Design Optimization for Enhanced Fuel Mixing and Reduced Combustion Instability
Enhancing Swirler Performance of a Small Turbojet Engine Combustor

Phillip Venter

26 August 2015

Faculty of Aerospace Engineering · Delft University of Technology
Design Optimization for Enhanced Fuel Mixing and Reduced Combustion Instability
Enhancing Swirler Performance of a Small Turbojet Engine Combustor

Master of Science Thesis

For obtaining the degree of Master of Science in Aerospace Engineering at Delft University of Technology

Phillip Venter

26 August 2015

Faculty of Aerospace Engineering · Delft University of Technology
The undersigned hereby certify that they have read and recommend to the Faculty of Aerospace Engineering for acceptance a thesis entitled “Design Optimization for Enhanced Fuel Mixing and Reduced Combustion Instability” by Phillip Venter in partial fulfillment of the requirements for the degree of Master of Science.

Dated: 26 August 2015

Head of department: Prof.dr.ir. Piero Colonna

Supervisor: Dr.ir. Wilfried Visser

Reader: Dr. Matteo Pini

Reader: Dr. Rene Pecnik
Summary

Aero-engine performance is becoming an increasingly regulated aspect in aerospace industries with tighter restrictions on emissions, greater expectations for efficiency and thrust as well as broader requirements for the range of the operating flight envelope. With an increasing consciousness toward these factors during the design of combustors, research led development and improvement of every single aspect of the combustor design needs to be considered in this modern era of aerospace technology. A major contributor to such performance enhancement is the design of flow swirlers used to induce central recirculation zones in the primary fuel/air mixing region. In the current study, the effect of modification to a swirler’s vane blade angle on mixing effectiveness and combustion stability is investigated, using flow properties such as turbulent kinetic energy, fuel distribution and pressure losses as a measure of combustor performance. The study takes a sensitivity analysis approach and makes use of an existing combustor design that acts as a benchmark for verification of results. A cold flow computational fluid dynamics analysis is used to test the effect of blade angle modifications based on a ‘cause and effect’ methodology. The computational fluid dynamics model is validated against experimental data from a similar combustor. It was found that optimal fuel/air mixing occured in a 70° blade angle swirler however large pressure losses and excessive vortex shedding directly behind the center body indicated a strong likelihood of combustion instability. Good fuel atomisation through strong shear layers and excellent pressure recovery seen in a 30° blade angle swirler was accompanied by poor fuel/air mixing. A swirler design featuring 50° blade angles was found to be the optimum, with good fuel atomisation, stable recirculation zones, promising flame anchoring potential, dispersive but orderly homogenous fuel/air mixing and desireable pressure recovery characteristics.

Keywords: Swirler, Combustion Stability, Mixing, CFD, Cold Flow, Optimisation, $K - \epsilon$ turbulence model, Precessing Vortex, Toroidal Recirculation
Acknowledgements

I would like to acknowledge with huge gratitude the selfless dedication and help offered by Monty Morris and Karl Du Plessis at Esteq in providing knowledge and computational resources.

The collaboration with the South African Council for Scientific Interest and Research (CSIR) was appreciated with special thanks to Dr Glen Snedden for his academic support.

Thank you to Wilfried Visser for supervising this thesis through to the end.

To my family, I cannot thank you enough for your endless support and relentless faith in my capacity to obtain this Master’s degree.

Finally I wish to thank Grant Bubb, who’s inspiring words gave me the courage to embark on this challenge.

Dankie O Here my God dat U my hier deur gedra het...

Delft, The Netherlands
26 August 2015

Phillip Venter
## Contents

**Summary**

**Acknowledgements**

1 **Introduction**
   1.1 Study Overview .......................... 3
   1.2 Document Structure ........................ 4
   1.3 Project Definition ........................ 5
   1.4 Combustion and Burner Technology Overview ........................ 8
      1.4.1 Combustor Types ........................ 9
      1.4.2 Combustion Difficulties ......................... 10
   1.5 Swirler Parameterisation ........................ 12

2 **Literature Review**
   2.1 Chapter Overview .......................... 15
   2.2 Fundamental Theory of Turbulent Flows .......................... 15
      2.2.1 Velocity Fluctuations .......................... 15
      2.2.2 Turbulent Flow Parameters ......................... 17
      2.2.3 Length Scales ............................... 18
      2.2.4 Time Scales ................................. 19
   2.3 Research Findings .......................... 20
   2.4 State of the Art Research ........................ 23
   2.5 Recommendations for Computational Analysis ........................ 23
3 Analytical Methods

3.1 Analysis ................................................................. 27
3.2 APA 350 Engine and Model Specification .................................. 27
  3.2.1 Design Specification .............................................. 28
  3.2.2 Parameterisation .................................................. 28
  3.2.3 CAD Cleanup ...................................................... 29
3.3 Computational Fluid Dynamics ............................................... 35
  3.3.1 Mesh Independence Study ....................................... 36
3.4 Flow Analysis Procedures .............................................. 39
  3.4.1 Modelling Swirlers ............................................... 39
  3.4.2 Preparation of FloEFD Model ..................................... 41

4 Model Validation ......................................................... 45

4.1 Motivation for Validation ................................................ 45
4.2 Experimental Approach .................................................. 45
  4.2.1 PIV Verification Measurements ................................... 51
4.3 CFD Validation Approach ................................................. 58
4.4 Results of Validation Study .............................................. 58
  4.4.1 Baseline Plane ..................................................... 58
  4.4.2 Primary Zone Plane ............................................... 59
  4.4.3 Secondary Zone Plane ............................................. 59
  4.4.4 Dilution Zone Plane ............................................... 59
4.5 Conclusions for Validation Study ....................................... 59

5 Results and Discussion .................................................... 63

5.1 Post Processing and Results Overview ................................... 63
5.2 Results ................................................................. 65
  5.2.1 30 Degree Blade Angle Summary .................................. 65
  5.2.2 50 Degree Blade Angle Summary .................................. 65
  5.2.3 70 Degree Blade Angle Summary .................................. 66
5.3 Discussion .............................................................. 67
  5.3.1 Analysis of Dynamic Flow Structures .............................. 67
  5.3.2 Pressure Loss ...................................................... 69
  5.3.3 Recirculation and Flame Anchoring ............................... 69
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Typical axial swirler</td>
<td>6</td>
</tr>
<tr>
<td>1.2</td>
<td>Swirler parameters [20]</td>
<td>13</td>
</tr>
<tr>
<td>2.1</td>
<td>Reynolds velocity decomposition</td>
<td>16</td>
</tr>
<tr>
<td>2.2</td>
<td>Axial flow RMS of fluctuations</td>
<td>17</td>
</tr>
<tr>
<td>2.3</td>
<td>Illustration of vorticity [6]</td>
<td>17</td>
</tr>
<tr>
<td>2.4</td>
<td>Effect of swirl number upon pressure loss coefficient. Taken from Syred N and Beer [29]</td>
<td>22</td>
</tr>
<tr>
<td>3.1</td>
<td>Cross section of combustor</td>
<td>28</td>
</tr>
<tr>
<td>3.2</td>
<td>APA 350 combustor</td>
<td>30</td>
</tr>
<tr>
<td>3.3</td>
<td>Breakdown of Components</td>
<td>31</td>
</tr>
<tr>
<td>3.4</td>
<td>Removal of barrel flange holes</td>
<td>32</td>
</tr>
<tr>
<td>3.5</td>
<td>Modification of flange Profile</td>
<td>33</td>
</tr>
<tr>
<td>3.6</td>
<td>Torch ignitor gap</td>
<td>33</td>
</tr>
<tr>
<td>3.7</td>
<td>Typical flow swirler injectors</td>
<td>34</td>
</tr>
<tr>
<td>3.8</td>
<td>Fuel injectors</td>
<td>34</td>
</tr>
<tr>
<td>3.9</td>
<td>Inner flame tube</td>
<td>35</td>
</tr>
<tr>
<td>3.10</td>
<td>Modified swirler blade angles</td>
<td>39</td>
</tr>
<tr>
<td>3.11</td>
<td>Swirlers with varying vane angle</td>
<td>40</td>
</tr>
<tr>
<td>3.12</td>
<td>Fuel injection cone</td>
<td>42</td>
</tr>
<tr>
<td>4.1</td>
<td>Combustor used for validation study, from Meyers [15]</td>
<td>46</td>
</tr>
</tbody>
</table>
List of Figures

4.2 Experimental apparatus for combustor PIV measurements from Meyers [15] 47
4.3 PIV optical instrumentation configuration from Meyers [15] 48
4.4 Experimental combustor liner from Meyers [15] 49
4.5 Fabricated perspex combustor Lining from Meyers [15] 50
4.6 Illustration of planes used for laser sheet from Meyers [15] 50
4.7 PIV verification velocity vector plot for primary zone from Meyers [15] 52
4.8 Photograph showing obstructions to laser that cause lack of vector information from Meyers [15] 52
4.9 PIV verification velocity vector plot demonstrating broken patches from Meyers [15] 53
4.10 PIV verification velocity vector plot demonstrating characteristic flow structures from Meyers [15] 54
4.11 PIV verification velocity vector plot demonstrating subtle fluid interactions from Meyers [15] 55
4.13 PIV verification velocity magnitude plot from Meyers [15] 57

5.1 Reference planes 64
5.2 Surface plot planes 65
5.3 Snapshots of an iso-vorticity surface at $\omega = 75,000 \, \text{s}^{-1}$ for two different swirl numbers taken from Huang and Yang [54] 68

A.1 Summarised results for meshes 4, 5 and 8 respectively 80
A.2 Evolution of mesh quality 81

B.1 Velocity magnitude for base plane 84
B.2 Velocity vector plot for base plane 85
B.3 Axial velocity profile for base plane 86
B.4 Velocity magnitude for mixed planes 88
B.5 Velocity vector plot for mixed planes 89
B.6 Axial velocity profile for mixed planes 90

C.1 Velocity magnitude with streamlines displayed 92
C.2 Close-up velocity magnitude 93
C.3 Pressure magnitude 94
C.4 Turbulent kinetic energy magnitude 95
C.5 Kerosene mass fraction 96
List of Figures

C.6 Velocity vectors ......................................................... 97
C.7 Swirler pressure loss ................................................... 98
C.8 Turbulent kinetic energy summary .................................... 99
C.9 Turbulence intensity summary ......................................... 100
C.10 Vorticity summary ..................................................... 101
C.11 Stoichiometric fuel/air ratio volume ................................. 102
C.12 Kerosene mass fractions of 2.5% and 5% .......................... 103
C.13 Iso-vorticity plot at 337501/s ........................................ 104
C.14 2D visualisation of spiraling vortex based on vorticity XY-plots ... 105
C.15 XY-Plots of axial velocities ........................................... 106
C.16 XY-Plots of combustor pressure ...................................... 107
C.17 XY-Plots of vorticity .................................................. 108
C.18 XY-Plots of kerosene mass fraction (logarithmic axis) ............. 109
C.19 XY-Plots of circumferential velocity ................................ 110
Nomenclature

\( \delta P_{sw} \) = Swirler pressure loss (\( Pa \))
\( \delta P_{L} \) = Liner pressure loss (\( Pa \))
\( \dot{m}_{sw} \) = Swirler mass flow rate (\( kg/s \))
\( \rho \) = Density (\( kg/m^3 \))
\( k_{sw} \) = Constant for flat bladed swirlers
\( \theta \) = Blade vane angle (\( ^\circ \))
\( A_{sw} \) = Swirler fluid area (\( m^2 \))
\( A_{L} \) = Liner area (\( m^2 \))
\( D_{sw} \) = Swirler outer diameter (\( m \))
\( D_{hub} \) = Swirler hub diameter (\( m \))
\( n_{\nu} \) = Number of vanes
\( t_{\nu} \) = Vane thickness (\( mm \)), Kinematic viscosity (\( m^2/s \))
\( S_{a} \) = Ribeiro and Whitelaw’s geometric swirl number
\( S_{g} \) = Khanna’s geometric swirl number
\( S_{N} \) = Beer and Chigier’s geometric swirl number
\( S_{a} \) = Aerodynamic swirl number
\( U_{z} \) = Axial velocity (\( unit \))
\( U_{\theta} \) = Tangential velocity (\( unit \))
\( r \) = Radial distance (\( m \))
\( A_{t} \) = Total area of tangential inlet (\( m^2 \))
\( A_{e} \) = Swirler inlet area (\( m^2 \))
\( \dot{m}_{\theta} \) = Tangential mass flow rate (\( kg/s \))
\( \dot{m}_{tot} \) = Total mass flow rate (\( kg/s \))
\( R_{0} \) = Radius of the inlet quarl (\( m \))
\( r_{e} \) = Burner exit radius (\( m \))
\( r_{i} \) = Radius upon which tangential jets fire (\( m \))
\( \Phi \) = Equivalence ratio
\( L \) = Integral eddy length scale (\( m \))
\( \eta \) = Kolmogorov eddy length scale (\( m \))
\( \lambda \) = Taylor eddy microscale (\( m \))
\( \epsilon \) = Energy dissipation rate per unit mass (\( m^2/s^3 \))
\( \tau \) = Eddy turnover time (\( s \))
\( Re \) = Reynolds number
\( C_{p} \) = Specific heat at constant pressure (\( kJ/kgK \))
\( C_{v} \) = Specific heat at constant volume (\( kJ/kgK \))
\( R \) = Specific gas constant (\( J/kgK \))
\( \gamma \) = Ratio of specific heats
\( V_{inj} \) = Fuel injection velocity (\( m/s \))
Chapter 1

Introduction

1.1 Study Overview

Combustors have been developed and improved for decades with design requirements shifting according to social, geopolitical and environmental concerns. With increasing pressure from regulations on minimum performance and emission control, recent efforts have focused on creating more fuel efficient burners as well as producing less harmful greenhouse gases such as NOx. This has in part been achieved by developing lean burning combustors, however operating at lean limits leads to a greater chance of instabilities. Aside from combating combustion stability, designs need to feature improved mixing effectiveness of the reacting flows.

Hence three issues can be identified that need to be accounted for in modern combustor design: The first is effective mixing and circulation of reactants in the combustion chamber. This has a large influence on combustion efficiency, production of harmful emissions such as nitrous oxides, and optimal heat release for maximized thrust and performance. Secondary consequences are the effects of pressure fluctuations and flow perturbations that can lead to instability.

The second is flame anchoring and thermo-acoustic coupled combustion instabilities. Problems arising from these phenomena include dangerous flash back, blow out and unsuccessful ignition attempts. Furthermore, pertaining to combustion instability, there are major concerns for excessive vibration leading to catastrophic damage or structural fatigue to the combustor caused by low amplitude resonance. Secondary effects are also incomplete combustion or diminished performance due to heat release fluctuation.

A third consideration for the design of combustors using flow swirlers to improve mixing effectiveness is the consequences that the swirlers have on pressure losses in the combustor; steeper blade angles will impart more rotational velocity in the reacting flow but will also act as an obstruction the axial flow and therefore induce severe pressure losses.

Studies have found [54] that the use of swirlers in burners can be made to increase fuel and air mixing, whilst also serving to increase combustion stability by anchoring the flame.
and deliberately inducing vortex breakdown through ingenious aero-mechanic mechanisms that will be discussed in this report. The current work carried out computational experiments to investigate how design parameters of swirlers influence their ability to resolve the three identified combustor issues.

