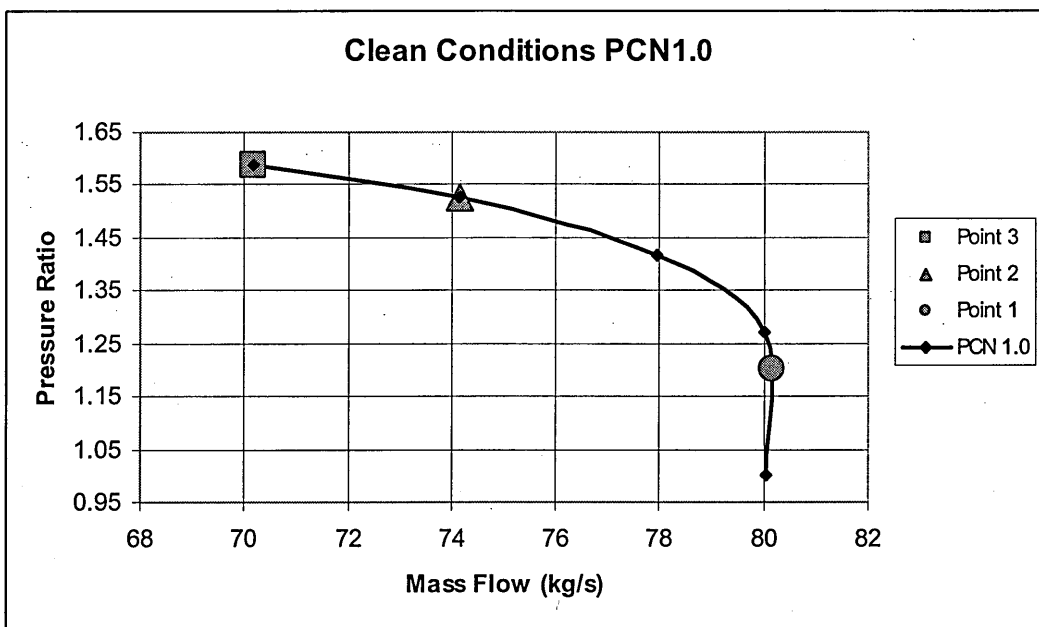
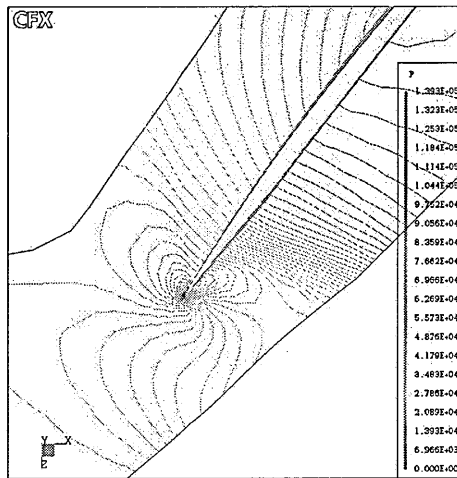
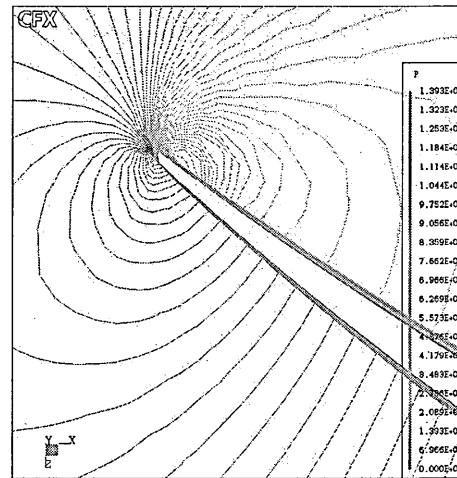


Figure 3.17 Clean design speed line

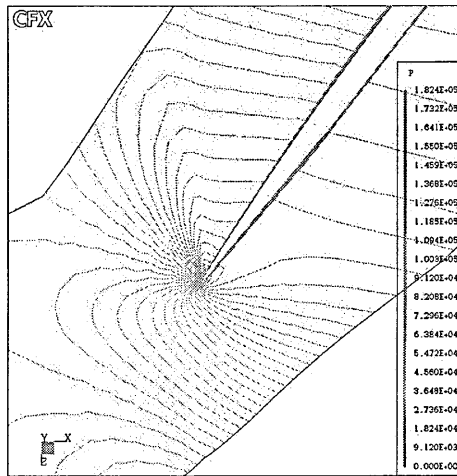
As mentioned earlier the second point of analysis is the one that corresponds more to the design point. Therefore, for the purpose of this analysis the second point will be referred to as the design point.



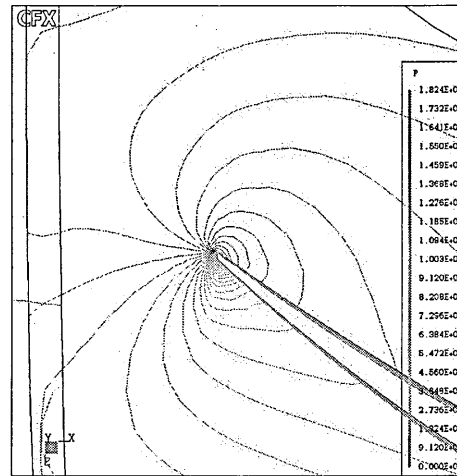
Point 1, rotor



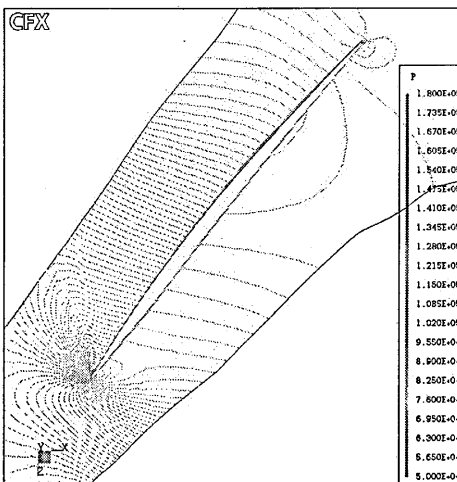
Point 1, stator



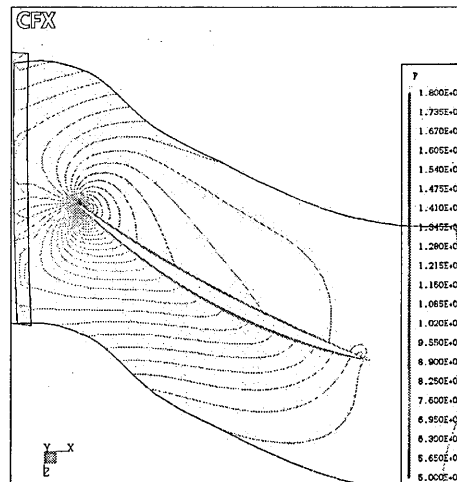
Point 2, rotor



Point 2, stator



Point 3, rotor



Point 3, stator

Figure 3.18 Static pressure distribution at the 50% of the rotor and stator blade

In terms of flow characteristics analysis one of the interesting features to investigate is the position of the stagnation point for both rotor and stator. Figure 3.18 above presents the stagnation points for both rotor and stator at the three points analysis mentioned earlier.

Since this is a transonic compressor, shock waves are expected to appear moving from the hub toward the tip. The pictures below show the contours of the relative Mach number at the design point.

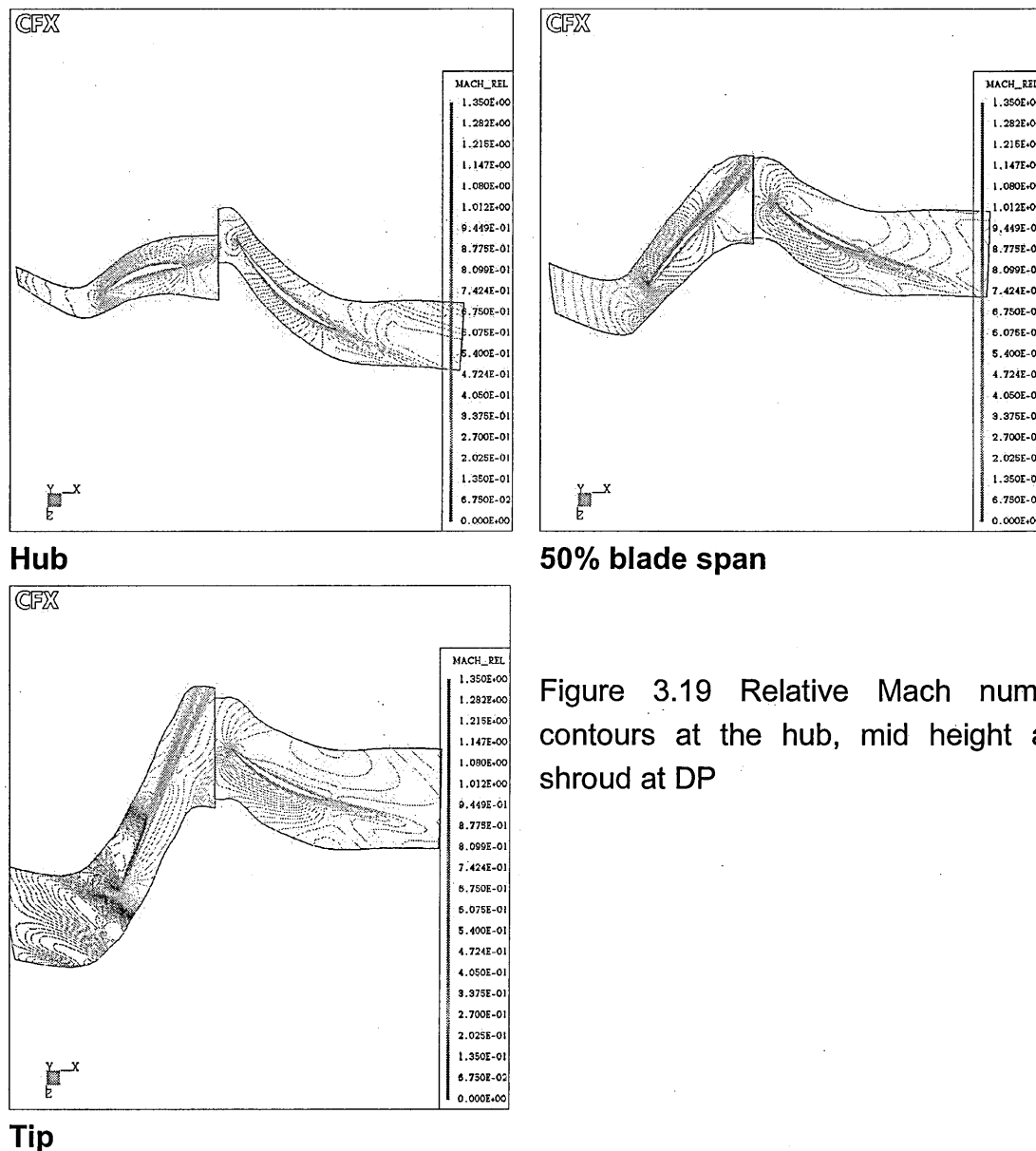


Figure 3.19 Relative Mach number contours at the hub, mid height and shroud at DP

The Mach number increases and the flow become supersonic around mid height of the blade span. Moving up to the tip, the shock waves become stronger and more evident. The shock wave appears at midway of the blade chord and it develops in front of the leading edge of the following blade. The shock wave structure can be seen in a J plane view of the passage, including both rotor and stator.

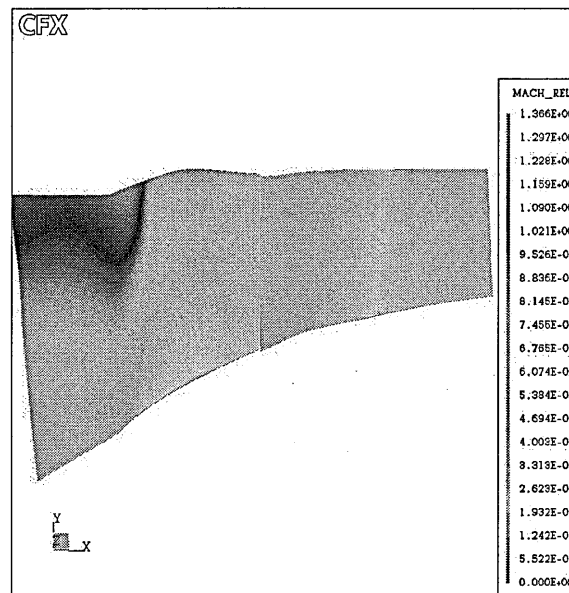


Figure 3.20 Relative Mach distribution at DP along the first stage

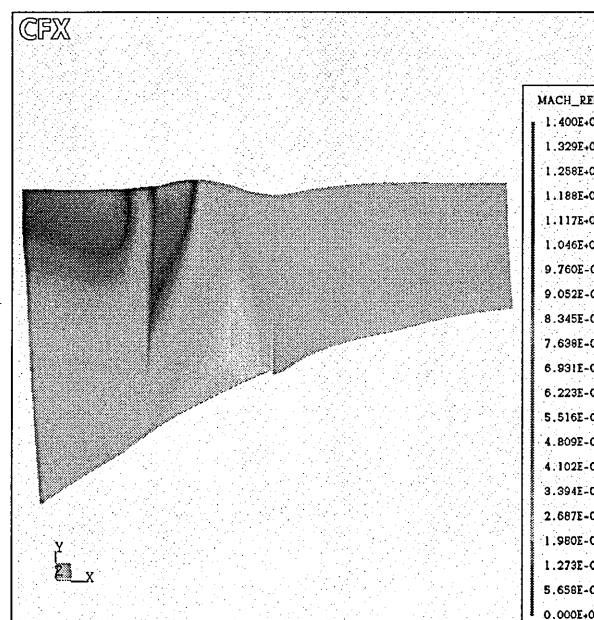


Figure 3.21 Shock wave structure at DP

Figure 3.20 shows the shock wave and the supersonic flow in front of the rotor. The shock wave begins from the shroud and, as mentioned before, penetrates in the flow field until almost mid height of the blade span. Figure 3.21 underlines the structure of the shock waves along the rotor blade.

An interesting flow characteristic obtained from the simulations is the direction of the flow when leaving the rotor trailing edge. The air velocity tends to increase leaving the trailing edge at the hub and moving towards the mid of the field. This can be seen in Figure 3.22.

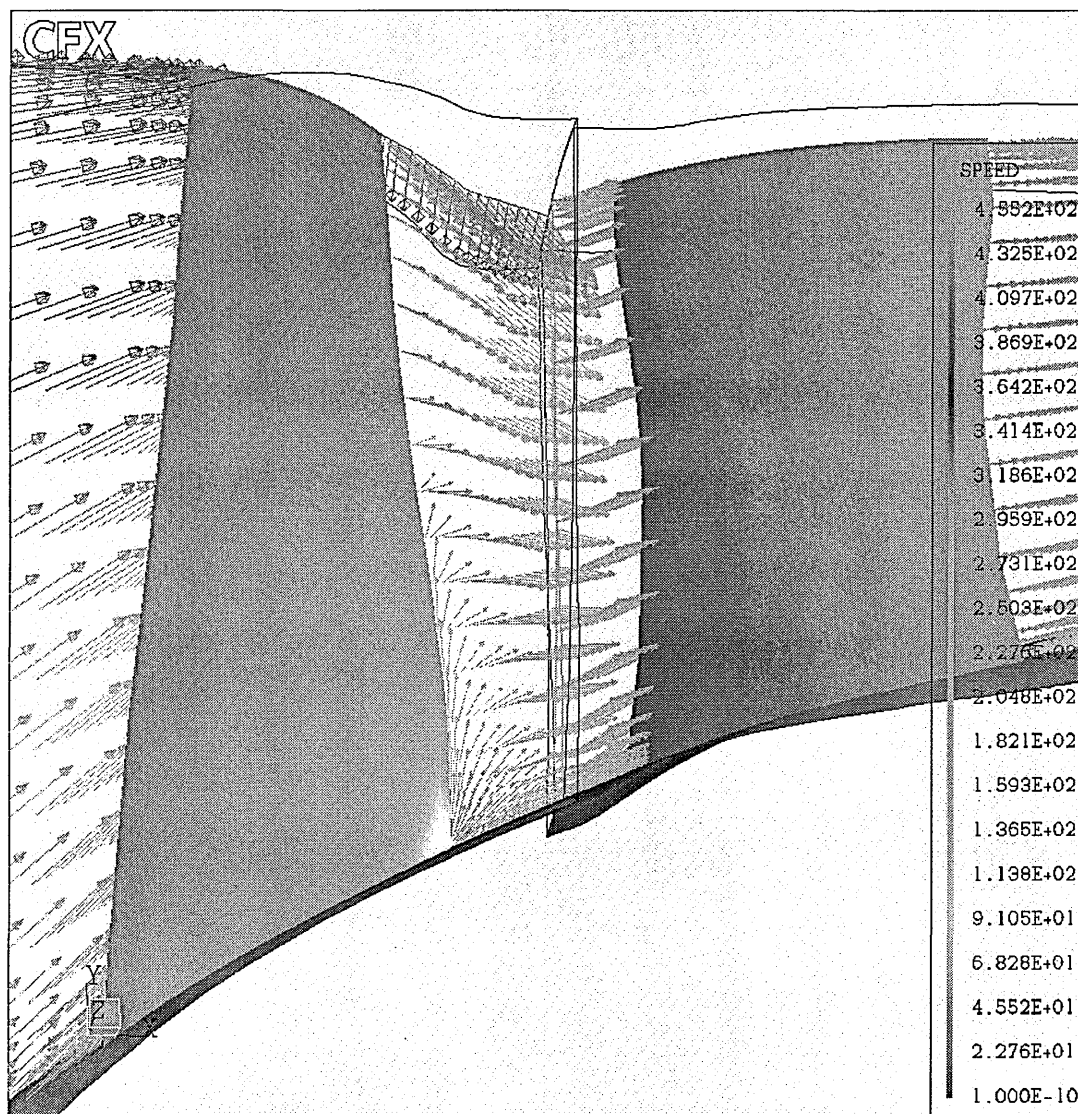


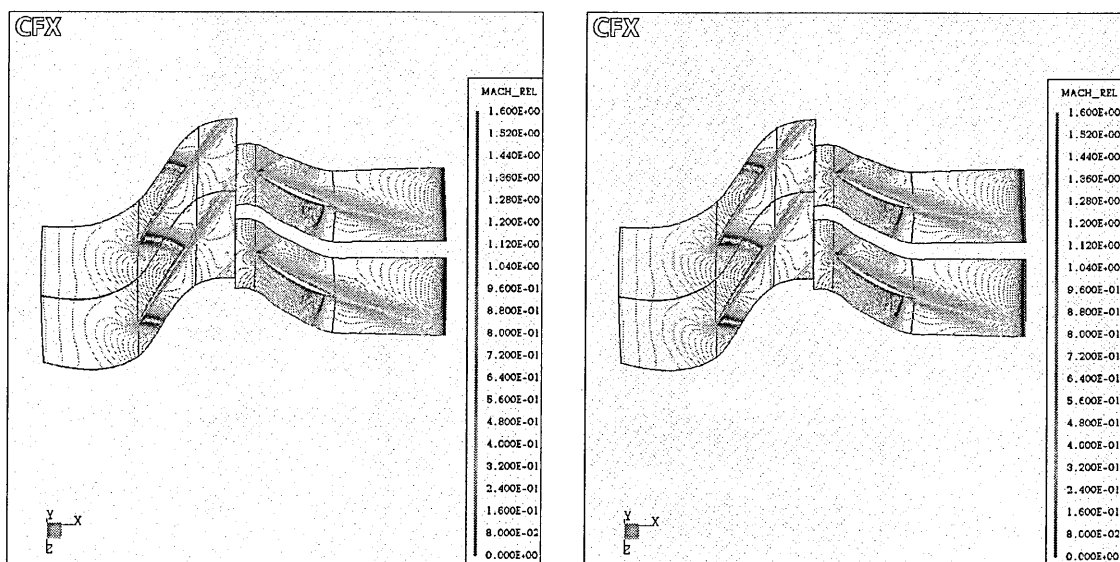
Figure 3.22 Flow speed at DP

3.5.2 Choke analysis

In terms of a compressor speed line, the first point mentioned earlier corresponds to the condition where the flow is almost choked. Since the static pressure is the outlet boundary condition set in Tascflow, this analysis point out the effects in terms of compressor performance and flow field if the static pressure is gradually reduced once choking is reached.

The reduction of static pressure makes more mass flow to pass through the compressor because of an increasing favourable pressure gradient, until it chokes. From a computational point of view, the software gets convergence much more easily while the static pressure reduces.

As Figure 3.23 shows, the mass flow changes with changes to the outlet conditions. In particular, there is a degradation of the overall compressor performance while the static pressure reduces (reduction in PR and efficiency). The flow field shows shock waves from hub to tip along both rotor and stator blades. The shock waves become stronger approaching to the shroud. The reduction of the static pressure makes the shock waves to be slightly stronger, but not great variations are observed in the flow field as the following Figures show.



$p_{out} = 50000 \text{ Pa}$

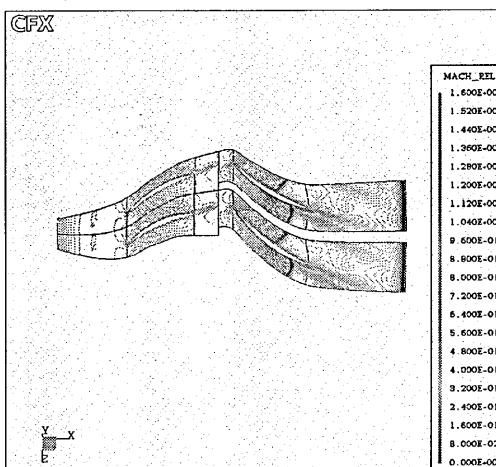
$p_{out} = 20000 \text{ Pa}$

Figure 3.23 Shock waves structure at 50 % blade span for two different outlet conditions

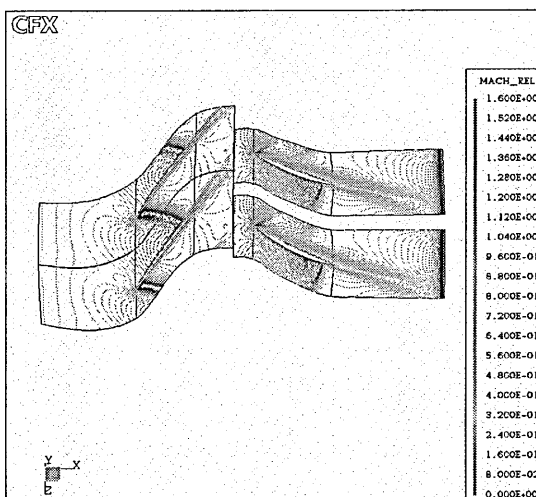
Passage shock waves become visible at the rotor and stator.

The Mach number at the outlet of the stator increases and the flow becomes supersonic. Naturally this is not physical, but it is due to the numerical simulations. The boundary conditions at the inlet and outlet are fixed and they do not change. The rest of the flow inside the computational domain develops while the simulation goes towards convergence. Obviously the solution adapts itself to the boundary conditions and since the outlet static pressure is very low, the flow turns supersonic when the code solves the solution in the nodes close to the outlet plane.

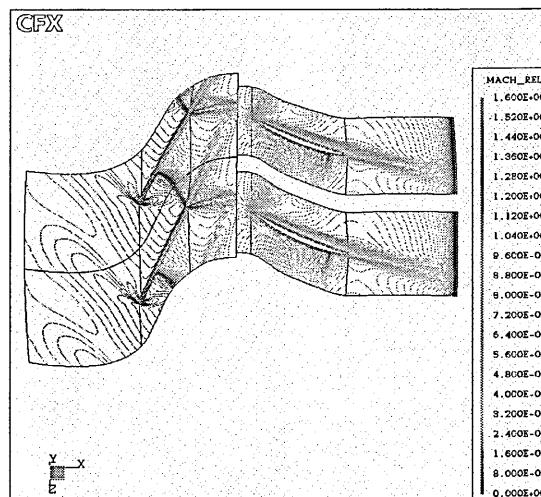
Apart from this region, the rest of the flow is reasonable and reliable. Unfortunately the numerical solutions cannot be independent from the boundary conditions. Figure 3.24 shows the shock waves structures from the hub toward the tip.



Hub



50% blade span

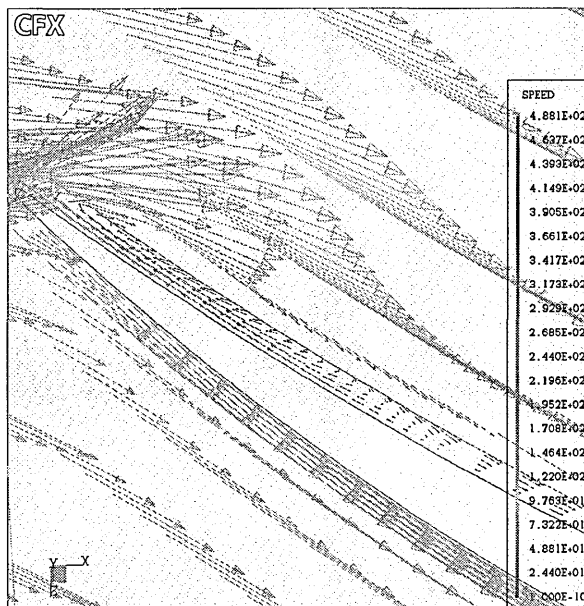


Tip

Figure 3.24 Shock waves structure from hub to tip, choke analysis, fine grid

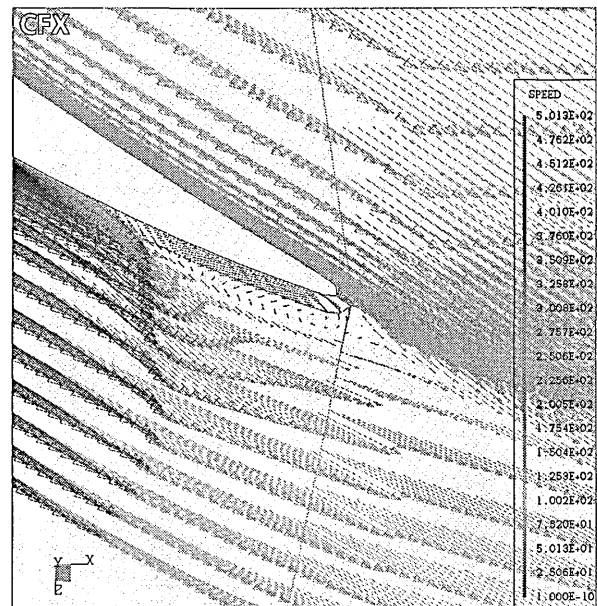
At the hub the shock waves appear along the stator blades. Moving toward the tip, the shock waves become less strong on the stator but stronger on the rotor.

Looking at the speed vectors, it is possible to observe flow recirculation on the stator next to the leading edge close to the shroud and next to the trailing edge close to the hub. That is shown in Figures 3.25 and 3.26. This behaviour is observed for each condition of pressure analysed. The recirculation is more noticeable as the outlet static pressure lowers.



90% blade span, stator LE

Figure 3.25 Flow recirculation at the stator leading edge



10% blade span, stator TE

Figure 3.26 Flow recirculation at the stator trailing edge

The flow separation is due to the shock waves. The rapid flow compression through the shock wave surface, together with the loss of total pressure makes the air to decelerate as much as to produce breakaway.

It is interesting to notice how the recirculation occurs close to the wall and how it is captured by the O-grid around the blade. The use of a finer grid would allow looking at details not marked in the coarse grid.

CHAPTER FOUR

Engine Performance Simulation

4.1 Introduction

Gas turbine modeling and performance simulation is an important area for both economic and safety issues. This is one of the four main topics that govern this study. In this section a small introduction will be given to the aspects of engine modeling and performance simulation. Saravanamuttoo [44] has provided extended information on gas turbine theory issues. Walsh and Fletcher [45] introduced methods of engine modeling for various engine configurations. Pilidis [46] has given a review in gas turbine theory and performance analysis. Mattingly [34] has provided a fundamental discussion of gas turbine propulsion.

The size of a gas turbine engine is usually defined by the mass flow and overall pressure ratio. From preliminary cycle calculations it is fairly straightforward to establish the mass flow requirement for a given power output, and the pressure ratio, which for any given maximum cycle temperature will give the maximum overall thermal efficiency. After determining mass flow, pressure ratio and turbine entry temperature from preliminary calculations, other suitable design parameters for a particular gas turbine system may be chosen. Individual engine components can then be designed in detail so that the complete system will give the required performance when running at the design point.

Apart from the design point performance of the gas turbine, its overall performance over the entire operating range of speed and power output needs to be established as well. This is normally referred to as the Off-Design performance.

Experimental data from actual rig tests or previous experience can be useful in determining the performance characteristics of individual engine components. Such data can be plotted together in a diagram to form a 'component characteristic' or 'component map' that describes the overall component performance under a wide range of operating conditions. Such a component map can be seen in Figure 4.1.

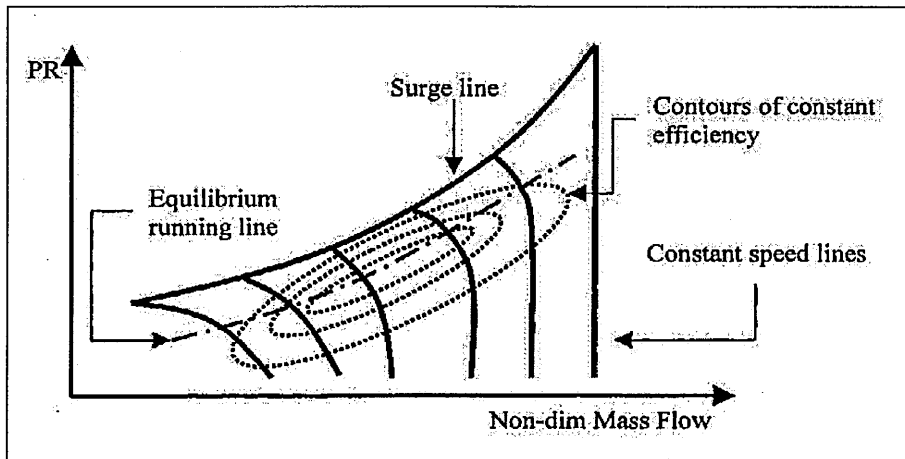


Figure 4.1 Compressor map

However, when individual components are linked together in an engine system, the range of possible operating conditions for each component is considerably reduced. When the engine is running at a steady state, corresponding operating points on the map of each component can be found. These equilibrium running points may be plotted on the compressor characteristic diagram, for a series of speeds, and joined up together to form a running line.

From such an equilibrium running diagram, the compressor surge margin can also be obtained (the proximity of the operating line to the surge line). The operation of the gas turbine engine will become unstable if the running line is displayed beyond the compressor surge line. Moreover, the equilibrium running diagram shows the region of compressor efficiency in which the engine operates at a given pressure ratio and non-dimensional mass flow. Preferably, the equilibrium running line should lie close to the region of points of maximum compressor efficiency.

Once the operating conditions of an engine have been determined various performance curves of power output, thrust, specific thrust, specific fuel consumption, etc. can be obtained.

Ambient conditions can also have a big impact in the performance of a gas turbine engine. The effects of high and low ambient temperatures and pressures must all be taken into account in gas turbine performance.

Therefore, the variation of engine performance with operating conditions is clearly a very important economic and safety issue and hence the need for an accurate gas turbine modeling and performance simulation.

4.2 Two spool low bypass turbofan engine modelling

For the purpose of this work the simulation of a two spool low bypass turbofan engine has been modeled with the use of Turbomatch. Turbomatch is a computer program developed by the School of Mechanical Engineering in Cranfield to reproduce design point and off-design performance of gas turbine engines using various pre-programmed subroutines to simulate the behavior of each engine component. The engine used for this study is based in a similar configuration to the EJ200 engine used for the Eurofighter aircraft and the main engine characteristics are summarized in Table 4.1.

EJ200 Engine specifications (uninstalled, ISA, SLS)	
Thrust, reheated	90 kN
dry	60 kN
Fan pressure ratio	4.2:1
Overall pressure ratio	26:1
Bypass ratio	0.4:1
Specific fuel consumption,	
reheated	47-49 g/kNs
dry	21-23 g/kNs
Air mass flow	75-77 kg/s
Overall length	4000 mm
Inlet diameter	740 mm
Weight	1040 kg

Table 4.1 Rolls-Royce EJ200 general specifications

The engine has been simulated at steady state performance, dry and reheats conditions and the influence of intake pressure recovery and bleed off-takes have been assessed. Simulation of pseudo-transient conditions has also been carried out. The results that follow represent the performance characteristics of the engine at various operating conditions.

4.2.1 Engine performance simulation

Based on the engine characteristics obtained from the previous Table, the first step in modeling the engine is to simulate the design point performance of the engine in order to produce the thrust provided under the appropriate specific fuel consumption. Once this is complete then the engine model can be used to simulate off-design conditions at various operational points.

The clean conditions simulation of the engine has been carried out and the equilibrium running line can be seen in Figure 4.2, plotted on the compressor characteristics map.

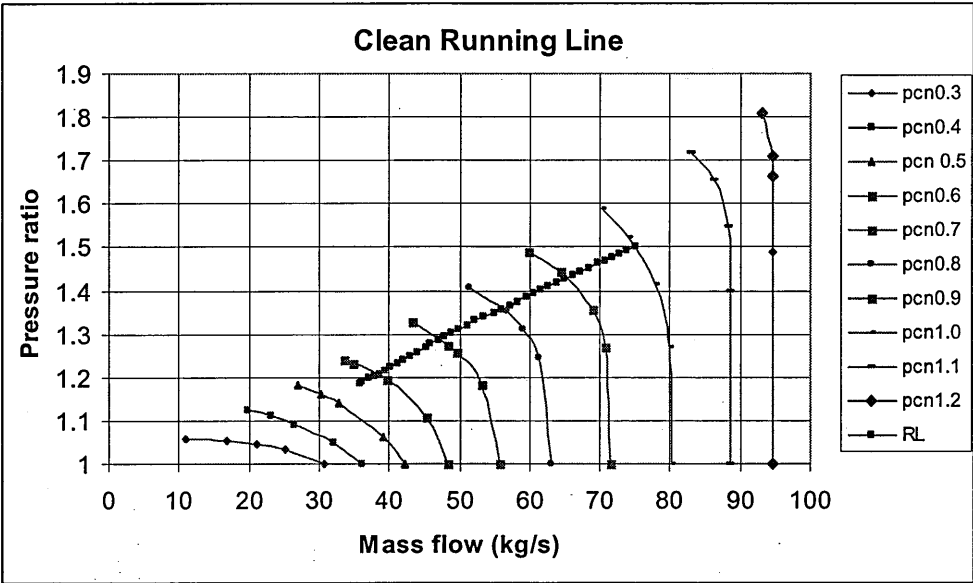


Figure 4.2 Clean running line

Clean conditions correspond to a uniform airflow entering the engine. In the sections that follow an analysis of engine performance at distorted and flow control conditions will be carried out.

