Measurements of the flow field in a modern gas turbine combustor

This item was submitted to Loughborough University’s Institutional Repository by the/an author.

Additional Information:

- A Doctoral Thesis. Submitted in partial fulfilment of the requirements for the award of Doctor of Philosophy of Loughborough University.

Metadata Record: https://dspace.lboro.ac.uk/2134/12714

Publisher: © J.P. Griffiths

Please cite the published version.
This item was submitted to Loughborough University as a PhD thesis by the author and is made available in the Institutional Repository (https://dspace.lboro.ac.uk/) under the following Creative Commons Licence conditions.

**Attribution-NonCommercial-NoDerivs 2.5**

You are free:
- to copy, distribute, display, and perform the work

Under the following conditions:

**Attribution.** You must attribute the work in the manner specified by the author or licensor. 

**Noncommercial.** You may not use this work for commercial purposes. 

**No Derivative Works.** You may not alter, transform, or build upon this work. 

- For any reuse or distribution, you must make clear to others the license terms of this work. 
- Any of these conditions can be waived if you get permission from the copyright holder. 

Your fair use and other rights are in no way affected by the above. 

This is a human-readable summary of the Legal Code (the full license).

For the full text of this licence, please go to:
http://creativecommons.org/licenses/by-nc-nd/2.5/
Please note that fines are charged on ALL overdue items.
Measurements of the Flow Field in a Modern Gas Turbine Combustor

by

J.P. Griffiths

Submitted in partial fulfilment of the requirements for the award of
Doctor of Philosophy of Loughborough University
September, 1999
Dedication

To family, friends, and MCFC.
Abstract

A detailed investigation into the aerodynamics of a modern gas turbine combustor is reported in this thesis. The main objectives of this work were to examine the interactions between the various features of the internal flow field, and between the external and internal aerodynamics, and to obtain sufficient flow field data for validation of CFD codes. A new experimental facility was developed to allow optical access for high quality internal and external measurements of the isothermal flow field in a three sector segment of an annular gas turbine combustor whose geometry is typical of the combustors in use in current turbofan engines. A specialised traverse system was designed to enable measurements of the flow field by a three component Laser Doppler Anemometry (LDA) system, and a considerable effort was made to maximise the accuracy of the LDA system. Measurements of three orthogonal mean velocity components and all six Reynolds stresses were obtained throughout a burner sector of the combustor. A set of data has been obtained that is sufficiently extensive for use as a benchmark data set for CFD validation.

Measurements in the feed annuli showed that the behaviour of the flow was as expected. Internal measurements revealed a strong coupling between the flow in the feed annuli and the flow entering the flame tube through primary and secondary ports. Differences in the geometries and flow splits in the inner and outer annuli caused significant differences between the opposed jets inside the flame tube. The initial pitch angle and axial and radial momentum components of the jets were found to be strongly dependent on the ports' feed conditions. Differences between the opposed jets, due to differences in their feed conditions, affected the location of their impingement and the trajectory of the jet fluid after impingement. The impingement process was also found to be unstable.

The centre primary jets, which are downstream of the fuel injector, displayed a dramatically increased sensitivity to their feed conditions, caused by the low pressure in the recirculation induced by the swirler. This caused the jets to be deflected in opposite directions, with no impingement. The flow field in the primary zone was thus substantially altered, with serious implications for the performance of the combustor. These results also demonstrate the importance of coupling the internal and external flows in all experimental and computational models.
Acknowledgements

This work was carried out in the Airflow Laboratory at the Department of Aeronautical and Automotive Engineering and Transport Studies at Loughborough University and was financially supported by Rolls-Royce plc.

The research was supervised by Dr. Jon Carrotte, whose guidance throughout the project is greatly appreciated. In particular, Jon’s enthusiasm for the project, his approachability and his much-valued advice were a huge help. Thanks must also go to Professor Jim McGuirk for his guidance as Director of Research.

Thanks must also go to Jon Carrotte, Jim McGuirk and Adrian Spencer for proof reading the thesis, and for their constructive criticism.

Thanks also to the Airflow Lab technicians, Les Monk, Dave Glover and Bill Niven, for their help in the construction and installation of the rig, traverse and LDA alignment equipment, and for some highly entertaining tea breaks. I would also like to thank several other members of the research group for their assistance during the course of the project, including Paul Denman, Adrian Spencer and Stan Stevens.