Experiments were carried out on swirler designs using computational fluid dynamics (CFD) analyses with a sensitivity approach: Vane blade angles were varied and modelled in a fully defined combustor geometry based on a benchmark design. Various resulting flow properties (discussed in section 1.3) were observed to investigate the resulting performance. The APA 350 engine combustor was used in the current study. The APA 350 is a small scale, low cost and short life engine typically used in long range cruise missiles or high altitude drones. It features an axial compressor and turbine with a through flow annular combustion chamber operating a non-premixed flame. It has a nominal design thrust of 3.3\(kN\) and an air flow rate of 5.62\(kg/s\) (static sea level conditions). It was designed and built through an extensive research programme that was initiated in the early 80’s in South Africa, driven by the increased demand of gas turbine manufacturing and testing facilities as air transport and use of turbine based power plants increased. Original engineering drawings had been converted to CAD files and some of the original technical design reports were recovered.

The combustor and its swirler geometry are discussed in further detail in chapter 3. Parameterisation of the APA 350 engine was carried out to obtain boundary conditions for the CFD analyses.

The end objective of the study was to investigate the effect of swirler blade angle on the resulting fuel/air mixing properties as well as the combustion stability in the APA 350 combustor. This research could be used to design and optimise swirlers in future applications.

1.2 Document Structure

The remainder of this chapter is split into several sections. Section 1.3 introduces the topic of swirlers and provides an overview of the functions of swirlers in combustor designs. The significance of swirler operation is thus highlighted. In this subsection the problem statement is revealed, motivating the need for such a study. Following a summary discussing the research questions that were investigated, a conclusion is reached of what the study aimed to achieve in terms of quantitative data gathering and qualitative discussion. Finally to end this section, the planned methodology is summarised, discussing the intentional use of CAD and CFD to fulfil the study.

Section 1.4 gives an overview of combustors within the field of aerospace propulsion, reviewing types of combustors and their merits. Furthermore this section discusses special phenomenon such as blowout, flash back and acoustic instability which are common problems that are solved through the use of swirlers.

Finally section 1.5 examines the design of swirlers and how their parameters are characterised.

Chapter 2 is a summary of an extensive literature review that was performed as an independent study [33]. An overview of the findings is presented and related in particular
to the research questions posed in section 1.3. A discussion follows of how the findings were used to influence the methodology of the current study. The points raised in this discussion were also used in the final conclusions (chapter 6) during a reflection of the results obtained and how they compared to results from previous research. Chapter 2 goes on to discuss typical CFD approaches to investigate flow through combustors, highlighting fundamental concepts that were reviewed during the the literature study.

Chapter 3 contains comprehensive outlines of analysis procedures. In section two the APA 350 engine combustor is introduced in more detail. Here a parameterisation was carried out to obtain boundary conditions needed for CFD analysis, and the process of preparing the CAD model for flow analysis was described. The third section of chapter 3 introduces the software used for CFD analyses, FloEFD. A mesh independence study [32] is presented which was used to select appropriate mesh settings for the APA 350 geometry, and investigate the sensitivity of results based on the mesh used. The final section describes and justifies the analysis procedure.

Chapter 4 deals with the validation of the CFD turbulence model used in this study: A similar combustor featuring a single can and swirler was tested for flow parameters using FloEFD software, and the results were compared to data obtained from experimental methods in an independent Master’s thesis by Meyers [15].

Chapter 5 contains the results and discussions for the CFD analyses. Discussions relate to post-processing figures found in appendix C.

Finally chapter 6 features the conclusions of the study as well as recommendations for future or extended work.

1.3 Project Definition

Swirler Technology

Two important aero-mechanical considerations of reacting mixtures in sustained quality combustion are ensuring adequate mixing of reactants and the ability to anchor the flame for stable combustion. Swirlers have been introduced to gas turbine combustors for decades as a means achieve this. A swirler found inside a combustion chamber is usually housed in a construction known as the dome. Typically placed circumferentially at the entrance to the primary reaction zone, air is passed through angled vane blades analogous with air flowing over stator blades in a turbine. An angular momentum is imparted onto the flow causing the centrifugal force to expand the air in the radial direction [38]. Fuel is typically injected from the centre of the swirler and enters the core of the swirling flow. The high degree of swirl in the flow leads to excellent mixing of the fuel with the air so that a more homogeneous mixture of reactants is achieved. Furthermore a central recirculation zone results that produces a pool of active species, increased residence time as well as heat and mass transfer. Resulting high turbulence intensities lead to better atomisation of liquid fuel sprayed from injectors [12]. All these are known characteristics
that improve flame stability and combustion efficiency, which has been shown to reduce the production of NOx, increase heat release and burn fuel more efficiently [39].

**Figure 1.1:** Typical axial swirler

Swirler construction has traditionally consisted of a core and a shroud section, with the air flowing through the shroud needing to pass over blades fixed at a set angle that induces circumferential rotation to the flow. Low swirl designs feature a larger core and smaller shroud. In these designs the deliberately lower swirl intensities lead to an aerodynamic phenomenon known as flow divergence and prevent vortex breakdown which is a precursor of flow reversal and recirculation [38]. The principle of low swirl burners (LSB) is that the flow divergence causes decreasing axial velocity; thus the flame settles at an axial distance where the bulk flow velocity is equal and opposite to the flame speed [38]. High swirl designs on the other hand rely on the fact that at above a critical swirl number (0.4 according to Beer and Chigiér [22]), core vortex breakdown results in flow reversal [54]. Asymmetry of flow is more readily obtained for flows with high swirl than flows with intermediate to low swirl [35], although large radial pressure gradients are caused by highly rotating flow. It is found that flows with higher inlet swirl numbers have a less steep turbulent energy dissipation gradient along the axial direction [35]. High swirl often causes traverse acoustic oscillations whereas low swirl tends to prevail longitudinal modes of wave motion.

The above content highlights the importance of swirlers in combustor design. Despite their small contribution to the overall construction their importance cannot be underestimated. Swirlers featured in the APA 350 gas turbine engine were constructed (as with the rest of the engine) with limited manufacturing capability. In fact the originally flat bladed vanes were literally hand-filed to add a degree of curvature to their profile. For this reason there is a huge potential for improvement of these swirlers’ performance. An opportunity was presented to design an enhanced performing swirler for the APA 350 combustor and in the process of doing so investigate the factors influencing their performance.

The study aimed to investigate the effect of changing swirler blade angle on the mixing performance of reactants as well as examining the potential reduction of combustion instabilities and promoting successful flame anchoring. Furthermore pressure losses resulting due to excessively high swirl were investigated. The modification of blade angle was known to affect all of these factors [54] and tests were performed using CFD software called FloEFD to investigate this correlation. A cold flow CFD analysis was used to verify the mixing aspect in which air, as well as injected fuel was modelled.
Research Objective

Following an in-depth literature review several research questions were presented, sparking inquisition into what would be observed throughout the progression of the study.

Studies suggested a form of core vortex breakdown upon increasing swirl in the flow. It was unknown and therefore desirable to establish at what swirl intensity this vortex breakdown begins to occur. This would be assessed based on the swirl number (section 1.5). Accompanying this vortex breakdown should be the forming of a recirculation zone. One objective of the study was to confirm this, through the observation of an increasing adverse pressure gradient and thus a decreasing axial velocity. It should be also be possible during post processor to illustrate the flow velocity vectors as well as flow streamlines to demonstrate recirculation zones.

The turbulence was expected to increase with increasing swirl number. Therefore it was reasonable to assume the mean kinetic energy of the flow would increase too. Post processing and inspecting parameters such as turbulent kinetic energy (TKE) and turbulence intensity (TI) would give insight into this. More specifically the study hoped to plot the TKE on a 2D contour plot of the combustor to see where regions with higher TKE occur and if this is correlated to the influence of swirl. Vorticity and turbulence intensity (TI) were also be investigated to see if a potential recirculation zone affects their strength properties.

Within the context of combustion stability, the study hoped to reveal a correlation between swirl intensity and flow characteristics such as flame anchoring, blowout, flashback and turbulent fluctuations. Together with knowledge of the flame speed for typical fuels an assessment would be made of the likelihood for a particular swirler configuration to encourage or combat flame travel, in other words pass judgement on how well the flame would be anchored. It would then be investigated whether changing swirl intensity has an effect on this characteristic and if so with what correlation (is there an optimal swirl intensity that is most successful in anchoring the flame?). The research objectives listed above led to the formulation of specific investigative queries that would be answered through the progression of this study. The aim of the analyses were to answer those queries.

A CFD study made use of existing CAD geometry of the APA 350 combustor. CAD cleanup was carried out where necessary preceding the CFD solving. Post processing analyses of the combustor would reveal flow distribution and aerodynamic structures resulting from a particular swirler design. Several iterations of swirler configuration with varying blade angles of 30°, 40°, 50°, 60° and 70° were then used in CFD runs. Through this sensitivity approach an optimal swirler was realised that offers the best tradeoff between mixing quality, pressure recovery and flame anchoring. Parameters relating to the quantification of these aspects are discussed in greater detail in chapter 2.
1.4 Combustion and Burner Technology Overview

Basic Requirements

It is assumed that the reader is familiar with the concepts of how a combustor works. A summary of combustor types and their design requirements are however briefly communicated.

As the component in a gas turbine (GT) engine which converts chemical energy to thermal energy to produce useful power output, the combustor is an essential part of any GT. Combustors (or burners) also feature in other applications such as domestic boilers. The main objectives of combustors are:

- Produce heat for the expansion of gases that drive turbine blades
- Minimise stagnation pressure drop throughout the construction
- Cater for a wide range of operating conditions
- Maintain flame anchoring
- Ensure flame stability (avoid extinction)
- Be light, durable, easy to maintain and readily manufacturable.

Air leaves the compressor at very high pressure, temperature and velocity. Naturally these high velocities will cause blow out (defined in section 1.4.2) so the first feature in the combustor is the diffuser which slows the air entering the burners. Another reason to slow the air flow is that pressure losses are proportional to velocity squared. Since combustion reaction occurs in the gas phase, atomising injectors ensure liquid fuel drops are adequately fine so that evaporation can occur almost instantaneously upon the fuel entering the primary reaction zone. The use of radial or axial swirlers for admitting air encourages turbulence which amplifies the process of mixing in the reacting flow.

Combustor inlet conditions are dictated by Mach number and operating altitude; increasing altitude at a fixed Mach number decreases the inlet temperature, pressure and mass flow. Conversely increasing Mach number at a fixed altitude has the opposite effect. Reduced temperature and pressures make it difficult for sufficient heat release to sustain combustion, therefore recirculation zones are introduced to increase residence time. This can be done by introducing bluff bodies, backward facing steps or flow swirlers. An undesirable consequence of having recirculation zones is that the increased duration of exposure to high temperature encourages the formation of thermal NOx. Two ways in which this effect is combated is either by having an excessively high equivalence ratio which starves the reaction of oxygen (results in unburnt hydrocarbon and formation of carbon monoxide) or excessively low equivalence ratio which reduces the flame temperature. Modern design opts for the latter option since no fuel is wasted however operating at lean limits promotes combustion instabilities. Such lean combustion burners have led to the so-called “lean direct injection” (LDI) combustor types.

Significant differences between the design of conventional and LDI combustor are the fuel to air ratio (FAR) and the spacing between the fuel injector and ignitor tip: In a
conventional combustor, the distance between ignitor and fuel injection is small and there is a rich fuel mixture zone. Comparatively in a LDI configuration there is a large distance between fuel injection and ignitor tip location, and the FAR is comparatively lean. As a result of lean FAR as well as higher flame speeds, the latter configuration hinders the ability to easily carry out the relight process.

Another alternative design is the lean pre-mixed, pre-vaporised (LPP) combustor. These exhibit fuel/air mixtures at the limits of flammability (equivalence ratio of approximately 0.6) so have cold flames and hence low NOx. Good pre-mixing largely inhibits production of carbon monoxide and unburnt hydrocarbons (UHC), although the time required for vaporisation and mixing of fuel can exceed the autoignition delay time. Furthermore feedback of pressure and heat release limit the use of LPP combustors due to resulting thermo-acoustic instabilities.

Rich-burn, quick-quench, lean-burn (RQL) combustors are another promising concept. A primary combustion zone is rich so that low reaction temperatures and therefore minimal NOx are present. A secondary lean mixture provides sufficiently low exit temperatures as well as ensuring any remaining fuel is burnt. This gives RQL combustors superior ignition and lean-blowout performance.

1.4.1 Combustor Types

Returning to conventional combustor types several configurations can be found in engines:

Can

These are mechanically robust and allow for easy rig testing. Some advantages are high strength design (high slenderness ratio) giving good survivability characteristics. They can be more economically tested since only one can needs to be ignited. Similarly if one can fails during operation it is relatively cheap to replace just one can. Finally the FAR can be controlled more easily due to the confined nature of the combustion zone. Downsides however are that they are bulky and heavy, require inter-connectors, struggle with light around and demonstrate higher pressure losses (about 7% [37]) than those seen in annular or can-annular combustor configurations. Further downsides are that flow temperatures are usually less uniform entering the turbine leading to local ‘hotspots’ and fluctuating thermal loads.

Probably the most simple and commonly used type of combustor, the can combustors were the earliest type of design. Their robustness and mechanical simplicity made them popular for early aircraft however their many downsides soon led to development of improved combustor designs. Typically an engine could feature anything from 7 to 14 cans, spaced equally around the circumference. The very first part of the combustor (the snout) has a diverging nozzle shape to slow down the air. Some air then passes through a swirler into the primary zone and the remainder is fed around the flame tube within the air casting. There are typically three zones where further mass flow of air is admitted into a flame tube. They have several functions including acting as a thermal jacket to keep the combustor lining temperature down to a tolerable value. This is done by impinging cool air onto the metal surface as well as by forming an insulative film layer of cool air. Slits
are usually used to form this film layer. Another function of the extra air admitted is to physically 'blow' the combustion away from the flame liner walls. This air is admitted by circular or oval holes. The secondary and tertiary (dilution) air inlets are to lower the temperature of the combustion products to an acceptable value for entry into the turbine blades. They can however also be used to make alterations to the FAR such as is done in RQL combustor designs. As the holes are positioned further down the length of combustor, they increase in diameter so that a constant mass flow of air is admitted.

Typical velocities within a can combustor are approximately: 5 to 10 m/s in the primary zone (2500K), 25 to 75 m/s in secondary zone and 150 to 200 m/s in dilution zone [37].

Annular

Minimum specific weight, minimum surface area, easy light around and low pressure losses are some advantages of this type of combustor. Annular combustors are even more simple than can combustors, featuring one large circumferential vessel that acts as a flame tube wherein the combustion takes place, giving only about 5% pressure drop. They give highly sought after combustion efficiencies and effective fuel/air mixing. Complicated rig testing however is a big disadvantage. These combustor designs are scrutinised because of their even more difficult circumferential temperature distribution management and low strength. Perhaps the largest disadvantage of this combustor type is its difficulty to service and its price to replace. Despite this, annular combustors have grown in popularity in modern engines especially since there have been excellent progresses in the field of material science producing stronger designs than earlier annular combustors.

Can-Annular

As a hybrid of the above two types, can-annular combustors share advantages and disadvantages of both: Can-annular or canular combustors have a smaller cross section and therefore a larger thrust to weight and strength to weight ratio. This can be key in aircraft design due to the lower mass. Circumferential temperature management is more difficult although they are naturally good at demonstrating good temperature distribution at the entry to the turbine. Used on both turbofan and turbojet engines for combining advantages of can and annular combustors, can-annular combustors can unfortunately encounter high thermal expansion stresses due to the way the individual flame tubes are connected together. Compared to high can combustor pressure losses, can-annular combustors have about 6% pressure drop [37].