4.2.1.1 Effect of flight velocity

Performance simulation and analysis of the engine at dry and reheat conditions has been carried out and Figures 4.3 and 4.4 present the effect of flight velocity on engine thrust and specific fuel consumption.

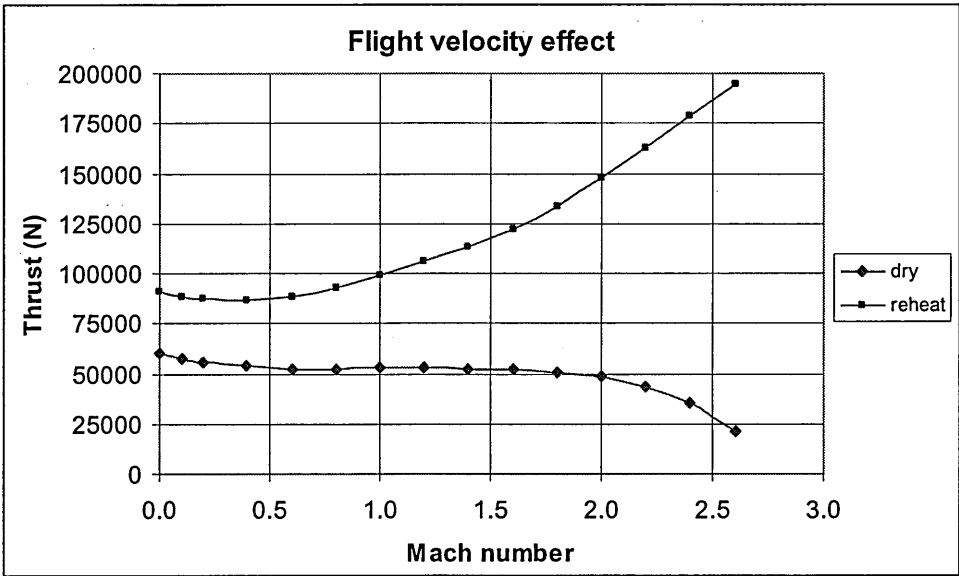


Figure 4.3 Effect of flight velocity on engine thrust

As the flight velocity of an aircraft changes, the performance of the engine is affected. Three main effects are taking place with increase in speed:-

- Momentum drag
- Ram compression
- Ram temperature rise

Momentum drag increases with Mach number and reduces the net thrust of the engine. Ram compression, on the other hand, has a double effect. It increases the pressure at the inlet and therefore flow density and mass flow into the engine rise. The second effect is of increasing nozzle pressure ratio and gross thrust. In terms of net thrust, ram compression and momentum drag are the primary effects. At low speeds, up to Mach numbers of 0.3-0.5, the effects of ram compression are relatively small and momentum drag predominates leading to reduced thrust. As Mach No increases, ram compression becomes more significant leading to increased thrust. In the case of a turbofan engine, however, the momentum drag effect is more pronounced due to the lower jet velocity and the larger inlet area.

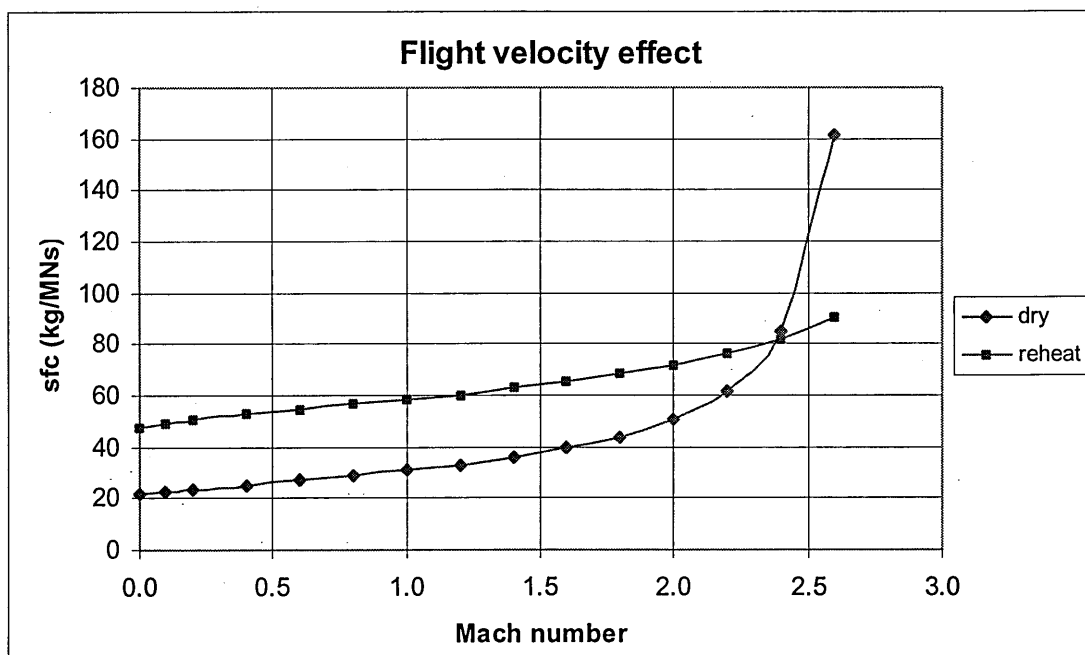


Figure 4.4 Effect of flight velocity on engine specific fuel consumption

In terms of non-dimensional performance, ram compression and temperature rise will result in reduced non-dimensional shaft speed, mass flow, pressure ratio and therefore specific thrust and thermal efficiency. In accordance with its

natural tendency the specific fuel consumption curve rises as Mach number increases. As far as compressor work is concerned, although pressure ratio reduces, ram compression and temperature rise are the dominant factors leading indeed to increased compressor work.

4.2.1.2 Effect of altitude

The effect of altitude on engine performance both at dry and reheat has been considered. The ambient static temperature falls linearly with altitude. Ambient static pressure and density also fall with altitude. With the continuous reduction in pressure and temperature, the mass flow reduces and this is the dominant effect on thrust and engine power. For a given Mach number the total temperature at the inlet of the compressor is proportional to the ambient static temperature. Therefore, the non-dimensional rotational speed and mass flow rise with altitude. Consequently, the engine behaves as its rotational speed rises giving increased pressure ratio. This in turn partly offsets the effect of reduced mass flow. The effect is experienced up to 11.5 Km and from then on the inlet temperature remains constant so that specific thrust no longer improves with increase in altitude, thus leading to continuous thrust reduction. Figure 4.5 presents the thrust variation with altitude change at a constant Mach number of 1.2.

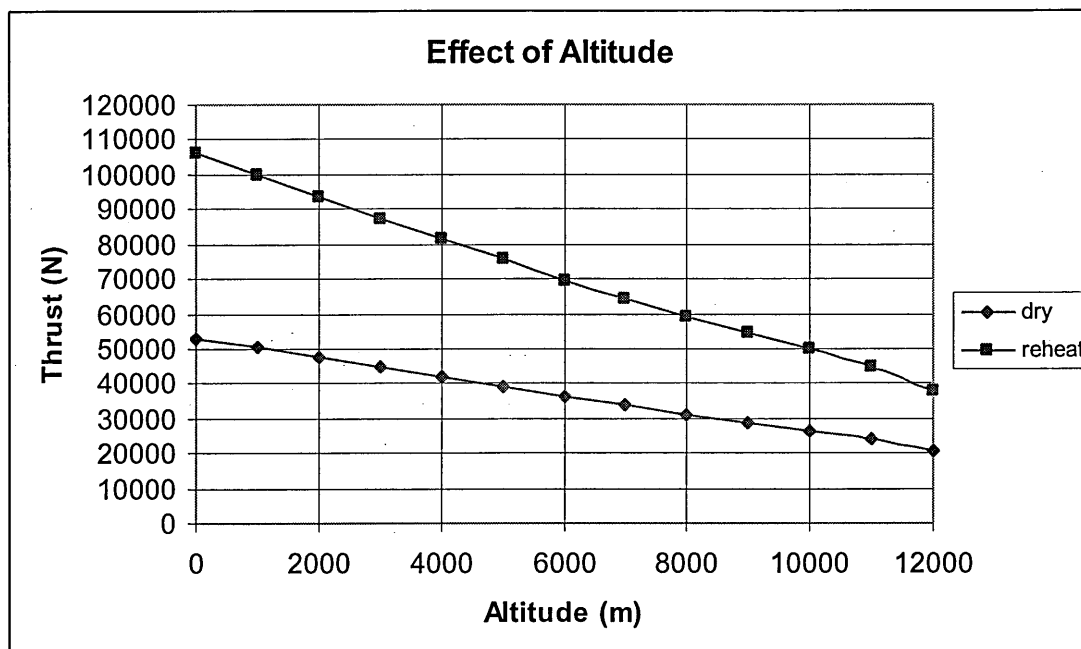


Figure 4.5 Effect of altitude on engine thrust

The rise in non-dimensional mass flow and rotational speed, experienced in the troposphere, increases the pressure and temperature ratio of the compressor causing an increase in thermal efficiency. Although the propulsive efficiency of the engine falls, the former effect predominates and the sfc falls; remaining constant for altitudes higher than 11.5 Km. The fuel flow, on the other hand, reduces continuously with altitude. Once again, the dominant effect in compressor work is the reduction in mass flow and as a result the work produced by the turbine falls with altitude. Figure 4.6 presents the specific fuel consumption variation with altitude change at a constant Mach number of 1.2.

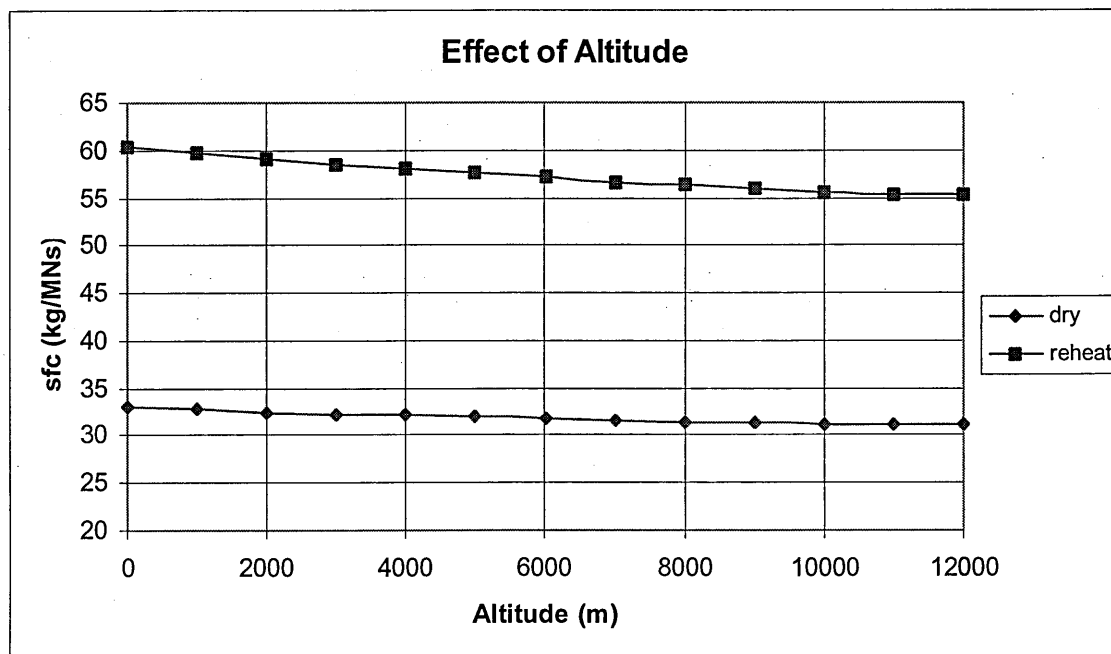


Figure 4.6 Effect of altitude on engine specific fuel consumption

4.2.1.3 Effect of shaft rotational speed

The next analysis that has been carried out is the effect of shaft rotational speed to the engine performance. For a properly matched engine operating at constant flight conditions and with no off-takes shaft rotational speed rises with turbine entry temperature (TET). In Figure 4.7 the effect of shaft rotational speed on engine thrust can be seen.

As the TET hence the rotational speed increases, both the mass flow and the non-dimensional mass flow rise. This action also increases the pressure and temperature ratio of the compressor. In terms of thrust, the rise in mass flow combined with the increased pressure ratio and TET lead to a rapid increase of net thrust. From the above it follows that specific thrust increases as well.

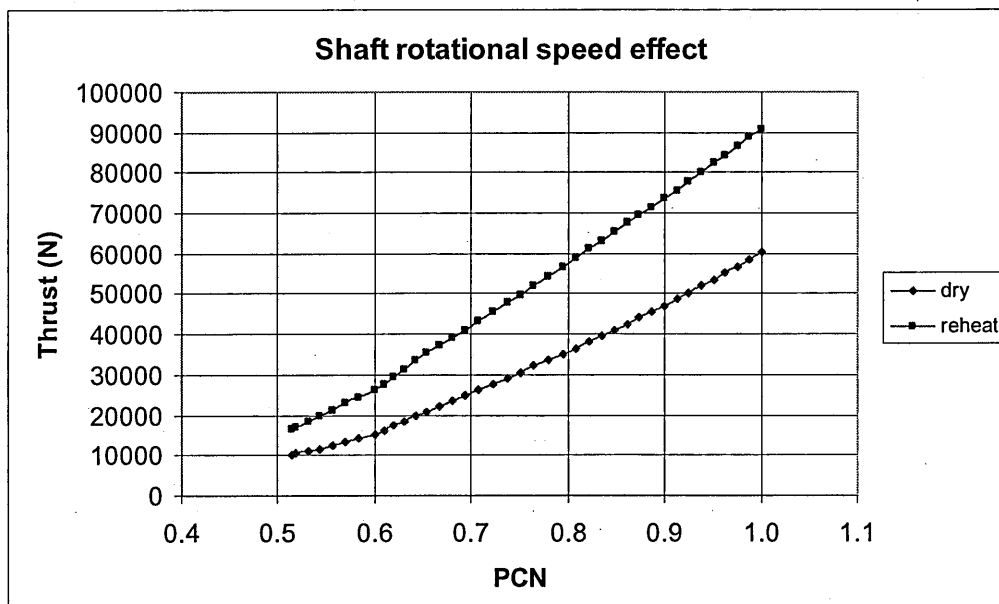


Figure 4.7 Effect of shaft rotational speed on engine thrust

Although fuel flow rises with TET, there is no mass flow effect on specific fuel consumption because its effects on thrust and fuel flow cancel out. However, the rise of TET and pressure ratio will improve thermal efficiency.

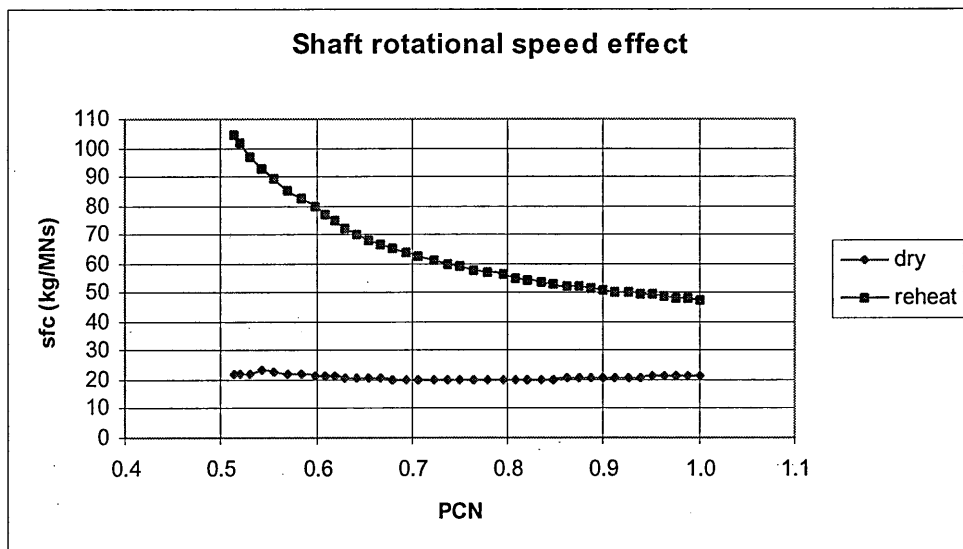


Figure 4.8 Effect of shaft rotational speed on engine sfc

Although propulsive efficiency falls, specific fuel consumption will experience a generally dropping tendency. The opposing effects of propulsive and thermal efficiencies cancel out as TET continues to increase resulting in increased

specific fuel consumption. These effects on specific fuel consumption can be seen in Figure 4.8.

4.2.2 Intake pressure losses

The purpose of this study as mentioned in previous chapters is to analyze the effect of intake airflow to the overall performance of the engine. One of the parameters that can affect engine performance is the pressure losses that occur within an intake duct. For this reason a preliminary study has been carried out in order to investigate this effect and the percentage penalty introduced on engine thrust and specific fuel consumption. This is analyzed in a variation of operational conditions.

4.2.2.1 Effect of flight velocity

Engine performance simulation has been carried out for different flight velocity conditions at a constant altitude of 10km. The cases considered include 5% and 10% intake pressure losses. Figure 4.9 shows the thrust penalty that the engine suffers with a reduction on intake pressure recovery.

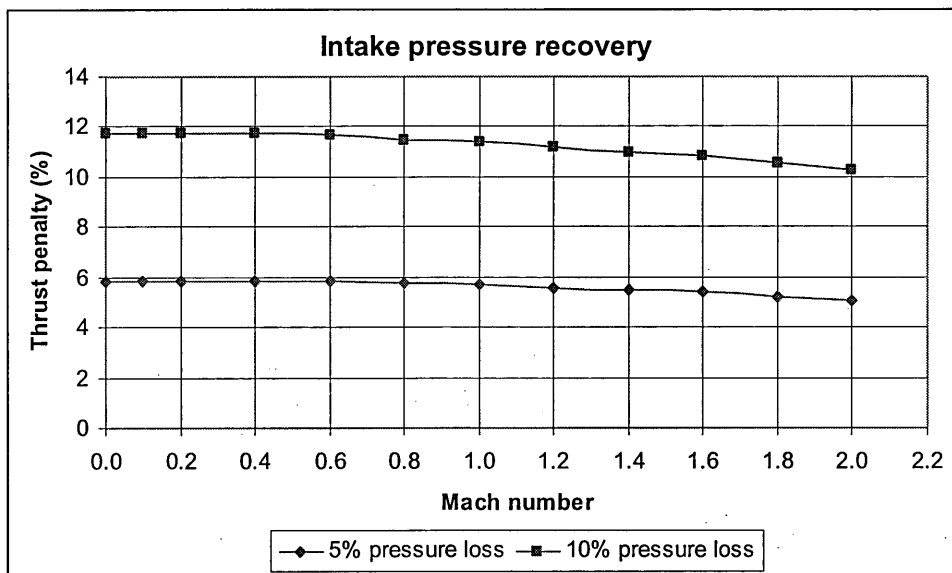


Figure 4.9 Effect of Mach number on PR penalties

The results show that with an intake pressure loss of 5% there is a 6% loss in engine thrust. By increasing the intake pressure losses to 10% the thrust penalty of the engine increases to 12%. This shows a linear relationship between engine thrust and intake pressure losses. In both cases thrust penalty drops with an increase in Mach number. The reason for this is because when

Mach number is increased, the effect of the intake pressure losses becomes smaller.

A similar trend can be followed for the penalty on specific fuel consumption, which is shown in Figure 4.10.

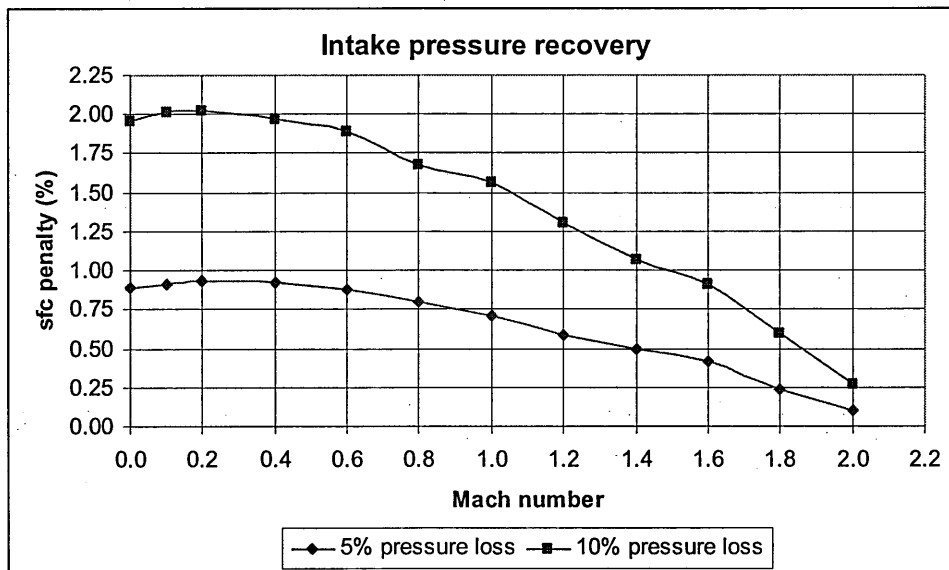


Figure 4.10 Effect of Mach number on PR penalties

Again there is a linear relationship between specific fuel consumption change with pressure losses change and the effect of the pressure losses decrease with an increase in Mach number.

4.2.2.2 Effect of altitude

In order to analyze the thrust and specific fuel consumption penalties of the engine with 5% and 10% pressure losses at different altitude the simulation of the engine model has been carried out from a range of sea level to 12000 meters at a constant Mach number of 1.2.

Figure 4.11 shows the thrust penalty at a range of altitudes for both intake pressure loss cases. It can be seen that the effect of pressure losses on engine thrust drops with an increase in altitude. The reason for that is that the pressure losses considered here are a percentage of the inlet pressure. With an increase in altitude the inlet pressure drops, corresponding to a decrease on pressure drop. Therefore, the pressure losses applied for every altitude are different, corresponding to a smaller pressure losses effect.

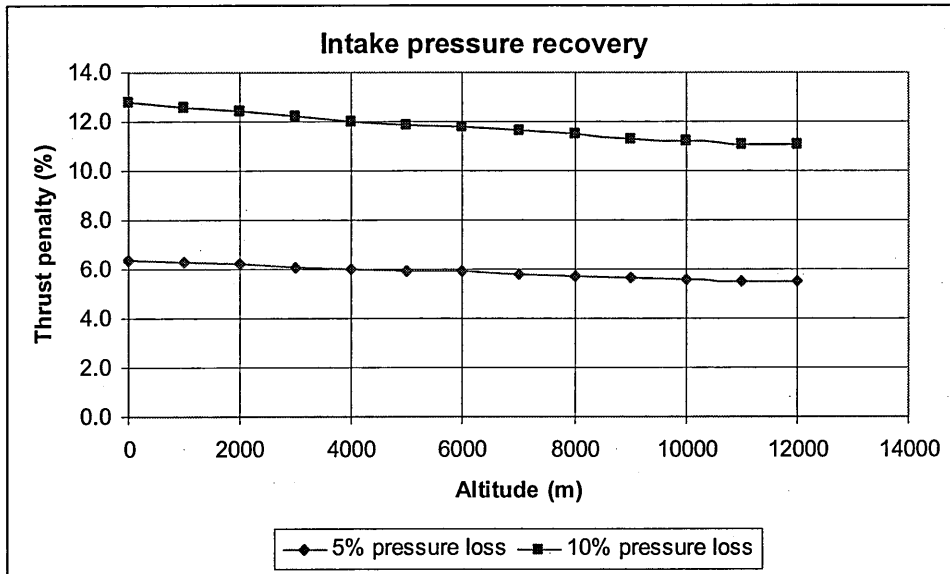


Figure 4.11 Effect of altitude on PR penalties

Figure 4.12 shows a similar decrease of specific fuel consumption for both cases. Again here the pressure losses applied vary with a change in altitude, hence giving a decrease of specific fuel consumption penalty.

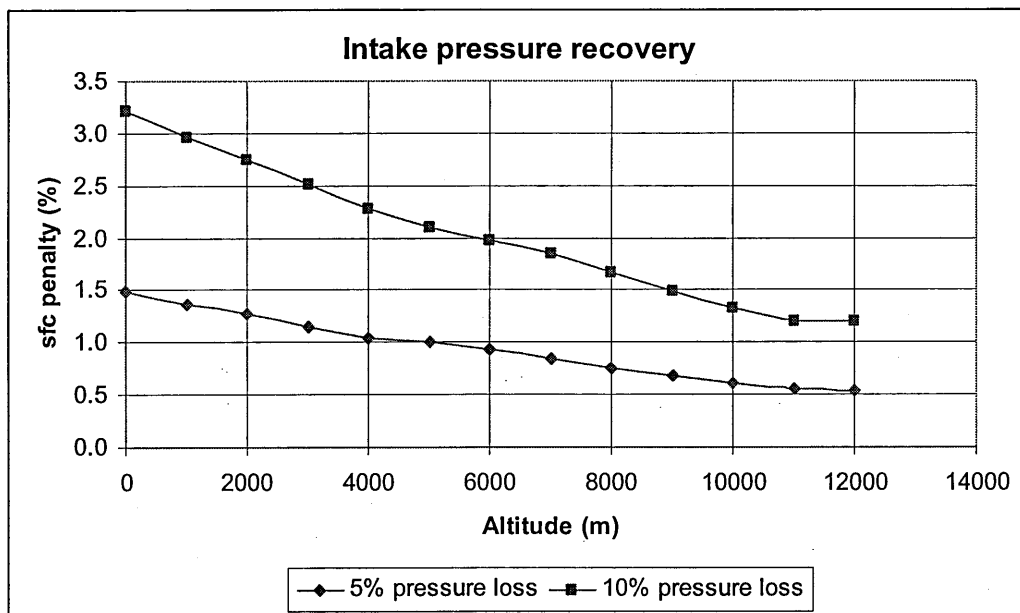


Figure 4.12 Effect of altitude on PR penalties

4.2.2.3 Effect of shaft rotational speed

The effect of intake pressure losses on different shaft rotational speed has also been investigated. The performance simulation has been carried out by changing the turbine entry temperature at a constant altitude and Mach number.

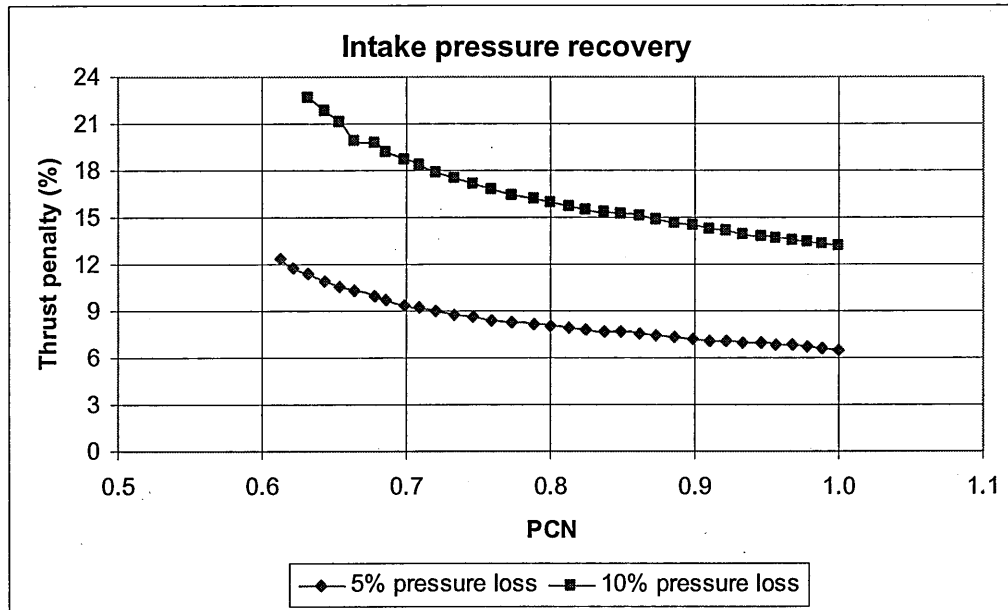


Figure 4.13 Effect of shaft rotational speed on PR penalties

Figures 4.13 and 4.14 show the thrust and specific fuel consumption respectively. For both engine performance parameters the penalty drops with an increase in turbine entry temperature.

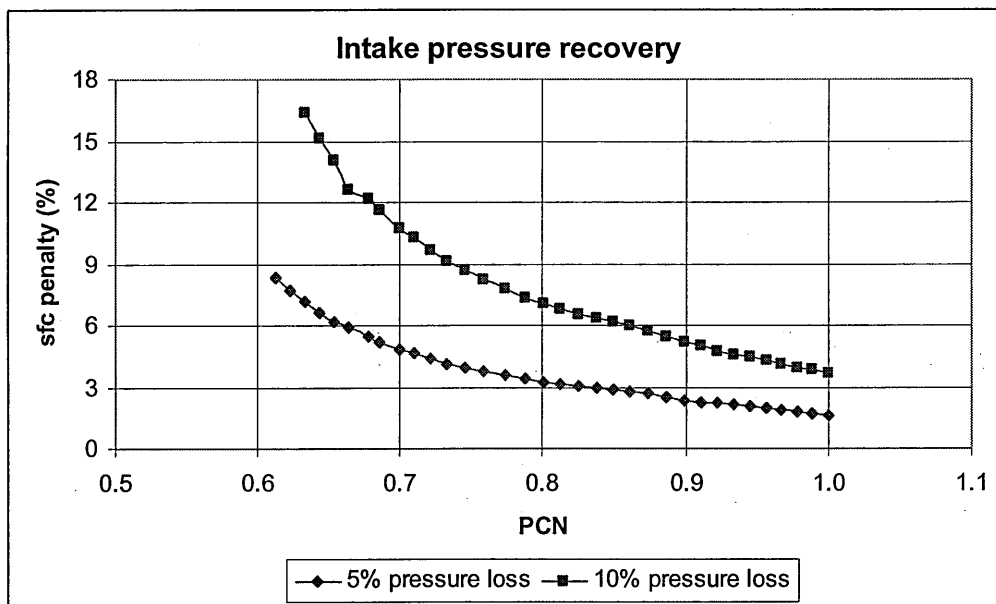


Figure 4.14 Effect of shaft rotational speed on PR penalties

In this case with an increase in turbine entry temperature, the mass flow and pressure ratio both increase while the pressure losses remain the same. For this reason the effect of pressure losses both for thrust and specific fuel consumption decrease.

4.2.3 Bleed off-takes

An important issue that this study is concerned with is the use of bleed air from the engine compressor. Off-takes have been previously studied by Laskaridis [47]. The objective of this study is to analyze the effects of distortion on overall engine performance and improve this distortion with the use of flow control methods. One of the flow control methods used for this study is injection. For this purpose air is extracted from the low pressure compressor and injected into the intake. This bleed air has a negative effect on a significant number of engine parameters like turbine entry temperature (TET), specific fuel consumption and thrust. Furthermore, the operation of the compressor and the handling qualities of the engine are also affected.

Compressor characteristics: For all but very low power settings the HP turbine will operate between choked nozzles. Therefore, by definition the non-dimensional mass flow at the inlet and exit will be constant. This also fixes the pressure and temperature ratio across the turbine. Taking bleed air from the compressor an increase in fuel flow and hence temperature is required. Since the compatibility of work between the compressor and turbine must be satisfied, the change in mass flow through the turbine will require an increase in TET. To ensure the correct value of turbine non-dimensional mass flow, the compressor pressure ratio will fall.

Effect on TET: Taking bleed air from the outlet of a compressor causes the delivery pressure and mass flow to drop by approximately the same percentage as the air off-take. Assuming constant TET and turbine pressure ratio, i.e. turbine operates between choked nozzles, the work output of the turbine drops immediately in proportion to the percentage of the air off-take. If turbine power is to remain constant an increase in TET is required. This action will also determine the operating point of the compressor re-establishing a new mass flow and pressure ratio, as explained above.

Effect on Net Thrust: When the engine is operating well away from maximum output, whether this is defined by a limit on TET or aerodynamic speed limit, it is possible to maintain constant thrust as well as provide the off-takes. However, if the engine is operating at rated conditions, such as maximum take-off, maximum climb, maximum cruise thrust etc. the need to produce extra power for off-takes will reduce the power available for thrust.

Effect on sfc: For the engine to maintain a specified thrust output and also provide bleed air, extra energy input is required. This is provided by burning fuel at higher rate. Thus, sfc increases. The increase in sfc will be a function of the size of the off-take relative to the total power delivered by the turbine and the characteristics of the engine, i.e. overall pressure ratio, specific thrust, TET, etc. Thus for a given off-take the effect will depend on the size of the engine, its configuration and finally its operating conditions.

4.2.3.1 Effect of Flight Velocity

The effect of flight Mach number on the performance of the engine model operating at rated TET and with one percent off-takes of the overall mass flow is shown in Figure 4.15 and 4.16. The bleed air is extracted from the end of the first stage of the low-pressure compressor.

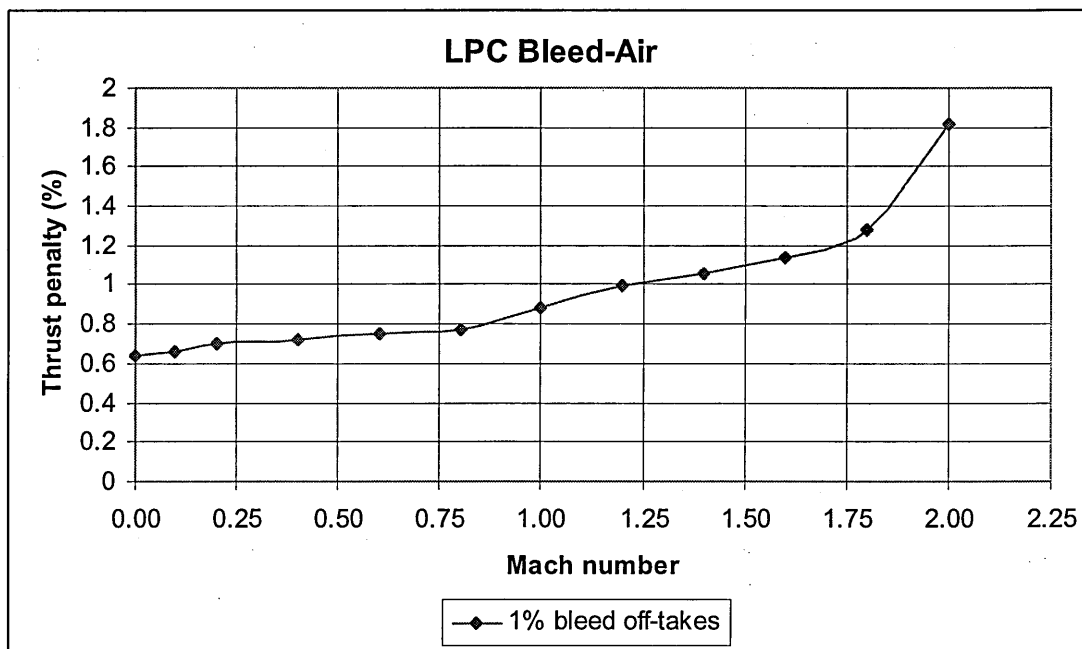


Figure 4.15 Effect of Mach number on Bleed Air Penalties

As previously discussed, increase of Mach No leads to more air pumped into the engine reducing the proportion of the off-take relative to the amount of air through the core. Therefore, performance penalties for fixed off-takes reduce with Mach number. However in the case of constant percentage of bleed air the effect of compressor inlet temperature and pressure on pressure ratio, non-dimensional power setting and specific thrust implies increased performance penalties with Mach number.