Finally, thanks to all my family and friends, who have endured my company during the writing of this thesis. Thanks must also go to Manchester City FC, for providing useful stress relief every other weekend.
List of Contents

Contents

Dedication i
Abstract ii
Acknowledgements iii
List of Contents iv
List of Figures viii
Nomenclature xv

Chapter 1 Introduction

1.1 Evolution of the gas turbine combustion system
1.2 Features of the combustion system
   1.2.1 The diffuser system
   1.2.2 The flame tube
1.3 Combustor external aerodynamics
1.4 Combustor internal aerodynamics
   1.4.1 Flow issuing from the swirler
   1.4.2 Jet flows
      1.4.2.1 Single jet in cross flow
      1.4.2.2 Multiple jets and opposed jets
      1.4.2.3 Parametric investigations
      1.4.2.4 Symmetry and stability of opposed jets
      1.4.2.5 Influence of feed conditions
   1.4.3 Combustor flow field interactions
1.5 Prediction of aerodynamic processes within the combustion system
1.6 The current investigation

Chapter 2 Test Facility Definition

2.1 Combustor geometry
   2.1.1 The nominal VULCAN Phase 5 combustor geometry
   2.1.2 Instrumentation requirements
   2.1.3 Sidewall design and flame tube mounting
   2.1.4 Turbine cooling flow simulation
List of Contents

2.1.4.1 Design of the metering plate
   for the inboard bleed duct 34
2.1.4.2 Design of the metering plate
   for the outboard bleed duct 37
2.1.5 Deviation of flame tube from nominal design 38

2.2 Test facility installation 41
   2.2.1 Inlet and exhaust ducting 41

3. Instrumentation and experimental procedure 43
   3.1 Laser Doppler anemometry 44
      3.1.1 Introduction to Laser Doppler Anemometry 44
      3.1.2 The LDA system 47
      3.1.3 The Burst Spectrum Analyser (BSA) 49
      3.1.4 Seeding 50
      3.1.5 LDA traverse system 51
      3.1.6 LDA alignment and transformation matrix calculation 54
         3.1.6.1 Alignment of the probes to obtain coincident measurement volumes 54
         3.1.6.2 Calculation of the transformation matrix 57
   3.2 External flow field measurements 58
      3.2.1 Button hook probes 58
      3.2.2 Hot wire anemometry 60
      3.2.3 Flow visualisation 61
   3.3 Data acquisition and reduction 62
      3.3.1 Operation of the test facility and traverse systems 62
      3.3.2 Operation of the LDA system 63
      3.3.3 Data reduction and analysis 66
         3.3.3.1 Data measured by button hook probe 66
         3.3.3.2 Data measured by hot wire anemometry 67
         3.3.3.3 Data measured by Laser Doppler Anemometry 68
   3.4 Estimate of experimental errors 71
      3.4.1 Pressure measurements 71
List of Contents

3.4.2 LDA measurements 72
  3.4.2.1 Statistical errors 72
  3.4.2.2 Systematic errors 73
  3.4.2.3 Alignment errors 78
  3.4.2.4 Processor resolution 78
  3.4.2.5 Overall estimate of LDA errors 79

4. The Experimental Programme 80
  4.1 Test Facility Commissioning Tests 80
    4.1.1 Pre-diffuser 80
    4.1.2 Outer feed annulus 83
    4.1.3 Inner feed annulus 86
    4.1.4 Annulus bleed flows 87
  4.2 LDA system tests 88
  4.3 The experimental programme 91

5. Results and Discussion 96
  5.1 External flow field 96
  5.2 Internal flow field 98
    5.2.1 Area traverse results 99
      5.2.1.1 Primary zone 99
      5.2.1.2 Secondary zone 103
      5.2.1.3 Flow development downstream 106
        of the secondary ports
    5.2.2 Port exit measurements 108
      5.2.2.1 Central primary jets 108
      5.2.2.2 Sector-edge primary jets 116
      5.2.2.3 Secondary jets 119
    5.2.3 Jet trajectory measurements 125
      5.2.3.1 Analysis of outer centre primary jet trajectory 131
    5.3 Turbulence in the internal flow field 139
    5.4 Closure 144
# List of Contents