1.4.2 Combustion Difficulties

“A flame may be defined as a rapid chemical change occurring in a very thin fluid layer, involving steep gradients of temperature and species concentrations, and accompanied by luminescence” [20].

Without delving too deeply into combustion fundamentals a quick recap follows to clarify the definition of key ratios: A stoichiometric fuel to air ratio in a reaction is that at which
there is just enough air to ensure complete combustion of all fuel molecules and no fuel is left over. Depending on the FAR specific burning characteristics can be controlled such as the flame speed, flame temperature as well as by-products of combustion.

The equivalence ratio is a measure of how near to stoichiometry a certain reaction is occurring:

\[ \Phi = \frac{FAR}{FAR_{stoch}} \]  

Equivalence ratios below unity signify lean combustion and above unity signify rich combustion. Combustion only occurs under certain required conditions, specific to the reactant mixture in question. The conditions for which successful deflagration occurs are dictated by upper and lower flammability limits. These depend on equivalence ratios, temperature and pressure. Flammability limits can be widened by increasing pressure as well as temperature, with less effect from the latter. This is especially true for hydrocarbon fuels. The lowest (lean) limit of equivalence ratio for successful combustion in most fuels is approximately 0.5 with the highest (rich) limit being about 3 [20]. Similarly the lowest temperature is that at which the fuel’s vapour pressure allows the lean limit volume concentration to be formed (known as the flashpoint) and the upper temperature limit that at which formation of the rich limit volume concentration occurs [20].

Deflagration waves in hydrocarbons usually occur at velocities below 1m/s and it is widely accepted that the most important factors affecting this property are pressure, temperature and equivalence ratio. According to Lefebvre and Ballal the equivalence ratio and temperature have approximately equal effects on flame speeds, with the maximum flame speed for most fuels usually occurring at an equivalence ratio between 1.05 and 1.10 [20]. Although operating at lean limits tends to slow the flame speed, secondary effects such as high temperature, pressure and turbulence intensity conversely increase flame speed.

Two phenomenon associated with combustion instability that are important to this study are “flashback” and “blowout”. These are briefly discussed:

**Flashback**

Flashback occurs mainly in premixed systems but can also occur in non-premixed systems under certain circumstances. Essentially it arises when the flame speeds exceed the flow approach velocity so that the flame is able to travel ‘upstream’. There are three main scenarios where flashback occurs as identified by [20]:

1. In the free stream when the bulk flow velocity is too slow
2. In the boundary layer where wall shear stresses decreases velocity to almost zero
3. Due to a central toroidal recirculation zone (CTRZ) resulting in combustion induced vortex breakdown (CIVB) and hence axial velocity reduction
The third scenario is most prevalent in non-premixed flames such as in the burner of the APA 350 combustor and therefore is of most interest in the current study. In low swirl burners where advantage is taken of aerodynamic flow divergence, CIVB is not significant and therefore there exists a distinct axial velocity gradient. Hence the flame kernel can travel upstream to an axial position where the increasing approach velocity equals the flame speed and hence anchors the flame. In high swirl burners however, CIVB is significant and a more homogenous velocity distribution in the axial direction is seen. Therefore given conditions which allow the flame speed to exceed the approach velocity, the flame front will effectively travel backwards and attach to the point directly downstream of the swirler where the fuel and air first meet. This is highly undesirable because the fuel and air will not have had sufficient mixing chance to ensure the desired equivalence ratio is achieved.

**Blowout**

Flame Blowout has the opposite effect to flashback: It occurs when the rate of heat release is insufficient to heat up the incoming fresh mixture to the required flashpoint temperature [20]. Of course the rate of heat release also needs to account for rapid evaporation of liquid fuel droplets. Larger droplets require more time to vaporise hence finer atomisation of droplets is one method of reducing this ‘combustion delay’. More effective atomisation is directly achieved by creating higher shear in the flow through increased turbulence intensity. This highlights another useful function of flow swirlers in combustors. In summary, finer atomisation and increased residence time through recirculation (both attributed to higher swirling flows) can prevent blowout occurrence. The sometimes loosely used term “flame anchoring” refers to maintaining a steady flame position, avoiding upstream or downstream movement caused by flashback and blowout respectively.

### 1.5 Swirler Parameterisation

One of the most important considerations when designing an axial swirler is that the necessary mass flow rate admitted be done at an acceptable swirler pressure loss, \( \delta P_{sw} \). The Aeronautical Research Council of Great Britain [14] defines the mass flow rate through the swirler as:

\[
\dot{m}_{sw} = \left[ \frac{2\rho_3 \Delta P_{sw}}{k_{sw} \left( \frac{\sec \theta}{A_{sw}} \right)^2 - \frac{1}{A_L^2}} \right]^{0.5}
\]

(1.2)

\[
A_{sw} = \left( \frac{\pi}{4} \right) \left( D_{sw}^2 - D_{hub}^2 \right) - 0.5n_{\nu}t_{\nu} \left( D_{sw} - D_{hub} \right)
\]

(1.3)

where

\[
\Delta P_{sw} \sim \Delta P_L
\]

\( k_{sw} \) is a constant with value 1.3 for flat vanes and 1.15 for curved vanes. \( \Delta P_{sw} \) should be no more than 3-4% of \( P_3 \). The number of vanes should be 8-16 and the blade thickness should be \( 0.7 - 1.5 \text{mm} \) [20].
When designing a swirler the first characteristic that is decided is the hub diameter, dictated by the size of the fuel injector. Equations 1.2 to 1.3 are then used to calculate the outer diameter necessary for the required mass flow rate.

Swirlers are usually characterised by a non-dimensional quantity called a swirl number. The aerodynamic swirl number is in essence the ratio of axial flux of the angular momentum to the axial flux of the axial momentum. Many variations of formula have been used [36] [11], for example Ribero et al [26] simplify the equation to:

\[
S_a = \frac{\int_{R_1}^{R_0} \rho U_z U_\theta 2\pi r^2 dr}{\int_{R_1}^{R_0} \rho \left( U_z^2 - \frac{U_\theta^2}{2} \right) 2\pi R_0 r dr} \tag{1.4}
\]

The problem with the aerodynamic swirl number is that it would need to be explicitly calculated at every single spatial point where it is required. For this reason a geometric swirl number also exists which accounts for the entire flow domain [50]:

\[
S_g = \frac{R_0 \pi r_e \left( \frac{m_\theta}{\dot{m}_{tot}} \right)^2}{A_t \left( \frac{\dot{m}_{tot}}{\dot{m}_{tot}} \right)^2} \tag{1.5}
\]

Another definition of geometrical swirl number is [49]:

\[
S_g = \frac{\pi \times r_e \times r_i}{A_e} \tag{1.6}
\]

Perhaps the most convenient version of a formula for the swirl number is demonstrated by Beer and Chigier [22]:
\[ S_N = \frac{2}{3} \left[ 1 - \left( \frac{D_{hub}}{D_{sw}} \right)^3 \right] \tan \theta \] (1.7)

Since it is solely expressed in terms of the swirler geometry and not dependent on the flow conditions [22].

According to this definition Beer and Chigier found that for a swirl number below 0.4 no flow recirculation is seen and that for swirl to become significant a swirl number of at least 0.6 is required. This is useful to know as it allows the designer to determine the size of swirler required according to desired swirl number.
2.1 Chapter Overview

This chapter presents work obtained from previous studies in the field of optimising swirlers for improving mixing quality and combating combustion instability problems. It refers to work carried out in a separate literature review [33] that identified prominent issues in dealing with these combustion difficulties and looked at ways of overcoming them. The chapter closely relates to the research objectives section in chapter 1 and highlights some of the answers that previous researchers have uncovered to research questions similar to those posed for this study. The literature review provided an overview of aspects of combustion stability and fuel/air mixing performance. It described factors associating the parameterisation of swirlers as well as addressed important design considerations.

Since one of the main purposes of the literature study was to build on knowledge of all aspects involved in the study, some fundamental (but limited) theory is included in the first part of this chapter that is considered paramount pre-requisite knowledge for the content that is to follow in later sections.

Later in the chapter recommendations are explored for carrying out computational analyses for cold flow and combustion simulations with a review of the various turbulence models that can be selected for such purposes.

In summary the most state of the art research by influential figures such as Syred, Beer, Gupta, Chigier and Lefebvre were examined and a well rounded knowledge base of understanding the concepts relating to the study was gained.

2.2 Fundamental Theory of Turbulent Flows

2.2.1 Velocity Fluctuations

Small eddys create turbulent fluctuation in velocity in the longitudinal direction that comprises of two components, \( u = \bar{u} + u' \). \( \bar{u} \) is the time average component and \( u' \) is the
fluctuating component. Hence for laminar flow it is found that \( u = \bar{u} \) for all values of time, \( t \) because there are no fluctuating components. Expressing the velocities in these constituent components is often referred to as Reynolds decomposition.

To determine the mean velocity, one must integrate over a time interval from \( t \) to \( t + T \) where \( T \) is much longer than any turbulence time scale, but smaller than the timescale for unsteadiness in the flow such as tidal fluctuation for example.

\[
\bar{u} = \frac{1}{N} \sum_{1}^{N} u_i
\]  

(2.1)

Turbulent fluctuation:

- \( u'(t) = u(t) - \bar{u} \) continuous

- \( u_i'= u_i - \bar{u} \) discrete

Turbulence strength (continuous):

\[
u_{RMS} = \sqrt{\bar{u}'^2(t)}
\]  

(2.2)

Turbulent strength (discrete):

\[
u_{RMS} = \sqrt{\frac{1}{N} \sum_{1}^{N} (u_i'^2)}
\]  

(2.3)

\[
\text{Turbulence intensity} = \frac{\text{Turbulence strength}}{\bar{u}}
\]  

(2.4)
2.2 Fundamental Theory of Turbulent Flows

2.2.2 Turbulent Flow Parameters

Turbulent kinetic energy is the mass specific kinetic energy associated with eddies in turbulent flow. Physically the property can be expressed by the root mean squared value (RMS) of the velocity fluctuation. The closure method is used in Reynolds Averaged Navier Stokes equations to quantify TKE as the mean of the turbulence normal stresses, shown in equation 2.5 adapted from [3]:

\[ TKE = \frac{1}{2} \left( \langle \bar{u}'_1^2 \rangle + \langle \bar{u}'_2^2 \rangle + \langle \bar{u}'_3^2 \rangle \right) \]  

(2.5)

TKE is produced by low frequency (integral scale) eddies through fluid shear or other external forces such as friction. TKE is transferred down a turbulent energy cascade, being dissipated down to Kolmogorov scales by viscous forces. It is essentially a measure of turbulence, with turbulence intensity (TI) being directly proportional to TKE. TKE and TI are associated with the quantities pertaining to rectilinear kinetic energy. Vorticity however is a pseudo-vector field that describes local spinning motion. The mathematical definition of vorticity for 3D flow is given by:

\[ \vec{\omega} = \nabla \times \vec{V} = \left( \frac{\partial V_z}{\partial y} - \frac{\partial V_y}{\partial z}, \frac{\partial V_x}{\partial z} - \frac{\partial V_z}{\partial x}, \frac{\partial V_y}{\partial x} - \frac{\partial V_x}{\partial y} \right) \]  

(2.6)

Figure 2.3: Illustration of vorticity [6]
2.2.3 Length Scales

The largest eddys given by $L$ in the integral length scale account for most of the transport of momentum and energy. The smallest eddys given by $\eta$ are found in the Kolmogorov scale. As eddys approach smaller scales viscous forces become more important. The smallest length scales are those at which kinetic energy is dissipated into heat.

From Kolmogorov’s first hypothesis, the only factors influencing the behaviour of the small scale motions are the overall kinetic energy production rate (which equals the small scale dissipation rate) and viscosity. Furthermore dissipation rate itself is not directly dependent on viscosity however the scales at which the energy is dissipated does depend on both dissipation rate and viscosity [31].

The Kolmogorov scale, being the smallest possible hydrodynamics scale in turbulent flow is defined by [31]:

$$\eta = \left\{ \frac{\nu^3}{\epsilon} \right\}^{\frac{1}{4}} \quad (2.7)$$

It is more practical to express the dissipation rate in terms of large scale flow features. Since the dissipation rate equals the kinetic energy (KE) production rate it is simply a matter of determining the rate at which KE is added to the small scales; KE is defined as $KE = \frac{1}{2}mU^2$ so clearly KE is proportional to $U^2$ (the bulk flow velocity squared). Now by defining the time scale for large eddys (turnover time) as $\tau = \frac{L}{U}$ it is reasonable to assume the KE supply rate is related to the inverse of the time scale, i.e. $\frac{U}{L}$.

The dissipation rate is thus estimated by

$$\epsilon \sim \frac{UUL}{\nu} \sim \frac{U^3}{L} \quad (2.8)$$

hence

$$\eta = \left\{ \frac{\nu^3L}{U^3} \right\}^{\frac{1}{4}} \quad (2.9)$$

Thus the ratio of the largest to smallest length scales in the flow is [31]:

$$\frac{L}{\eta} \sim \left\{ \frac{UL}{\nu} \right\}^{\frac{1}{2}} = Re^{\frac{3}{4}} \quad (2.10)$$

Notice the term for the dissipation in equation 2.8 is not dependent on viscosity as mentioned earlier.

Another length scale often used is the Taylor microscale, $\lambda$ defined by

$$\left( \frac{\partial u'}{\partial x} \right)^2 = \frac{u'^2}{\lambda^2} \quad (2.11)$$
2.2 Fundamental Theory of Turbulent Flows

where \( u' \) is the RMS of the fluctuating velocity field [31]. \( \lambda \) is useful for providing an estimate of the fluctuating strain rate field. \( \lambda \) is also sometimes known as the turbulence length scale, giving rise to the turbulence scale Reynolds number

\[
Re_\lambda = \frac{u' \lambda}{\nu}
\]

(2.12)

2.2.4 Time Scales

The time scales for large eddys associated with integral length scales are referred to as “eddy turnover time”, given by:

\[
t_L = \frac{L}{u}
\]

(2.13)

The timescale for small eddys associated with the Kolmogorov scale is given by

\[
t_\eta = \left(\frac{\nu}{\epsilon}\right)^{\frac{1}{3}} = \left(\frac{\nu L}{u^3}\right)
\]

(2.14)

Time scales for large eddys are much, much larger than those for small eddys. From the above two equations, the ratio of time scales is then

\[
\left(\frac{t_L}{t_\eta}\right) = \left(\frac{uL}{\nu}\right)^{\frac{1}{3}} = Re^{\frac{1}{2}} L
\]

(2.15)

The turnover time is the time taken for a large integral scale eddy to reduce down to the smallest kolomogorov scale eddy. The following explanation is taken from a lecture by the University of Utah [31]: "The turnover of a size l eddy \( \tau_l \) is the time taken to traverse the inertial range \( tl \). From dimensional analysis:

\[
\frac{dl}{dt} \sim \frac{l}{t_l}
\]

(2.16)

The characteristic time scale of a size l eddy is related to the large eddy turnover time \( \tau_L \) using Kolmogorov scalings applicable to the inertial range

\[
t_l \sim \left(\frac{l}{L}\right)^{\frac{2}{3}} \tau_L
\]

Now substituting this expression back into equation 2.16 and integrating from the integral scale to the Kolmogorov scale gives the time it takes for an integral scale eddy to be reduced to a Kolmogorov scale eddy:

\[
\frac{tl}{\tau_L} = 1 - \left(\frac{\eta}{L}\right)^{\frac{2}{3}} = 1 - Re^{\frac{1}{2}}
\]

(2.17)
where

\[ \frac{L}{\eta} \sim Re^{\frac{3}{4}} \]

for the inertial range

Equation 2.17 demonstrates the dependance of Reynolds number on the process, where the dependency disappears for high Re flows” [31].