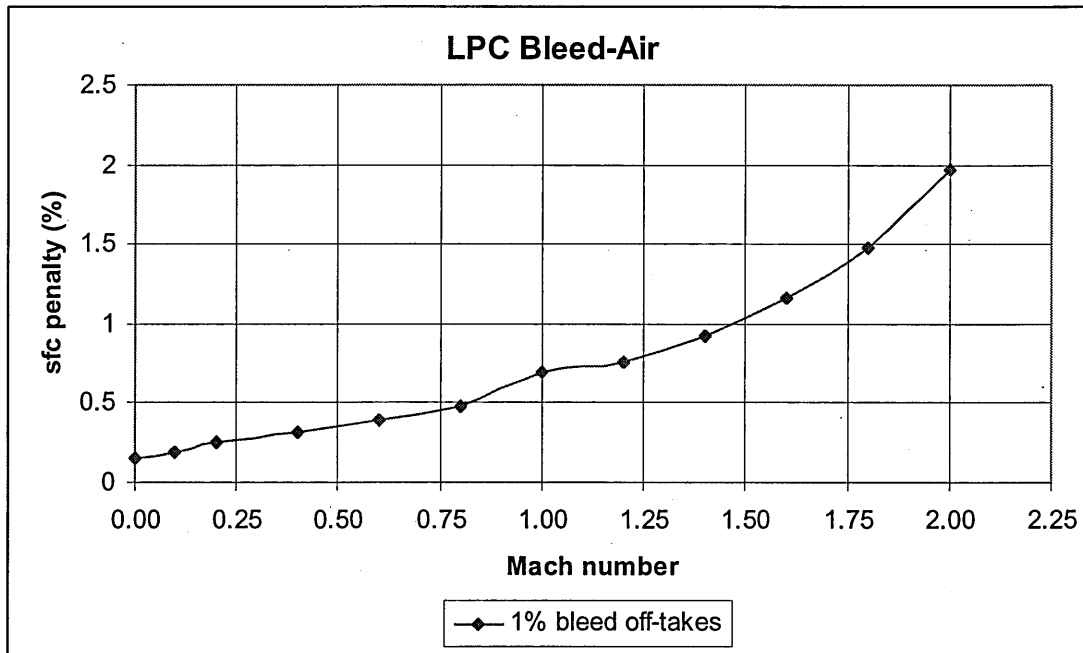


Figure 4.16 Effect of Mach number on Bleed Air Penalties

4.2.3.2 Effect of Altitude

The effect of altitude on the engine model operating at rated TET, 1.2 Mach and one percent of bleed air is shown in Figures 4.17 and 4.18.

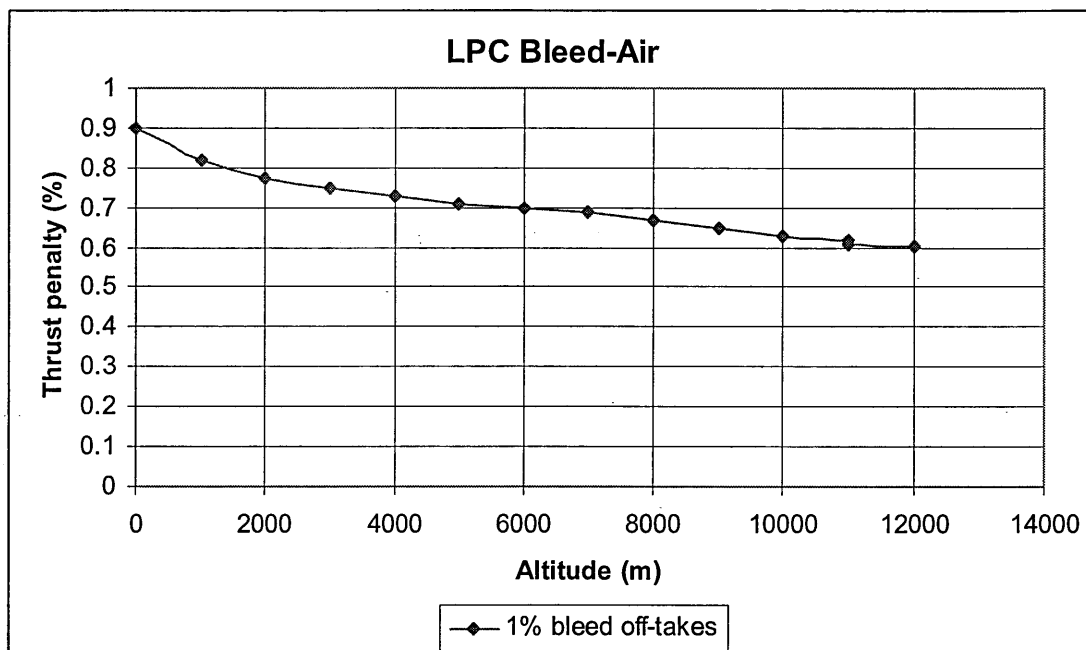


Figure 4.17 Effect of Altitude on Bleed Air Penalties

These plots indicate that for a fixed amount of bleed air engine performance penalties increase with altitude. However, these penalties are the net effect of two main counteracting factors: increased TET to engine inlet temperature ratio and increased percentage of bleed air. Once again the size of the off-take relative to the power of the engine is the predominant factor leading to higher performance penalties as altitude increases. Yet, it can be seen that up to an altitude of 11.5 km the effect of increased TET to engine inlet temperature ratio (hence, increased pressure ratio, specific thrust and thermal efficiency) partly offsets the effect of rising percentage of bleed air. This conclusion is also supported by the performance of the engine under constant percentage of bleed air.

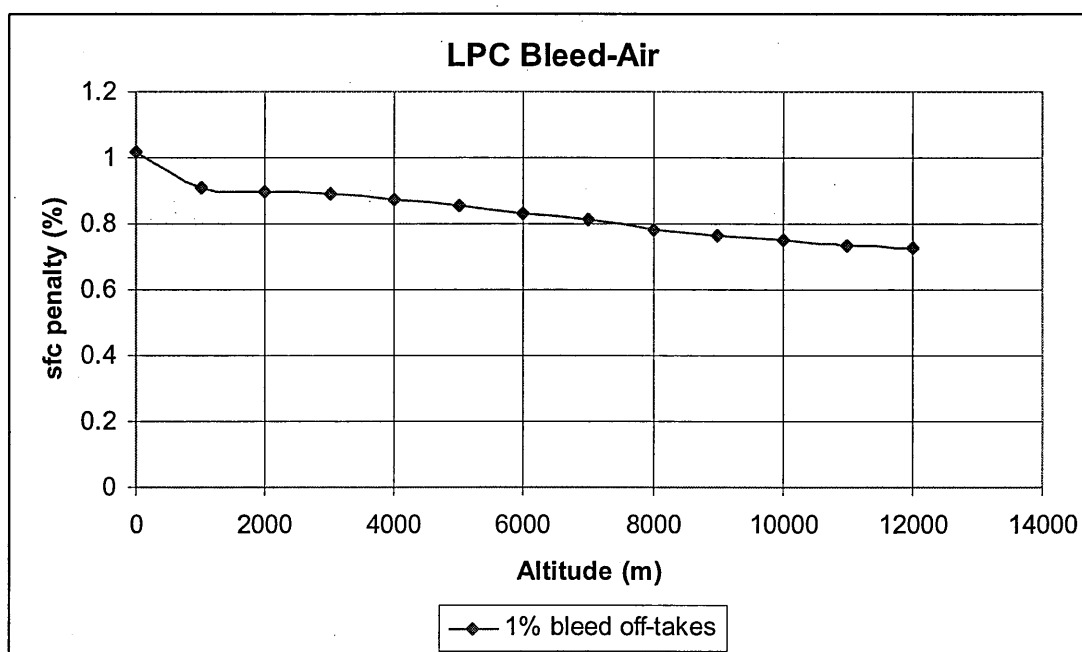


Figure 4.18 Effect of Altitude on Bleed Air Penalties

4.2.4 Power off-takes

When the engine accelerates work has to be done against the shaft inertia, to increase speed and to also increase the temperature of the various elements. This is exactly analogous to taking a shaft off-take and causes the same upward motion of the running line towards surge. Deceleration, on the other hand, has the opposite effect. When designing the engine the running line will be positioned to allow sufficient surge margin for acceleration. With a shaft power off-take, however, the surge margin is reduced with the subsequent danger of surge. Although taking bleed can control this effect, this action restricts the thrust output and the efficiency of the engine.

Compressor characteristics

The effect of taking bleed air from the compressor has been explained earlier and the movement of the running line can be seen in Figure 4.19. In the same Figure the effect of shaft power extraction can also be seen.

When shaft power is extracted the turbine will have to produce a higher level of work to cover the higher load. If engine speed is to be maintained the fuel control system will increase fuel flow and thus TET. To satisfy the choked flow condition the increase in TET will cause pressure to increase.

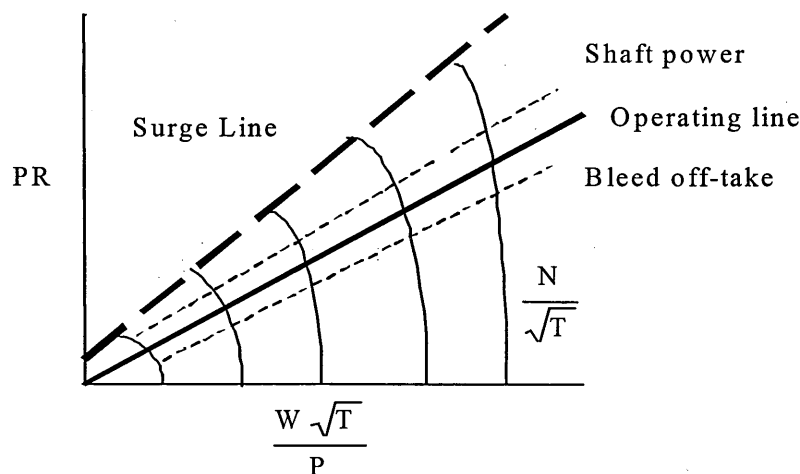


Figure 4.19 Compressor characteristics map

While extraction of bleed air increases the surge margin, shaft power off-takes have the opposite effect. Ideally the running line of the compressor will be positioned for maximum efficiency. Movement of the running line as bleed or shaft power is extracted will thus result in a change of component efficiency.

4.2.4.1 Effect of Altitude

The effect of flight velocity to the engine performance under power off-takes conditions is presented in Figure 4.20 that follows. The cases that were considered, extract two different amounts of power from the low pressure turbine and an analysis is carried out on the engine performance penalty that this has.

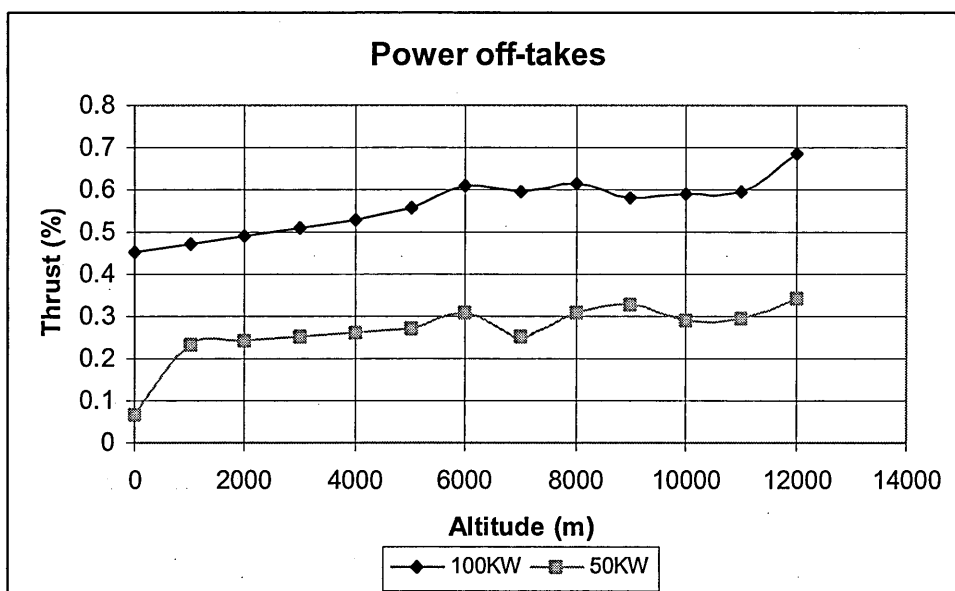


Figure 4.20 Power off-takes thrust penalty

It can be seen both from the above and below Figures that thrust and sfc have a considerable penalty when power is extracted from the engine. For both cases the power is extracted from the low pressure turbine. The power extracted is 100 KW and 50 KW respectively. The results obtained show that the higher the power extraction the higher the engine performance penalty becomes.

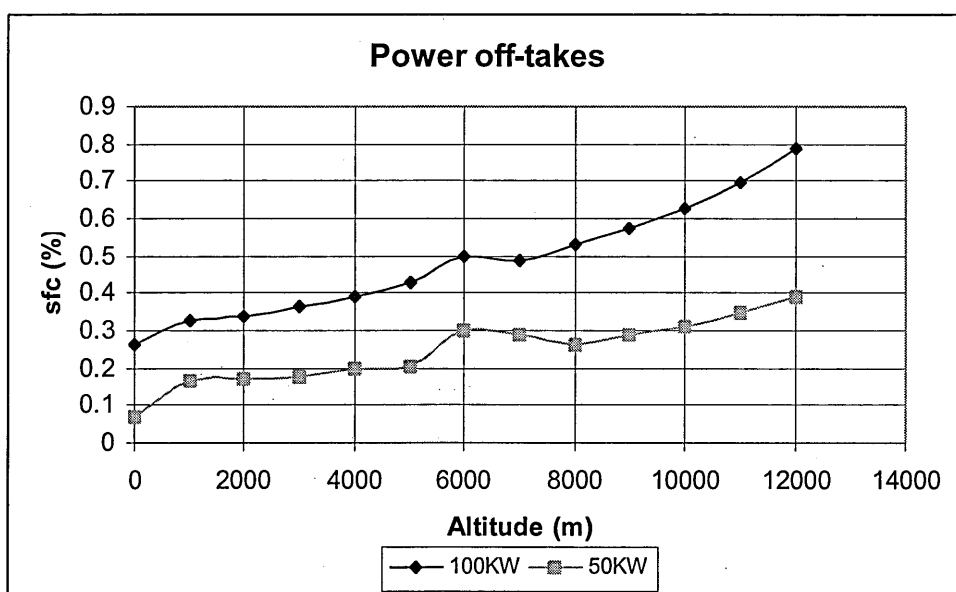


Figure 4.21 Power off-takes sfc penalty

Considering the engine operation at sea level conditions by extracting twice the power, the thrust penalty is increased by four times.

4.3 Engine modeling at distorted conditions

Earlier in chapter two results have been presented that correspond to the flow field within the RAE M2129 Intake duct at an axial flow case. These results presented the distorted flow field of the particular intake and the distorted pressure profile at the inlet to the compressor. Using these distorted profiles the compressor performance has been analysed and the compressor characteristics map has been obtained. This procedure has been presented in chapter three. Using this map of the axial distorted case, an engine performance analysis has been carried out and this section presents the results of this analysis.

4.3.1 Engine performance simulation

The modeling and simulation of this low bypass, two-spool turbofan engine for clean conditions have been presented in the previous sections of this chapter. Figure 4.22 presents the operating line of the engine on the original clean map.

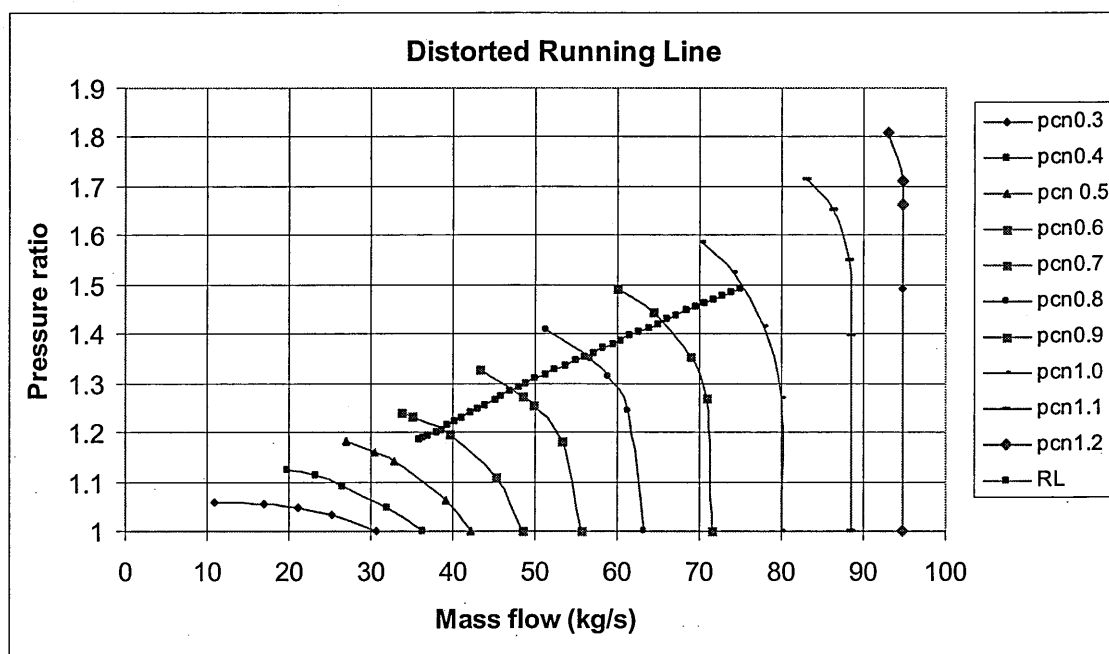


Figure 4.22 Distortion running line

As mentioned earlier distorted conditions for this case correspond to the simulation of the axial flow to the intake, giving a pressure profile to the compressor inlet where the DC60 increases with respect to the mass flow entering the engine.

4.3.1.1 Effect of shaft rotational speed

The effect of shaft rotational speed has been analyzed earlier for a variation of operational conditions. For the engine performance at distortion simulation the penalties both for thrust and specific fuel consumption can be seen in Figures 4.23 and 4.24.

Figure 4.23 shows the thrust penalties with a change in turbine entry temperature, hence shaft rotational speed. This clearly shows that by operating at higher power settings the engine has a higher thrust penalty. Taking into consideration also the results from the second chapter where the distortion increases with an increase in mass flow it can be concluded that the results follow a similar trend.

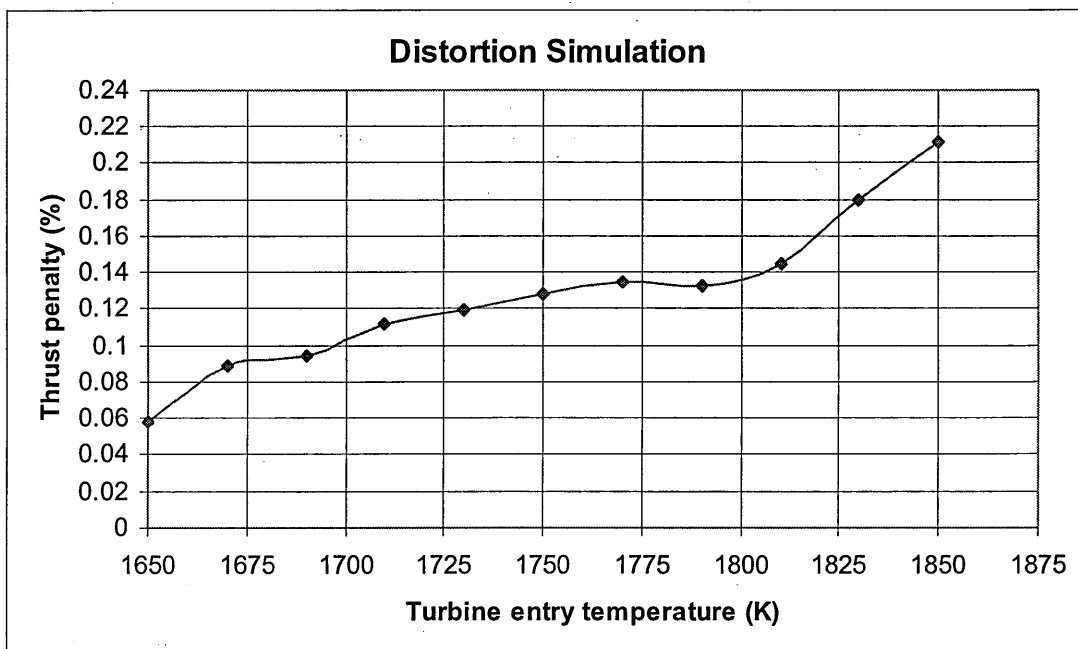


Figure 4.23 Effect of TET on engine thrust

Figure 4.24 shows the specific fuel consumption penalties that the engine suffers with an increase in power settings. The trend for the specific fuel consumption is similar to the engine thrust results which again concluded that the engine when operating under distorted conditions not only suffers performance penalties but these penalties increase with an increase in engine power settings.

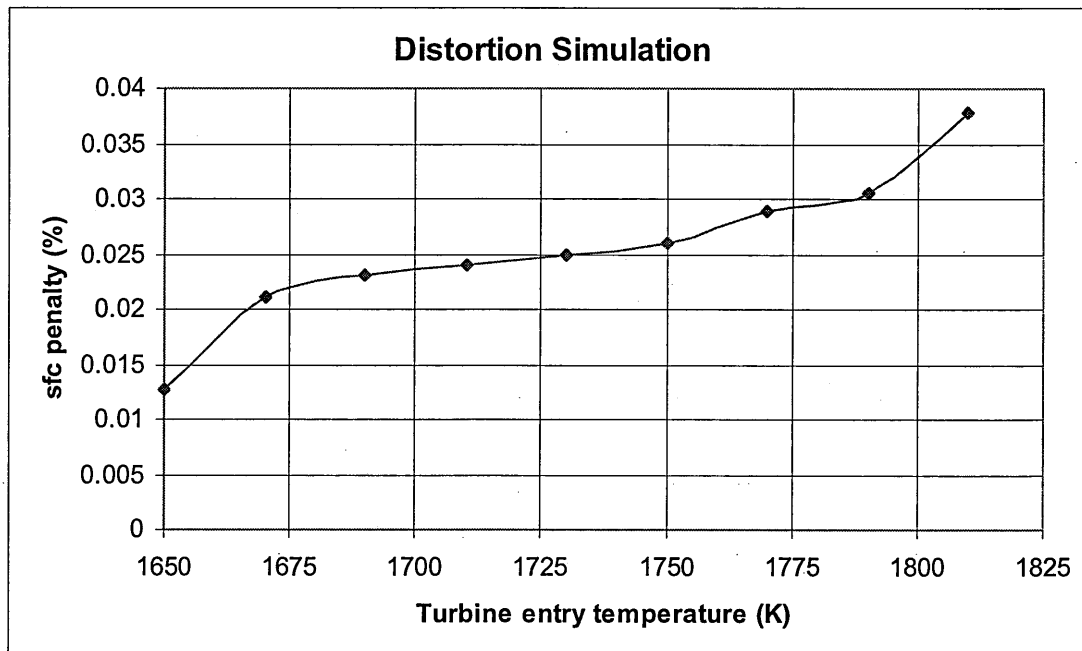


Figure 4.24 Effect of TET on engine specific fuel consumption

4.3.1.2 Effect of altitude

Further studies have been carried out with the engine operating at distortion conditions. A simulation of the engine at a range of altitudes has been carried out and the results can be seen in Figures 4.25 and 4.26 for thrust and specific fuel consumption.

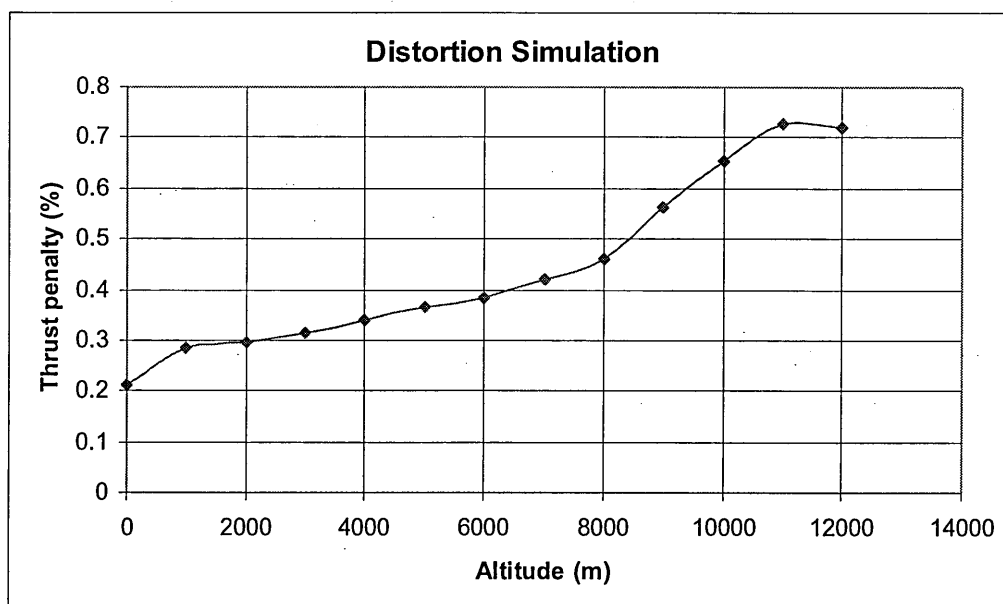


Figure 4.25 Effect of altitude on engine thrust

The simulation of the engine has been carried at sea level conditions and the altitude increased up to 12000 meters. Both thrust and specific fuel consumption show an increase of distortion effect with an increase in altitude.

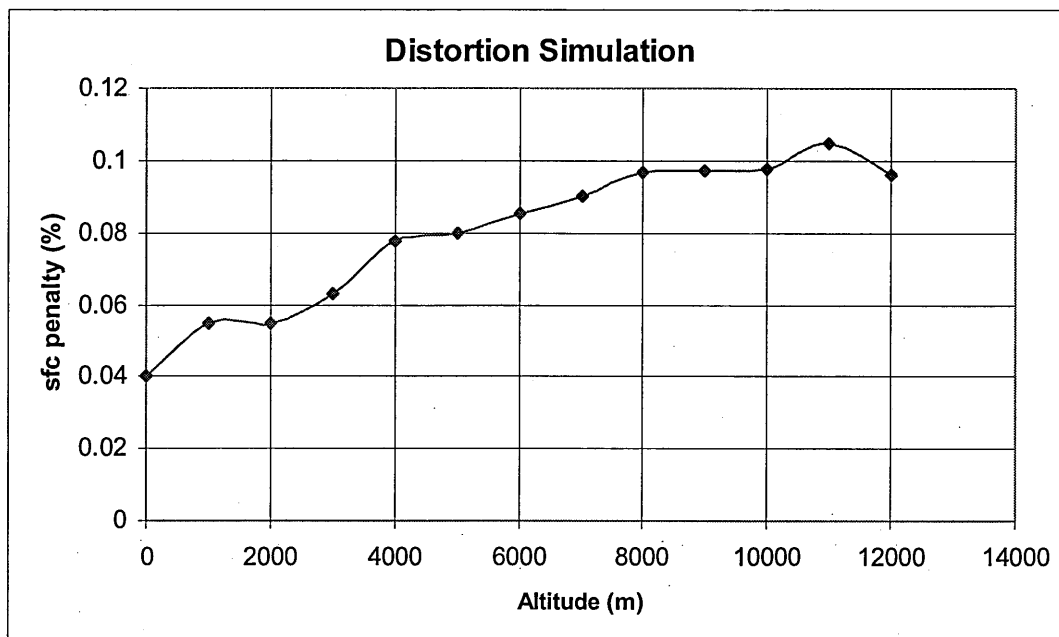


Figure 4.26 Effect of altitude on engine specific fuel consumption

4.4 Engine modeling at flow control conditions

The purpose of this study is to be able to analyze the effect of distortion to the overall performance of the engine and introduce flow control techniques in order to improve the engine performance and minimize the engine performance penalties.

In chapter two, simulation and analysis of the intake duct has been carried out and for the axial cases injection has been used in order to minimize the distortion at the compressor face. Based on the improved profiles the compressor model has been used to simulate the compressor characteristics map that correspond to the flow control simulations. Using the flow control map, this section of chapter four presents the modeling and simulation of the engine performance under improved operational conditions using flow control injection.

4.4.1 Engine performance simulation

As mentioned earlier the improved performance of the engine has been carried out using the results obtained from the introduction of flow control into the intake duct.

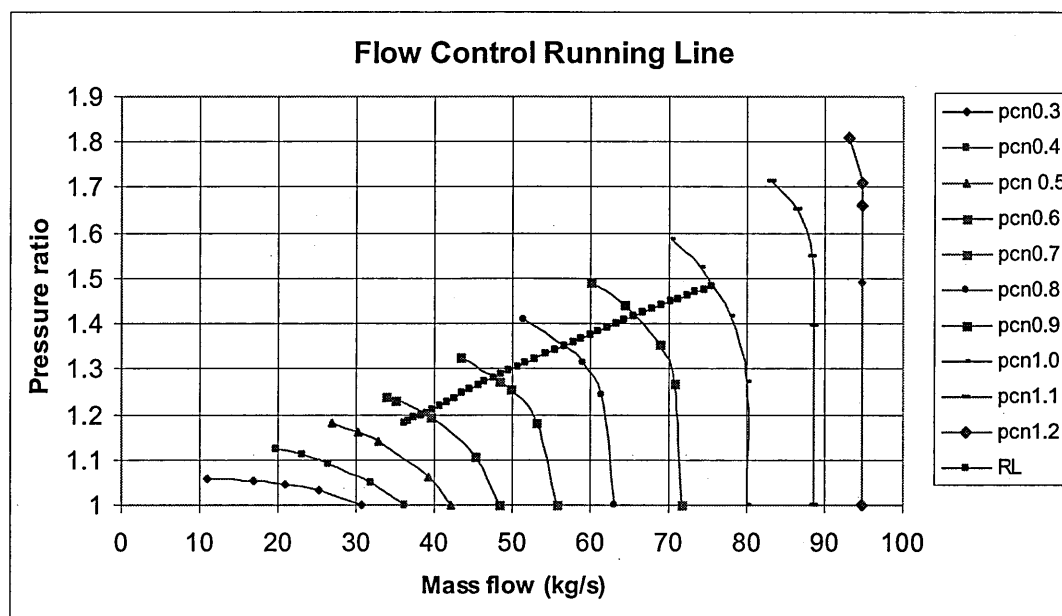


Figure 4.27 Flow control running line

The flow control method used for this purpose is injection where air is injected into the main flow of the duct to improve the flow field by avoiding or delaying boundary layer separation. This air has been provided from the end of the first stage of the low pressure compressor. Therefore, the engine simulation for the improved operational conditions, include the bleed of air from the engine in order to supply the system for the flow control operation. As mentioned earlier in this chapter and also in chapter two the amount of air used for the flow control corresponds to 1% of the overall engine mass flow. The pressure that the air has been injected into the intake duct is the same with the pressure of the air at the end of the first stage of the low pressure compressor, taking into consideration any pressure losses.

Figure 4.27 presents the operating line of the engine again on the original clean map with the introduction of the bleed air, which corresponds to 1% of the overall mass flow of the engine.

4.4.1.1 Effect of altitude

A similar study to the previous one has been carried out in order to analyze the effect of altitude to this low bypass engine performance operating at the flow control case conditions. Figure 4.28 presents the results with a variation on altitude with and without bleed air extraction.

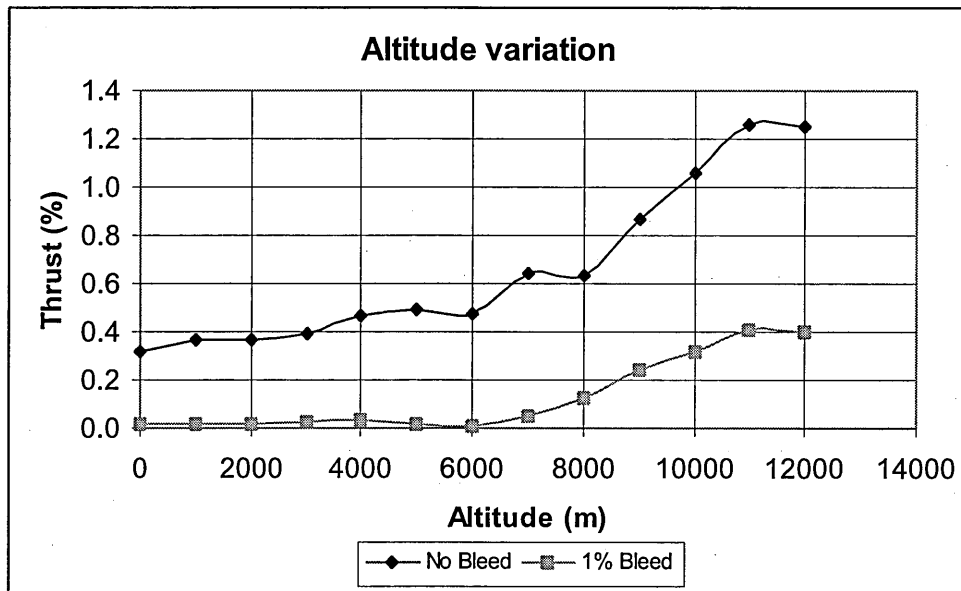


Figure 4.28 Effect of altitude on engine thrust

Figure 4.29 presents the results for two different flow control applications. One uses a 45 degrees angle for the secondary flow injection and the second one uses 90 degrees flow injection.