6. Conclusions and Recommendations for Further Work 146
   6.1 Conclusions 146
   6.2 Recommendations 148

References 150
Figures 157
Appendix 291
List of Figures

<table>
<thead>
<tr>
<th>Number</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig. 1.1</td>
<td>Gas generator</td>
<td>158</td>
</tr>
<tr>
<td>Fig. 1.2</td>
<td>Application of gas generator in aircraft engines</td>
<td>158</td>
</tr>
<tr>
<td>Fig. 1.3</td>
<td>Evolution of the combustion system</td>
<td>159</td>
</tr>
<tr>
<td>Fig. 1.4</td>
<td>Alternative forms of the combustion system</td>
<td>160</td>
</tr>
<tr>
<td>Fig. 1.5</td>
<td>Features of a typical combustion system</td>
<td>161</td>
</tr>
<tr>
<td>Fig. 1.6</td>
<td>Alternative diffuser systems</td>
<td>161</td>
</tr>
<tr>
<td>Fig. 1.7</td>
<td>Dependence of combustion efficiency on combustor loading</td>
<td>162</td>
</tr>
<tr>
<td>Fig. 1.8</td>
<td>Primary zone with opposed jets and swirler</td>
<td>162</td>
</tr>
<tr>
<td>Fig. 1.9</td>
<td>Pre-diffuser stability chart</td>
<td>163</td>
</tr>
<tr>
<td>Fig. 1.10</td>
<td>Effect of pre-diffuser cant angle on outer wall pressure gradient</td>
<td>163</td>
</tr>
<tr>
<td>Fig. 1.11</td>
<td>Three dimensional pre-diffuser inlet flow due to OGV wake</td>
<td>164</td>
</tr>
<tr>
<td>Fig. 1.12</td>
<td>Flow around head of flame tube</td>
<td>164</td>
</tr>
<tr>
<td>Fig. 1.13</td>
<td>Recirculation in a swirling flow field</td>
<td>165</td>
</tr>
<tr>
<td>Fig. 1.14</td>
<td>Vortex systems associated with a single jet in cross flow</td>
<td>165</td>
</tr>
<tr>
<td>Fig. 1.15</td>
<td>Shear layer vortices</td>
<td>166</td>
</tr>
<tr>
<td>Fig. 1.16</td>
<td>Vortex systems associated with multiple jets in cross flow</td>
<td>166</td>
</tr>
<tr>
<td>Fig. 1.17</td>
<td>Impingement of strongly opposed jets in a cross flow</td>
<td>167</td>
</tr>
<tr>
<td>Fig. 1.18</td>
<td>Definition of variables in the jet</td>
<td>167</td>
</tr>
<tr>
<td>Fig. 1.19</td>
<td>Impingement of weakly opposed jets</td>
<td>168</td>
</tr>
<tr>
<td>Fig. 1.20</td>
<td>Flow in tubular combustor</td>
<td>169</td>
</tr>
<tr>
<td>Fig. 1.21</td>
<td>Sector rig configuration</td>
<td>169</td>
</tr>
<tr>
<td>Fig. 2.1</td>
<td>VULCAN Phase 5 combustion system</td>
<td>170</td>
</tr>
<tr>
<td>Fig. 2.2</td>
<td>VULCAN Flame Tube</td>
<td>170</td>
</tr>
<tr>
<td>Fig. 2.3</td>
<td>Fuel Injector and Swirler</td>
<td>171</td>
</tr>
<tr>
<td>Fig. 2.4</td>
<td>Circumferential distribution of flame tube ports</td>
<td>171</td>
</tr>
<tr>
<td>Fig. 2.5</td>
<td>Heat shield cooling flows</td>
<td>172</td>
</tr>
<tr>
<td>Fig. 2.6</td>
<td>Angled effusion cooling</td>
<td>172</td>
</tr>
<tr>
<td>Fig. 2.7</td>
<td>Principal regions of interest</td>
<td>172</td>
</tr>
<tr>
<td>Fig. 2.8</td>
<td>Pre-diffuser stability chart</td>
<td>173</td>
</tr>
<tr>
<td>Fig. 2.9</td>
<td>3D LDA probe arrangement</td>
<td>173</td>
</tr>
</tbody>
</table>
List of Figures