### 2.3 Research Findings

**Mixing**

Swirlers introduce an angular momentum into the fluid flow entering the primary burn region. Thus large scale unsteady motion arising from shear layer instability and vortex breakdown influences the heat release by influencing pressure conditions [54]. Directly behind the centre body large coherent structures are formed through aerodynamic mechanisms such as vortex shedding. These large structures cause the majority of turbulent momentum transport and mass and heat transfer making them very significant to the combustion dynamics. Geometry as well as flow regimes play a big role in determining these coherent structures [49]. Vigueras-Zuniga et al report that “Recent studies performed by Bouremel et al [53] attribute the excellent mixing to the interaction between large structures especially on their boundaries where major exchange of mass and energy occurs” [49].

When the rotational forces become large enough, critical breakdown of the vortex structure causes a reduction in axial velocity and therefore increase in pressure. Studies have shown that this adverse pressure gradient caused by high swirl intensity can lead to flow reversal and vortex breakdown [36]. The disturbance in vortex moves upstream and merges with the wake recirculation zone produced by the centre body [54]. As observed by Cheng this results in a “fully developed central toroidal recirculation zone” or sometimes referred to as a “toroidal vortex core” (TVC) [38]. Further studies [21][30] demonstrate the vortex instability initiating with a small closed bubble which returns momentarily to stable flow before breaking down again. This fluid dynamic phenomenon is usually followed by a second breakdown taking the form of either an axisymmetric, spiral or double helix structure [45].

According to Syred and Beér the Re number may also be useful in defining the onset of vortex breakdown which has to be taken into account during the design of combustors [29]. Apparently for low swirl numbers vortex breakdown will only occur once a certain Re number has been reached. In the CTRZ flows are associated with higher shear rates and strong turbulence intensities resulting from the vortex breakdown [54]. Increasing turbulence intensity leads to better atomisation of liquid fuel sprays [12] which in turn lead to more complete combustion. Higher turbulence intensity furthermore increases flame speed, which has the effect of decreasing the length of flame (resulting in shorter more compact combustor designs) but no effect on heat release [54].

Second to the CTRZ another form of large coherent structure known as a precessing vortex core (PVC). O’ Doherty et al [44] suggested that ”the appearance of vortices such as the PVC improve the fuel/air mixing because of the creation of large turbulent scales
2.3 Research Findings

which translate through into the dissipation range of the energy cascade”. They can be excited or damped merely by altering the mode of fuel entry [27][28].

Much work [29] [26] [54] [38] has been carried out in past research to study the shape and characteristics of the recirculation zone arising from vortex breakdown. Some studies have suggested that as swirl number is increased, the length of the recirculation zone increases until a maximum length of $5 \times D_e$ is reached at a swirl number of about 1.5 [29]. Syred further suggests that only after a swirl number of approximately 2 does the shape of the recirculation zone start to contract in length and broaden its radius substantially.

Instability

Thermo-Acoustic Coupling

Combustion is a broadband noise, but a Fourier transform of instabilities shows that they occur at crisp definitive frequencies. According to Huang and Yang “swirl number has little influence on the frequencies of the acoustic oscillations but plays a dominant role in determining the amplitudes of wave motions” [54]. Furthermore they state that “according to Bradley et al, instabilities in swirl combustors are explained as a periodic extinction of the flame produced in the outer recirculation zone by fluctuations in the stretch factor due to periodic vortex shedding in this area” [16]. Further factors that can cause increased instability include entropy waves in inlet nozzle causing pressure oscillation feedback, as well as the fact that the precession of a vortex core can couple acoustically with the chamber to produce resonance [54]. Huang and Yang also investigated Azimuthal instability modes occurring in annular combustors [34]. According to Stow and Dowling [41], Krebs et al [52] and Schuermans et al [13] azimuthal modes are usually the strongest acoustic mode for annular combustors, appearing as standing waves for low oscillatory amplitudes, with a possibility of rotating mode for high oscillatory amplitudes [34].

Several theories on how to suppress thermo-acoustic coupled oscillations that cause instability have been proposed:

- Eliminate the coupling between combustion and acoustics (i.e. so that they are not in phase)
- Change timescales by changing geometrical dimensions, e.g. increasing combustion chamber length, diameter of injector nozzles, alter injection velocities, etc
- Introduce passive damping such as acoustic lining to absorb pressure waves
- Disturb time delays such as those demonstrated by Lieuwen [43]
- Use anti-sound to suppress waves (active control) [23]
- Use secondary fuel inlet with designed pulses to control heat release oscillations
Flame Anchoring

Not all combustion instability is due to thermo-acoustic coupling. As touched upon in chapter 1, flashback and blowout instabilities also present serious challenges for engineers. The literature study [33] identified that these two issues motivating the intention of so-called 'flame holding' are primarily a derivative of the mixing and flow dynamics which are dictated by aspects covered in mixing considerations. Being disconnected from the duct acoustics, the ability to anchor the flame comes down to providing the correct recirculating zone for the flow mixture. For example Cheng suggests the use of low swirl burners results in the flow diverging due to centrifugal forces, causing decreasing axial velocity; thus flame settles at an axial distance where the flow velocity is equal and opposite to the flame speed [38].

Swirler Design

Much attention has been given to the design considerations for swirler geometry and configuration. It is found that flat vanes (typically angled between 30° and 70°) are clearly easier and cheaper to manufacture. Profile vaned swirlers however have much smaller pressure losses attributed to them (straight vaned swirlers only exhibit low pressure losses at low swirl numbers) with loss coefficients being approximately halved [10][51]. Typical pressure loss coefficients are given in figure 2.4 where the line with black diamond markers represents an axial, flat-bladed swirler such as used in the current study.

![Figure 2.4: Effect of swirl number upon pressure loss coefficient. Taken from Syred N and Beer [29]](image)

Improved performance by profiled blades is due to a Rankine tangential velocity distribution [29] as apposed to a forced vortex tangential distribution such as that produced
by flat vanes [22]. Due to their in-efficiencies, some studies [25] [29] suggest that the use of flat vaned swirlers be limited to combustors featuring low swirl since their efficiency deteriorates for high swirl.

2.4 State of the Art Research

As a summary of this literature review, a concluding overview of previous research relating to the effect of swirl number on global flow patterns, mixing and stability is reviewed. Mestre compares swirling combustion with non-swirling combustion and demonstrates that the existence of swirl helps increase combustion efficiency, decrease pollutants and increase flame temperature [8]. Work by Tangirala et al finds that mixing and flame stability is increased up to a swirl number of approximately unity after which turbulence level and flame stability actually decrease [48]. Broda et al [42] and Seo [40] find from experimental studies on combustion of lean-premixed swirl stabilized combustor that the “dominant acoustic motion corresponds with the first longitudinal mode of the combustor” [54]. As swirler intensity is increased, above a certain critical swirl number, a vortex-breakdown-induced CTRZ emerges. As swirl number increases further, the recirculation zone moves further upstream, accompanied by an increasing adverse pressure gradient in the positive axial direction. This adverse gradient results in further flow reversal. Eventually the recirculation zone approaches the annulus region, introducing a risk of flashback [54]. Syred and Beér found that for swirl numbers up to 0.6 a satisfactory lower limit of Re number required to maintain operation outside the region of initial vortex breakdown is 18,000. Any value lower than this would lead to small perturbations of Re number causing large disruption to the recirculation and mixing zone and therefore alter the flame characteristics [29].

Huang and Yang found through interpreting TKE distributions that a high turbulent intensity region develops downstream of the center-body and backward facing steps where shear layers formed between incoming and recirculating flows produced large velocity gradients and high fluctuating flow [54].

2.5 Recommendations for Computational Analysis

Carrying out computational investigations rather than experimental methods can be a cheaper and more time saving approach. A brief introduction to CFD turbulence models follows which were investigated during the literature review:

Several types of turbulent models have been created and used in the past:

- (RANS) Reynolds Averaged Navier Stokes Simulation
- (DNS) Direct Numerical Simulation
- (LES) Large Eddy Simulation
• (RSTM) Reynolds Stress Turbulence Model
• \((K - \epsilon)\) model

RANS models are time averaged and make use of the Reynolds decomposition principle discussed in section 2.2.1. It is found that a RANS approach can cause an excess of turbulence dissipation. It is regarded as insufficient for complex flow fields where swirl introduces factors such as flow separation, wall effects and recirculation zones [17].

Two time-dependent numerical simulation techniques are DNS and LES. DNS resolves all of the turbulent scales from the largest integral length scale down to the smallest Kolmogorov scale where energy is dissipated whereas the LES model however considers only the largest, most energetic scales for calculation. The influence of the smaller scales with LES are modelled through a typical viscosity approach. DNS is exact, therefore not requiring to model the turbulent flow. Due to resolving all the turbulence DNS models are hugely demanding of computer memory and are limited to low Reynold’s number flows. LES on the other hand is less computationally demanding, making use of the Navier Stokes equations to reduce the range of length scales in the solution. Users need to ensure a very fine mesh near the wall in order to capture the most ‘energetic region’ close to the wall [18].

Two remaining turbulence models are RSTM and \((K - \epsilon)\). RSTM is considered more simplistic than \((K - \epsilon)\) however it is not as widely validated as \((K - \epsilon)\) models and is much more computationally expensive. Despite this, studies by Yang et al indicate that Reynolds Stress Turbulent Model (RSTM) does a better job than the \((K - \epsilon)\) model at resolving recirculation zones with high velocity gradients [24]. For solving complex swirling flows that give rise to flow separation and strong anisotropy, RSTM models with higher order are preferred [24]. RSTM models naturally resolve instantaneous strain rate changes, flow rotations and general flow anisotropy due to the fact that each Reynolds stress component has its own transport equations. Furthermore RSTM models source terms do not need to be modelled since they are exact [24].

\((K - \epsilon)\) models have grown in popularity for use in high turbulence models due to ease of use and relatively low computational costs. Because of its isotropic nature however the \((K - \epsilon)\) code does tend to produce inconsistent and diffusive results [24].

Perisetty et al performed computational studies on a commercial burner using \((K - \epsilon)\) turbulence models and Reynolds Stress Turbulence Models (RSTM) [36][29]. Difficulties in solving swirling flow are experienced due to large degree of coupling between momentum equations, when the rotational components are large. Special strategies had to be developed to cope with the solution instabilities that resulted. Strategies included but were not limited to

• Suitably refining mesh to deal with very large flow gradients
• Starting initially with small rotational and axial velocities and gradually increasing velocity
• Using a \((K - \epsilon)\) model to obtain initial conditions and the RSTM model for actual swirl flow solving.
It was observed that below a critical swirl number of 1 no vortex formed, hence the axial velocity was always positive (no recirculation). Above 1 a vortex formed, which remained constant with further increase in swirl number, a swirl number of 1.38 was reached at which point the vortex grew suddenly [36].

Due to the turbulent nature of reactive flow being so computationally expensive, most studies [7][9] in the past have used cold flow simulations to try and evaluate ignition probability and sustained combustion by observing local flow characteristics as well as global flow patterns. Issues with this are that the transient nature of the flame kernel are not considered, further to the neglecting of significant effects that the heat release has on the flow properties and behaviour.

As far as investigation combustion stability goes, pressure fluctuations can be computed as a function of time if a time-dependent CFD code is used by making use of the Poisson equation either in differential form or integral form for incompressible flow. For compressible flow however the Navier stokes equations have to be solved simultaneously with the energy equations. Post-processing will give access to all of the necessary statistical quantities needed to calculate the solution. Even with advances in codes such as RANS, due to intrinsic limitations involved, previous researchers have attempted to calculate wall pressure fluctuations directly from time-dependent simulations. The number of turbulent scales which need to be computed make it extremely difficult which is why many researchers assume fully developed channel flow rather than consider the boundary layer. This massively reduces computational demand.
Chapter 3

Analytical Methods

3.1 Analysis

This chapter introduces two main aspects of the methodology by which this study was conducted; namely a presentation of the APA 350 turbojet engine combustor which is used as a benchmark combustor in this test, and an introduction to FloEFD, the CFD software used for flow simulation.

Next a mesh independence study [32] will be discussed briefly in which extensive work was done to optimise a mesh grid and solver settings for the APA 350 combustor. This was done in order to prepare the model in the current study for computational fluid dynamic simulation and ensure accurate results that were not mesh sensitive. A further prerequisite to the actual CFD analysis was preparation of the CAD model in a process commonly referred to as "CAD cleanup". This will also be discussed in this chapter. Furthermore following a more comprehensive introduction to the APA 350 combustor and it’s design specifications a parameterisation study was performed. This was done in order to acquire technical data pertaining to its performance, boundary conditions and initial operating conditions; all required by the CFD solver in order to calculate a flow solution. The chapter concludes with a description of the procedure to set up the model for flow analysis.

3.2 APA 350 Engine and Model Specification

An introduction to the APA 350 combustor geometry is discussed firstly, after which a thermodynamic parameterisation is reviewed, followed by a discussion on the CFD prerequisite work to prepare the CAD geometry for the CFD analysis.

Figure 3.1 shows a cross section of the APA 350 combustor assembly. As pictured, the flow enters from the left side. The spool shaft has been removed for clarity of the internal components. A diffuser slows the air entering the combustor. Thirteen air swirlers are
placed radially on the domed swirler housing (pictured in green). Each swirler assembly features a fuel injector.

![Figure 3.1: Cross section of combustor](image)

As seen in figures 3.7a and 3.8a, a complex set of cavities and channels allows the fuel to be ejected as a vaporised volume at a cone angle of 60° from the center of the swirler assembly. Meanwhile the cold mixing air passes through the swirler blades where it receives and angular momentum which is instantly passed onto the fuel molecules. The fuel/air mixture is then ignited by a torch ignitor in the central combustion zone. An inner and outer flame tube cone serves to accelerate the diffusion air into the combustion zone somewhat further down the length (figure 3.9). The purpose of the circular and oval ‘punched’ holes is to provide a nozzle effect that will admit the diffusion air into the combustion with higher velocity than would occur with plane holes. Air then travels further down the combustion chamber before exiting through a convergent nozzle.

### 3.2.1 Design Specification

The APA 350 Engine features an axial compressor, through-flow, annular combustion chamber, a non-premixed burner, an axial turbine, an exhaust nozzle and some additional units such as fuel pumps and alternators. Operating between $-30^\circ C$ and $+65^\circ C$ and up to an altitude of 10,000m the engine was designed for a nominal sea level thrust of $3.3kN$. The flow swirlers currently featured in the APA 350 combustor have vane blades angled at 48°.

### 3.2.2 Parameterisation

In preparing for CFD studies on the APA 350 combustor boundary conditions for the computational domain had to be calculated. The FloEFD solver required a static pressure to be specified at the exit. At the inlet the fluid mass flow rates as well as static temperature had to be specified. A pressure is preferred for an exit boundary condition instead of using mass flow again because a pressure allows the solver to dictate the velocity vectors of the flow due to the “back-pressure” whereas simply stating a mass flow
would result in the flow being assumed completely axial. Furthermore by placing a lid at the exit to create a fluid volume, FloEFD recognised this lid as a physical boundary and hence was inclined to create small recirculation zones at these points. Therefore the geometry was extended with a constant cross sectional cylindrical tube. Although a cold flow CFD simulation was carried out, pressure calculations still had to take account of the contribution to pressure rise of fuel combustion. Therefore the exit pressure specified was that from test reports[4] of the APA 350 combustor for normal combustion operation. The following engine parameters were obtained from archived design reports for the APA 350 combustor and applied to the CFD model:

**Ambient Conditions**

Static ambient temperature was taken as 288.15\(^{\circ}\)K corresponding to sea level altitude.