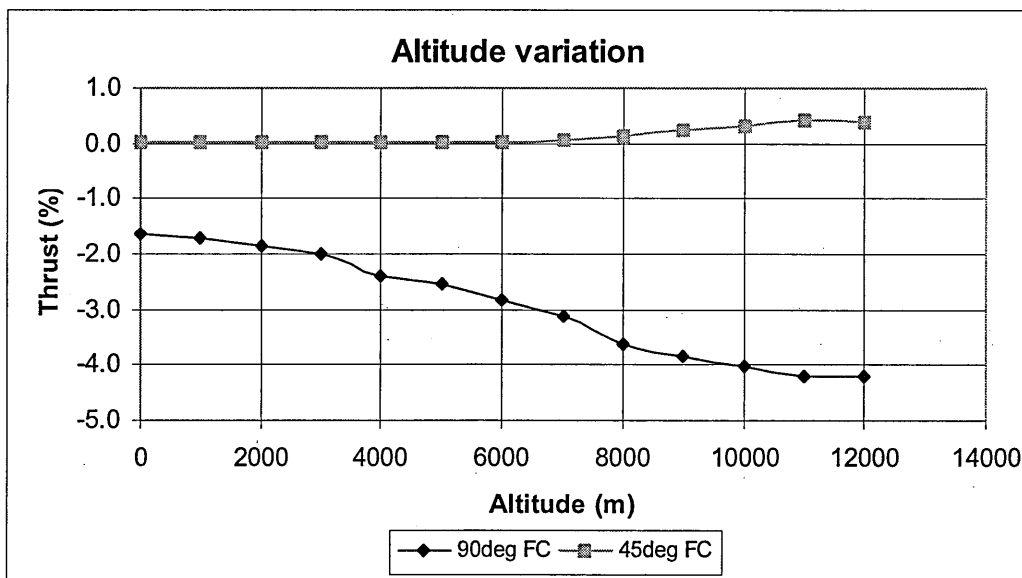


Figure 4.29 Effect of altitude on engine specific fuel consumption

The trends for all cases show that when the amount of air used for the secondary flow injection is bled from the engine the improvements on engine performance are minimized due to the effect of the air extraction. The results also show that depending on the optimization of the flow control application, the

method could easily degrade engine performance as the 90 degrees case show in Figure 4.29.

Therefore as a conclusion the application of flow control can improve engine performance but it is crucial to analyze the penalty that the engine will suffer in order to run the flow control mechanism and run a flow control optimization based on the pressure losses within the intake duct.

CHAPTER FIVE

Results and Discussion

5.1 Introduction

Chapter five is involved with the implementation of the tools and models mentioned in previous chapters and presents complete case studies based on the methodology followed throughout the project.

The results presented in this chapter have been obtained with the work contribution of Muleri [48] and Tentorio [49] and are separated into three main areas, which are the axial flow, pitch flow and yaw flow cases.

- The axial flow cases include the simulation of the intake flow normal to the boundary layer giving the distorted case and the use of secondary flow injection for the flow controlled case.
- The pitch angle cases include a 40 degrees flow entering the intake duct, which substantially alters the flow characteristics of the intake duct and increase flow distortion. For the purpose of controlling the flow, two cases are considered. One using secondary flow injection in order to enhance the flow mixing between the boundary layer and free stream flow, hence increasing the flow momentum and the second using suction in order to increase the velocity of the boundary layer flow just upstream of the flow separation region.
- The third case study area is concerned with the yaw angle flow characteristics. The case studies here cover the intake duct flow characteristics for the 10 and 30 degrees yaw angles and the use of secondary flow injection in order to reduce the engine face distortion for the 30 degrees yaw angle case. The reason that flow control is not considered for the 10 degrees yaw angle is because for that flow angle the characteristics of the intake duct flow improve in relation to the axial case. This can be explained by analysing the shape of the intake duct, which given the 10 degrees flow angle the air runs almost parallel to the bottom s-shaped wall of the duct. This can be seen in the 10 degrees yaw angle case study.

For all the cases mentioned above the complete methodology has been followed through for both intake and compressor flow simulation. Engine performance results are compared for all cases taking into consideration any

bleed-air or power extractions from the engine in order to operate the flow control systems.

5.2 Axial flow case studies

This section of chapter five presents the axial flow case studies. These studies include the analysis of the flow characteristics of the intake duct at normal operating conditions, the pressure distortion introduced to the compressor inlet face due to the shape of the duct and an analysis of the effect of this distortion to the first stage of the low pressure compressor and the overall engine performance. Once the analysis of the distortion is complete flow control method is implemented in order to improve the flow characteristics of the intake duct, hence improving both the compressor and engine performance.

The case studies in this chapter will be presented in a way similar to the structure of the thesis. The case study will be separated into three sections based on the three main areas, which are the intake flow analysis, the compressor characteristics analysis and the overall engine performance results. Comparison and discussion of the results presented here will be carried out for each individual section.

5.2.1 Intake flow simulation

Intake duct modelling and flow simulation has been discussed in detail in the second chapter of the thesis. The duct used for the case studies has been designed and modelled based on the engine face diameter used for this study. The engine model based on the mass flow requirements for a particular operating point has provided the boundary conditions. The flow simulation for the axial case has been carried out using different turbulence models and settings. Finally the results have been validated with experimental case studies based on the engine face distortion and total pressure recovery for that particular intake duct.

Figure 5.1 that follow presents the pressure distribution of the intake duct. It can be seen that flow separation occurs at the lower half of the diffusing part of the duct and increases until it reaches the recirculation area at the higher part of the s-bend. This highly recirculated flow area continues up to the engine face introducing a pressure distortion to the inlet to the compressor.

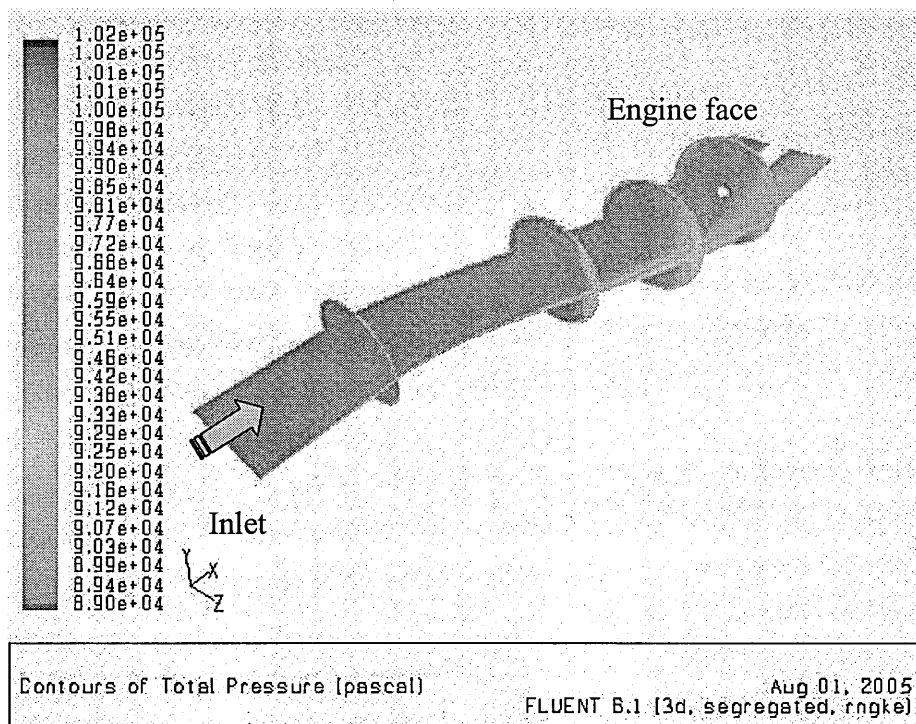


Figure 5.1 Intake duct total pressure distribution

The compressor face distortion that has been produced by the flow simulation of the intake duct had to be translated into the compressor model code in terms of total pressure distribution. This has been achieved by registering a total pressure value to each grid node that correspond to the compressor face. Figure 5.2 below shows an example of the intake outlet profile produced by the flow simulation and the profile transferred to the compressor model.

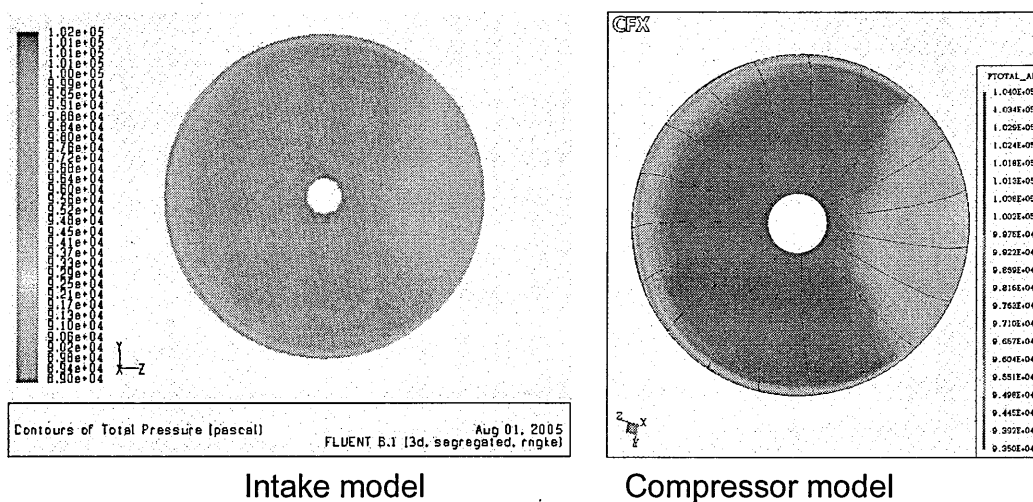
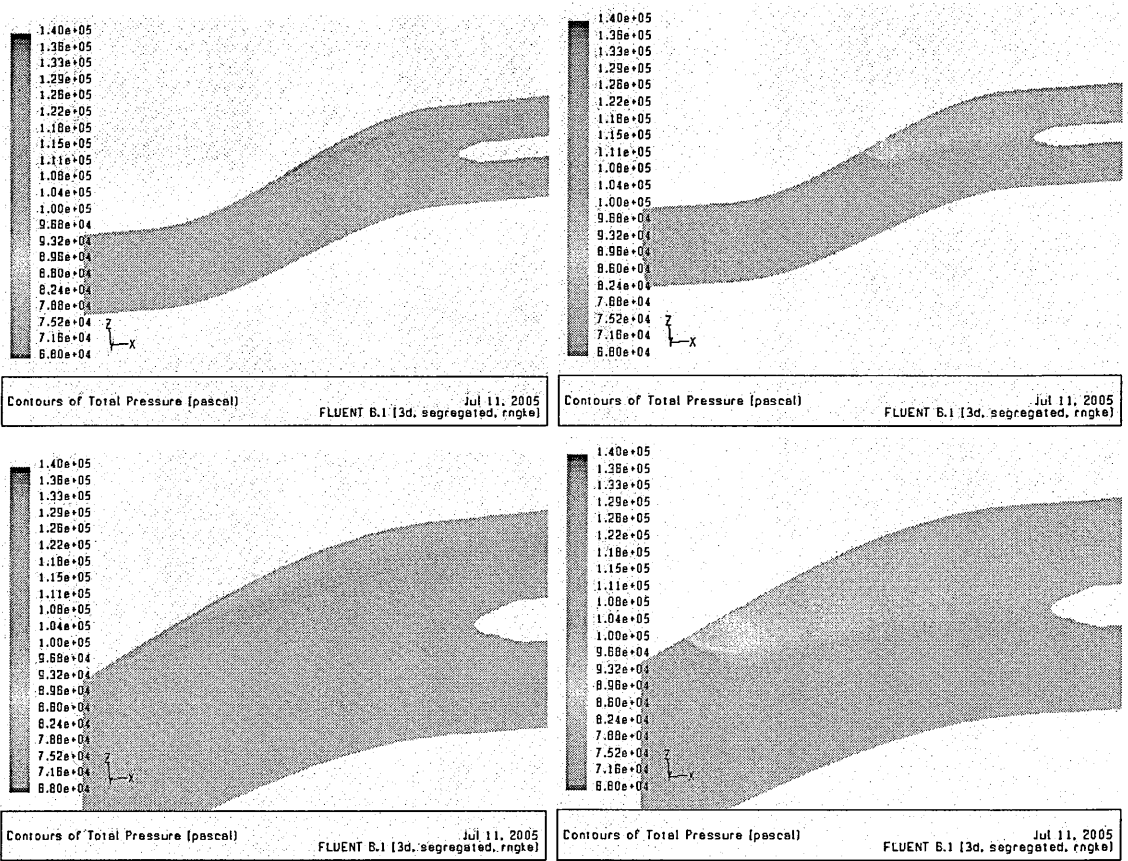


Figure 5.2 Total pressure distortion profiles

With the intake axial flow simulation completed and the distortion profile successfully translated to the compressor model, the next step was to introduce secondary flow injection in order to improve the flow characteristics of the duct. Previous studies mentioned in chapter two have shown that the most effective way of suppressing flow separation is by applying injection at a point upstream of the separation area. An important parameter for flow control using injection is the angle at which the flow is injected into the main stream. This can affect both the recirculation area and the overall pressure recovery of the duct.

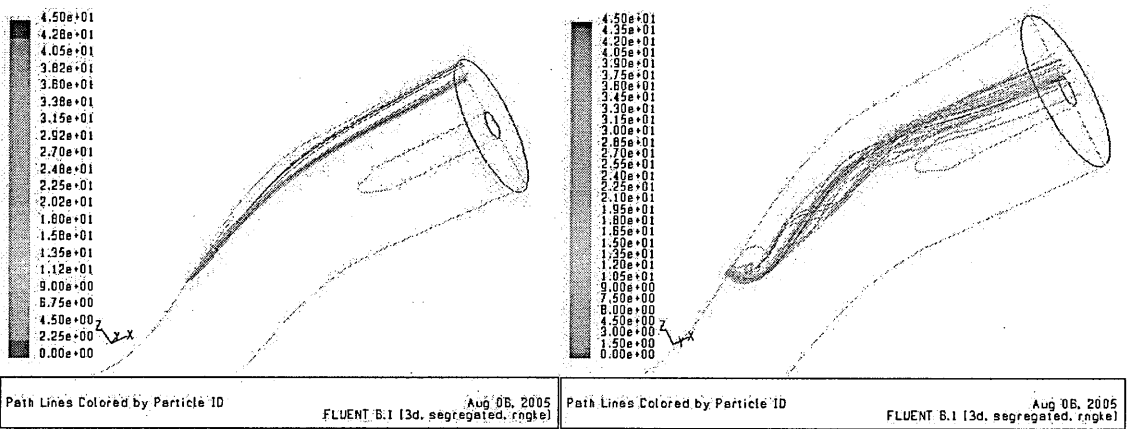
This study is concerned with two angles of injection and the effect that they have into the flow characteristics and overall performance. The angles considered are 45 degrees and 90 degrees to the boundary layer. Figure 5.3 gives an illustration of the secondary flow.



45 degrees injection 90 degrees injection
 Figure 5.3 Secondary flow injection angle

At a 90 degrees injection the secondary flow penetrates the main flow stream introducing further disturbance to the main flow. In the case of the 45 degrees injection the secondary flow penetrates the primary flow just enough in order to

produce a flow mixing between the main stream and the boundary layer. Figure 5.4 presents the flow particles pathlines for the secondary flow penetrating the main air stream.



45 degrees injection 90 degrees injection
Figure 5.4 Secondary flow pathlines

From the Figure above it can be seen that by introducing a 90 degrees injection to the main air stream, the mixing procedure for the two flows introduces further flow distortion. The 45 degrees injection introduces the flow mixing needed between the main flow and the boundary layer flow without introducing further distortion to the main flow.

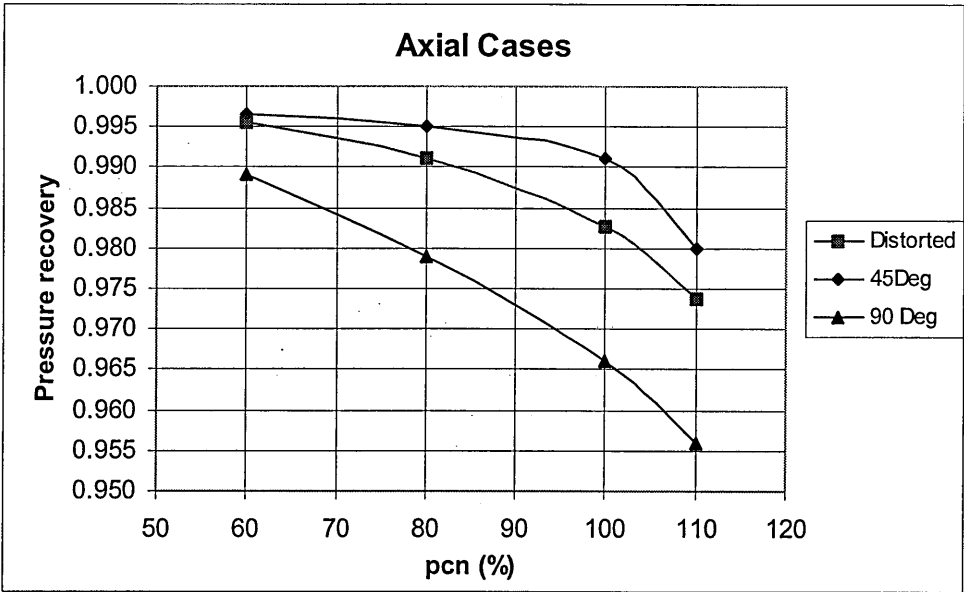


Figure 5.5 Axial cases pressure recovery

In the case of the two injection angles there is always an improvement to the engine face distortion with the 45 degrees injection giving the best results. However, when taking into consideration the pressure losses of the intake duct for the two case studies, the 45 degrees flow injection gives high improvements to the pressure losses, whereas the 90 degrees injection shows an increase in pressure losses compared with the distorted case. Figure 5.5 presents the pressure losses results for different pcn.

The change of injection angle seems to have a major effect to the engine face total pressure profile. Figure 5.6 below presents the total pressure profiles for the engine face using the two secondary flow injection angles. It clearly shows that there is an improvement in engine face distortion with the 45 degrees injection. In terms of DC60 parameter the improvement is as high as 50%. A similar improvement is shown in terms of duct pressure losses in Figure 5.5.

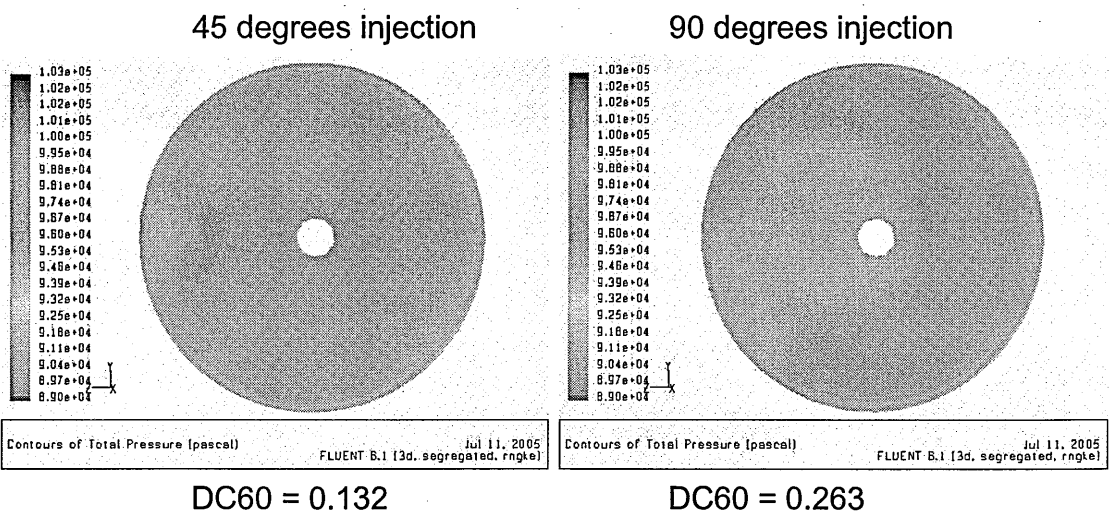


Figure 5.6 Engine face total pressure profiles at different injection angles

From the results presented earlier it can be seen that the engine total pressure distortion given from the intake flow simulation can be improved with the use of secondary flow injection just upstream of the separation region. The angle of this flow injection is an important parameter to the improvement of the flow characteristics achieved with flow control injection. Intake flow simulations have been carried out at different operating points and the results of these simulations can be seen in Figure 5.7 in terms of DC60 with a pcn.

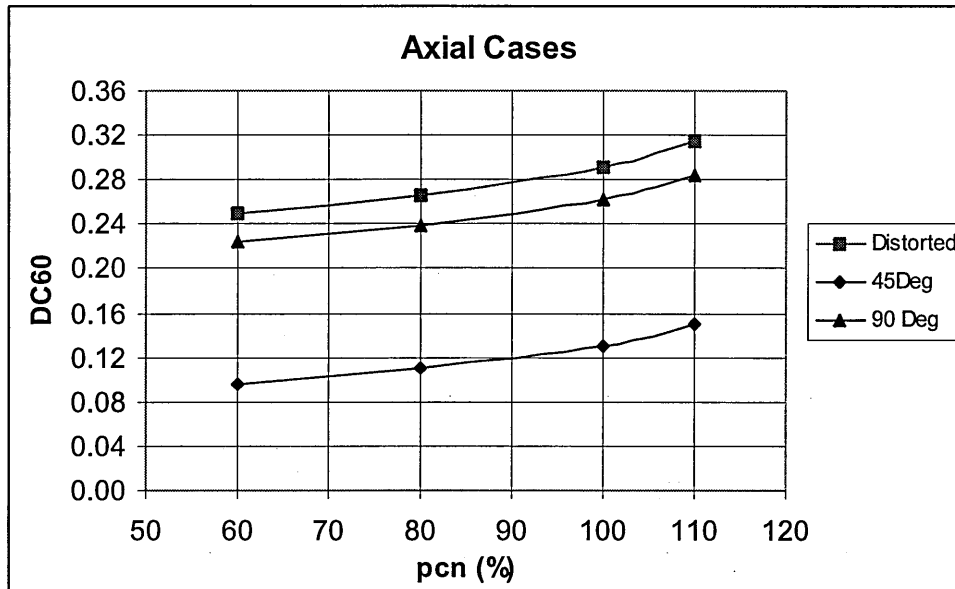


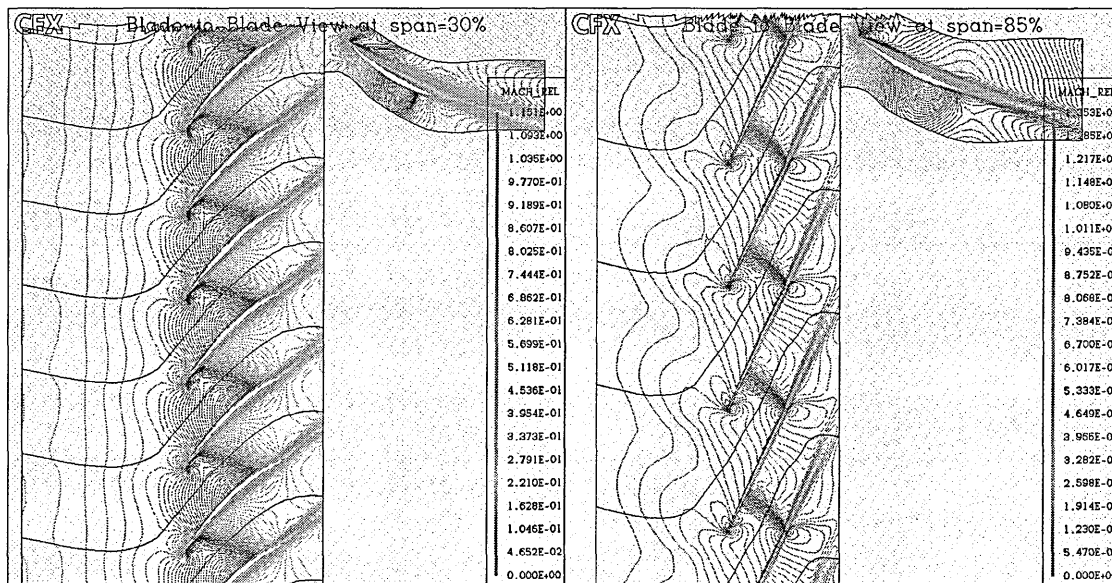
Figure 5.7 Axial cases engine face distortion

From Figure 5.7 it can be seen that considering the design point operating conditions of the engine at sea level the intake flow characteristics give a DC60 of 0.291. Introducing a secondary flow injection at a 90 degrees angle to the boundary layer, DC60 has a drop of 9.5%. When the secondary flow injection angle is changed to 45 degrees the improvement given is close to 55% from the distorted case. As far as the 90 degrees injection, although the engine face distortion tends to improve, the pressure losses of the duct increase giving a lower pressure recovery than the distorted case.

5.2.2 Compressor flow simulation

Having produced the engine face total pressure distortion profiles from the intake flow simulation the next step is to translate the data into the compressor model. This has been presented earlier. The results that follow the compressor flow simulation are presented in this section.

The flow visualisation of the compressor model shows similar characteristics to the clean operating conditions analysed in chapter three. There are no extreme effects through the compressor system with the introduction of the total pressure distortion. An interesting flow pattern that can be seen is when the flow simulation is carried out at low outlet static pressures where the system is close to choke conditions. At those conditions shock waves appear in the compressor passages on both rotor and stator. Figure 5.8 shows these flow characteristics. The shock waves tend to increase when moving closer to the tip of the blades.



30% Blade Span

85% Blade Span

Figure 5.8 Compressor passages shock waves formation

The compressor flow simulation introduced a procedure where the flow was simulated for four engine rotational speeds and each rotational speed had a case of five outlet static pressures. These simulations produced the compressor characteristics maps for each case study.

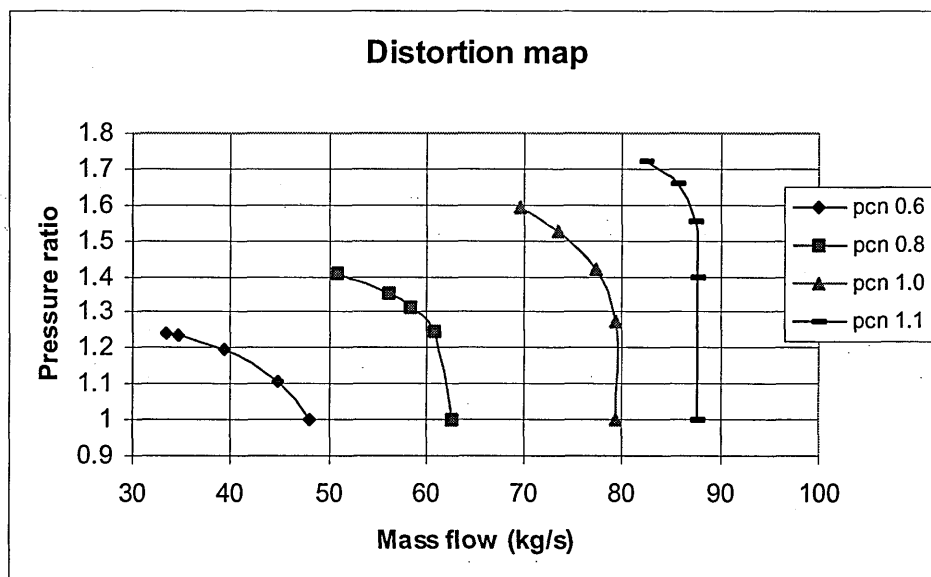


Figure 5.9 Compressor pressure ratio map

For the axial case, the engine face profiles obtained from the intake flow simulations have been used to produce the inlet boundary conditions for the

compressor model. The rotational speed was set for each speed line simulation and altering the static pressure controlled the outlet boundary conditions.

The compressor characteristics maps for the distorted case can be seen in Figures 5.9 and 5.10 for the pressure ratio and efficiency respectively. The maps present the compressor performance for four different compressor rotational speeds.

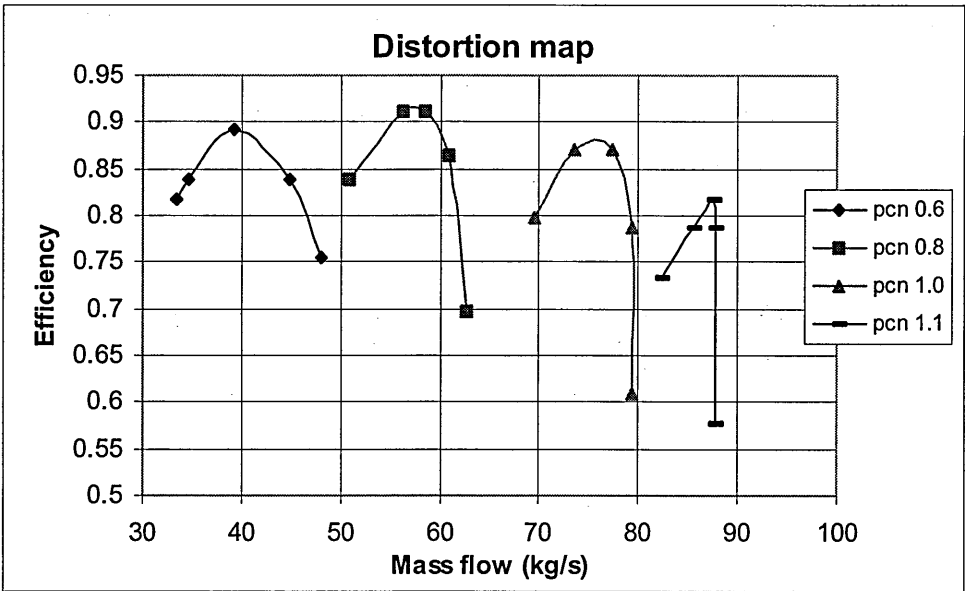


Figure 5.10 Compressor efficiency map

A comparison between the distorted case and the two flow control cases can be seen in Figure 5.11.

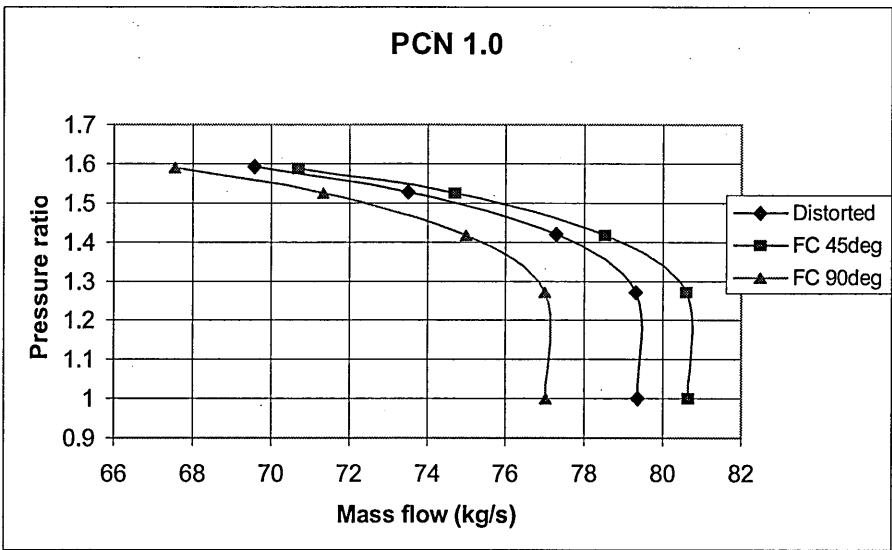


Figure 5.11 Axial cases pcn 1.0 pressure ratio results

The Figure presents the results in terms of pressure ratio and mass flow for the speed line pcn 1.0. It can be seen that the introduction of secondary flow injection can produce a movement to the constant speed line in either way depending on the way that the secondary flow is injected into the main air stream. The 45 degrees case gives an improvement in compressor performance characteristics by moving the speedline further to the right. In the case of the 90 degrees injection degradation seems to appear on the compressor performance. This seems to have been affected by the intake pressure losses introduced due to the aggressive secondary flow injection.

5.2.3 Engine performance

The axial case studies continue with the simulation and analysis of the overall engine performance. Having completed the flow simulations for both the intake and compressor models, flow characteristics have been obtained and analysed in terms of engine face distortion and pressure losses for the intake model and performance characteristics maps for the compressor model. This section analyses the performance of the engine model when the compressor characteristics maps are introduced for all the axial case studies.

Performance results for the engine model running at clean conditions and an analysis of air and power extraction effects have been presented in chapter four. For the axial case studies that follow all these operating settings will be used in order to achieve a complete picture of the interaction between the engine and the intake models.

This section starts with a comparison between the intake results in terms of pressure losses and engine face distortion and the engine thrust. All results are presented in percentage change in Table 5.1.

	Distorted	45 Deg FC	90 Deg FC
Intake PR	0.9825	0.865%	-1.679%
Engine Face DC60	0.291	55.3%	9.8%
Engine Thrust	59921 (N)	0.32%	-1.255%

Table 5.1 Axial case studies comparison

From the Table above it can be seen that although both flow control cases show an improvement in engine face distortion, the dominant effect for engine performance in terms of thrust is the intake pressure recovery. The thrust results follow a similar trend to the pressure recovery results. Either way this is a representation of the dominant effect parameter. The improvement shown on thrust for the 45 degrees injection is obtained without considering the bleed air needed for the flow control. The results in Table 5.1 do not take into consideration the effect of bleed air used for the flow control injection.

Analysis and comparison of the axial cases results have been carried out for different engine operational conditions. The Figures that follow present the engine performance at these conditions where the engine operates with air or power extracts whenever requested by the flow control system.