**Inlet**

Based on design reports for the APA 350 engine [47] the design mass flow is 5.620\(\text{kg/s}\) but since it also features 2% bleed, a mass flow rate of 5.5\(\text{kg/s}\) was selected. The inlet nozzle efficiency was obtained from [4] and resulted in a 5% pressure drop.

**Compressor**

All of the relevant compressor parameters were obtainable from references [47][46] in which a rotor speed of 27800\(RPM\) is specified. A nominal pressure ratio of 3.56 was found from compressor maps and tables in [47]. To correspond with combustor inlet pressure ratios used in early rig tests, an effective compressor pressure ratio of 2.75 was selected (low power setting) for use in the CFD analysis.

**Combustor**

For the fuel a standard blend of Jet A-1 fuel was selected with a lower heating value of 43,031\(kJ/kg\). The specific heat of the fuel was taken as 2,093\(J/kgK\) at a temperature of 298.15\(K\). The design reports also specify a fuel mass flow rate of 0.1375\(kg/s\). Inlet total pressure for the combustor was taken as 2.79\(bar\) and the static exit pressure (needed as a boundary condition for CFD) was taken as 2.4565\(bar\) [4].

### 3.2.3 CAD Cleanup

It is not uncommon to have to dedicate quite some effort in preparing a CAD model for finite element analysis (FEA) or CFD simulations due to errors in the geometry. Some of these errors are physical topographical errors made during the drawing or issues arising when switching between CAD formats such as gaps created in the model structure. This subsection identifies the errors found in the CAD model for the APA 350 combustor and demonstrates how the errors were corrected.
The original APA 350 engine drawings were produced before 3D modelling capabilities were freely available for most consumers. Only recently have the drawings been copied and redrawn in the modern CAD package Solid Works 2014 by draughtsmen at the Council for Scientific and Industrial Research (CSIR) in Pretoria. Since this software package was not obtainable to the author the CAD manipulation and any further work on the geometry was carried out in PTC Creo 3.0. This software has the ability to open up geometry files from most modern CAE packages, as well as the standard STEP, IGES and Parasolid files.

The figures below show the original CAD geometry as obtained from the CSIR. The isometric, axial front and axial rear views are shown respectively. Furthermore figure 3.3 illustrates some of the features of the combustor. For clarity the shaft is not shown:

![Figure 3.2: APA 350 combustor](image)

Due to a combination of factors the original CAD geometry was not ready for the CFD analysis as is, but required as usual with these types of analyses some "clean-up" to prepare the model. Geometry preparation can account for up to 80% of the overall time of the CAE process. It is usually tedious but necessary in order to obtain a closed fluid volume.

Specific examples of issues that are traditionally associated with CAD geometry in order to carry out CFD applications are discussed next.

**External Components**

The first step was to remove any unnecessary parts and features on the model. This involved any external parts which were irrelevant for the internal fluid study of the combustor. In particular this included the torch igniter and thermocouple assemblies shown in Figure 3.3a. Since the flow analysis was carried out on the internal fluid volume external components were irrelevant in the study and could therefore be removed.
3.2 APA 350 Engine and Model Specification

Figure 3.3: Breakdown of Components

(a) External parts

(b) Internal parts
Model Simplification

The model had to be simplified as much as possible without removing geometrical properties essential to the fluid study. For this reason, any voids and holes that were in present in solids (for example those made for bolts or pins to fit into) were filled in. An example of such a case can be seen in the intake strut components shown in Figure 3.3b. These struts serve the aerodynamic purpose of implementing a degree of axial direction to flow leaving the compressor. The holes shown in transparent view are used for mounting purposes and hence do not experience the air flow into combustor passing over the strut body. Most CFD and meshing packages will still try to produce a mesh over the surface of the inside of the hole since it does indeed interpret it as a “surface”. This unnecessary meshing leads to excessive time and computational requirements that are discussed in detail in [32]. Hence, the holes in the struts were filled in so that the entire strut was represented by a single solid part.

Another example of simplifying geometry to improve the mesh effectiveness and relevance was for the front and rear barrel flanges. A view of the rear flange in Figure 3.4a shows its original geometry, featuring 48 location holes which are used for assembly and structural fastening purposes. These were removed to give the resulting flange in Figure 3.4b. Furthermore Figure 3.5a shows the original flange cross section which was again concluded to be unnecessarily complex. The edges were irrelevant to the results of air flow. The cross section was therefore modified to that shown in Figure 3.5b.

Furthermore the surfaces of the outer barrel and two end flanges were booleaned together to achieve the surface continuity that is required by the mesh.

Overlapping Geometry

Overlapping geometry or intersecting solids needed to be mended. This is because even though the overlapping solids would in physicality mean that they are one solid object, a
solver or mesh builder may interpret them as two distinct solids, each featuring its own surfaces. Therefore once again unnecessary mesh will be constructed. The solution to this problem was to merge the solids together so that the mesh builder recognised them as one part.

**Other CAD Errors**

Figure 3.6 is a good example of topographical CAD errors that were inherited from original models drawn. Despite the torch igniter assembly being removed in any case, it can clearly be seen how there was a large gap between the barrel and the fastening point of the igniter assembly. These sorts of gaps were found in numerous places on the model and had to be filled in. This is either done by extending the geometry of the part until it mates successfully with the corresponding assembly, or redefining the spatial constraints so that it is fixed to the barrel.

The swirlers constituted a major part of the CAD cleanup: The fuel injector within the swirler assembly was of very complex design containing many cavities for fuel flow as well as featuring some rounded surface protrusions. The first modification to the injectors was to remove an unnecessary void created by the mating of the injector rounded face and
swirler blade assembly hub: It was assumed that any recirculation or eddies formed due to or within this cavity would not affect the flow on a macro scale.

The fuel injectors within the swirler assembly contained channels to feed the fuel to the atomiser. In order to simulate fuel release only the surface from which fuel enters the mixing zone was important for the CFD solver to know. For this reason the fuel lines and internal volumes in the fuel jets were not needed and could be filled in so that the fuel injector components were also one solid part. Small cones (seen in red in figure 3.7b) were drawn onto each injector’s fuel release surface that were specified to the solver as the fuel release surface. The angle with which the cones were drawn resulted in a simulated atomised spray angle of 60°.

Shaft seals were introduced to block off the flow since in the real life combustor the spool is sealed off. Furthermore the rear lid for the shaft seal was extended to close cavities which the CAD model featured but the real combustor didn’t.

![Figure 3.7: Typical flow swirler injectors](image)

The single most time consuming CAD error to fix was the inner combustor flame tube; the diffusion air which passes around the flame tubes bypass the initial fuel/air mixture that is ignited and flows through the cut-outs into the combustion zone downstream of the flame kernel. The cut-outs are shaped to form a nozzle in order to purposefully accelerate the flow of diffusion air into the combustion zone. The admittance of diffusion air serves
3.3 Computational Fluid Dynamics

3.3.1 Inner flame tube

![Before modification](image1)
![After modification](image2)

**Figure 3.9:** Inner flame tube

many purposes such as cooling the air for an acceptable turbine inlet temperature (avoid damage to stator blades) and controlling emission concentrations. The design of the cutouts are therefore a sensitive design aspect which needs to be finely determined for optimising mixing concentrations of air and fuel. The original Cad geometry obtained from the CSIR had the nozzles pointing radially inward (such as is featured on the outer combustor tube). This would cause the nozzles to act as diffusers, slowing the air entering the combustion zone down rather than accelerating as per design intent.

During the import process of the original Solid Works geometry into Creo 3.0, all feature functions were lost in the features tree so to correct the flame tube error meant manual re-drawing of the entire cone. This took a lot of time and careful measuring of the original (incorrect) flame tube to capture design parameters.

One final major CAD alteration which was done in preparation for the CFD analysis was to extend the barrel rear flange 270mm in order to prevent vortices and recirculated flow occurring at the exit: It is common practice to extend the outlet of an internal fluid assembly to ensure the solver does not base its calculation on the perceived outlet plane as a physical boundary. Additional frictional and pressure loss considerations arising from the increased flange length was accounted for by applying an ideal wall boundary condition to the inside surface of the flange [32].

3.3 Computational Fluid Dynamics

Computational fluid dynamics is the study of heat transfer, fluid flow and reacting mixtures based on a numerical process of solving mathematical equations. The CFD process involves a discretisation of the geometry into millions of individual cells to form a grid or mesh. This discretisation allows governing fluid equations to be simplified into approximate algebraic equations in the regions of interest. These equations are then solved numerically for the flow field of each cell. During post processing quantities of interest are extracted and values obtained. The mesh forms an essential part of the CFD analysis and is crucial to the accuracy of the results.
The basic outline of any CFD setup requires the following:

- Obtain correct and physically accurate CAD model
- Prepare the geometry as described in section 3.2.3 so that it is ready for fluid analysis
- Determine the most suitable turbulence model for the flow type
- Select appropriate physical model such as flow regime, turbulence level, etc
- Define material properties and fluid type; molecular compositions, specific heats, viscosity, etc
- Specify mixture quantities if more than one substance type is to flow through the same fluid volume.
- Operating conditions are to be defined in case of external fluid flows
- Boundary conditions need to be defined as well as initial values proposed
- Choosing solver controls and setting up a convergence criterion (discussed in more detail later)

The different turbulence models available for CFD studies were discussed in chapter 2. It was decided for this study to use a $K-\epsilon$ model. As a confirmation that the $K-\epsilon$ selected could accurately resolve the swirling flow in the APA 350 combustor, a model validation study was carried out, presented in chapter 4.

The accuracy of a solution depends on the quality of the original geometry used, the mesh refinement, the appropriateness of turbulence model chosen and the strictness of the convergence criteria selected. The correlation between mesh quality and solution accuracy was investigated and discussed more comprehensively in [32].

FloEFD is a fully featured CFD software that embeds itself into major CAD packages such as PTC Creo, Siemens NX and CATIA V5. It was designed to be useable by engineers from disciplines other than the simulation industry, making it revolutionarily simple to use [2]. The engineering behind the development of FloEFD allows it to not only deal with very complicated geometry but also solve complex three-dimensional turbulent flows. FloEFD uses a $K-\epsilon$ turbulence model designed to simulate a variety of turbulent flows accurately. Together with its pioneering full body immersed cartesian meshing methods it is capable of extensively validated ([1]) accuracy of flow field resolution.

### 3.3.1 Mesh Independence Study

Discretisation of geometry that is to be used for any parametric fluid analysis is a necessary requirement for calculation of fluid properties. Effective discretisation is paramount for increased accuracy, cost effective computational resource management as well as acceptable computing times.

A mesh independence study was undertaken [32] to investigate the effects of changing mesh properties on the quality of results, as well as the time taken for convergence to be
achieved. The study was done as a pre-requisite for the full cold flow simulation of the APA 350 combustor.

Mesh types, resolution and quality measures were among the aspects investigated in this mesh study. Furthermore potential effects such as oscillations in convergence arising due to instabilities and truncation errors were examined. Finally a mesh optimisation was carried out on the APA 350 combustor in preparation for full computational fluid analyses. The various mesh refinement settings available in FloEFD were reviewed and conclusions drawn as to which mesh configuration would provide the most reliable solution for the most optimal computational demand. Due to the size of the report only a summary is included in this document. For the full version the reader is referred to [32].

The basis of CFD studies on a component or assembly using the finite volume method relies on formulating fundamental fluid flow equations across a large number of surfaces or vast volumes while ensuring accurate conservation of properties across the computational domain. To do this the computational domain must be split up into many smaller control volumes. These are referred to as cells. The governing partial differential equations that are formulated for flow through each control volume are discretised and converted into algebraic form. First and second order derivatives are approximated by truncated Taylor series expansions. The resulting linear equations are then solved iteratively or simultaneously.

Performing these calculations over complete geometrical surfaces which may change vastly in shape from one part of an assembly to the next would render the process virtually impossible; stream functions and other flow gradients may change immensely in a very small area of ordinates, which requires that the geometry must be split up into thousands of individual cells so that each cell element can capture the flow properties at a particular point.

In CFD one should strive to select starting values for parameters such as pressures and mass flow rates that are as close as possible to the real values which will be calculated. Choosing arbitrary (but realistic) boundary conditions will still lead to convergence of the same solution but may take significantly longer to reach a convergence plateau. Selecting values which are close to the final solution however will decrease computing time and reduce iterations considerably. It is equally desirable to have an optimal mesh (grid) before starting analyses; the grid has a significant impact on the accuracy of results, the solution stability, CPU expenses and time taken to reach convergence. It is therefore very important to have an excellent initial mesh rather than waiting for a computation to complete then having to make changes to the grid due to insufficient resolution in areas which need increased attention. For this reason a pre-analysis mesh optimisation study was carried out to create the ideal mesh. This allowed for viewing the mesh and inspecting for errors or irregularities before allowing the solver to do a full, time consuming CFD analysis.

Mesh Optimisation

Several mesh configurations were constructed for the APA 350 combustor’s geometry. The methodology incorporated was to begin with the most coarse and basic mesh to demonstrate the capabilities with limited cell elements on the current geometry. The meshes were then increased in complexity, adding more control to each one by utilising
the settings discussed. Mesh size in terms of number of cells was also thus increased. The various mesh control settings available in FloEFD are comprehensively discussed and demonstrated in [32].

A summary of the control settings used for various meshes is as follows:

- Mesh 1 - automatic mesh refinement
- Mesh 2 - minimum gap size specified
- Mesh 3 - partial cells, narrow channels and small features refinement
- Mesh 4 - as mesh 3 but with higher refinement resolution
- Mesh 5 - local mesh refinement
- Mesh 6 - as mesh 5 but with additional refinement of small features and partial cells
- Mesh 7 - solution adaptive refinement limited to 3 million cells
- Mesh 8 - as mesh 7 but with double the allowed number of cells to 6 million

Meshes 4, 5 and 8 were tested with full flow simulations. Figures relating to the mesh independence study are presented in appendix A. For example figure A.1 shows results summaries for three different meshes. The variation in results with increased mesh resolution can be seen from the tables. Furthermore figure A.2 visually illustrated the change in velocity (left) and vorticity (right) magnitudes as the mesh quality improved.

Evaluation of Mesh Independence Study

This image clearly motivates the need for such a mesh sensitivity study; the results (figure A.1) are very dependent on the quality and resolution of the mesh.

When comparing the velocity and vorticity cut plots for mesh 4 and 5 the differences were quite subtle. Despite having over three hundred thousand cell elements more than mesh 4, the variation in the plots for mesh 5 were not very significant compared with mesh 4. The cut contour plot for mesh 8 however was far more developed than the former two meshes. The distinct disparity between the cut contour plot for mesh 8 comes down to a more accurate answer since a further developed mesh refinement was used for solving. The optimal mesh which used solution adaptive refinement settings was mesh 8 and it was decided that these settings would be used for the full flow simulations on the APA 350 combustor.

From the normalised standard deviation values it was find that of the three flow parameters used for comparison of meshes, the vorticity values had the most deviation with additional mesh refinements. The vorticity values had consistently decreased with mesh refinement whereas the pressure and velocity terms had increased. This shows that refinement steps have led to definite movements in a specific direction of results and that there was indeed a correlation between the results and the steps taken to refine the mesh (i.e. results are mesh-sensitive).
3.4 Flow Analysis Procedures

3.4.1 Modelling Swirlers

The benchmark swirlers were modified in Creo 3.0 to feature varying vane angles of 30°, 40°, 50°, 60° and 70°. They were then replaced into the full combustor geometry and imported into FloEFD. As seen in figure 3.10a flat blades were modelled to correspond with the real swirlers in the APA 350 combustor.

![Figure 3.10: Modified swirler blade angles](image)

The leading and trailing edges of blades were rounded to prevent additional pressure loss or vortices forming. In the real APA 350 combustor these sharp edges were filed to a more rounded profile.
Figure 3.11: Swirlers with varying vane angle
3.4 Flow Analysis Procedures

3.4.2 Preparation of FloEFD Model

With the combustor imported into FloEFD and a computational domain established, all settings were carried out. The inlet mass flow rate and exit static pressure boundary conditions discussed in section 3.2.2 were applied, as well as an ideal wall established on the inside of the dilution zone extension. Calculation control options were selected to achieve a trade off between the most accurate, high resolution results and the least computationally demanding model. After several initial attempts at iterations using the mesh and calculation control settings determined in [32] the settings were altered slightly to improve result accuracy and calculation time even further: The global cell refinement level was set to 7 and a maximum number of allowed cells of 12 million specified.

In preparation of the post processing methodology for assessing the mixing effectiveness a fluid was introduced into the model that would represent the fuel. One method to evaluate the mixing quality would be based on analysing cut contour plots that illustrate mass fraction density of the fuel. To introduce this fuel a separate boundary condition was established for a gas to be expelled from the injector cones drawn on each fuel injector component. Extracts from an APA 350 combustor design document [4] report that the fuel injectors were "of the vaporizing type where hot combustion products envelop the fuel injection/vaporizing tube which is situated in the primary combustion zone". For this reason a gaseous fluid was selected. Although the APA 350 was designed to use Jet A-1 fuel the fluid library for FloEFD did not contain this as a pre-set substance. For productive comparison of the mixing with Jet A-1 fuel, the fluid chosen had to feature a similar density to that of Jet A-1 (constituting mostly of Kerosene, the relative vapor density of Kerosene to air was used as a reference). For that reason Xenon was chosen as a gas to represent the fuel. Air has a molar density of 0.0291 kg/mol at 443 K. With a relative density of 4.5, Kerosene vapor hence has a molar density of 0.131 kg/mol, and the Xenon used, a molar density of 0.130 kg/mol. Thermochemical properties of the pre-set Xenon had to be modified accordingly in order for its thermofluid dynamic behaviour to match that of Jet A-1 fuel. For instance the specific heat at constant pressure \( C_p \) was required. To calculate this the molecular weight of Kerosene \((C_{12}H_{26})\) gives a specific gas constant \( R \) of 48.9 J/kgK. The ratio of specific heats \( \gamma \) was taken as 1.03 as per a data sheet for JP-4 fuel [5]. (JP-4 is a military classification of jet fuel similar to Jet B, an enhanced cold weather variant of Jet A-1 fuel).

\[
\gamma = \frac{C_p}{C_v} \quad (3.1)
\]

\[
C_p - C_v = R \quad (3.2)
\]

Equations 3.1 and 3.3 were then used simultaneously to obtain \( C_p = 1679.18 \text{kJ/kgK} \) and \( C_v = 1630.28 \text{kJ/kgK} \).

The model was set to inject Xenon gas at a mass flow rate of 0.010577 kg/s (the design point fuel flow rate divided into thirteen injectors). In order to quantify and control the simulation of the mixing of the two gases in the combustor an injection velocity had to be specified for the Xenon. Data for the injection fuel injection velocity was not available in design reports and therefore had to be calculated using fluid dynamic and geometric relationships.
Calculation of Fuel Injection Velocity

A certain mass flow rate of fuel per injector is released into a stream of hot air at $443K$. With a relative vapor density of gaseous Kerosene to air of 4.5, the density of the Kerosene (in this case replaced by Xenon) is $3.825 kg/m^3$. Hence the volumetric flow rate of fuel per second per injector is found by:

$$\dot{Q} = \frac{\dot{m}_f}{\rho} = 0.002765 m^3/s$$  \hspace{1cm} (3.3)

For which a conical spray of fuel is produced. As seen in figure 3.12 this volume of fuel is illustrated as the difference of two separate conical volumes:

$$Vol_B = \frac{\pi h^2 \{ h \cos(30) \} \{ \sin(30) \}^2}{3} = \frac{\pi h^3 \{ \cos(30) \} \{ \sin(30) \}^2}{3}$$  \hspace{1cm} (3.4)

Where $h$ represents the distance travelled by a two dimensional plane of fuel in one second (equal to the injection velocity $V_{inj}$ that is to be calculated) perpendicular to the surface of the fuel injector cone (drawn in brown).
Then from geometry:

\[
Vol_A = \left( \frac{\pi}{3} \left[ \left[ h + p \right] \cos(30) \right] \left[ h + p \right]^2 \sin(30) \right)^2 - 2 \times \left( \frac{\pi \{ p \cos(30) \}}{3} \right) r^2 \right) \tag{3.5}
\]

Since the length of the injector cone \( p \ll h \) it can be said that \( p + h \approx h \) hence \( V_{inj,A} \approx V_{inj,B} \).

\[
\therefore \text{Total volume of fuel spray per second is found by:}
\]

\[
Vol_{total} = Vol_A - Vol_B = \dot{Q} = 0.002765 m^3/s \tag{3.6}
\]

Upon measuring and substituting in the required quantities \( p \) and \( r \) (labelled in the figure), equation 3.6 could be used to solve for \( h \) which equates to the desired fuel injection velocity \( V_{inj} \).

The solution simplified to

\[
0.01277 = (h + p)^3 - h^3 = h^3 + 2h^2p + hp^2 + h^2p + 2hp^2 + p^3 - h^3 \tag{3.7}
\]

With the cancellation of cubic \( h \) terms the remaining equation was be solved using the quadratic formula:

\[
h = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} \tag{3.8}
\]

where \( a = p \), \( b = (p^2 + 4p^2) \) and \( c = p^3 - 0.01277 \).

\[
\therefore 0.01277 = ph^2 + (p^2 + 4p^2)h + p^3 \tag{3.9}
\]

\[
h \approx V_{inj} = 2.183 m/s \tag{3.10}
\]

Another boundary condition necessary for the CFD analysis was the static temperature of the inlet air. This was calculated based on the compressor exit Mach number [47], the combustor inlet total temperature [4] and the equation:

\[
\frac{T_o}{T_s} = 1 + \frac{1}{2} \{ \gamma - 1 \} M^2 \tag{3.11}
\]

A static temperature value of 427.2K was calculated.

The first CFD analysis carried out was for the swirler configuration featuring a vane with blade angles of 30°, followed by successive analyses with 10° incremental blade angle increase up to 70°. Boundary conditions as well as mesh and calculation control settings were kept constant for all blade angle configurations.
Chapter 4

Model Validation

4.1 Motivation for Validation

In order to rely on the accuracy of the results obtained with CFD it was important to validate that the CFD turbulence models selected for the CFD study could accurately resolve highly turbulent, swirling flow. For this reason a validation study was conducted. The standard approach is to validate CFD results against experimental data collected during tests with physical hardware such as combustion rigs. To this end, use was made of an existing Master’s thesis [15] in which rig tests were carried out on a single-can, single-swirler combustor solely for the purpose of providing a CFD validation test case. In this Master’s research, a non-reacting flow test was done using a perspex replica of a combustor.

4.2 Experimental Approach

In the experimental study carried out by Meyers, stereoscopic particle image velocimetry (PIV) measurements were performed at different planes for each of the sections of the combustor (primary, secondary and diffusion).

In the experimental study carried out by Meyers, stereoscopic particle image velocimetry (PIV) measurements were performed at different planes for each of the sections of the combustor (primary, secondary and dilution). By analysing velocity profiles, jet penetration depth and size and location of recirculation zones the data collected in the experimental result was compared to identical plots produced by FloEFD CFD results. A cross sectional view of the combustor used in the validation study is illustrated in figure 4.1.

A long list of hardware is required to conduct such measurements but the general process of obtaining the velocity vectors are as follows: A planar laser sheet is shone into the flow field in the plane which is of interest. The flow is seeded with particulates that follow the flow. In this case smoke from burning oil (Shell Ondina EL) was used. Seeding particles
Figure 4.1: Combustor used for validation study, from Meyers [15]
must ensure good light reflection, the ability to follow the mean flow field and distribute effectively throughout the bulk flow. Size and density of seeding particles must be considered. Excessively low densities lead to buoyancy forces causing seeding particles to not follow mean flow, whereas excessively large or dense particles will overshoot flow structures such as vortices due to their significant inertia. In order to effectively trace through turbulent flow, particulates in the order of microns must be used. Two cameras pointing at the sheet with different angles take photos that are synchronised to two consecutive laser pulses. Each camera hence takes two photos and thus four photos are produced in total. These are used for comparison of the displacement of individual seeding particles over a finite time spacing. Two-dimensional images are produced and after a combination of vector geometry analysis and a calibration functioning, three-dimensional velocity vectors are calculated. Many efforts for calibration of various hardware are done before experimental results can be collected. Details about the experiment used for validation will be discussed next but to avoid losing focus on the scope of this chapter the experimental procedures described will be restricted to the essential information and leave out content that is not of any relevance to the validation study. Figure 4.2 shows the layout of experimental apparatus used in conducting PIV measurements on the validation combustor.

Figure 4.2: Experimental apparatus for combustor PIV measurements from Meyers [15]

An 11kW motor ran a fan that supplied the inlet air. A long pipe section separated the fan outlet and combustor inlet so that the flow entering the combustor could be assumed fully developed. The combustor exhausted to the atmosphere so that the outlet pressure boundary condition required for CFD purposes was taken as the static atmospheric pressure. A brief description of the combustor follows: A single swirler with ten flat blades at 50° was featured, containing a fuel injector that injected fuel at a cone angle of 80°. There were six radially equi-spaced holes in the primary zone combustor liner with 9.5mm diameters. The secondary zone had eight equi-spaced holes of 5mm diameter. The dilution zone featured ten equi-spaced holes of 11.8mm diameter. A photograph and drawing of
the combustor liner is given in figure 4.4, taken from [15]. Cooling holes were present in the secondary and dilution zone, featuring fifty and thirty holes respectively, all with a diameter of 1.2\,mm. The inlet mass flow was measured using an orifice plate placed in the piping upstream of the combustor shown in figure 4.2. Temperature measurements were also taken in this location.

Due to the use of PIV it was necessary for the combustor annulus and flame tube to be transparent. Hence a perspex replica was manufactured for the experiment. Due to construction limitations however the rear and front (containing swirler) flanges were fabricated from Erthacetal C (low cost, easy manufacturing).

Due to the irrelevance to this report it was deemed unnecessary to include details about instrumentation used such as laser wavelengths and camera lens specifications, etc.

In order to get a multi-dimensional view of the velocity vectors, the PIV measurements were taken at four different planes that pass through the longitudinal axis and were normal to the inlet and outlet flange faces. The first (baseline) plane was the one that passed through the centre of all the liner holes (primary, secondary and dilution zones). The other three planes used were those which passed through the point exactly in between the primary zone holes, secondary zone holes and dilution zone holes respectively. This is clarified in figure 4.6 which illustrates the planes used with solid black lines. The illustration is from an axial perspective.
4.2 Experimental Approach

(a) Photographed flame tube

(b) Drawing of flame tube

Figure 4.4: Experimental combustor liner from Meyers [15]
Figure 4.5: Fabricated perspex combustor lining from Meyers [15]

Figure 4.6: Illustration of planes used for laser sheet from Meyers [15]
4.2 Experimental Approach

4.2.1 PIV Verification Measurements

Some initial measurements were done to verify that the PIV equipment had been set up correctly.

Figures 4.7 to 4.13 relate to the verification measurements to ensure the PIV was operating properly, hence only the primary zone was photographed. They were all using an air inlet mass flow of 0.056 kg/s and photographed on the plane passing through all holes.

Figure 4.7 is a velocity vector plot including a magnitude scale. The same image with the photograph background removed (figure 4.9) demonstrates how over-exposure of the photos taken during PIV measurements can result in localised areas that do not produce data on the plot. Further to the issue of over-exposure some other “patches” of missing vector data are attributed to the reflection from combustor lining, and obstruction to laser and camera view of the fluid by physical features such as the swirler. This is also seen and highlighted in figure 4.8. The same velocity vector plot is shown in figure 4.10 demonstrating some characteristic flow structures such as recirculation zones (encircled in purple) and highlights the returning flow in red. Furthermore, subtle flow occurrences are highlighted by [15] in figure 4.11.

These figures highlight examples of flow structures that will act as benchmark observations to look for in the data obtained by CFD analysis of this combustor. Further to global velocity magnitude plots and velocity vector analysis, axial velocity profiles were determined at various “slices” along the longitudinal direction.

Figure 4.12 was used by [15] to demonstrate the 3D nature of the flow. The image shows an axial view (viewed from upstream) of the velocity vectors measured in the same plane as for figures 4.7 to 4.11. It clearly demonstrates the positive Z-direction flow in positions where \( x > 0 \) and negative Z-direction flow in positions where \( x < 0 \) indicating swirl in the flow.

Finally, figure 4.13 shows the velocity magnitude plot obtained for the primary zone during the PIV verification measurements.
Figure 4.7: PIV verification velocity vector plot for primary zone from Meyers [15]

Figure 4.8: Photograph showing obstructions to laser that cause lack of vector information from Meyers [15]
Figure 4.9: PIV verification velocity vector plot demonstrating broken patches from Meyers [15]
Figure 4.10: PIV verification velocity vector plot demonstrating characteristic flow structures from Meyers [15]
Figure 4.11: PIV verification velocity vector plot demonstrating subtle fluid interactions [15] from Meyers
Figure 4.12: Axial view of longitudinal planar velocity vector plot from Meyers [15]
Figure 4.13: PIV verification velocity magnitude plot from Meyers [15]
4.3 CFD Validation Approach

The CAD geometry for the experimental combustor was obtained and a similar (but less extensive) clean-up process was carried out as discussed in chapter 3. The model set up and mesh generation were all carried out using similar methods as for the APA 350 combustor CFD: The model was imported into FloEFD and boundary conditions specified. An air inlet mass flow rate of $0.056\, \text{kg/s}$ was used and an atmospheric pressure outlet specified to match the experimental set up described above. A solution adaptive mesh setting was applied, limited to 2 million cell elements. An extension to the geometry with an ideal wall condition was applied to the combustor incase any downstream flow vortices were induced by the apparent lid required on the combustor exit for completing the fluid volume. The solver was initiated and the resulting mesh contained 1,991,216 cell elements. Planes were created that coincide with those shown in figure 4.6 for viewing velocity plots.

4.4 Results of Validation Study

In the following section results from the experimental combustor CFD are discussed. Cut plots and axial velocity profiles were produced exactly as was done by Meyers in order to compare the experimental results with the CFD results. Discussions follow according to the results plane in question. The baseline plane is discussed first, followed by the primary, secondary and dilution zones respectively. Due to the number of figures involved, all figures relating to the results discussion can be found in appendix B.

4.4.1 Baseline Plane

The velocity magnitude contours with streamlines are given in figure B.1b. The velocity vectors are given in figure B.2b. The axial velocities are demonstrated in the same fashion as by Meyers in figure B.3b.