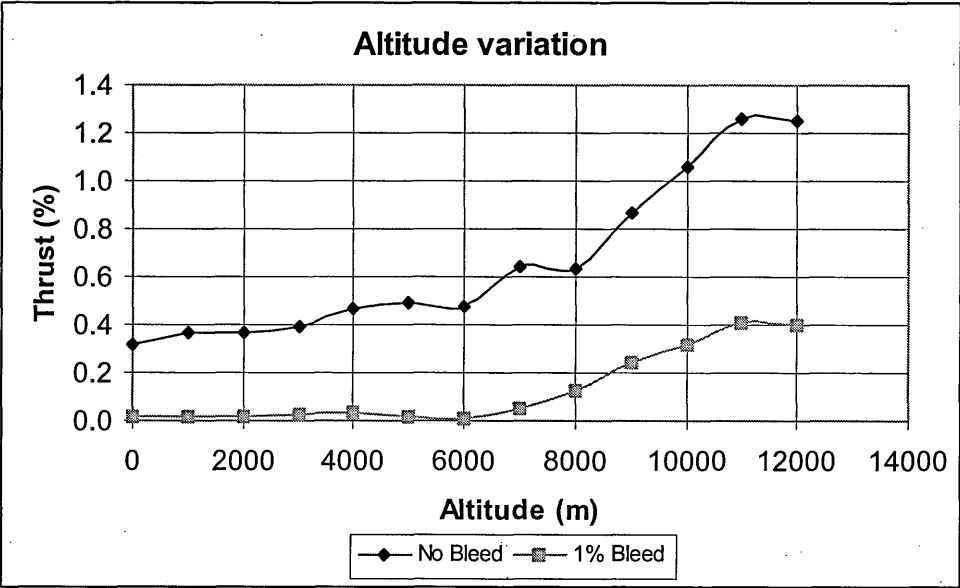


Figure 5.12 Percentage thrust with altitude variation

Figure 5.12 presents the thrust percentage change based on the distorted results for the 45 degrees injection cases. It can be seen that by introducing secondary flow injection to the intake, the improvements of the main flow, effect in a positive manner the performance of the engine. It is shown that even with air extraction from the engine the performance show an improvement.

The introduction of flow control gives an improvement to the engine performance in terms of thrust close to 0.35% with respect to the distortion case at sea level conditions. This result does not include the performance penalty

due to bleed air used for the flow control. With the bleed air taken into consideration the thrust improvement drops to just 0.015%.

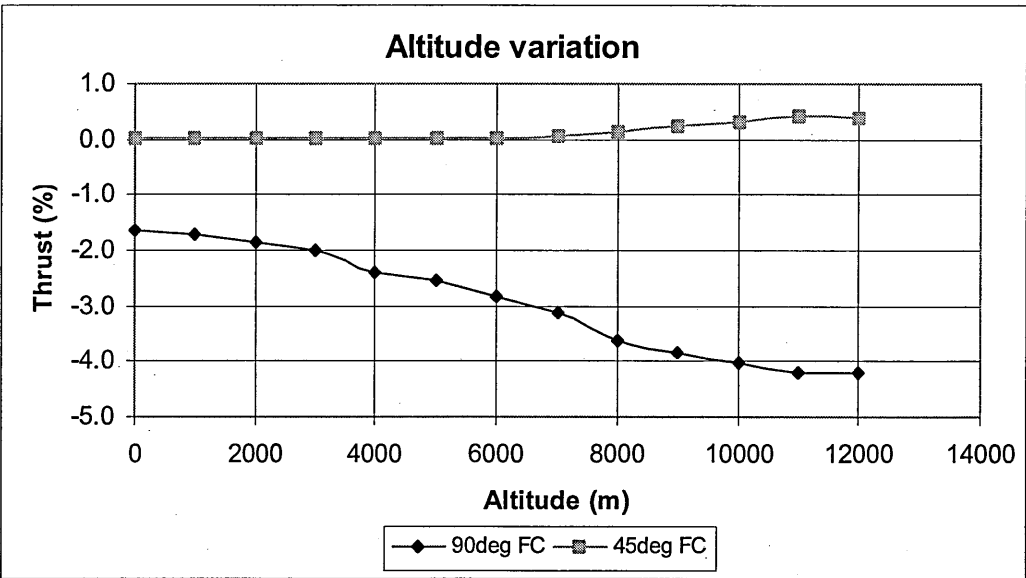


Figure 5.13 Percentage thrust with altitude variation

Figure 5.13 shows the comparison between 90 degrees and 45 degrees angle of injection and the effect that this has to the engine performance. Thrust is represented in percentage based on the distortion results. The air needed from the engine for the secondary injection flow is also been considered.

As already shown in Table 5.1, the introduction of 90 degrees injection into the intake main flow introduces high pressure losses. These pressure losses have a dominant effect to the engine performance, dropping the thrust of the engine as much as 1.5% at sea level conditions. The 45 degrees injection shows a very small improvement after the air is extracted from the engine.

5.3 Yaw flow case studies

This second section of the chapter presents the case studies for the intake inflow at a variation of yaw angles. The flow is introduced into the intake duct at yaw angles of 10 and 30 degrees. An analysis is carried out for the three components involved being the intake, compressor and overall engine performance. The effect of this type of flow within the intake duct is investigated in order to examine the characteristics of the flow and performance effects.

This section will cover case studies that have been carried out for 10 and 30 degrees inflow and the use of flow control to improve any flow discrepancies that these conditions introduce. A flow control optimisation has also been studied for the 30 degrees yaw angle. This case is where the highest distortion is introduced into the flow. This section of yaw cases has been split in two parts that examine the duct inflow at 10 and 30 degrees respectively.

5.3.1 10 degrees yaw angle

5.3.1.1 Intake flow simulation

In the case of 10 degrees yaw inflow the flow shows an improvement as mentioned earlier in chapter two. The reason for this is the shape of the duct. When altering the inflow angle the air tends to flow parallel to the first bend of the s-shaped duct, hence improving the characteristics of the flowfield. Figure 5.14 presents the results of the intake flow simulation in terms of total pressure contours.

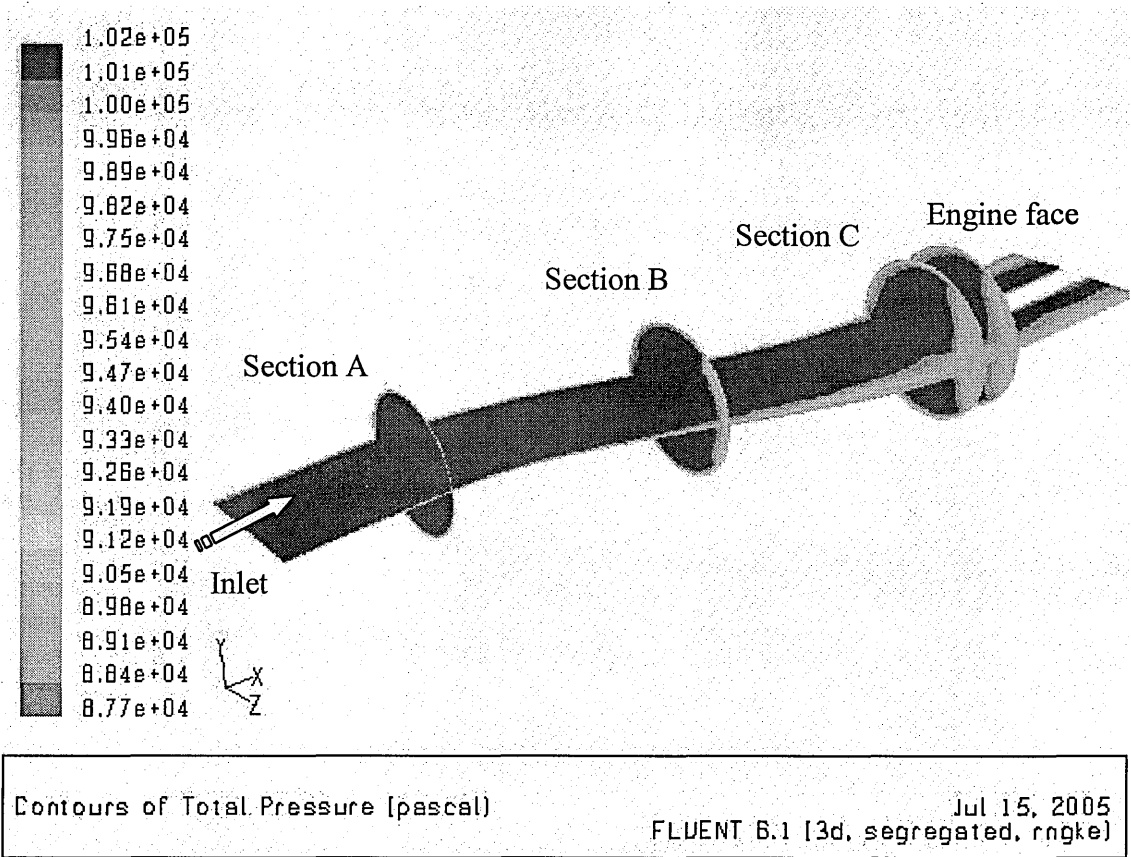


Figure 5.14 Yaw angle inflow

Figure 5.14 shows the plane on which the inflow angle is altered. The distortion that is shown on the second bend of the duct is lower in magnitude than the

axial case. This is because the flow coming into the duct avoids the first bend, moving parallel to the intake wall and minimising any flow deflections.

Figure 5.15 is another way of visualising the flow within the duct at a 10 degrees yaw angle. The Figure shows airflow results in terms of velocity vectors. A 10 degrees flow is introduced and the air alignment with the intake curvature is clearly shown.

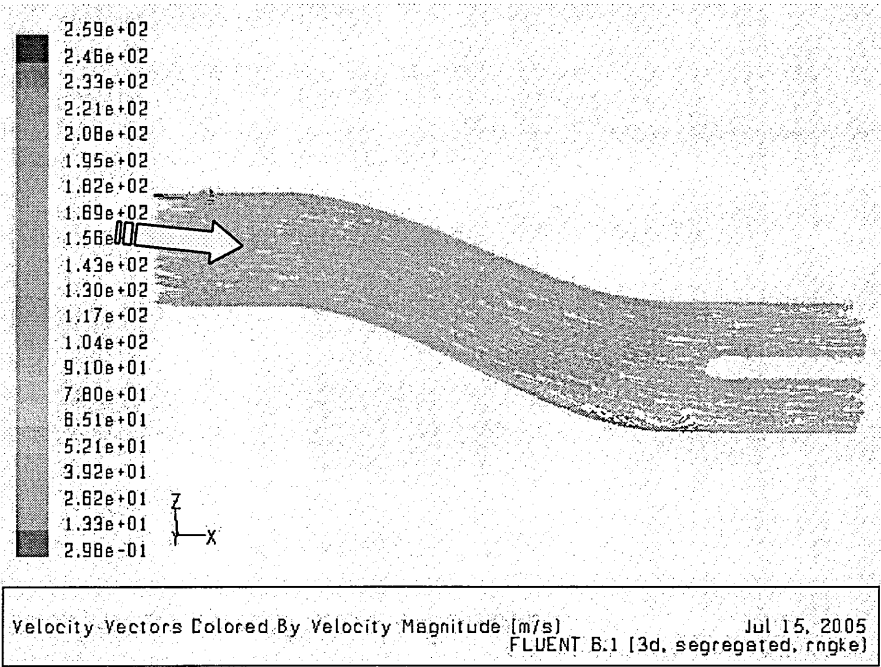
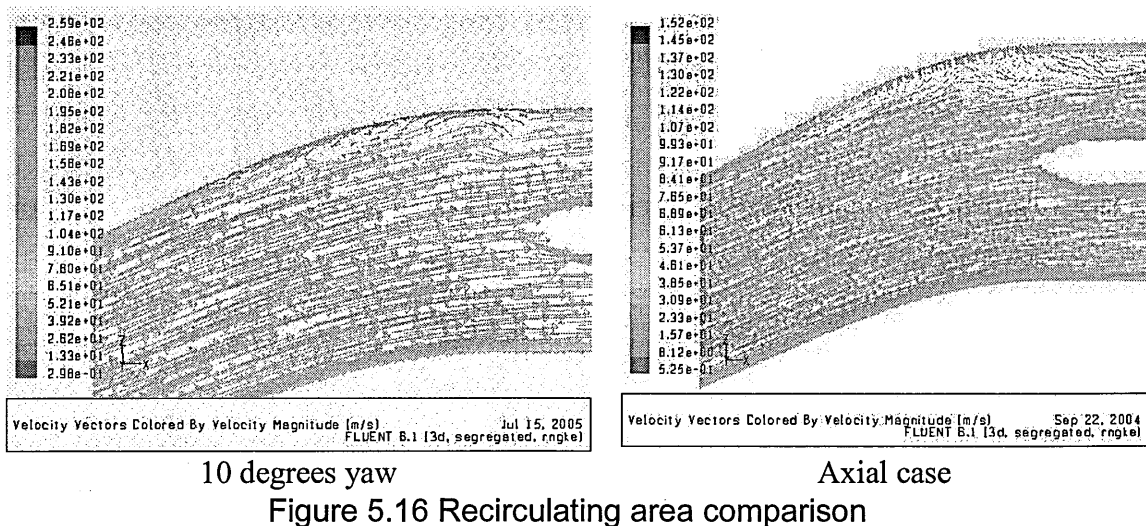


Figure 5.15 Velocity vectors flow visualisation

As mentioned above the 10 degrees inflow shows an improvement to the flowfield due to the nature of the intake curvatures. The area of major importance for this duct is the second bend where separated flow creates a flowfield of recirculation, hence introducing distortion to the engine face. Figure 5.16 presents a comparison of the axial case and the 10 degrees yaw angle case. The flow recirculation area of the axial case is greater than the 10 degrees yaw angle case. This in turn effects the engine face distortion, introducing a more uniform flow to the compressor stage.



An interesting flow visualisation is presented in Figure 5.17 where the flowfield is analysed in various sections throughout the duct. The position of these section with respect to the duct can be seen in Figure 5.14.

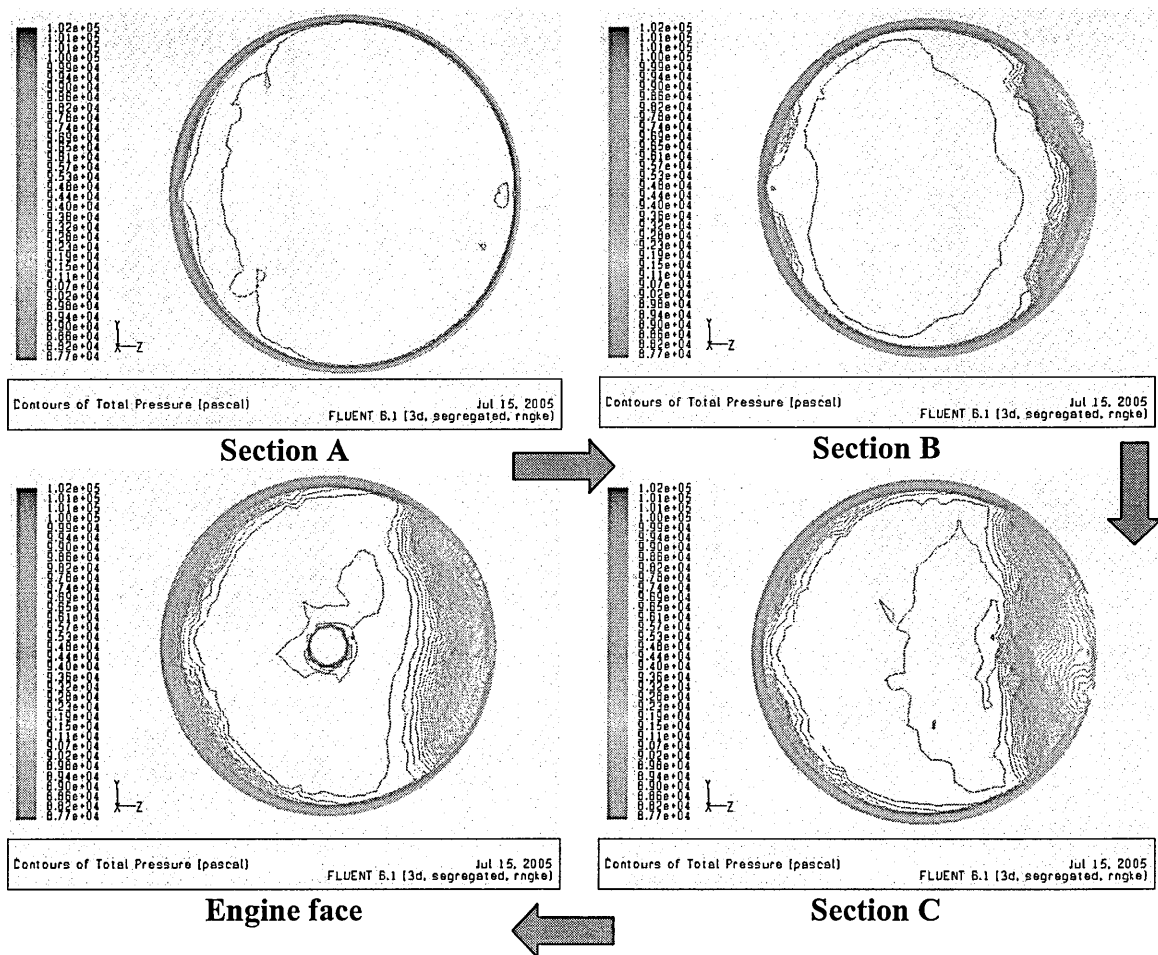


Figure 5.17 Total pressure contours along the intake

The flow visualisation in Figure 5.17 shows a small pressure increase at the lower wall of the first bend in section A. Moving to the middle of the diffusing part of the duct at section B the pressure distortion is translated to the upper wall of the duct where separation occurs. This flow separation introduces a recirculation area, which reaches its maximum at section C just upstream of the engine face. The final section is the total pressure profile that compressor stage is introduced with. This is the section that will be analysed when flow control is introduced into the system.

Having analysed the flow characteristics of the 10 degrees yaw angle case the next step is introducing flow control in order to improve the flow and introduce a more uniform profile at the engine face.

For this case study the flow control method introduced is a secondary flow injection method at the first intake duct. The air is injected through eight holes placed in equal distance around the intake duct. This type of injection is set in order to attack the flow separation problem at an early stage, before further progression occurs.

Figure 5.18 shows the effect of the secondary flow injection into the primary flowfield. The flow visualisation is presented in terms of total pressure contours showing the effect of the injected flow close to the wall at the mid-plane of the duct.

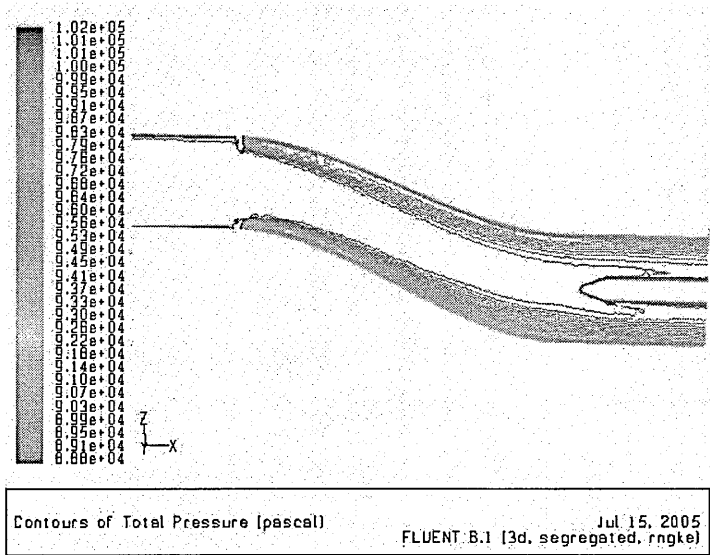


Figure 5.18 Injection flow visualisation

By introducing the secondary flow injection into the main flow, a mixing is introduced between the main and secondary flow, re-energizing the recirculating area and improving the flowfield.

This improvement of the flowfield in return gives a better engine face total pressure profile. Figure 5.19 show a comparison between the engine face pressure profiles of the 10 degrees yaw angle with and without flow control.

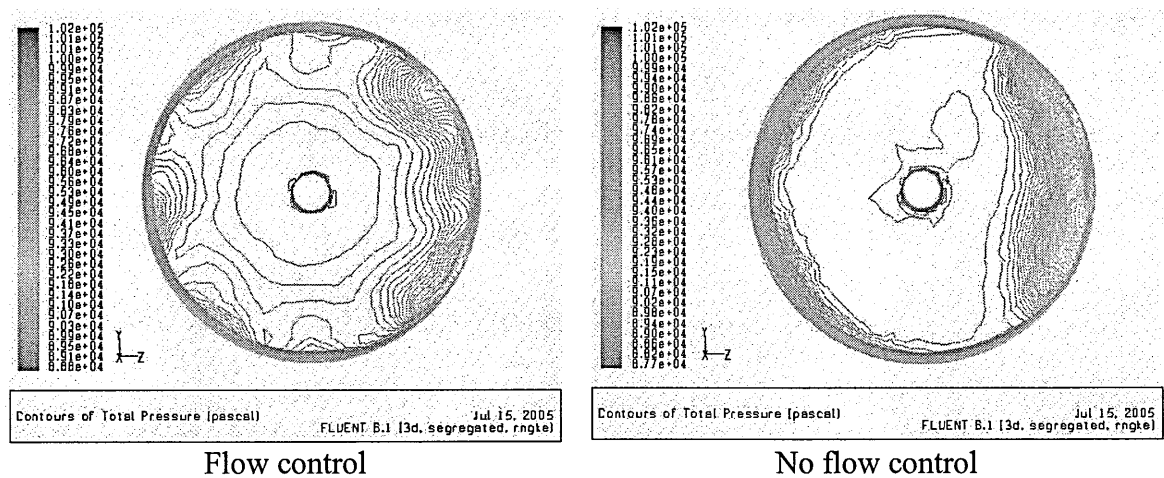


Figure 5.19 Total pressure profiles at engine face

The Figure shows that although the pressure profile is not totally uniform, with the introduction of flow control the low pressure sections of the profile have been eliminated.

A similar improvement of the flow characteristics when flow control is introduced can be seen in the intake high recirculation area. Figure 5.20 shows the flow visualisation of this area in terms of velocity vectors for both cases.

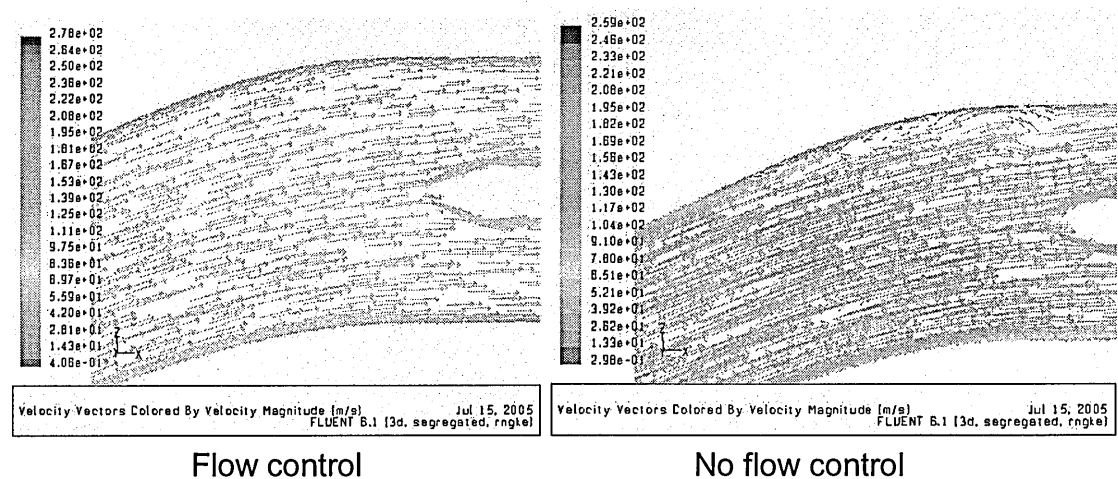


Figure 5.20 Recirculating area comparison

The final analysis of the intake duct performance has been carried out at a variation of operating conditions in order to examine the flow characteristics in terms of engine face distortion and intake pressure recovery.

Figure 5.21 that follow presents the engine face distortion for different engine operating conditions.

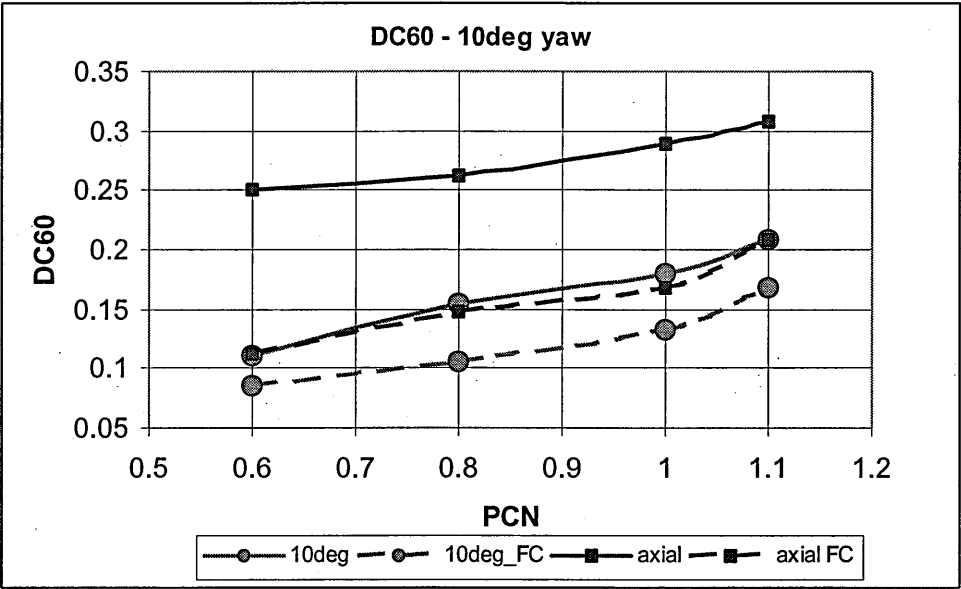


Figure 5.21 DC60 for 10 degrees yaw

The above Figure presents a comparison between the axial and 10 degrees yaw cases with and without flow control. The flow improvement of the 10 degrees cases in comparison to the axial cases can clearly be seen with the engine face distortion improving up to 40%. The introduction of flow control for the 10 degrees cases further improves the engine face distortion up to 27%.

Figure 5.22 presents the pressure losses within the intake duct at the same operating conditions as earlier. Again the pressure losses for the axial cases are higher than the pressure losses given by the 10 degrees yaw cases. This once more points out the improvements introduced by the 10 degrees inflow. The improvement of the pressure losses for the yaw cases in comparison to the axial cases is about 0.15%. With the introduction of flow control for the yaw cases the intake pressure recovery improves up to 0.2%.

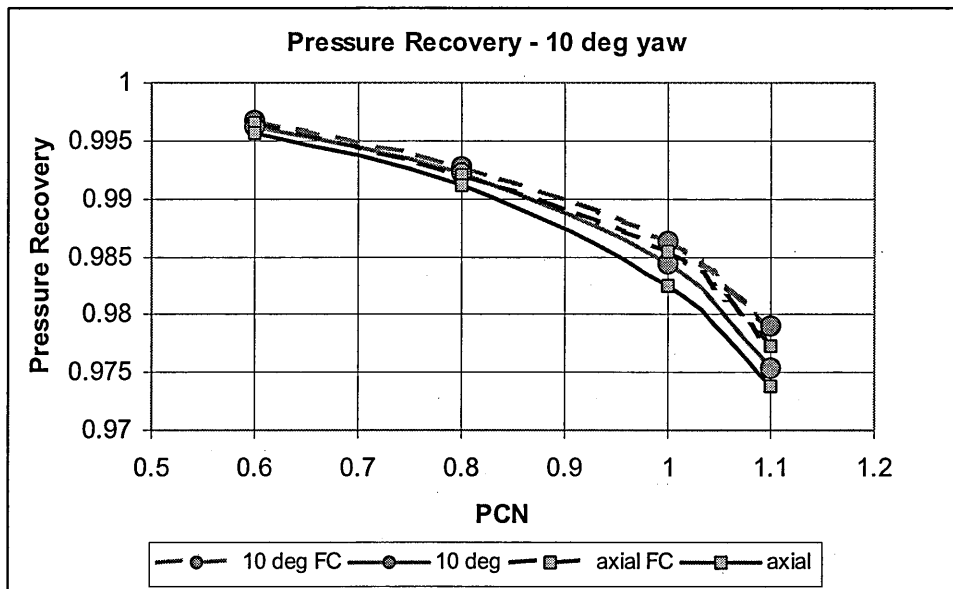


Figure 5.22 Pressure recovery for 10 degrees yaw

In general, all cases for the 10 degrees yaw inflow show an improvement to the intake flowfield characteristics due to the nature of the duct geometry. Further improvement can be achieved with the introduction of secondary flow injection, which helps to minimize the separation flow and improve the recirculation area within the duct.

5.3.1.2 Compressor flow simulation

Once the flow simulation of the intake is completed the next step is to transfer the intake outlet profile data to the compressor stage model and simulate the flow within the first stage.

The results of the compressor stage flow simulation can be seen in Figure 5.23 where a comparison is shown between the cases with and without flow control. Figure 5.23 shows the flow changes at three positions through the rotor. Each picture has its own local scale for the total pressure contours in order to highlight better the pressure distribution through the rotor stage. It can be seen that the pressure distortion moves through the stage until the flow reaches the exit of the rotor. The comparison between the cases with and without flow control show an improvement to the flow characteristics of the rotor minimising high pressure spots at the exit of the rotor.

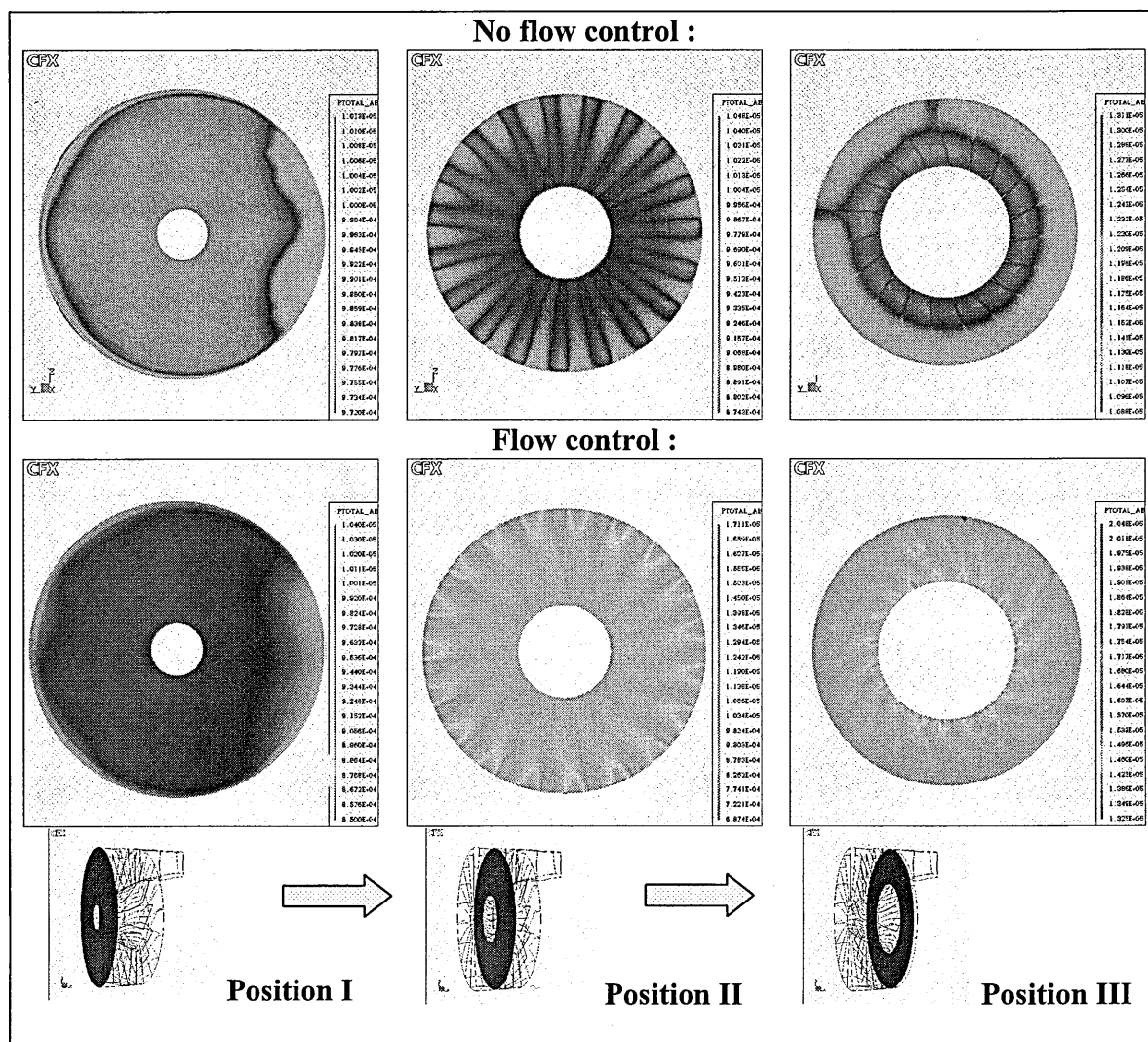


Figure 5.23 Compressor flow simulation results

Based on the profiles obtained from the intake results, the compressor flow simulations have been carried out for a variation of operating conditions in order to create the compressor map characteristics used for the engine performance model. Again the maps have been obtained by setting the engine operation at a variation of rotational speeds. Then for each constant speed the outlet static pressure is modified in order to obtain the whole speed line.

Based on the procedure mentioned above the flow characteristics maps have been created in terms of mass flow, pressure ratio and efficiency for a variation of engine rotational speeds.

The compressor stage performance in terms of pressure ratio and mass flow is presented in Figure 5.24 below. This Figure shows the map characteristics for the 10 degrees yaw cases both with and without flow control.

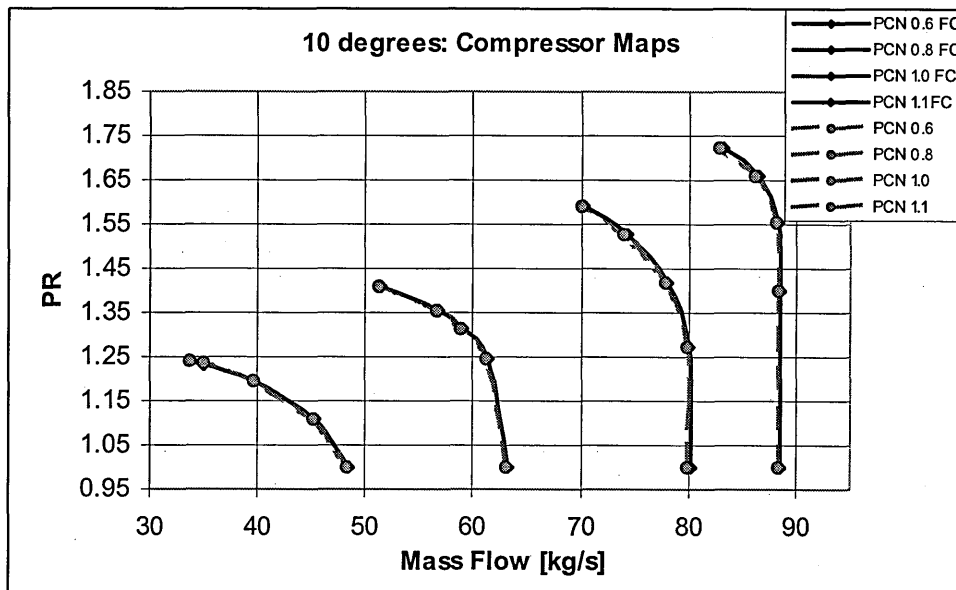


Figure 5.24 Compressor pressure ratio 10 degrees yaw

From the Figure above it can be seen that although the differences between the cases with and without flow control are very small, there is a movement of the speedlines to the upper right of the graph showing a compressor performance improvement when flow control is implemented.

The same improvement can be seen in terms of compressor efficiency in Figure 5.25. By implementing flow control to the system the compressor efficiency seems to improve due to the improved inlet conditions to the compressor stage.

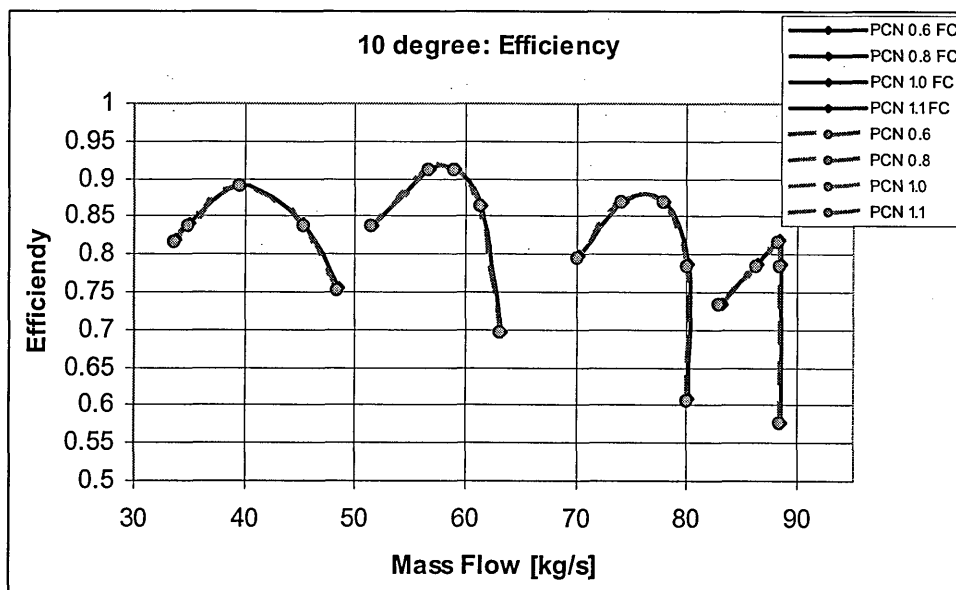


Figure 5.25 Compressor efficiency 10 degrees yaw

5.3.1.3 Engine performance

Having produced the compressor characteristics maps that were presented above, the next step was to implement the maps into the engine performance simulation tool in order to model the engine performance under the influence of yaw flow conditions both with and without flow control.

Table 5.2 presents the thrust in percentage difference between the 10 degrees yaw cases with and without flow control compared to the intake duct flow simulation results.

	10 degrees	Flow Control
Intake PR	0.984	0.2%
Engine Face DC60	0.179	27%
Engine Thrust	59952 (N)	0.17%

Table 5.2 10 degrees yaw engine performance

The Figure that follows show engine performance in terms of thrust with a variation in turbine entry temperature. The comparison is between the 10 degrees yaw cases with and without flow control. The differences between the two are quit small with the flow control giving an engine performance improvement of no more than 0.17%. this is for the same operation condition that give a DC60 improvement of almost 27%.

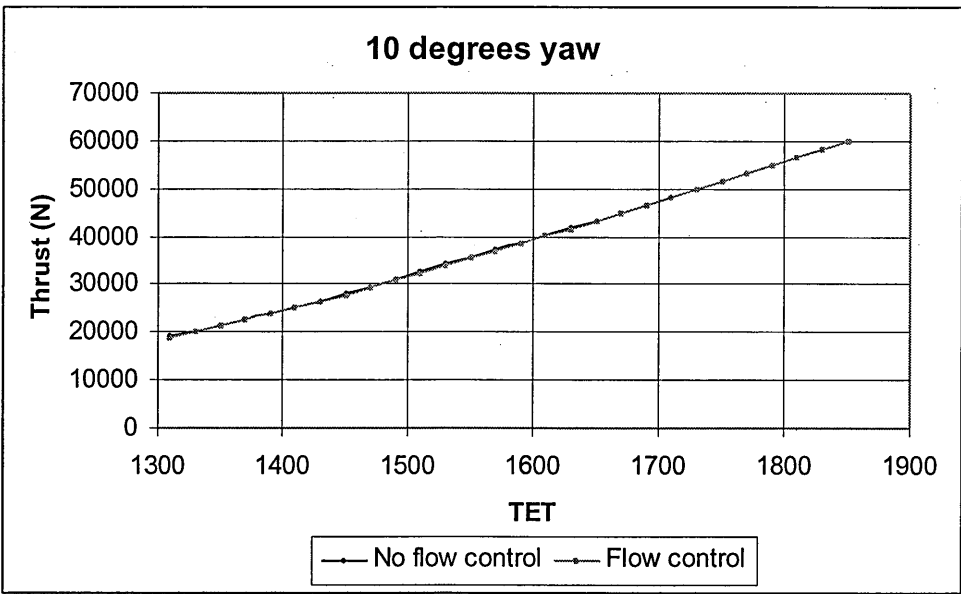


Figure 5.26 Engine performance at 10 degrees yaw

A more clear difference between the engine performance of the 10 degrees yaw angle cases with and without flow control can be seen in Figure 5.27. This Figure presents the specific fuel consumption for different thrust outputs for both the distorted and flow control case. It can be seen that flow control improves the fuel consumption of the engine at a particular thrust setting.

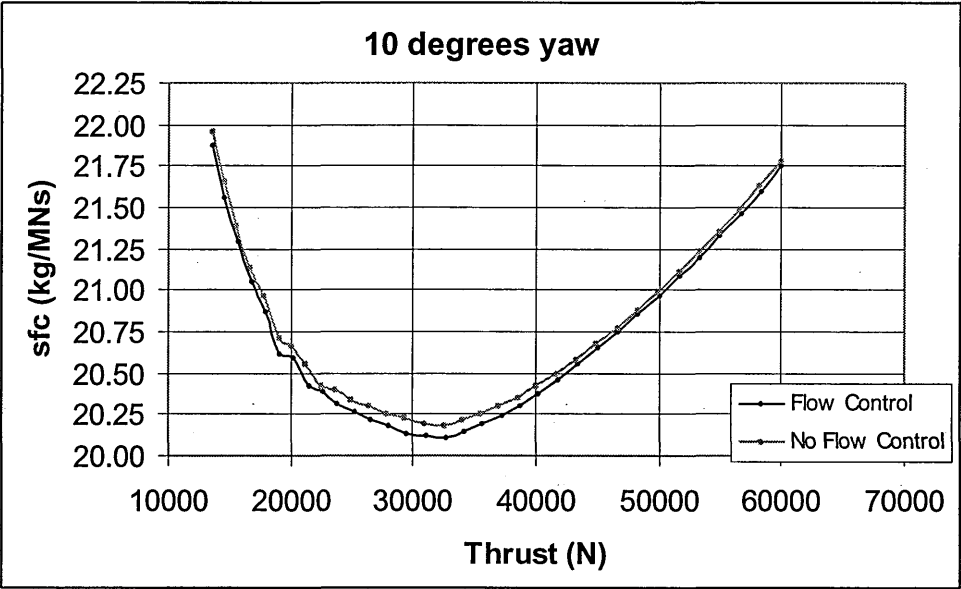


Figure 5.27 Thrust vs sfc for 10 degrees yaw

In conclusion, it has been shown that the implementation of flow control for the 10 degrees yaw cases improves both the flow characteristics of the intake and compressor and the engine performance. It seems though that the magnitude of improvement is not similar. At a 30% improvement in engine face distortion the engine thrust shows only a 0.2% improvement.

5.3.2 30 degrees yaw angle

5.3.2.1 Intake flow simulation

In the previous section of this chapter air flow has been introduced into the intake duct at a 10 degrees yaw angle. This type of flow generated a flow pattern that improved the flowfield characteristics in comparison to the axial case. The flow distortion dropped and that in turn gave an improvement to the engine face distortion, compressor characteristics and overall engine performance.

This second section of the yaw angle cases introduces a higher distortion into the intake flowfield by increasing the air inflow to 30 degrees yaw angle. This

enhances any vortices within the main flow introducing higher recirculating areas. In Figure 5.28 the flow characteristics of the intake first bend are presented.

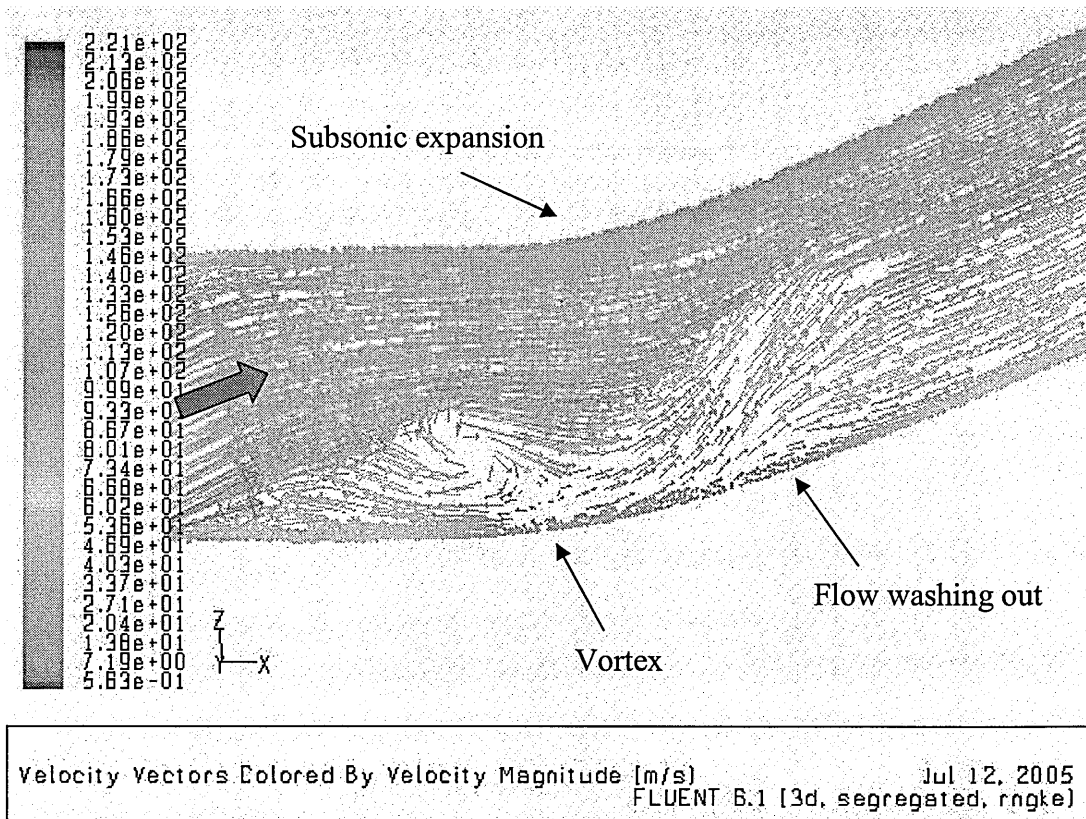


Figure 5.28 Flow characteristics at 30 degrees yaw inflow

As the flow enters the intake, it separates and a large vortex appears in the first segment covering almost half of the duct cross sectional area. Downstream of the vortex, the slower particles of air mix with the faster ones washing out and swirling the flow. This effect enlarges the distortion up to the engine face.

Analysing the flowfield of the intake duct at different sections throughout it can be seen that a high pressure distorted profile is introduced at the inlet of the duct up to the first bend where the diffusing part starts. As the air flow moves further into the duct the high pressure distortion is dissipated until it reaches the engine face where the pressure profile although not uniform has a lower distortion than the inlet profile. Figure 5.29 shows the flow visualisation carried out at different sections of the duct. The sections shown are separated in sections A, B and Engine face representing the flow at the first bend, the mid-section and engine face of the intake duct respectively.

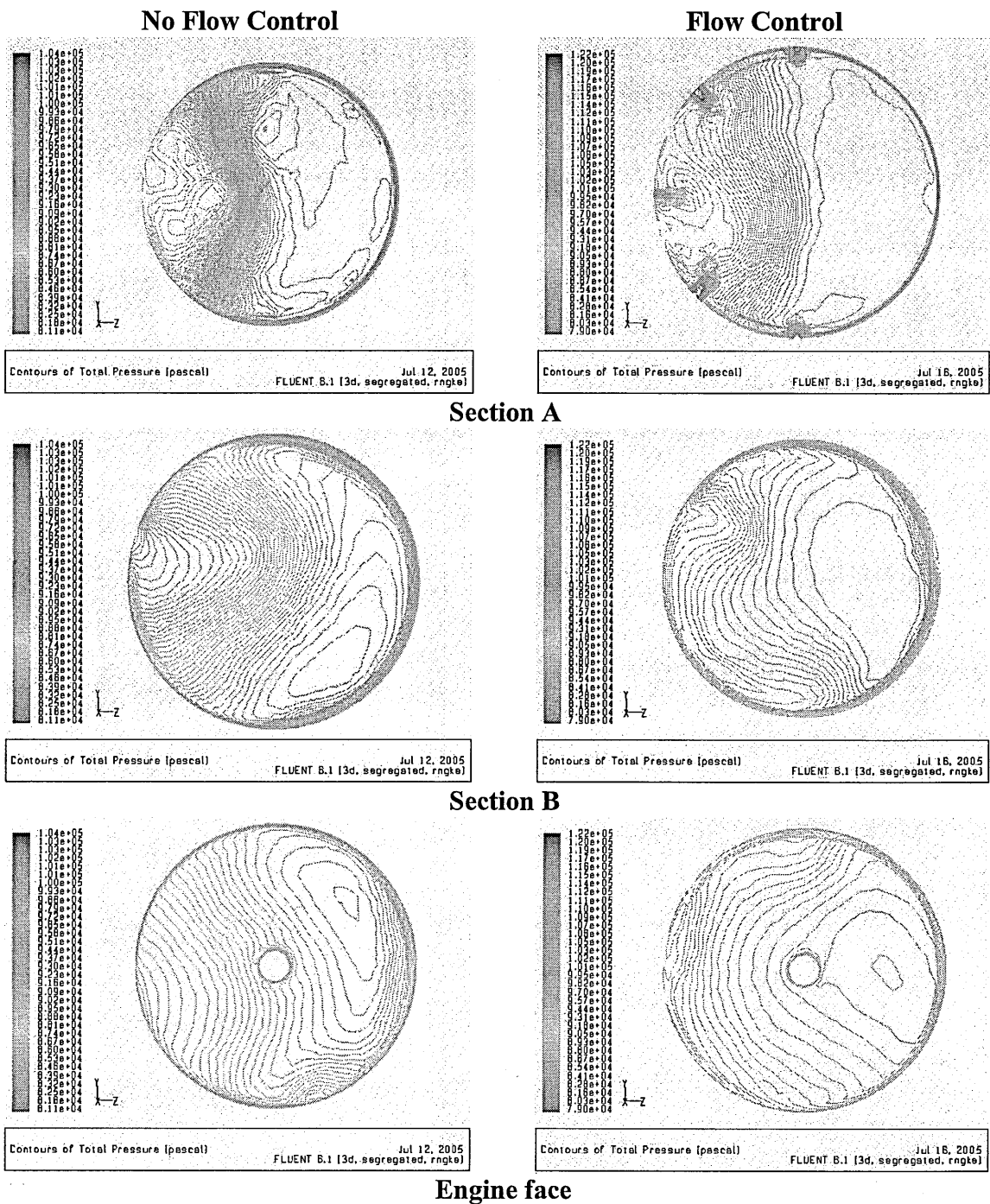


Figure 5.29 Flow visualisation throughout the intake duct

Figure 5.29 also present the flow visualisation for different sections throughout the intake duct for the flow control cases. At section A the flow injection of four jets is shown. This type of flow control has been decided after a flow control optimisation was carried out.

The first decision on flow control implementation has been made based on the effected area of the inflow. As mentioned earlier the flowfield of the duct up to the first bend is introduced with a vortex that covers almost half of the cross sectional area of the duct. This can be seen in Figure 5.28. From this it has been concluded that for the flow control application in this case the injected holes used would be only the holes that cover the area where the vortex is generated.

The next consideration was the angle of secondary flow injection into the main flow. A flow optimisation has been carried out and the results have been analysed both in terms of engine face distortion and pressure recovery. Figure 5.30 presents flow results in terms of intake pressure recovery for three angles of secondary flow injection.

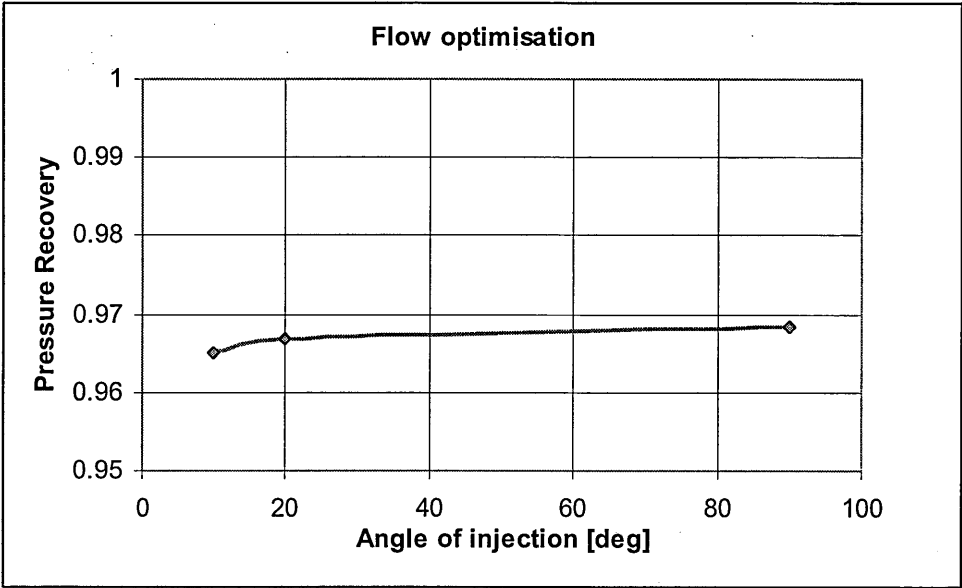


Figure 5.30 Flow optimisation for intake pressure recovery

Figure 5.31 that follows presents the flow optimisation on engine face distortion for the same angles of injection as above. Analysing both Figures it can be seen that at low angles of injection the engine face distortion improves but there is also a small drop in pressure recovery. At 20 degrees injection the pressure recovery improves but the engine face distortion increases to a DC60 of 0.355. this increase in engine face distortion is due to the mixing between the main and secondary flow. The secondary flow tends to penetrate deep into the main flow, introducing an unwanted mixing that increases the flow discrepancies. The

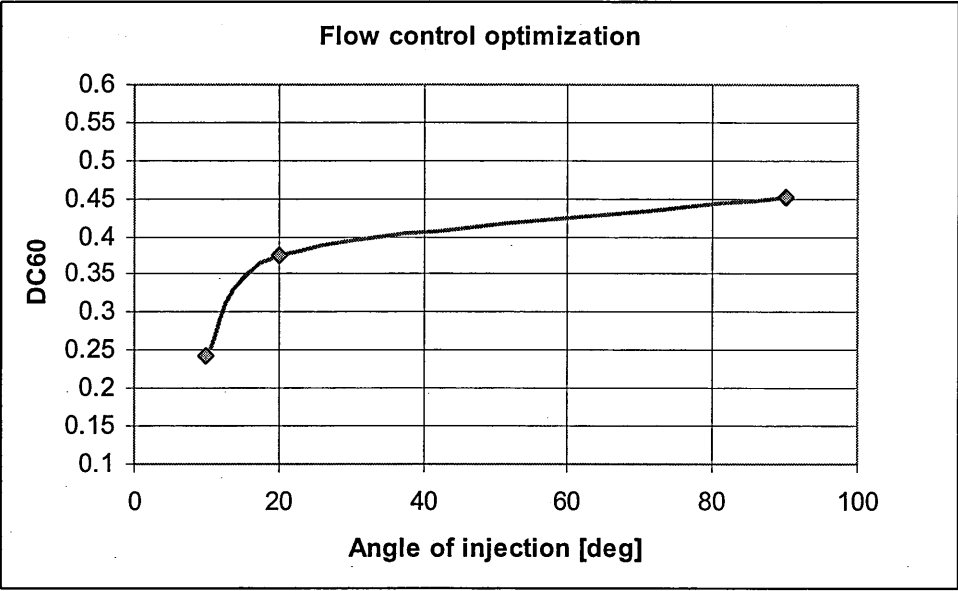


Figure 5.31 Flow optimisation for engine face distortion

In terms of engine face distortion, total pressure contours flow visualisation can be seen in Figure 5.32. the engine face profiles are analysed for three different angles of injection and the value of DC60 is also presented.

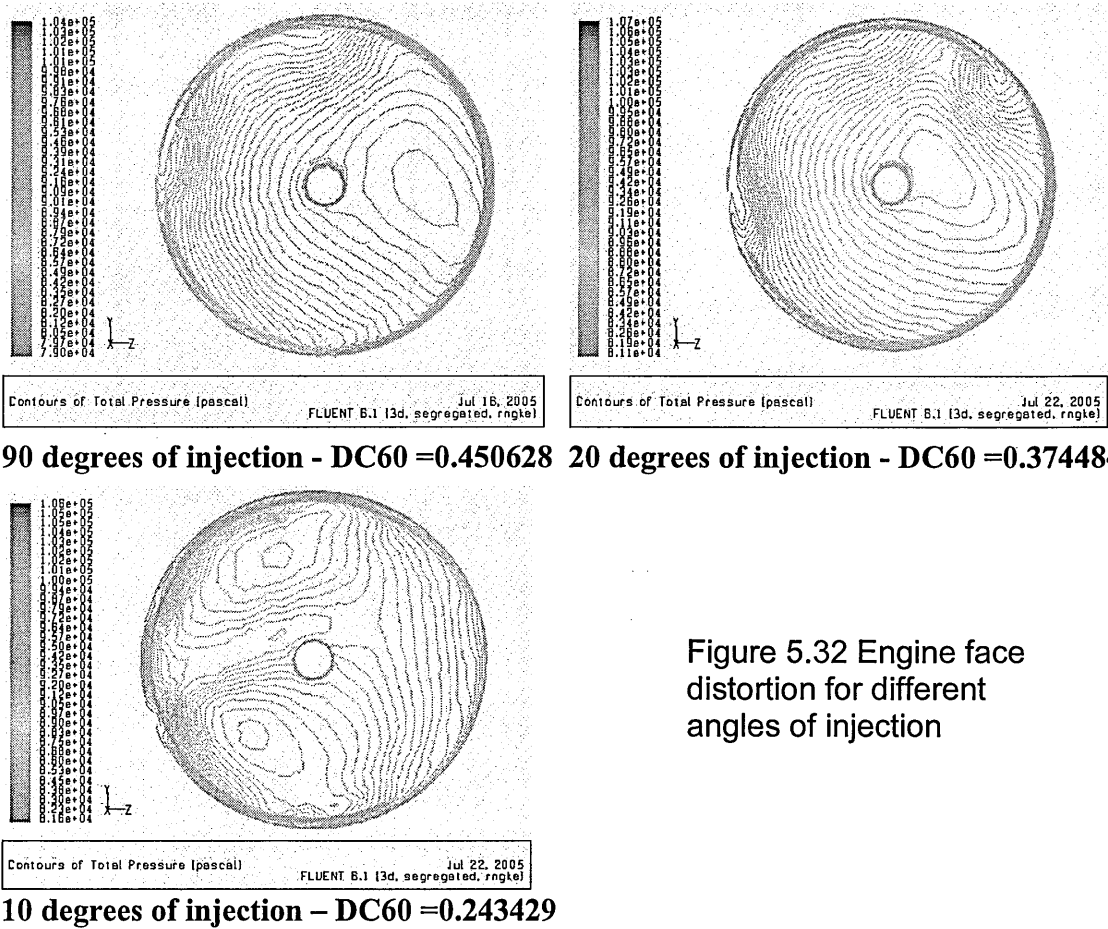


Figure 5.32 Engine face distortion for different angles of injection

Once the flow optimisation was complete injection has been implemented into the 30 degrees yaw case in order to improve the intake duct flow characteristics. The results of the flow control cases have been compared with the distorted cases and Figure 5.33 presents the flow visualisation of the duct for both settings.

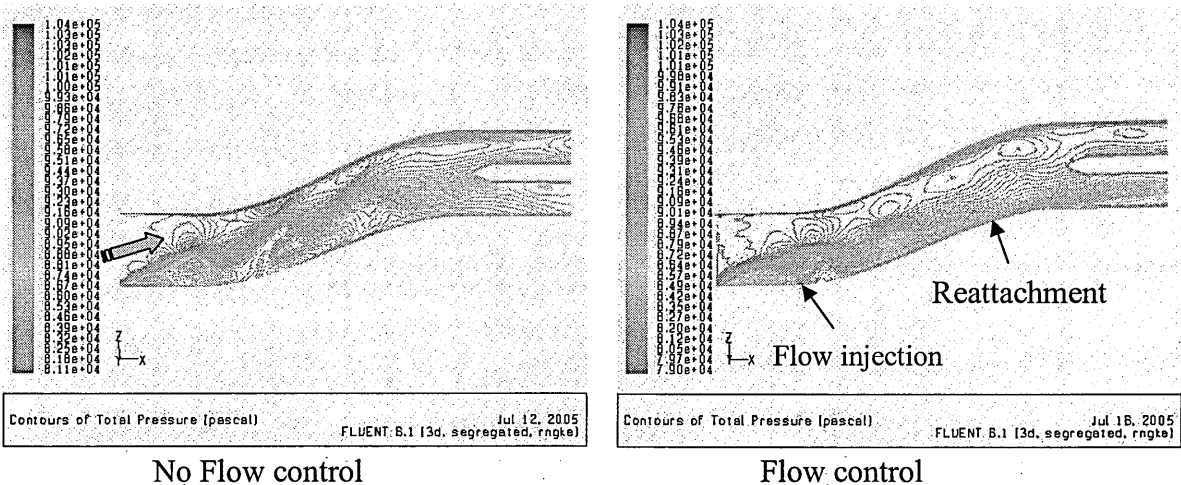


Figure 5.33 Flow visualisation with and without flow control

The Figure above clearly shows the improvement on flow characteristics with the introduction of flow control. The high vortex area is minimised and further downstream there is a flow reattachment.

The improvement of the flow characteristics can also be seen in Figure 5.34 where the engine face total pressure is presented for cases with and without flow control.

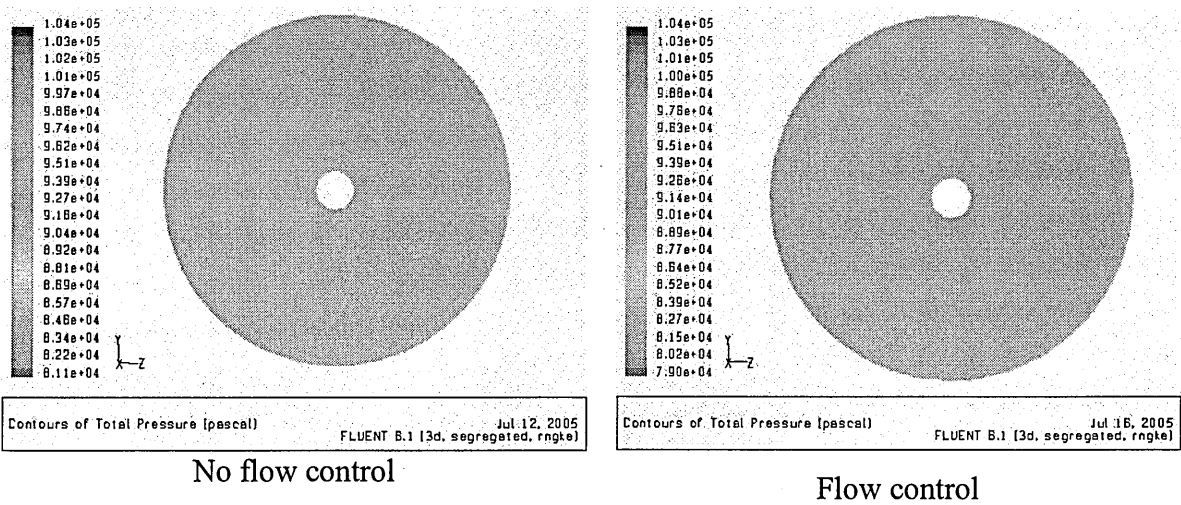


Figure 5.34 Engine face profile with and without flow control

Different engine operating conditions have been analysed with a 30 degrees inflow and the results have been presented in engine face distortion and intake pressure recovery. Figure 5.35 presents the results of axial and 30 degrees yaw angle inflow in terms of engine face distortion.

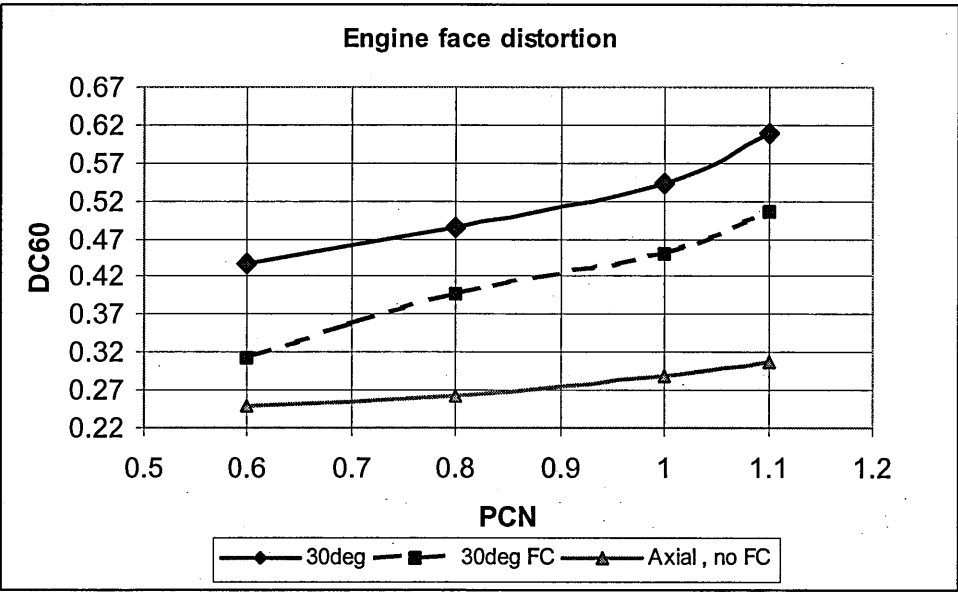


Figure 5.35 Engine face distortion for 30 degrees yaw

The Figure above shows the distortion increase at the engine face when a 30 degrees inflow is considered. DC60 is increased by 80% with a 30 degrees yaw inflow. With the introduction of flow control there is an improvement of engine face distortion by 18%.