The recirculation zone that anchors the flame and which forms a pool of hot reacting species is present at (3) as in Meyers’ plot. By inspection of the velocity magnitude plots the velocity of the primary injection air is equal to that shown in Meyer’s results. The streamlines entering the first recirculation zone were not as rounded however, with a larger delay before the streamlines bend right into reversed flow. There appeared to be a definite deflection in the streamlines at (A). It is possible that this was due to the solution not converging far enough. At (4) the fully reversed flow due to recirculation in the primary zone was seen. Meyer states that "the reverse flow zone is quite rectangular in shape" [15] which was even more true in the CFD result. At (5) the stagnation zone cause by the recirculating flow, the main combustor flow and the injection air flow was seen. The injection air flow was clearly seen at (6). In the plot by Meyers, the balloon at (7) shows streamlines diverging upward toward the entry hole for the primary injection jet. This was not seen in the CFD results. The cause for this was attributed to the experimental setup having the cooling holes blocked off since the PIV recording equipment would not be able to visualise that flow. In the CFD model however cooling air did flow through the
cooling holes, preventing the streamlines from diverging upward as seen in figure B.1a. The streamlines from cooling air were clearly marked at (B) in the CFD result and absent in the experimental result. The secondary air injection holes created the accelerated air flow were seen at (8). The injection air causes the streamlines to diverge downstream of injection flow as seen at (9). At (10) the second recirculation zones were seen as in the experimental result. This recirculation zone was due to the high mass flow (50% [15]) of the total air flowing in through the dilution air holes as seen at (11). Finally at (12) another stagnation point existed due to the collision of the two dilution air injections.

4.4.2 Primary Zone Plane

The velocity magnitude contours with streamlines are given in figure B.4b. The velocity vectors are given in figure B.5b. The axial velocities are demonstrated in the same fashion as by Meyers in figure B.6b.

As confirmed by the experimental results, the CFD plot showed that the flow within the primary zone was very similar on the primary zone plane in between the injection holes as for the baseline plane that passes through the holes. The main difference was that the high velocity injection streams were not present. At (21) the first recirculation zone was seen once again, as well as the first stagnation point at (23). Although the jets were missing from their previous position at (22), the streamlines still followed this path as they were affected by the injection flow on the baseline plane.

4.4.3 Secondary Zone Plane

At (24) the divergence of the streamlines were now not deflected by the incoming secondary injection air jets. The streamlines were not affected by the injection holes at all as is the case in the primary zone since the holes were now further away. (25) points out another missing injection flow. At (26) the diverging flow straightened out to become more axial.

4.4.4 Dilution Zone Plane

The second central toroidal recirculation zone was shown at (27) and the second stagnation point due to the influence of dilution injection air jets colliding was shown at (29). At (28) a contrast from the baseline plane result: the streamlines in the dilution zone plane continued downstream along the walls of the combustor towards the outlet instead of diverging inwards such as in the baseline plane.

4.5 Conclusions for Validation Study

For each comparison of velocity magnitudes and velocity vector plots, every single balloon that fell within the CFD computational domain was identified by both CFD and experimental results. For the baseline plane axial velocity plots the plots were exactly the same from 81\textit{mm} onwards. From 29\textit{mm} to 72\textit{mm} the plots were slightly different. The
main difference was that the velocity profiles were much more exaggerated toward the combustor centre line. Judging by the scale on the figures the aspect ratio of the plots for the experimental results was clearly different to that produced by the CFD results. For this reason there were a few plots such as for the 81\,mm position which had the same result according to the scale but appeared visually emphasized due to the increased scale length. Overall the only plot which did not agree with experimental results on whether the net flow was positive or negative at a certain radial position was that for 72\,mm. The CFD plot showed the flow to be zero or positive for the entire axial plane but the experimental results showed a central reverse flow region with positive flow towards the outer radial regions. The CFD plot line had the same profile as that for the experimental result but was offset to the right by approximately 7\,m/s. Similar results were found for the mixed planes axial velocity plot, this time with only the 48\,mm line having significant emphasis of centered velocity with the rest agreeing with experimental results visually and quantitatively. The same velocity profile for 72\,mm was again offset to the right, this time only by approximately 4\,m/s, representing only zero or positive flow.

In summary the mixed plane axial velocity CFD plots are in better agreement with experimental results than for the baseline plot. Despite a very small difference in plots for the primary zone plane experimental axial velocities and those obtained by CFD, the velocity magnitude contour plots were in very good agreement which therefore supports the CFD model’s validity. By the streamlines displayed in the velocity magnitude contours is has been shown that the cooling air holes influence the streamlines’ divergence in a manner that was not shown by the experimental results, due to the cooling holes being blocked off in the experimental set up.

The mixed plane plots did not feature any dilution air injection flows as in the baseline plane. It was this additional interaction of the injection air that introduces an additional complexity to the main flow for the baseline plane case. A more severe stagnation zone due to colliding jets produced a stronger recirculation zone, as well as causing higher shear rates that led to vortex shedding. It can be argued that the disparity in results shown between the experimental and CFD axial velocity plots for the primary zone plane may be due to the fact that the $K-\epsilon$ model used could not sufficiently resolve the highly turbulent flow due to its swirling nature.

Perisetty et al\cite{36} found that with $K-\epsilon$ turbulence models \"difficulties in solving swirling flow are experienced due to large degree of coupling between momentum equations, when the rotational components are large\". They further report that by increasing the mesh fineness the results using this turbulence model could be improved. It is therefore a suggestion that the mesh for the experimental combustor should have been refined more.

At the time it was assumed that the 1,991,216 cell elements would suffice for such a basic geometrical model.

An alternative turbulence model that could have been selected is a Reynolds Stress Turbulence Model. Work by Yang et al indicate that RSTM’s are better at resolving recirculation zones with high velocity gradients such as is found in highly swirling flow\cite{24} however it is known that these turbulence models are much more computationally expensive.

In conclusion the CFD results obtained for this experimental combustor agreed well and were supported by the experimental results obtained by Meyers\cite{15}. Due to the advan-
tages of being less computationally expensive and more widely validated it was decided that a $K - \epsilon$ turbulence model would be used and that the CFD model used for the APA 350 combustor would be accurate. If more powerful computational resources were available then a RSTM would be considered for future studies.
5.1 Post Processing and Results Overview

Results are mainly focussed on the 30°, 50° and 70° blade angle swirlers since these provided sufficient data to observe trends and demonstrate the effect of changing swirler angle on the flow structures, flame anchoring and pressure losses.

For clarity of explanations this chapter uses planes that are referenced to the geometry. The planes and their names are shown in figure 5.1. Unless otherwise specified, when swirler numbers are quoted they are defined according to the definition of Beer and Chigier [22] as given by equation 1.7.

The post processing followed a methodical approach: CFD results for each swirler configuration were processed individually and a variety of data extracted pertaining to three characteristics that were investigated given below. All figures relating to post-processed results are found in appendix C.

Fuel/Air Mixing Effectiveness

2D cut contour plots were made for velocity, pressure, turbulent kinetic energy, fuel (kerosene) mass fraction, and turbulence intensity. 3D iso-surfaces simultaneously showing 1%, 2.5% and 5% mass fraction of kerosene demonstrated the dispersion of the kerosene within the primary zone of the combustor.

Pressure loss

For each swirler configuration the total pressure loss was measured. An XY-plot was constructed giving the pressure along the combustor length with the X-axis corresponding to the purple line in figure 5.1 (extending from the fuel injector cone and thus through the centre of the reacting zone). This line was used as the X-axis for all XY-plots.
Furthermore 2D cut contour plots in figure C.7 provided a visual as well as quantitative indication of swirler pressure losses arising from increased blade angle. Note the plane used for all 2D contour plots intersects the fuel injector cone for the bottom segment and passes through the centre of two swirlers for the top segment.

**Flame anchoring/Flow recirculation**

Finally for cut plots taken in the "top" view as shown in figures C.8 to C.10, the cuts were taken at planes along the combustor length as shown in figure 5.2 corresponding to offsets of 2mm, 27mm, 50mm, 75mm and 100mm from the exit face of the swirlers.
5.2 Results

5.2.1 30 Degree Blade Angle Summary

The 30° blade angle swirler had a swirl number of 0.46. A swirler pressure loss was calculated by taking the average pressures on an annular surface right in-front of the swirler inlet and directly after the swirler exit. The calculated value was 9.98%. The residence time was calculated for each swirler design by taking a sample of 500 seeding particles of one micrometer diameter. Due to some seeding particles becoming entrapped in recirculation zones for long periods of time (labelled by balloon 1 in figure C.1) an averaging method could not be used as this would result in misleadingly long residence times. Instead the median value was used in each case. The residence time calculated for this swirler design was 0.012 seconds. The average combustor total pressure loss was calculated as 26,776.49Pa (9.98%). This was expected as pressure losses in small combustors such as are found in micro gas turbines are usually bigger than those found in large gas turbines.

5.2.2 50 Degree Blade Angle Summary

The 50° blade angle swirler had a swirl number of 0.95. A swirler pressure loss was calculated as 9.96%. The residence time calculated for this swirler design was 0.0134 seconds. The average combustor total pressure loss was calculated as 29,136.69Pa (10.44%). A
larger combustor pressure loss than for the 30° swirler was expected due to the increased steepness of the vane blade angles.

5.2.3 70 Degree Blade Angle Summary

The 70° blade angle swirler had a swirl number of 2.19. A swirler pressure loss was calculated as 12.02%. The residence time calculated for this swirler design was 0.015 seconds. The average combustor total pressure loss was calculated as 33,376.11 Pa (11.96%). A larger pressure loss than for the 50° swirler was again expected due to the even further increased steepness of the vane blade angles.

Mixing Effectiveness

Figure C.5 shows a 2D contour plot of kerosene mass fraction relative to air. For the 30° blade angle swirler shown in figure C.5c a highly compact area represents the highest concentrations of kerosene by mass. This concentration is seen to be spread out much more for the 70° blade angle swirler shown in figure C.5a. The kerosene mass fractions downstream of the dilution air injection jets did not alter much for varying degree of swirl however upstream of these jets in the primary zone the differences were quite significant. By comparing the upper segment of these 2D contour plots it is found that for the 30° blade angle swirler there was only a mass fraction of kerosene of approximately 0.36%-0.8%. In the 70° blade angle swirler the same area of space contained a mass fraction in excess of 3% demonstrating a much better dispersion and fuel mixing performance by the 70° swirler configuration.

A 3D iso-surface plot for kerosene mass fractions is given in figure C.12. The blue surfaces represent 5 % mass fraction and the green surfaces represent 2.5 % mass fraction. The longitudinal and axial views clearly show the improved mixing and dispersion of fuel within the combustor volume for the 70° blade angle swirler as compared to the 30° configuration. The mixing increased with increased blade angle.

Pressure Loss

XY-plots demonstrating combustor pressure along the length of the combustor are given in figure C.16. A similar trend was observed in each of the swirler designs; first a strongly adverse pressure gradient approaching the swirler inlet. This was due to a stagnation zone just ahead of the fuel injector, labelled in figure C.7 by balloon 1. Next a gap in the plot represented the area where no fluid was modelled (inside the fuel injector). A sudden pressure peak corresponding to another stagnation zone caused by dilution air injection jets was followed by a gradual pressure decrease due to wall friction. The less prominent recirculation zone for the 30° blade angle swirler was due to a lower mass flow rate of dilution air injected. This is discussed in more detail in section 5.3.3.

Flame Anchoring and Recirculation

As the blade angles were increased from 30° through to 70° the calculated residence time for seeding particles increased. This was expected since increased recirculation zones
brought on by toroidal vortices would mean a longer path taken by reacting species. It is observed from the streamlines displayed in the velocity contour plots that large recirculating flow regions as well as several small localised recirculation zones due to complex interaction of main flow, dilution air injection and vortex structures are produced by the swirlers. Several recirculation zones are furthermore identified for all the swirler configurations by inspecting the XY-plots of axial velocity along the length of the combustor, given in figure C.15. These are discussed in greater depth in section 5.3.3.

5.3 Discussion

5.3.1 Analysis of Dynamic Flow Structures

The degree of flow swirl can be interpreted as the magnitude of azimuthal momentum in a flow. By inspection of XY-plots demonstrating circumferential velocity along the length of the combustor (figure C.19) it was seen that the height difference between peaks and troughs was much larger for the 70° blade angle swirler than for the 30° blade angle swirler. This represented a more significant change in azimuthal flow direction for the 70° blade angle swirler. Furthermore the plot for the 70° blade angle swirler featured three instances of azimuthal flow reversal whereas for the 30° blade angle swirler this only happened twice, hence the 70° blade angle swirler featured more directional fluctuation. The mean circumferential velocity was not very high (oscillated at or near zero) since the line which represented the axis at $Y = 0$ on these plots ran through the centre of the fuel injector and hence swirler assembly (in other words the radius of circumferential flow rotation was extremely small and thus small angular/azimuthal velocity). Therefore a high vorticity should indeed be expected. True enough this was seen in the XY-plots for vorticity in figure C.17. The circumferential flow reversal observed was therefore either due to constructive interference between adjacent swirler’s large scale spiralling vortices or breakdown of a precessing vortex core. The theory of constructive interference would be less valid for the 30° blade angle swirler since it caused less flow divergence. This was evident from comparison between figures C.13a and C.13c which are iso-vorticity plots for a vorticity of 33,750 1/s.

Further downstream there was a net positive growth in circumferential velocity. This was due to the mean circumferential velocity in the entire annulus driven by co-rotating swirlers. This mean azimuthal flow can be seen in figure C.11. The dilution air injection jets could not be blamed for azimuthal flow reversal since the jet flows entered the combustor symmetrically above and below at approximately equal velocities as seen in figure C.1. It was concluded that the azimuthal fluctuation was mainly driven by interactions of broken down vortex boundaries and local recirculation zones.

From the plot data of the vorticity XY-plots, values for points downstream of the fuel injection cones were extracted, inversed (i.e. 1/vorticity) and new lines plotted. These new inversed plots were copied and the copies were turned upside down and superimposed on the original plotted lines to produce a 2D visualisation of the large scale spiralling vortices formed by each swirler configuration (figure C.14).

Figure C.12 demonstrates that for high swirl flows large coherent structures formed, as confirmed by Vigneras-Zuniga et al [49] and Huang and Yang [54]. Figure C.10 shows
Results and Discussion

Figure 5.3: Snapshots of an iso-vorticity surface at $\omega = 75,000 \, s^{-1}$ for two different swirl numbers taken from Huang and Yang [54]

intense vortex shedding directly behind the swirler (at position $X = 2 \, mm$) for high swirl in contrast to the compact unbroken spiralling flow structure for the low swirl configuration. This flow behaviour was also observed in studies by Lee et al [42] who says that vortex shedding plays a large role in combustion stability and that excess shedding should be avoided. Despite this, vortex shedding encourages mixing due to resulting increased turbulent energy. Hence the 30° blade angle swirler had too little vortex shedding, and the 70° blade angle swirler had too much. In this assessment characteristic the 50° blade angle swirler would be the best choice for improved mixing and combustion stability. Furthermore from figure C.9 although the downstream turbulence intensities were similar, directly behind the swirler the turbulence intensity was much lower for the 30° blade angle swirler than for the 70° blade angle swirler, indicating much more stable flow. This low turbulence intensity can also be disadvantageous since the resulting lower shear rates did not aid in atomisation of the fuel droplets or dispersion of the fuel very well (confirmed again by figure C.12). Another advantage to increased turbulence intensities are that they facilitate a more compact flame and therefore smaller combustor design.

It was found that mixing (shown in figure C.5) and turbulence intensity (shown in figure C.9) both increased for increasing swirl number. Work by Tangirala et al [48] suggests that this trend only continues up until a swirl number of approximately unity. It was found in the current study however that the trend continued until at least a swirl number of 2.187 seen in the 70° blade angle swirler.