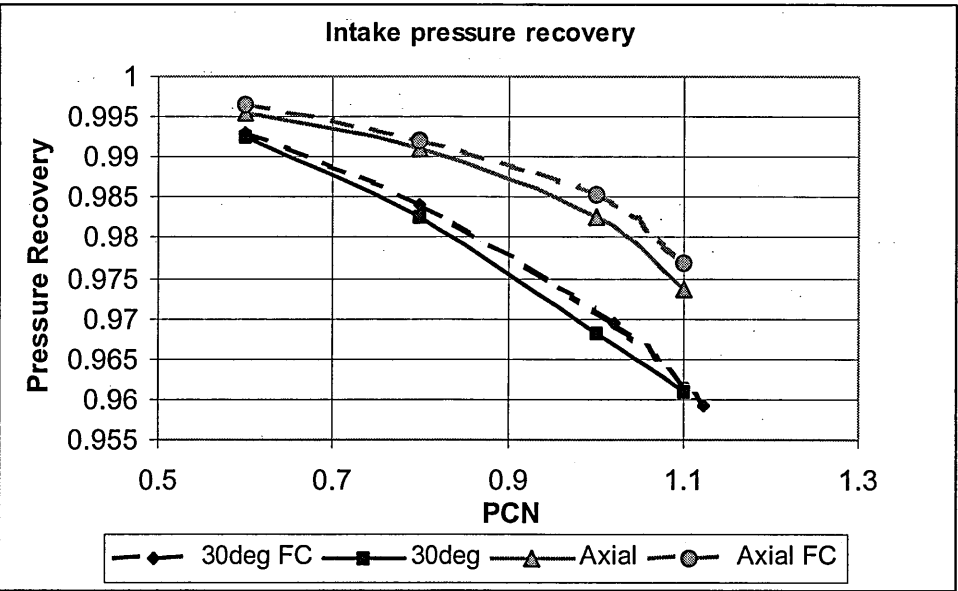
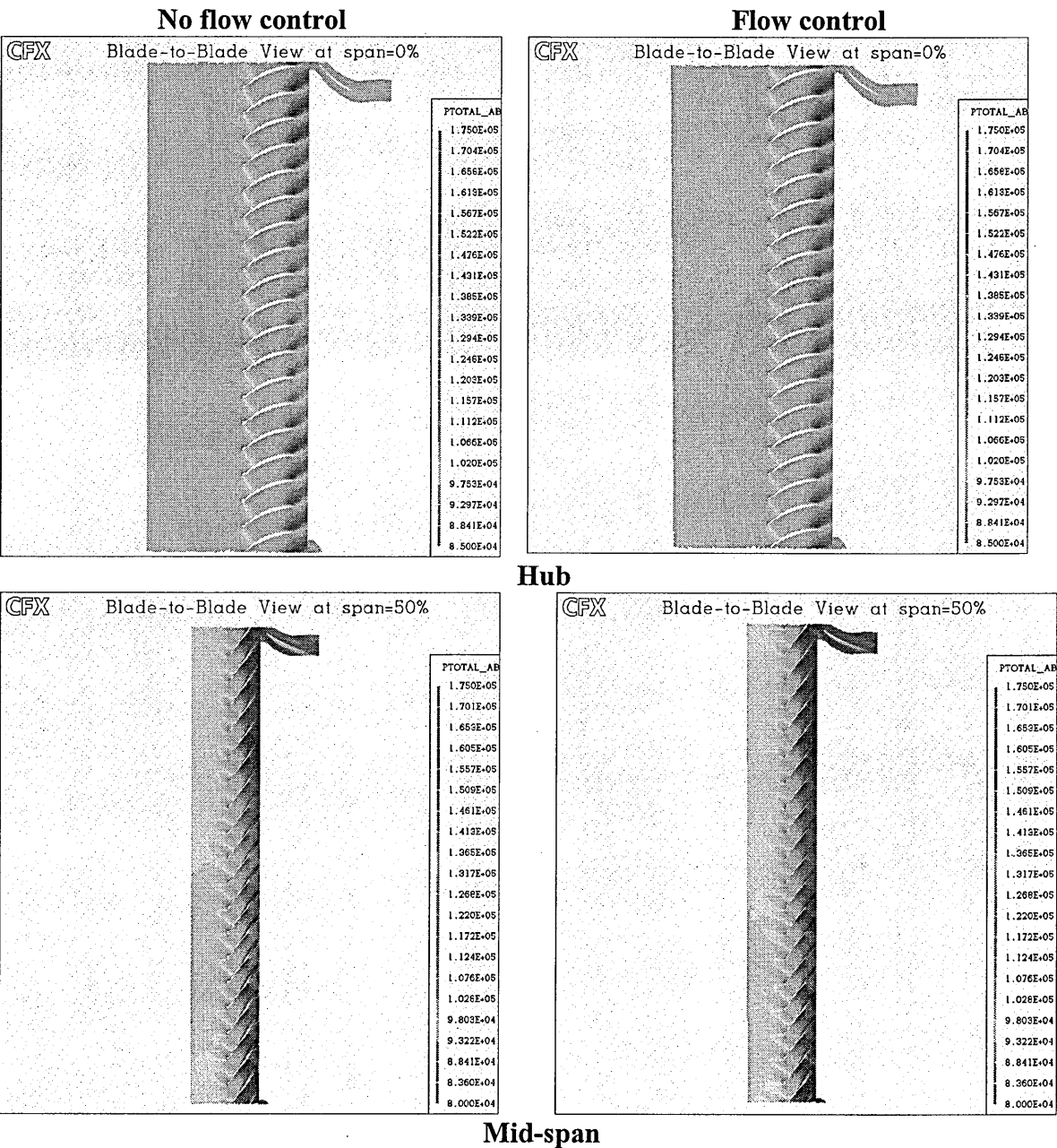


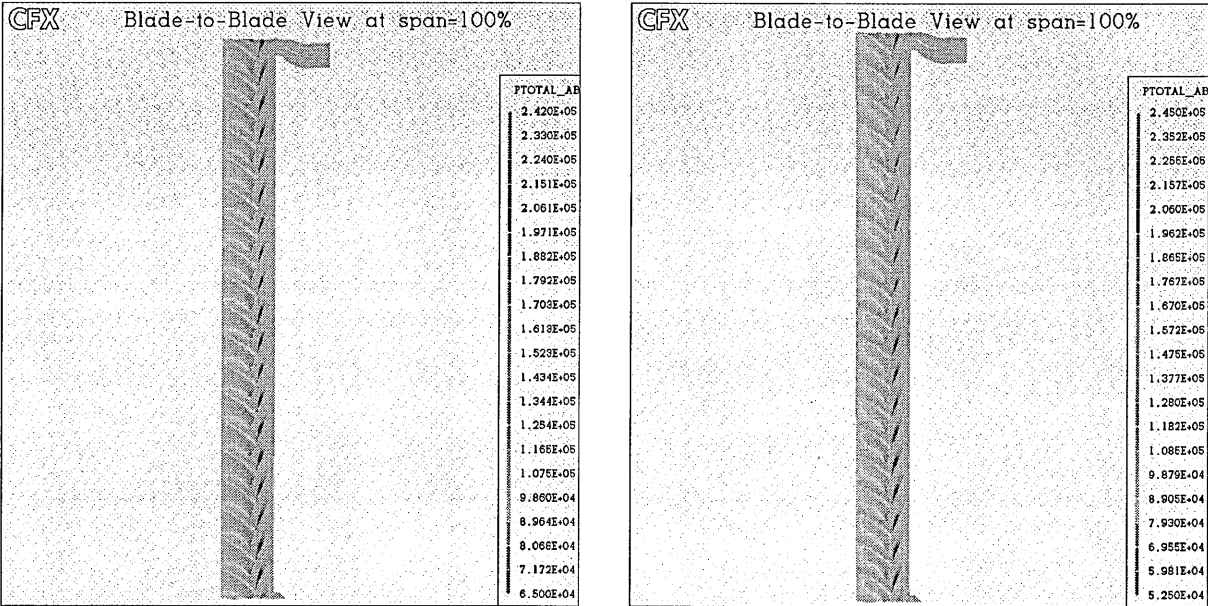
Figure 5.36 Intake pressure recovery at 30 degrees yaw

Figure 5.36 presents the changes in intake pressure recovery. For this case study the pressure recovery is improved by 0.3% when flow control is applied.

5.3.2.2 Compressor flow simulation

Having obtained the results for all the 30 degrees yaw angle inflow both with and without flow control the next step is to translate the intake results into compressor inlet conditions and run the compressor flow simulations. Figure 5.37 presents the total pressure distribution of the compressor stage at the hub, the mid-span and the tip for cases with and without flow control.





Tip

Figure 5.37 Flow analysis along the blade span

Running the flow simulation of variation of engine rotational speeds and a variation of stage outlet static pressures the compressor stage map characteristics have been produced. The maps represent the compressor performance characteristics in terms of pressure ratio, efficiency and mass flow. Figure 5.38 presents the map characteristics of pressure ratio with mass flow.

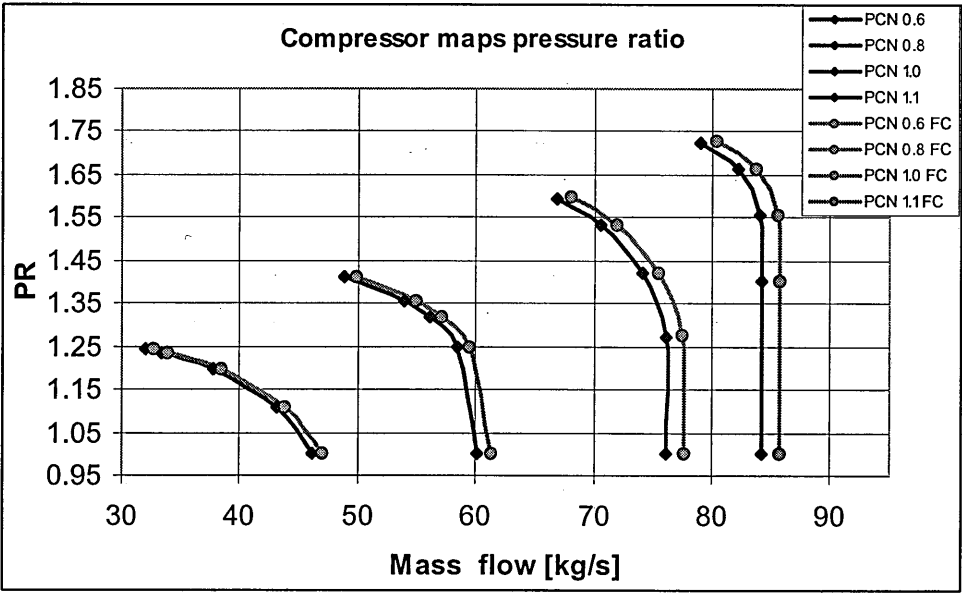


Figure 5.38 Pressure ratio map characteristics

The two compressor characteristics maps shown in the Figure above represent the cases with and without flow control. The results show a clear improvement when flow control is applied to the operation of the system. It can also be said that when compared to the 10 degrees yaw angle cases the improvements that flow control introduces to the system are higher.

The same improvement is shown for the efficiency of the compressor component in Figure 5.39. The flow control cases map tends to move to the upper right of the graph giving more efficient component for the same amount of mass flow.

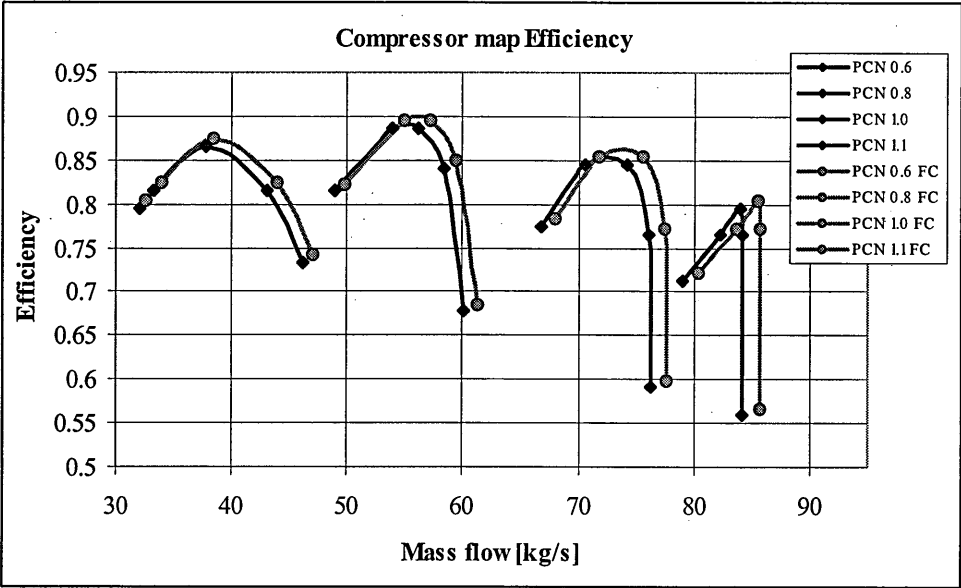


Figure 5.39 Efficiency map characteristics

The results of the compressor flow simulation cases give a clear picture of the improvements that the system has when flow control is applied in order to improve the intake duct flow characteristics.

5.3.2.3 Engine performance

The maps obtained from the compressor flow simulation cases can be used for the analysis of the engine performance under the conditions of 30 degrees yaw inflow. After post-processing the results of the compressor characteristics maps are implemented into the engine performance simulation tool in the form of matrix.

The characteristics maps are implemented into the performance tool and Table 5.3 presents the results of the intake duct in terms of engine face distortion

DC60 and pressure recovery, compared with the thrust obtained from the engine at particular operating conditions.

	30 degrees	Flow Control
Intake PR	0.968	0.21%
Engine Face DC60	0.544	20%
Engine Thrust	57932 (N)	0.81%

Table 5.3 30 degrees yaw results

The results in Table 5.3 give a similar picture to the results for the axial and 10 degrees yaw cases. The improvement achieved with the use of flow control on the engine face distortion is about 20% with an improvement in intake pressure recovery of 0.21%. These improvements in terms of engine thrust translate to just a 0.81%. Again this concludes that for a high improvement in the intake duct flowfield there is just a small improvement in engine performance. This improvement follows more the trend of the intake pressure recovery rather than the engine face distortion. Another point that can be concluded for these results is that for high distorted cases the improvements that flow control introduces to engine performance are higher.

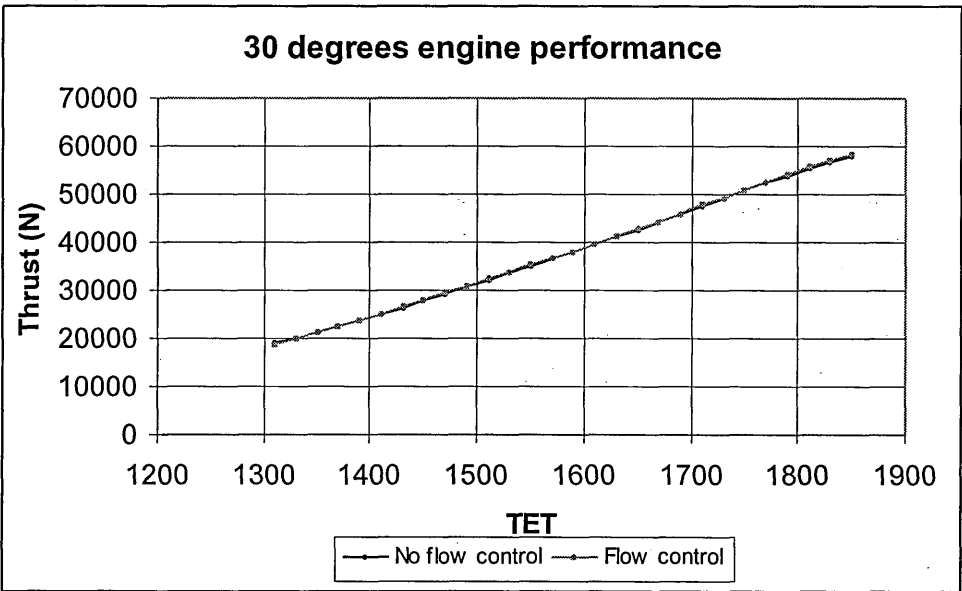


Figure 5.40 30 degrees yaw angle engine performance

Figure 5.40 present the variation of engine thrust with turbine entry temperature. The results show both cases with and without flow control for the 30 degrees

yaw inflow. The difference between the two trend lines is small but it is shown that flow control gives an improvement throughout the different turbine entry temperatures operating conditions.

The next Figure presents the variation of specific fuel consumption with thrust for cases with and without flow control. Again flow control cases show an improvement on engine performance with the engine operating with lower specific fuel consumption for variation of engine thrust.

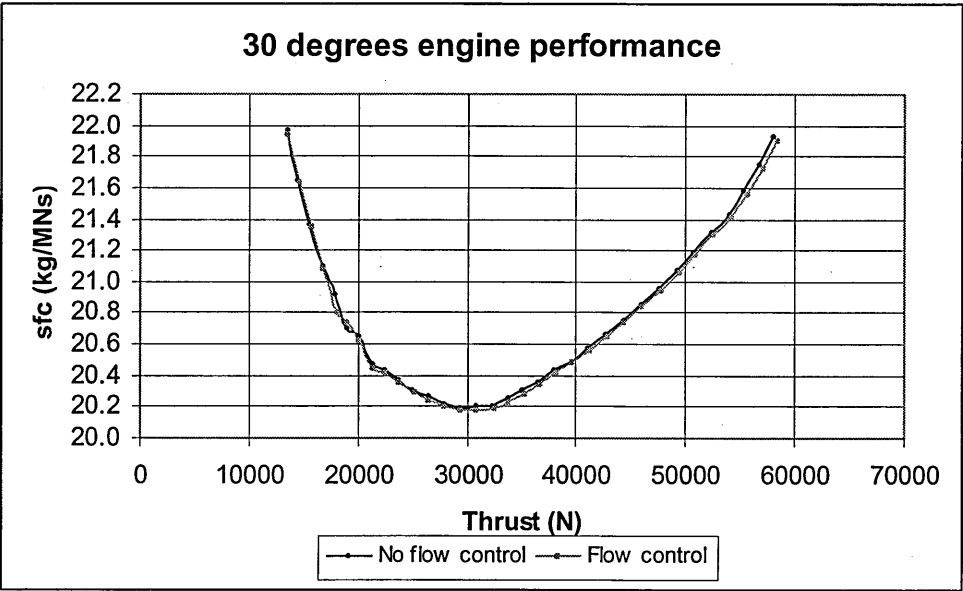


Figure 5.41 30 degrees yaw engine performance

5.4 Angle of attack case studies

In the previous case studies of this chapter an analysis has been carried out on the influence of intake flow to the performance of both the compressor component and engine. The case studies covered the analysis of axial airflow into the intake duct and a variation of yaw angles. The methodology has been carried out in order to analyse the flow characteristics for both the intake and compressor components and the overall engine performance. Different flow control settings have also been introduced to achieve an improvement to the performance of all components.

This final section of chapter five presents case studies for 40 degrees angle of attack operating conditions. The study analysis the influence that such an aggressive inflow has to the intake flowfield characteristics and how that affects the performance of the compressor component. The results of the flow

simulation analysis are then passed through to the engine performance tool in order to simulate engine performance characteristics under those particular operating conditions. For the purpose of improving the flow characteristics of the intake duct and compressor and engine performance in general, two flow control methods have been implemented. The methods used are injection and suction. for the improvement of flow characteristics in previous studies, injection was the only method used. For this case study an analysis of both methods was considered in order to draw some conclusions as to the feasibility of the application.

5.4.1 Intake flow simulation

For the purpose of this study airflow is introduced into the intake duct at 40 degrees angle of attack. Figure 5.42 presents the flow visualisation at the inlet of the duct.

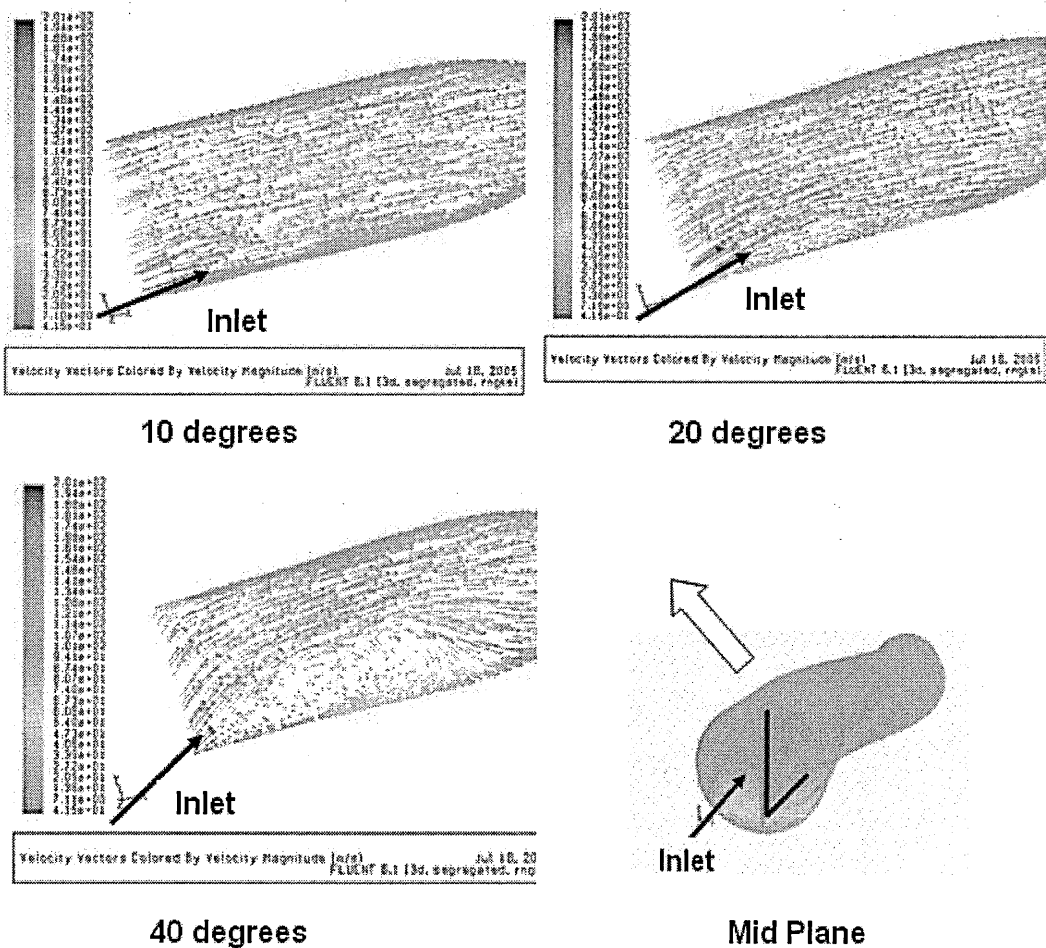


Figure 5.42 Intake flow visualisation at angle of attack

The Figure above presents the duct inflow for a variation of angle of attack cases. This presents an interesting comparison of flow characteristics for low and high angles of inflow air. The results show that for the 10 degrees case the flowfield upstream of the duct has no major changes. There are no flow separation or recirculation areas. When the air inflow angle is increased to 20 degrees angle of attack the flow separates from the wall and a flow recirculation area is introduced. Flow is reattached further downstream. In the case of a 40 degrees inflow the recirculation area is increased presenting the flowfield with a highly distorted flow which effects the air far downstream in the duct.

In Figure 5.43 this far downstream effect is shown with a comparison between the engine face total pressure profiles for the axial, 20 degrees angle of attack and 40 degrees angle of attack cases.

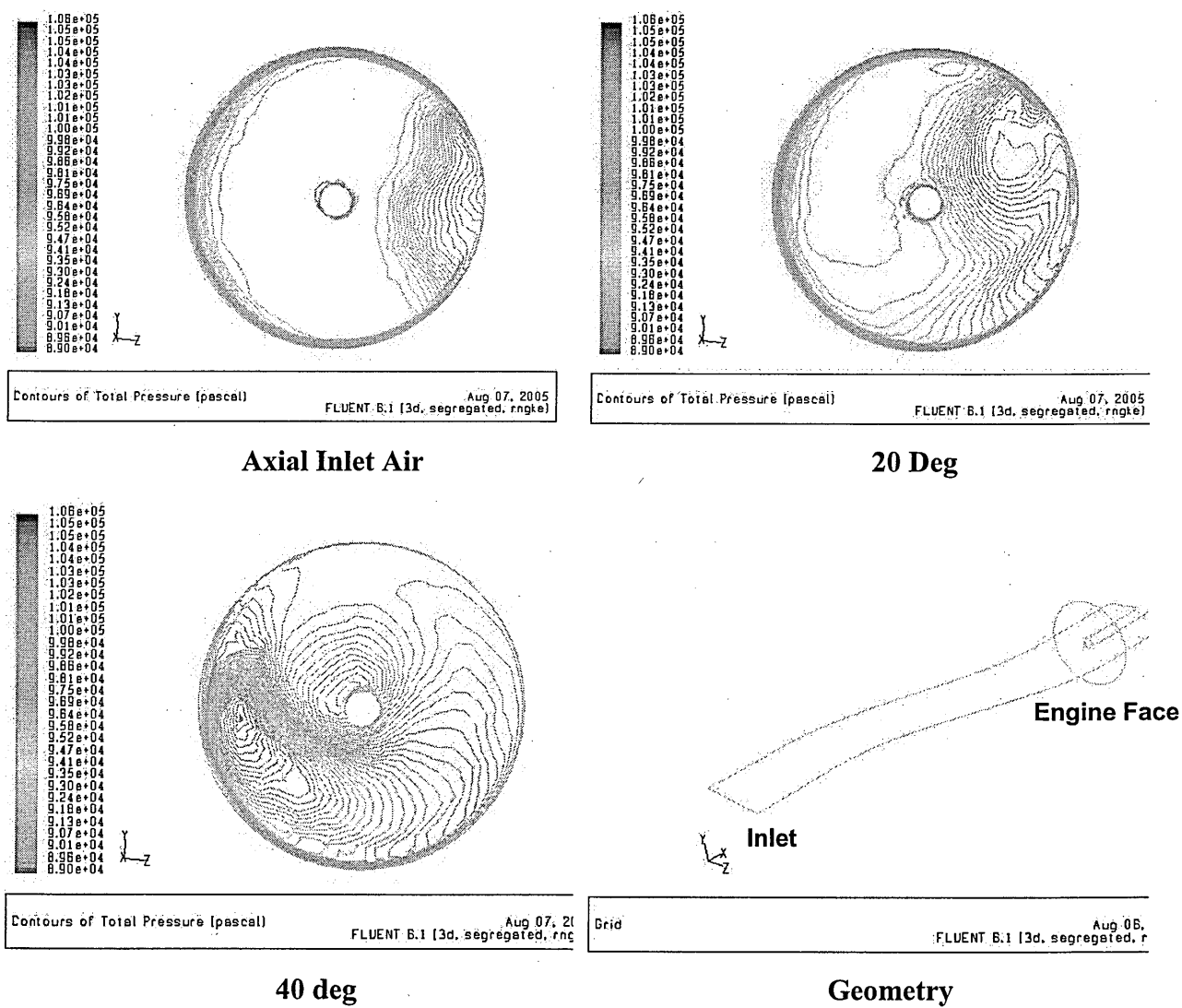


Figure 5.43 Engine face profiles for angle of attack cases

The results in Figure 5.43 show the highly distorted pressure profile of the 40 degrees case in comparison to the axial and 20 degrees case, at engine face. The flow characteristics had a considerable change at that high inflow angle which in turn effected the total pressure profile at engine face.

For improving this distorted flowfield within the intake duct two flow control methods have been used as mentioned earlier, which tackle the problem either by injecting secondary flow into the duct main flow or extracting flow from the intake duct. These flow control methods are applied just upstream of the intake first bend and Figure 5.44 presents the profiles at that particular section for both injection and suction.

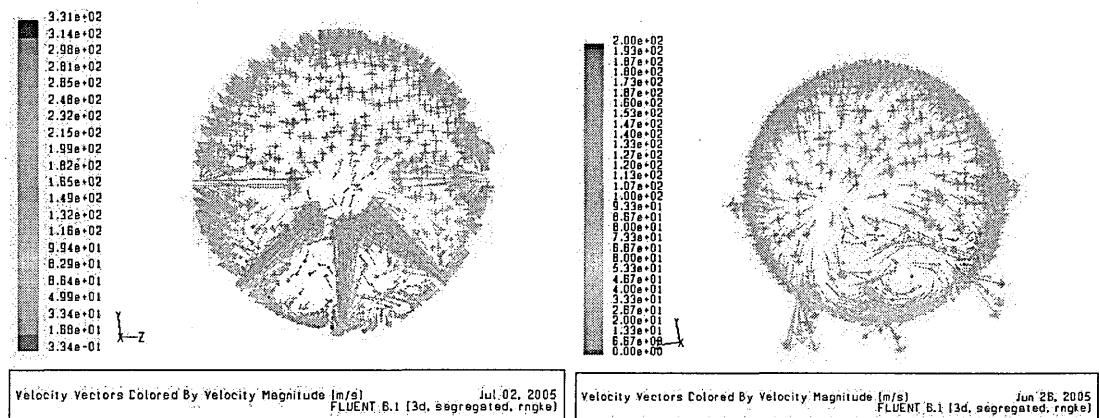


Figure 5.44 Injection and suction at flow control plane

Analysing the engine face total pressure profile for the 40 degrees inflow, the engine face distortion in terms of DC60 is 0.6. With the introduction of injection the DC60 drops to 0.436.

This improvement together with the total pressure contours at engine face can be seen in Figure 5.45 in which results of the two cases are presented. The total pressure contours show an improvement on the uniformity of the engine face profile and a reduction of the high-pressure areas through the engine face. This reduction of high-pressure areas will in turn improve the inlet conditions to the compressor component and the flow characteristics through the compressor rotor and stator stage.

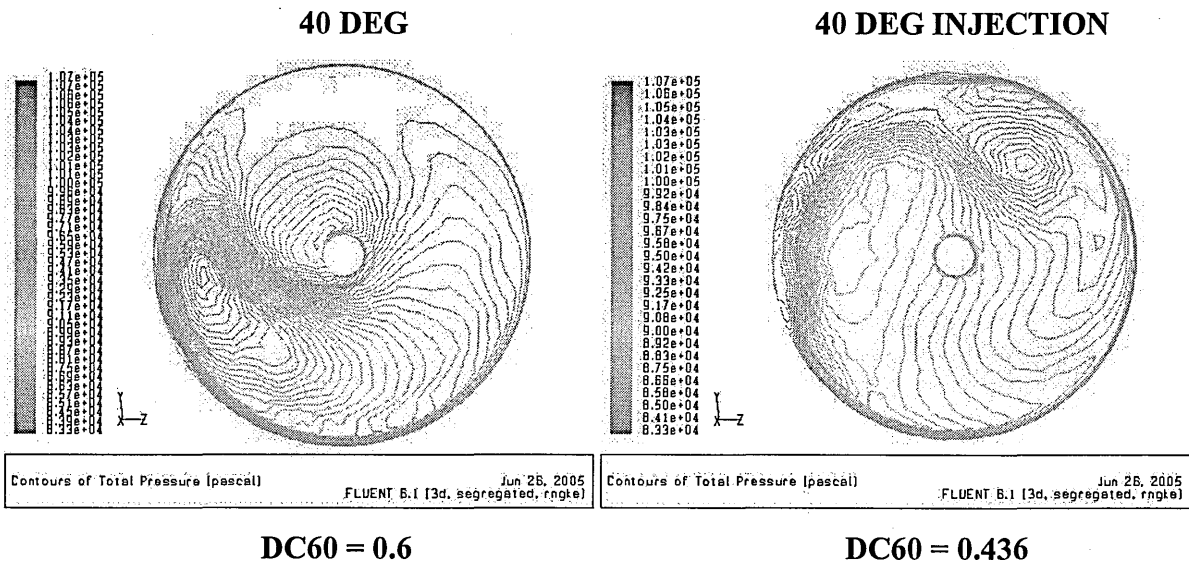


Figure 5.45 Engine face profiles with and without flow control

Running the flow simulation cases for a variation of engine operating conditions the intake results were analysed in terms of engine face distortion DC60 and pressure recovery. Figure 5.46 below show the results of pressure recovery at different engine rotational speeds for suction, injection and no flow control. The Figure that follows 5.47 presents the results analysed in terms of engine face distortion.

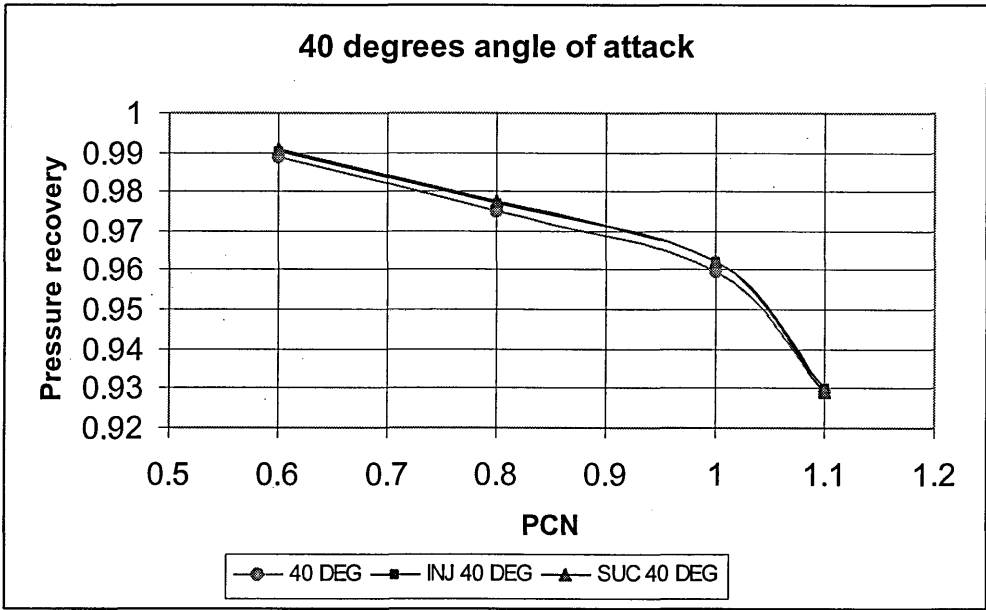


Figure 5.46 40 degrees angle of attack pressure recovery

Studying both Figures it can be seen that as far as engine face distortion is concerned the method that can give the best results of flow characteristics improvement is the injection method. Taking also into consideration the intake pressure recovery the injection methods seems to introduce more pressure losses to the duct than the suction. It can also be seen that at low engine power settings the results of both suction and injection flow control methods are almost the same.

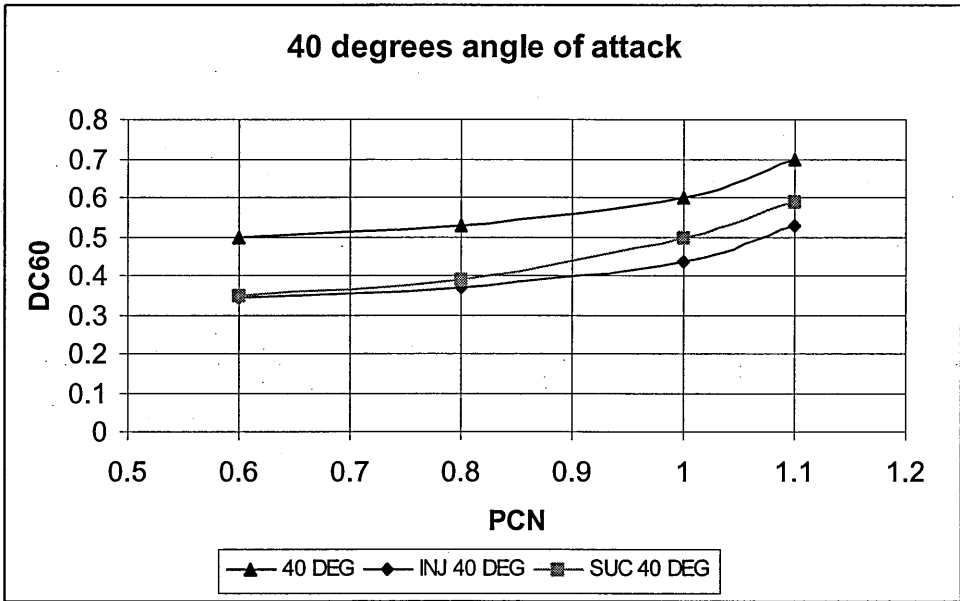


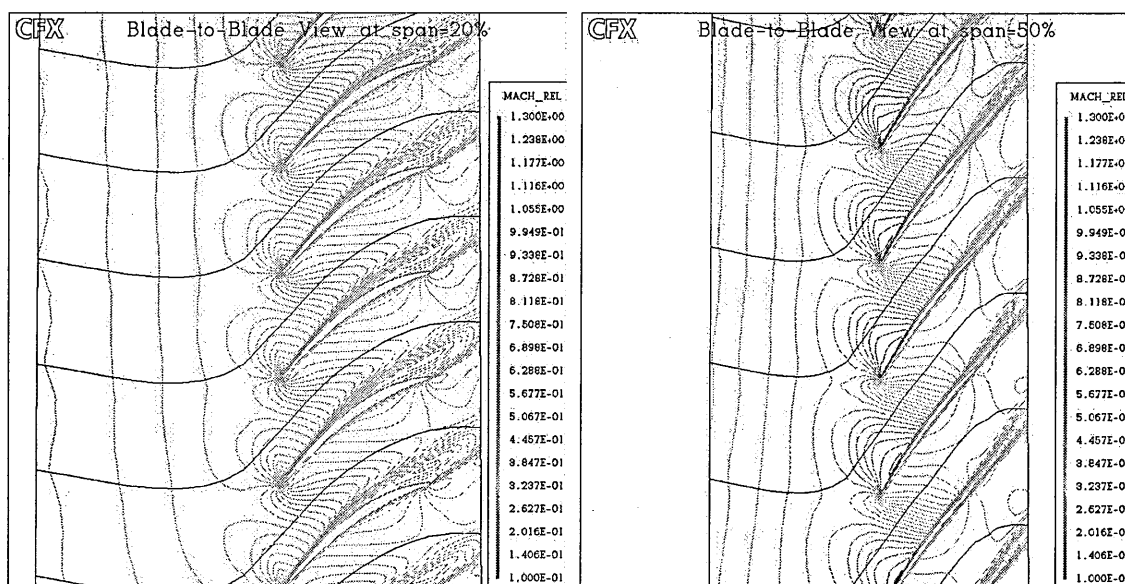
Figure 5.47 40 degrees angle of attack DC60

5.4.2 Compressor flow simulation

Having completed the intake flow simulations for all 40 degrees angle of attack case studies including the flow control cases with suction and injection the next step was to introduce the results of the engine face profile to the compressor stage model.

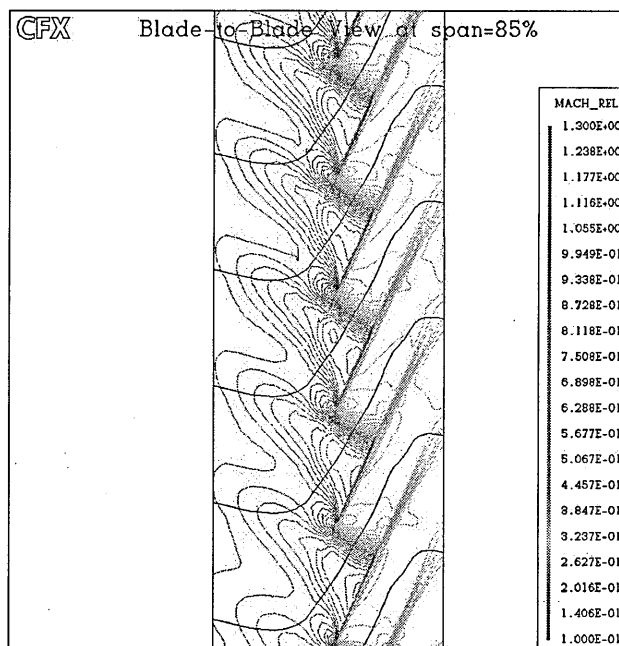
Setting the inlet conditions to the compressor model flow simulation cases have been considered at a variation of compressor rotational speeds and outlet static pressures. The flow simulation for the 40 degrees angle of attack case without flow control was analysed and results are presented in Figure 5.48 for relative Mach number at different points along the blade span. The results show that at 20% of the blade span the relative Mach number is subsonic with no shock waves appearing. Moving to 50% blade span the flow becomes supersonic and weak shock waves appear. Finally at 85% blade span the flow has reached high Mach numbers with the shock wave system being extremely evident.

40 Deg Angle of Attack



Rel Mach at 20 % Rotor Blade Span

Rel Mach at 50 % Rotor Blade Span



Rel Mach at 85 % Rotor Blade Span

Figure 5.48 Relative Mach number across the blade span

Applying suction as the flow control method to the intake flowfield, effected the compressor stage performance by improving the inlet conditions to the system. Figure 5.49 that follows presents compressor total pressure profiles at four

different sections of the rotor, starting from section one being the inlet to the compressor and section four being the exit of the rotor.

40 Deg Suction

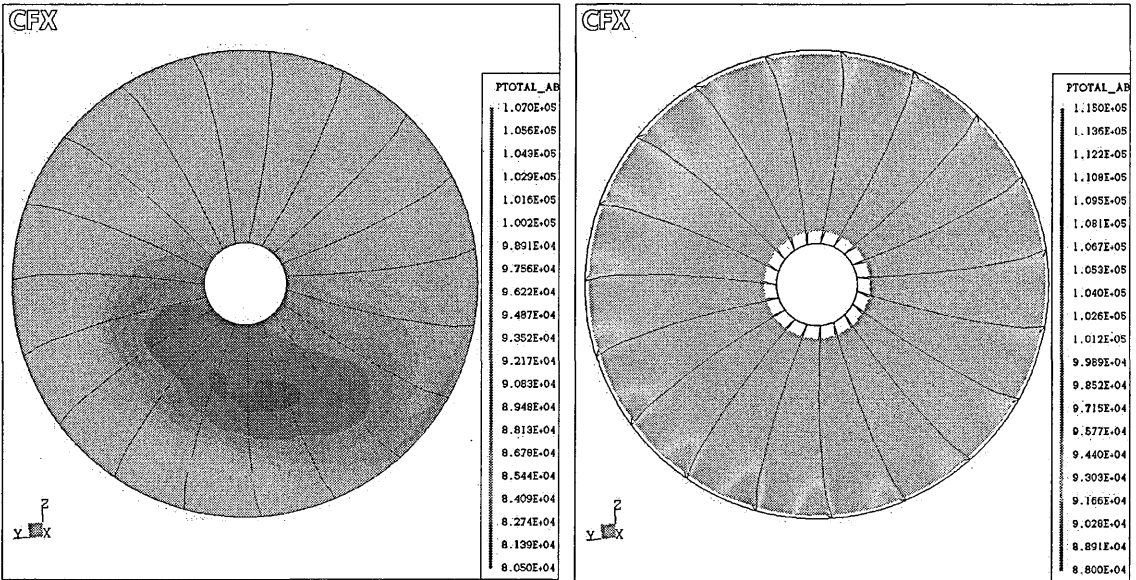
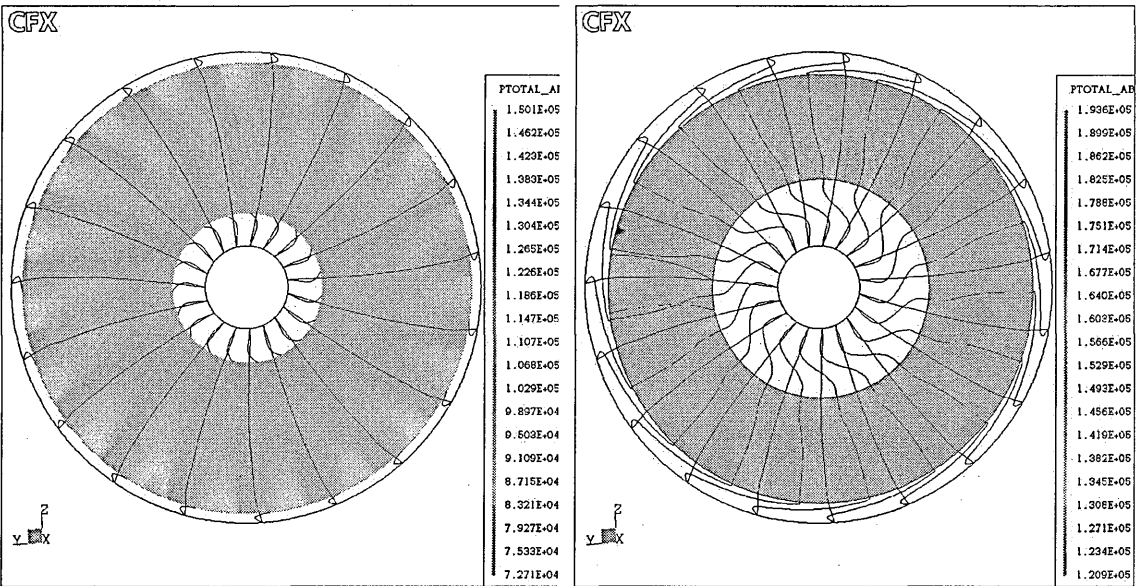
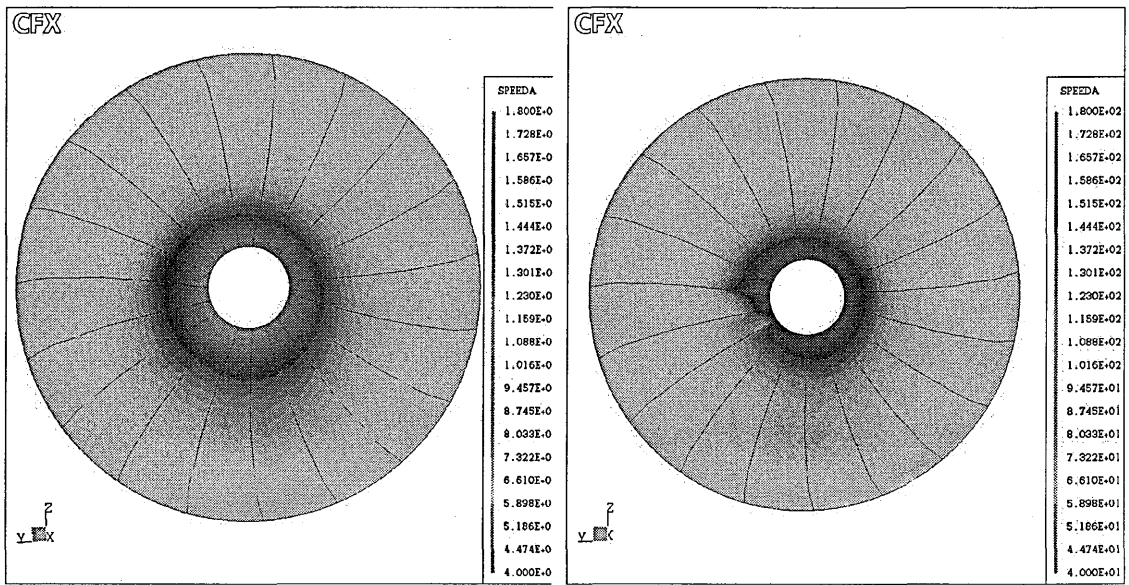


Figure 5.49 Compressor total pressure profiles for suction

The results from the previous Figure show how the inlet distortion is modified and minimised moving further into the compressor rotor section. Although most of the high distortion areas are dissipated there are still some high pressure

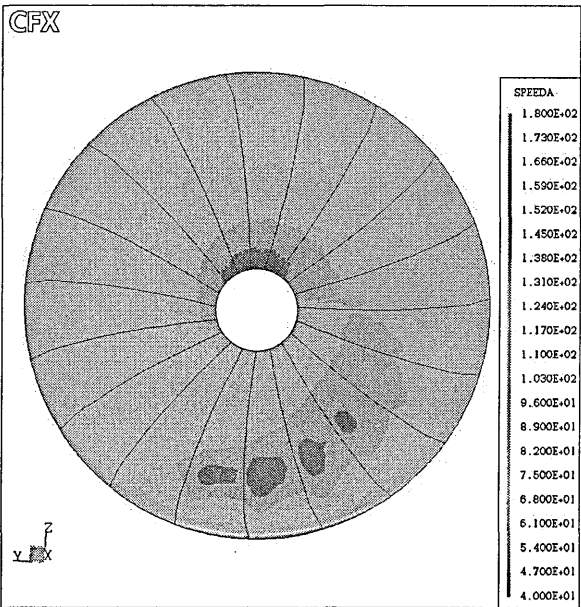
spots shown at the exit of the rotor and into the stator section. This again shows that most of the distortion introduced by the intake, even at high manoeuvres of the aircraft, is attenuated by the rotor section of the compressor stage.

Now in terms of axial velocity of flow coming into the compressor stage it can be seen from Figure 5.50 that the application of either suction or injection flow control tends to increase the axial velocity of the flow close to the hub and minimise the velocity distortion of the case without flow control.



Inlet Axial Vel, 40 Deg Pitch, Suction

Inlet Axial Vel, 40 Deg Pitch, Injection



Inlet Axial Vel, 40 Deg Pitch, Distorted

Figure 5.50 Inlet axial velocity profiles for 40 deg angle of attack cases

All flow simulations have been carried out in order to be able to produce the compressor performance characteristics maps that can be implemented into the engine performance simulation tool and give performance results under specified operating conditions. The Figures that follow present the compressor maps for all 40 degrees angle of attack cases with suction and injection as flow control methods and without flow control.

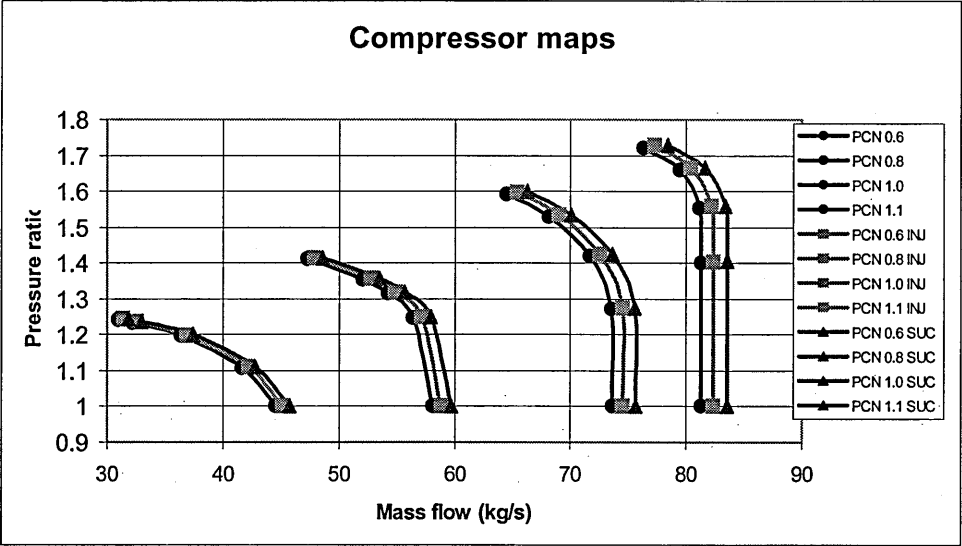


Figure 5.51 40 degrees compressor pressure ratio maps

Figure 5.51 above and 5.52 below show the compressor performance characteristics maps for the 40 degrees angle of attack cases. The distorted case without flow control shows the worst performance characteristics with improvement obtained from both suction and injection flow control case.

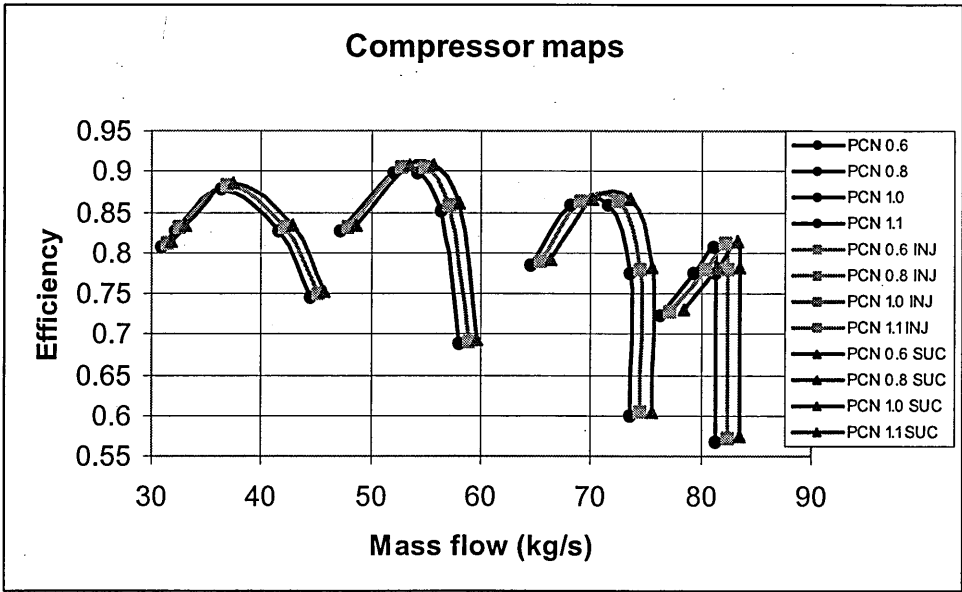


Figure 5.52 40 degrees compressor efficiency maps

The compressor characteristics maps show results that follow conclusions from previous case studies. The analysis of the intake flow simulation presented earlier showed that by applying injection as the flow control method, the engine face distortion is improved more than with the suction method but this is achieved with a small penalty in pressure recovery. The results that the compressor flow characteristics present give a better performance of the compressor system for suction rather than injection. This clearly is affected by the small penalty that injection method introduces to the intake component. Once more it can be seen that although engine face distortion is important for the stability of the compressor and the surge margin, the dominant effect in compressor performance is the pressure losses of the intake.

5.4.3 Engine performance results

By implementing the compressor characteristics maps presented above to the engine performance simulation tool, the engine model has been used for a variation of operating conditions in order to analyse the improvements that both flow control methods can introduce to the engine performance.

Table 5.4 that follows gives an idea of the improvement on engine performance and compare the results with the flow characteristics obtained from the intake simulations. The results are presented in terms of engine face DC60, intake pressure recovery and engine thrust.

	40 Degrees	Injection FC	Suction FC
Intake PR	0.961	0.318%	0.73%
Engine Face DC60	0.6	27.4%	16.7%
Engine Thrust	58034 (N)	0.38%	1.28%

Table 5.4 40 degrees angle of attack results

The results in Table 5.4 show what has already been concluded earlier. When considering injection as the flow control method applied to the system the improvement of engine face distortion is higher than suction method. But the best improvement in engine thrust comes from the application of suction. This can be explained by two parameters. The first parameter the pressure recovery of the intake duct. This seems to have a dominant effect on engine performance. The second parameter is the extraction of air used for the injection method. In the case of suction power is extracted from the engine to

drive the pump used to bleed air from the intake. The effect of power extracted from the engine is lower than the effect from bleed air. This in turn shows an improvement on engine performance.

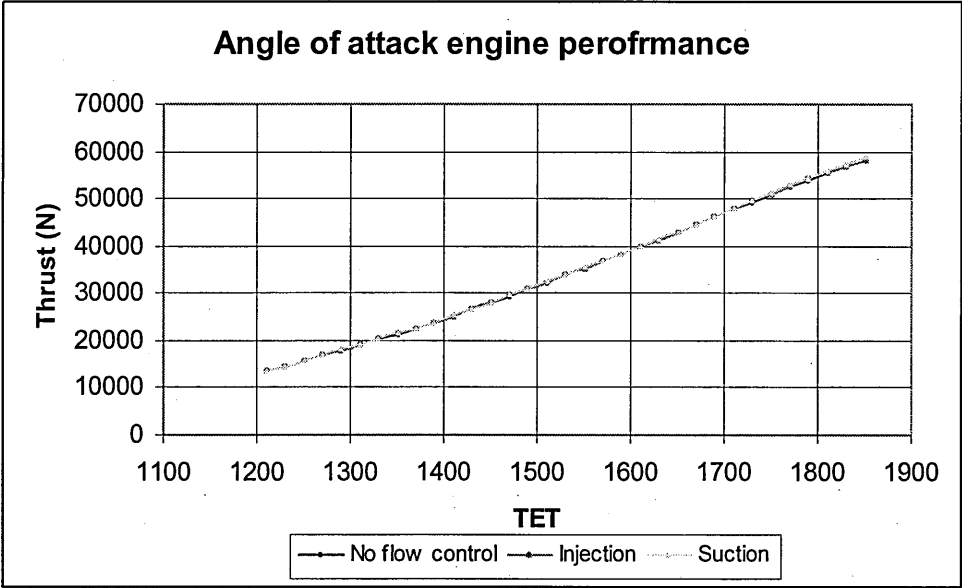


Figure 5.53 Thrust vs TET for angle of attack

The Figures that follow show engine performance comparison for all three cases at a variation of engine operating conditions. Figure 5.53 presents the results of engine thrust with a variation on turbine entry temperature and Figure 5.54 show specific fuel consumption variation with thrust.

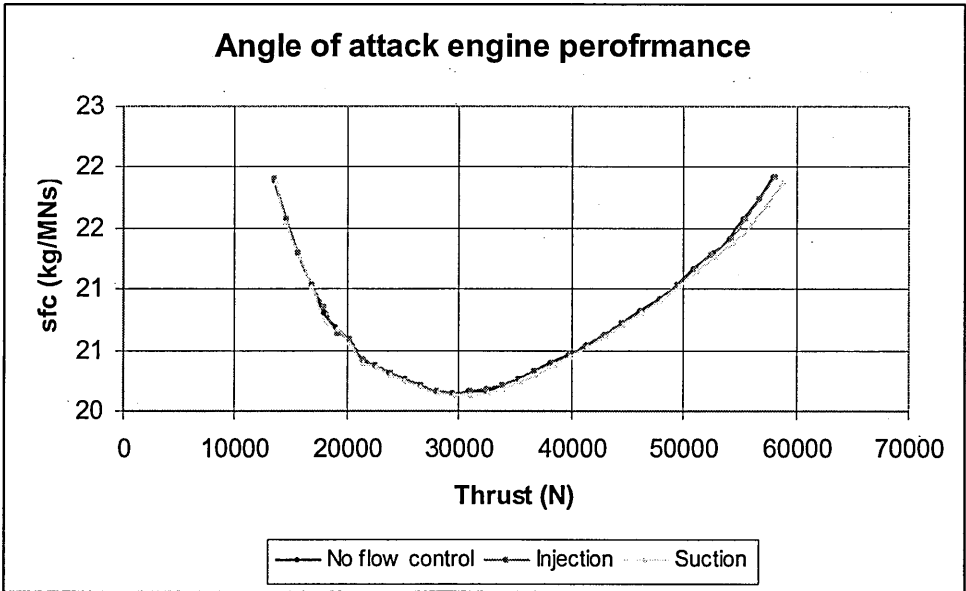


Figure 5.54 Thrust vs sfc for angle of attack

Both Figures show an improvement in engine performance throughout the operating conditions with the use of suction as a flow control method. The importance of limiting the pressure losses of the intake duct in order to improve engine performance has been presented throughout this work and these last Figures show a similar trend.

It is clear that the two parameters used to analyse the flow characteristics of the intake duct effect the engine performance in two different ways. The engine face distortion gives an idea of the effect that the flow will have on the compressor stability in terms of surge. The intake pressure losses give an indication to the effect that the distorted flow will have to the engine performance.

CHAPTER SIX

Conclusions and Future Work

6.1 Conclusions

Literature review carried out for this work showed that many studies throughout the years analysed the phenomenon of boundary layer separation both for internal and external flows. The studies both experimental and computational, used methods to produce distorted flows in order to create the conditions needed for the flow to separate from the wall surface. Having obtained flow separation regions within the flowfield, work was done on improving the characteristics either by delaying or altogether suppressing boundary layer separation. In order to achieve that methods have been used known as flow control methods. These methods that were either passive or active in nature were analysed in terms of improvements on the flowfield characteristics. The studies involved internal and external flows including intake components, wings, etc. The results of these studies showed a considerable improvement to the flow characteristics, which in some cases for intake flowfields analysed in terms of engine face distortion DC60 reached 50% improvement.

The objective of this work was to create a methodology that could carry out an analysis of a system that takes into consideration all the components involved on a flow control application. In other words rather than just analysing the advantages that flow control techniques introduce to the flow, be able also to include the penalties that flow control application will introduce into some systems, in this case the engine.

The work that this study was concerned with, involves the flow analysis of an s-shaped intake duct delivering air to a low bypass turbofan engine. The intake due to its geometry is introduced with a high distortion flow where pressure losses take place and provides the engine with a non-uniform pressure profile. Flow control methods were introduced in order to achieve an improvement of the intake flow characteristics, hence an improvement in engine stability and performance. The flow control methods that seemed more appropriate for this work have been decided after an assessment was carried out, which can be found in chapter one of this work.

For the purpose of this study a methodology has been created that includes all the components involved. These components are the intake duct, the first compressor stage, which is effected by the distortion and the overall engine system. The idea of the methodology is to be able to analyse the flow characteristics of the intake and compressor models and translate that in the

form of compressor performance maps into the engine performance tool. This will allow running an investigation into the engine performance parameters taking into consideration any air or power off-takes that are used to operate the flow control systems.

The methodology was used to analyse the effect of a variation of air inflow angles at both pitch and yaw conditions. The results obtained from the simulation of these cases gave some interesting conclusions, which are summarised below.

- Flow control can be used in this case to improve the internal flow characteristics of s-shaped intake ducts with complex flow characteristics. Flow control can introduce great improvements to the engine face distortion and also smaller improvements to the pressure losses of the flowfield.
- The magnitude of improvements between the intake flowfield and the overall engine performance is not the same. Improvements have been seen especially for the engine face distortion of up to 50% although these improvements translate in engine performance parameters into no more than 0.8% in terms of thrust.
- Further penalty to the engine performance is suffered from the off-takes in order to operate the flow control systems. An analysis has been carried out in chapter four on the effect that these off-takes have on engine performance.
- The improvement that flow control introduces to the system is sufficient in order to cover the request of the flow control mechanism and give a small improvement to the engine performance for all cases analysed. However, in some cases the improvements are very small in order to compensate for the cost and complexity of introducing a flow control mechanism to the overall system.
- The effectiveness of flow control in overall engine performance improvements seems to be higher for the cases of high distortion flows. It can be seen from results presented in chapter five that for high distortion flow cases the improvements that flow control methods introduce, give better engine performance results.
- Finally, the results showed that for the improvement of engine performance that this study is concerned with, the intake parameter that has the dominant effect is pressure losses rather than engine face

distortion. The results showed that a change in intake pressure losses could have a great effect to the engine performance parameters. Engine face distortion seems to affect more the stability of the engine.

6.2 Future work

As mentioned earlier for the purpose of this work a methodology has been created that used to analyse the interaction of intake/compressor/engine characteristics. The methodology is based on computational analysis and the only experimental results that could be found were used for the validation of the intake flow simulation. Furthermore, in order to carry out the work assumptions had to be made on the interaction of the three component models. Basic limitations of the methodology used are presented below.

- Steady CFD analysis has been carried out for both intake and compressor models, simulating rather unsteady flow characteristics.
- Although experimental results have been used to validate the flow characteristics of the intake, there were no data available to validate the compressor results. This resulted in a number of assumptions of the compressor grid, turbulence models, simulation procedures, etc.
- Averaging of inlet and outlet conditions to the models and compressor characteristics have been used due to computational time restrictions.
- Limitations to the analysis of compressor stability in terms of surge margin due to the modelling procedures and the nature of the phenomenon.
- Interaction between the intake and compressor model using two different commercial CFD tools to run the flow simulations.
- The averaging used for the creation of the compressor characteristics maps and the translation into the engine model.

The intake model used for this study has been validated based on experimental results and the flow characteristics obtained from the simulations has been translated into the compressor model. The compressor stage modelling and the settings used for the flow simulation are based on previous experience and studies that have been carried in earlier years. There were no experimental case studies that could help to validate the model and flow results. Finally, the engine model used for this study has been obtained from open literature and is an approximation to the actual engine with many assumptions on engine components efficiencies and specifications.

The results obtained from the modelling and simulation of the case studies presented throughout this work give a clear picture of the complexity of flow control applications and the parameters involved in order to optimise a flow control technique. This work covered some flow control optimisation studies analysing some of the parameters that influence the application of the methods. The complexity of applying flow control is based on the number of variables that need to be considered. These parameters are presented below.

- Injection holes size and shape: the holes used for either injection or suction can be optimised in terms of both size and shape in order to introduce a smooth secondary flow into the intake and achieve the highest flow effects.
- Position of the holes: although this depends primarily on the point of flow separation studies showed that flow control can be applied either upstream or downstream of the separation region.
- Boundary conditions: this is one of the two areas considered in this work and optimisation studies have been carried out on the mass flow and velocity of the secondary flow injection. The intensity of the secondary flow can introduce great pressure losses to the flow, which result in engine performance losses.
- Angle of injection: this is the second area considered for flow control optimisation in this work. The results showed that the angle at which secondary flow is injected into the intake duct could have great effects into the flow characteristics and minimise the flow control improvements.

In conclusion this work provided an inside view to the application of flow control in an overall system configuration giving some interesting results on the advantages and disadvantages of the system. Due to the lack of experimental data many assumptions were made in order to set and apply the methodology. Improvements can be made for future work on improving the methodology by tackling some of the points mentioned earlier and further optimisation studies on flow control specifications could help improve the effect of flow control methods on flowfield characteristics.

References

1. Prandtl, L. "Über Flüssigkeitsbewegung bei sehr kleiner Reibung", *Proceedings, 3rd International Mathematics Congress*, Heidelberg, Germany, 1904.
2. Gad-el-Hak. M., "Flow Control: passive, active and reactive flow management", Cambridge University Press, 2000.
3. Wallis, R., "The use of air jets for boundary layer control", *Aero Note* 110, 1952.
4. Johnston, J. and Nishi, M., "Vortex generator jets – A means for passive and active control of boundary layer separation", AIAA 89-0564, 1989.
5. Selby, G., Lin, J., Howard, F., "Control of low speed turbulent separated flow using jet vortex generators", *Experiments in fluids*, Vol. 12-6, 1992.
6. Lin, J., Howard, F., Bushnell, D., and Selby, G., "Investigation of several passive and active methods for turbulent flow separation control", AIAA 90-1598, 1990.
7. Ball, W., "Tests of wall blowing concepts for diffuser boundary layer control", AIAA 84-1276, 1984.
8. Senseney, M. B., Buter T. A., Bowersox R. D. W., "Flow structure and performance characterization of an offset diffuser with and without vortex generator jets", AIAA 95-2884, 1995.
9. Raghunathan, S., Watterson, J.K., Cooper R.K., Kelly, B., "A novel concept of passive vortex generator jets for separation control", AIAA 99-1004, 1999.
10. Vakili, A.D., Sauerwein S.C., Miller D.N., "Pulsed injection applied to nozzle internal flow control", AIAA 99-1002, 1999.
11. Sedor, J.L., "Experimental investigation of pulsed jet nozzle blowing for forebody flow control", AIAA 96-0004, 1996.
12. McManus, K.R., Lenger, H.H., Davis, S.J., "Pulsed vortex generator jets for active control of flow separation", AIAA 94-2218, 1994.
13. Johari, H., McManus, K.R., "Visualization of pulsed vortex generator jets for active control of boundary layer separation", AIAA 97-2021, 1997.
14. Amitay, M., et al., "Control of internal flow separation using synthetic jet actuators", AIAA 2000-0903, 2000.
15. Rabe, A., et al., "A Facility for Active Flow Control Research in Serpentine Inlets", AIAA 2002-0510, 2002.

16. McLean, J.D., Herring, H.J., "Use of multiple discrete wall jets for delaying boundary layer separation", NASA/CR-2389, 1974.
17. Hernandez, G., Schonfeld, T., Nicoud, F., "Numerical active control of two-dimensional boundary layer separation", Reports AIAA-96/2142, 1996.
18. Lin, John C., "Control of turbulent boundary-layer separation using micro-vortex generators", Reports AIAA-99/3404, 1999.
19. McKinzie, Daniel J., "Turbulent boundary layer separation over a rearward facing ramp and its control through mechanical excitation", AIAA-91/0253, 1991.
20. Haas, M., Lin, R.-S., Brogan, T., "Boundary layer separation control", AIAA-2002/2947, 2002.
21. McCormick, D. C., "Boundary layer separation control with directed synthetic jets", AIAA-2000/0519, 2000.
22. Erbsloh S.D., Crowther W. J., "Control of compressor face total pressure distortion on a high bypass turbofan intake using air-jet vortex generators", AIAA-2004-2206, 2004.
23. Goldsmith E. L., Seddon J., "Practical intake aerodynamic design", Blackwell Scientific Publications, 1993.
24. Miller D. S., "Internal flow systems", BHRA, Fluid Engineering Series 5, 1978.
25. Dardis W. J., Mayhew E. R., "An acceptance process for the evaluation of inlet distortion", AIAA Conference Paper, 92-3918, 1992.
26. Kral L. D., Recent experience with different turbulence models applied to the calculation of flow over aircraft components. Progress in Aerospace Sciences, 1998.
27. Harloff G.J., Smith C.F., Bruns J.E., and DeBonis J.R., "Navier-Stokes analysis of three-dimensional S-ducts", Journal of Aircraft, 1993.
28. Abrahamsen P.E.H., Pettersson-Reif B.A., Sætran L., and Fossdal J.B., "Air intake studies: Experimental measurements and computational modelling", RTO AVT Symposium on Missile Aerodynamics, Italy, May, 1998.
29. May N.E., "The prediction of intake/S-bend diffuser flow using various two-equation turbulence model variants, including non-linear eddy viscosity formulations", ARA, March 1997.
30. May N.E., "The prediction of intake/S-bend diffuser flow using the SAUNA CFD system with two-equation turbulence models", ARA, November 1995.

31. Zhang G., Assanis D.N., "3-D turbulent subsonic compressible flow predictions using high-order schemes and comparison with measurements", ASME Journal, 1996.
32. Menzies R.D.D., "Investigation of S-shaped Intake Aerodynamics using Computational Fluid Dynamics", Ph.D. Dissertation, University of Glasgow, 2002.
33. Fluid Dynamics Panel Working Group 13, "Test case 3 - subsonic/transonic circular intake", AGARD, Advisory Report 270, 1991.
34. Mattingly J. D., "Elements of gas turbine propulsion", McGraw-Hill, 1996.
35. Johnsen, I.A., Bullock R.O., "Aerodynamic design of axial flow compressors", Volume I, II, III, NACA Research Memorandum, 1956.
36. Ramsden K.W., "Compressor aerodynamic design", Cranfield University, 2001.
37. Cumpsty N.A., "Compressor aerodynamic", Longman scientific and technical, 1989.
38. Hirai K. et al, "Unsteady three-dimensional analysis of inlet distortion in turbomachinery", AIAA 97-2736, 1997.
39. Davies M., Hale A., Beale D., "An argument for enhancement of the current inlet distortion ground test practise for aircraft gas turbine engines", Journal of turbomachinery, 2002.
40. Ghisu T., "Axial compressor response to inlet flow distortion by CFD analysis: effect of different levels of distortion", MSc Thesis, Cranfield University, 2003.
41. Charalambous N., "Axial compressor response to inlet flow distortion by CFD analysis: effect of different frequencies of distortion", MSc Thesis, Cranfield University, 2003.
42. Iurisci G., "Axial compressor response to inlet flow distortion by CFD analysis: effect of different types and extends of distortion", MSc Thesis, Cranfield University, 2003.
43. Hill P. G., "An educational introduction to transonic compressor stage design principals"
44. Saravanamuttoo H. I. H., Rogers G.F.C., Cohen H., "Gas turbine theory", Pearson education limited, fifth edition, 2001.
45. Walsh P.P., Fletcher P., "Gas turbine performance", Blackwell science, 1998.
46. Pilidis P., "Gas turbine theory and performance", Cranfield University, 2001.

47. Laskaridis P., "Performance investigations and systems architectures for the more electric aircraft", PhD Thesis, Cranfield University, 2004.
48. Muleri M., "Effect of angle of attack on jet engine inlet flow distortion", Cranfield University, 2005.
49. Tentorio L., "Low bypass engine performance in a yawing aircraft", Cranfield University, 2005.