Snapshots of the breakdown of spiralling flow structures due to high swirl are illustrated in figure 5.3 with iso-vorticity surface plots by Huang and Yang [54]. The same type of iso-vorticity surface plots were made for the APA 350 combustor in figure C.13. Coinciding with the observations of Huang and Yang, the significantly increased centrifugal forces experienced by flow through the higher swirl configuration caused a much increased flow divergence and hence breakdown of the precessing vortex core.
5.3 Discussion

5.3.2 Pressure Loss

The 30° blade angle swirler was found to have a much lower pressure loss than for the higher swirl caused by the 70° blade angle swirler. This supported findings found by Ivanov [10] and Romadin et al [51].

5.3.3 Recirculation and Flame Anchoring

Breakdown of the central toroidal vortex (CTV) structures led to reduced axial velocities hence adverse pressure gradients and therefore recirculation zones. Another reason why recirculation zones were produced could be attributed to the varying strength of dilution air injection jets: For the swirlers with larger blade angles, a larger resistance of flow through the swirler meant a larger mass flow of air flowing through the dilution holes. These dilution air jets acted as a wall to the air flowing from the swirlers, causing a change in velocity vector and ultimately flow reversal. Figures C.1 and C.6 demonstrate this clearly. This variation in mass flow rate of air admitted through the swirlers affected the FAR and hence contributed in part to the change of stoichiometric fuel/air mixture positioning seen in figure C.11.

Syred and Beer [29] suggest that the length of the recirculation zone increases until a maximum of $5 \times D_{sw}$ at a swirl number of about 1.5 (according to their own swirl number definition). Inspecting the lower segment of the velocity plots as given in a close-up view in figure C.2, it is seen for the 30° blade angle swirler a stable toroidal recirculation zone (at balloon 3) which caused a stagnation zone centered approximately a length equal to one swirler diameter away from the fuel injection cone. The recirculation zones were very rounded and symmetrical. From the velocity vectors in figure C.6 the recirculation zone for the 30° blade angle swirler flowed in such a manner that the flow diverged radially outwards then longitudinally backward. In contrast, the 50° and 70° blade angle swirlers had much more obscure shaped, asymmetric main recirculation zones (balloon 4), with originally divergent flow curving radially inwards then longitudinally back. The asymmetric nature of the recirculation zones for the latter two swirler configurations was attributed to the difference in mass flow rate of primary holes injection air between the inner and outer flame tubes. Relating these observations to those of Syred and Beer, it is difficult to judge whether the recirculation zones would have increased up until $5 \times D_{sw}$ since they were broken down by the injection air from dilution holes. It could be observed however from inspection of the top segment of figure C.1b where the flow was not disturbed by the primary zone injection flow that the length of the recirculation zone was approximately $1.7 \times D_{sw}$. For the 70° blade angle swirler (C.1a) the lower segment which was influenced by injection air had a recirculation zone of approximately $0.9 \times D_{sw}$ and in the upper segment no distinct recirculation zone is observed at all. This indicated that for the 70° blade angle swirler the recirculation zone in the lower segment was mainly due to the stagnation zone caused by the collision of primary hole injection flow and not due to swirler induced toroidal structures as for the other swirler designs.

From the XY-plots of axial velocity (figure C.15) it was observed that both the 50° and 70° swirlers had an immediate deceleration of axial velocity, followed by a large acceleration, a second deceleration, a second acceleration and finally a third deceleration. The 30° blade angle swirler however had 4 decelerations. For the 50° and 70° blade angle swirler
the initial deceleration was caused by a recirculation zone brought on by main flow air hitting the jets of dilution air. Hence why this was followed closely by a large acceleration (due to injection of dilution air). The second deceleration was due to the same reason, a recirculation zone caused by air hitting the second dilution air jet. This was confirmed by the magnitudes of deceleration: the 50° blade angle swirler which had a smaller mass flow of secondary dilution hole air than the 70° blade angle swirler, caused an axial velocity reduction of approximately 64 m/s whereas the 70° blade angle swirler which had a larger mass flow of dilution air caused an axial velocity reduction of approximately 107 m/s. As before, the sudden acceleration that followed was due to the injection of secondary dilution air jets (again confirmed by magnitude of accelerations). For the 30° blade angle swirler plot however, the two recirculation zones discussed above caused the second and third decelerations. The first deceleration on this plot was attributed to the strong and sudden radial pressure gradient directly behind the fuel injector causing a very small and localised recirculation zone. This is seen in figure C.7. This pressure gradient caused an intense velocity gradient directly behind the fuel injector centre-body as seen at balloon 2 in figure C.1c. According to Cheng, swirlers featuring lower swirl intensity (such as the 30° blade angle swirler) deliberately have lower swirl intensity to prevent vortex breakdown and thus flow reversal by diverging the flow radially outward [38]. He furthermore states that “flow divergence causes decreasing axial velocity”. According to the observations in this study however, for the lower swirl 30° swirler configuration, indeed an initially decreasing axial velocity was observed but this was attributed to the small recirculation zone immediately after the fuel injector. The observations in figures C.1a to C.1c respectively (high to lower swirl) showed a reduction in flow divergence immediately after the swirler – a direct contradiction to the statements of Cheng.

Despite the large radial pressure gradients demonstrated behind the swirler in figure C.7, Perisetty et al ([35]) report that “large radial pressure gradients are caused by highly rotating flow”. The fact that the opposite to this was observed in figure C.7 (high gradients seen in low rotating flow and almost no gradient seen in high rotating flow) supported the hypothesis that central vortex breakdown had occurred due to the merging of an upstream translating CTRZ and a downstream extending PVC, as observed by Huang and Yang [54]. This was further supported by figure C.10 which for the X = 2mm position of the 30° blade angle swirler clearly shows a compact rotational vortex at each swirler whereas the 70° blade angle swirler plot had no complete rotational vortex structure but instead multiple shredded vortices. In figure C.10 the 50° swirler still had a mildly compact single spiralling vortex structure at this position. This shows that merging of the CTRZ and PVC had not happened yet, parted by the stable section of flow seen at the X = 27mm position where there is very little vorticity.

Stagnation points on the XY-plots for axial velocities indicated potential “flame holding” positions. The 30° swirler configuration had more than one stagnation point whereas the 70° swirler configuration had one main one around X = 0.21m. In the case of the 50° swirler configuration the second recirculation zone was only just strong enough to cause flow reversal and in the 70° swirler configuration there was not even enough recirculation to cause reversed flow, merely an adverse pressure gradient. This meant that the 50° blade angle swirler could help prevent blowout much better than the 70° blade angle swirler. In the case of the 30° blade angle swirler, significant flow reversal occurred for both these main recirculation zones, but this was not necessarily a good thing since it is
desirable to anchor the flame in one place and not several (a more compact flame means smaller combustor designs).

The operating equivalence ratio of kerosene for the APA 350 combustor is 2.56. According to Richards and Lefebvre the turbulent flame speeds for liquid kerosene droplets at this equivalence ratio (depending on the Sauter mean diameter) range between $1 - 4m/s$ [19]. From the velocity magnitude plots for the 30° blade angle swirler it was observed that this range of velocities corresponded to a longitudinal position of $D_{sw}$ away from the fuel injector in the lower segment (balloon 5) and a longitudinal position of $0.4\times D_{sw}$ from the fuel injector in the upper segment (balloon 6). This variation in longitudinal position meant that the flame cannot exist at the same position along the combustor length around the entire annulus. The 50° blade angle swirler was somewhat an improvement in this regard with a less significant variation in longitudinal flame position than for the 30° configuration (in terms of blowout limits). The 70° blade angle swirler (figure C.1a) had an even more promising envelope for combustion in terms of flame speeds and the prevention of flashback. According to the axial velocity XY-plots the 30°, 50° and 70° configurations had 4, 4 and 1 positions respectively at which the required flow velocity occurred for an anchored flame. The 30° blade angle swirler had its anchored flame speed positions at $0.65\times D_{sw}$, $D_{sw}$, $2\times D_{sw}$ and $5.6\times D_{sw}$. The 50° swirler had its anchored flame speed positions at $0.87\times D_{sw}$, $2\times D_{sw}$, $2.6\times D_{sw}$ and $3.6\times D_{sw}$. Finally the 70° blade angle swirler had its only anchored flame speed position at $1.7\times D_{sw}$. 

6.1 Conclusions

Figure C.5 shows that 50° blade angle swirler had good fuel/air mixing. 70° swirler had even better mixing however from figure C.10 it is seen that there was excessive vortex shedding directly behind swirler leading to combustion instability. Conversely it was seen that in the $X = 27\, mm$ position for the 30° swirler the vorticity was low. This meant that in this ideal position for the flame kernel a ‘restful’ stable flame could be anchored. Figure C.9 shows that the turbulence intensity for the 30° swirler was sufficient for good atomisation. This was also evidenced by the extremely intense shear layer found at balloon 2 in figure C.2c. Turbulent kinetic energy was lowest in the mixing critical (primary) zone for the 70° swirler as seen in figure C.4. Additional turbulent energy caused by increased blade angle was only observed far downstream of flame position which was not of any practical use.

As swirler blade angle was increased it was found that the combustor pressure loss increased. Large pressure losses typically found in micro-gas turbines of $10 - 12\%$ were calculated. These pressure losses could be reduced by replacing the flat bladed vanes with profiled ones.

A major goal in anchoring the flame is to reduce axial flow velocity and increase residence time. It was found that with increasing swirler blade angle the residence time increased. The most stagnant air flow was found for the 50° and 70° swirler configurations meaning more time for complete combustion and less likelihood of blowout. The 70° swirler had no distinct recirculation zones in the plane running between dilution holes, and only a very short one $(0.9\times D_{sw})$ in the plane running in line with dilution holes. The 50° swirler however had large recirculation zones in both these planes. The lack of orderly recirculation in the 70° swirler configuration and therefore increased turbulence intensity was an indication of complete toroidal vortex breakdown. This was advantageous for mixing but disadvantageous for combustion stability as well as pressure recovery.

Iso-surface plots of the required equivalence ratio for kerosene combustion showed an increased volume of potential combustion envelope for increased blade angle. For the 50°
swirler this envelope was satisfactory in size and maintained an orderly, relatively cylindrical space. For the 70° swirler, despite a larger spread, this volume became disordered with a much larger spread in the azimuthal direction. From axial velocity measurements the 30° and 50° swirler configurations both showed axial flow velocities that corresponded to turbulent flame speeds for kerosene droplets at longitudinal distances that are close (but not too close) to the swirlers of 0.65×Dsw and 0.87×Dsw respectively. For the 70° swirler however the same axial velocity only occurred at a point 1.7×Dsw which was further downstream than the desired position of flame anchoring.

Computational fluid analyses were carried out on 5 different swirler designs. Results were carefully analysed for each. Results for the 40° swirler and 60° swirler designs followed the trends discussed for the other swirler designs. Through a scientific evaluation of the desirable characteristics of efficient combustor swirler design, various parameters were weighed against each other to determine the optimum blade angle for the APA 350 combuster. The 50° vane blade angle was deemed to provide superior performance.

### 6.2 Recommendations

In case the study were extended, repeated or any other similar study conducted in the future, some recommendations are mentioned below.

1. Refine validation CFD model to have more cells for the possibility of improved ϵ turbulence model performance

2. Swirl number intervals were quite large between different chosen blade angle configurations. Hence it would be useful to increase the blade angles by smaller increments. Furthermore focus should be given to the change in configuration that initiates the toroidal recirculation zones: This will only be possible by increasing blade angle in smaller steps

3. The research could be extended to include studies that investigate the reacting region’s equivalence ratio and axial velocities, and comparing this with flame speeds and lean blowout limits to determine whether or not the fuel would actually combust. Furthermore a full combustion analysis can be carried out as an additional study

In terms of improving the APA 350 combuster even further, it is recommended to use (and test) profiled swirler vane blades rather than straight blades. It was shown by Mather and Fraser [25] as well as Syred and Beer [29] that profiled blades have reduced pressure loss and increased efficiency. These studies suggest that flat bladed vanes only be used for combustors with low degree of swirl. In case a low swirl design is selected, flat bladed vanes should be used in the interest of costs and ease of manufacturing. Finally counter rotating swirlers should be tested as appose to co-rotating ones. This is because the higher shear rates produced can contribute to even better mixing. The consequences on combustion stability of such modification should however be investigated fully.
References


[31] University of Utah. Length and time scales in turbulent flows.


Appendix A

Mesh Independence Study

This appendix contains figures relating to the mesh independence study [32] discussed in chapter 3.
Figure A.1: Summarised results for meshes 4, 5 and 8 respectively
Figure A.2: Evolution of mesh quality
Appendix B

Model Validation

This appendix contains pairs of figures for side by side comparison of the velocity cut plots obtained during experimental tests on a combustor with those obtained through CFD. In each pair the experimental result is illustrated above the CFD result. The appendix is organised into two sections, the first comparing results for the base plane (passing through the holes in the combustor liner for all three flow zones) and the second containing velocity cut plots for the three remaining planes discussed in 4.

B.1 Base Plane
Figure B.1: Velocity magnitude for base plane
Figure B.2: Velocity vector plot for base plane
Figure B.3: Axial velocity profile for base plane
B.2 Mixed Planes
Figure B.4: Velocity magnitude for mixed planes
Figure B.5: Velocity vector plot for mixed planes
Figure B.6: Axial velocity profile for mixed planes
Appendix C

Post-Processing Figures

This appendix contains figures relating to post-processing of the CFD study.
Figure C.1: Velocity magnitude with streamlines displayed
Figure C.2: Close-up velocity magnitude
Figure C.3: Pressure magnitude

(a) 70° Swirler blade angle

(b) 50° Swirler blade angle

(c) 30° Swirler blade angle
Figure C.4: Turbulent kinetic energy magnitude
Figure C.5: Kerosene mass fraction
Figure C.6: Velocity vectors
Figure C.7: Swirler pressure loss
<table>
<thead>
<tr>
<th></th>
<th>70 Deg Blade Angle</th>
<th>50 Deg Blade Angle</th>
<th>30 Deg Blade Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>X = 2mm</td>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>X = 7mm</td>
<td><img src="image4.png" alt="Image" /></td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>X = 75mm</td>
<td><img src="image7.png" alt="Image" /></td>
<td><img src="image8.png" alt="Image" /></td>
<td><img src="image9.png" alt="Image" /></td>
</tr>
<tr>
<td>X = 100mm</td>
<td><img src="image10.png" alt="Image" /></td>
<td><img src="image11.png" alt="Image" /></td>
<td><img src="image12.png" alt="Image" /></td>
</tr>
</tbody>
</table>

| SN    | 2.19 | 0.95 | 0.46 |

Figure C.8: Turbulent kinetic energy summary
Figure C.9: Turbulence intensity summary
Figure C.10: Vorticity summary
Figure C.11: Stoichiometric fuel/air ratio volume
Figure C.12: Kerosene mass fractions of 2.5% and 5%
Figure C.13: Iso-vorticity plot at 337501/s
Figure C.14: 2D visualisation of spiraling vortex based on vorticity XY-plots
Figure C.15: XY-Plots of axial velocities
Figure C.16: XY-Plots of combustor pressure

(a) 70° Swirler blade angle

(b) 50° Swirler blade angle

(c) 30° Swirler blade angle
Figure C.17: XY-Plots of vorticity

(a) 70° Swirler blade angle

(b) 50° Swirler blade angle

(c) 30° Swirler blade angle
Figure C.18: XY-Plots of kerosene mass fraction (logarithmic axis)
Figure C.19: XY-Plots of circumferential velocity

(a) 70° Swirler blade angle

(b) 50° Swirler blade angle

(c) 30° Swirler blade angle