Fig.2.10  Rotation of probes to maximise access to centre sector  174
Fig.2.11  Omission of skirts from test facility geometry  175
Fig.2.12  Modifications to inboard bleed offtakes and casing profile  176
Fig.2.13  Sidewall window cooling system  177
Fig.2.14  Section through sidewall  177
Fig.2.15  Outboard bleed offtakes  178
Fig.2.16  Bleed System  178
Fig.2.17  Section through outer bleed duct  178
Fig.2.18  Test Facility General Arrangement  179
Fig.2.19  Casing Specification  180
Fig.2.20  Window Specification  181
Fig.2.21  Bleed System Specification  182
Fig.2.22  Inboard bleed system  183
Fig.2.23  Orifice with chamfered inlet  183
Fig.2.24  Inboard metering plate  183
Fig.2.25  Outboard bleed system  184
Fig.2.26  Outboard metering plate  184
Fig.2.27  Distortion of sectored flame tube  184
Fig.2.28  Test facility geometry measurements  185
Fig.2.29  Circumferential location of primary ports  185
Fig.2.30  Circumferential location of secondary ports  186
Fig.2.31  Test facility installation  186
Fig.2.32  Inlet duct  187
Fig.2.33  Exhaust duct  187
Fig.2.34  Window cooling air inlet  188
Fig.3.1  Laser beams crossing to form interference fringes  189
Fig.3.2  Doppler signal  189
Fig.3.3  Direction of measured velocity component  189
Fig.3.4  Three component LDA system  190
Fig.3.5  LDA Optical arrangement  190
Fig.3.6  Backscatter and cross coupled arrangements  191
Fig.3.7  Seed delivery apparatus  191
### List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig.3.8</td>
<td>Sampling of the burst by the BSA</td>
<td>192</td>
</tr>
<tr>
<td>Fig.3.10</td>
<td>Rotation of probes to maximise access to centre sector</td>
<td>192</td>
</tr>
<tr>
<td>Fig.3.10</td>
<td>Rotation of probes to maximise access to centre sector</td>
<td>193</td>
</tr>
<tr>
<td>Fig.3.11</td>
<td>Slide ring system for circumferential traverse</td>
<td>194</td>
</tr>
<tr>
<td>Fig.3.12</td>
<td>Frame supporting test facility and traverse</td>
<td>194</td>
</tr>
<tr>
<td>Fig.3.13</td>
<td>Test facility support structure</td>
<td>195</td>
</tr>
<tr>
<td>Fig.3.14</td>
<td>Addition of rotary stage to LDA probe traverse</td>
<td>196</td>
</tr>
<tr>
<td>Fig.3.15</td>
<td>Support, rotation and movement of LDA probes</td>
<td>197</td>
</tr>
<tr>
<td>Fig.3.16</td>
<td>Pinhole meter</td>
<td>198</td>
</tr>
<tr>
<td>Fig.3.17</td>
<td>Probe alignment apparatus</td>
<td>198</td>
</tr>
<tr>
<td>Fig.3.18</td>
<td>Measurement volume locating plate</td>
<td>199</td>
</tr>
<tr>
<td>Fig.3.19</td>
<td>Location of probes and window for alignment of probes</td>
<td>199</td>
</tr>
<tr>
<td>Fig.3.20</td>
<td>Probe alignment procedure</td>
<td>200</td>
</tr>
<tr>
<td>Fig.3.21</td>
<td>Calculation of the transformation matrix</td>
<td>201</td>
</tr>
<tr>
<td>Fig.3.22</td>
<td>Button hook probe operation</td>
<td>202</td>
</tr>
<tr>
<td>Fig.3.23</td>
<td>Button hook probe calibration</td>
<td>202</td>
</tr>
<tr>
<td>Fig.3.24</td>
<td>Manual traverse for button hook and hot wire probes</td>
<td>203</td>
</tr>
<tr>
<td>Fig.3.25</td>
<td>Motorised traverse for button hook probe</td>
<td>203</td>
</tr>
<tr>
<td>Fig.3.26</td>
<td>Pressure measurement locations</td>
<td>203</td>
</tr>
<tr>
<td>Fig.3.27</td>
<td>Circumferential traverse of button hook probe</td>
<td>204</td>
</tr>
<tr>
<td>Fig.3.28</td>
<td>Hot wire probe</td>
<td>204</td>
</tr>
<tr>
<td>Fig.3.29</td>
<td>Wool tuft</td>
<td>204</td>
</tr>
<tr>
<td>Fig.3.30</td>
<td>Location of inlet pitot and static tapping</td>
<td>205</td>
</tr>
<tr>
<td>Fig.3.31</td>
<td>Typical velocity histogram</td>
<td>205</td>
</tr>
<tr>
<td>Fig.3.32</td>
<td>Bimodal velocity histogram</td>
<td>206</td>
</tr>
<tr>
<td>Fig.3.33</td>
<td>Effect of residence time weighting</td>
<td>206</td>
</tr>
<tr>
<td>Fig.4.1</td>
<td>Pre-diffuser inlet velocity profiles</td>
<td>207</td>
</tr>
<tr>
<td>Fig.4.2</td>
<td>Boundary layer on inner wall at pre-diffuser inlet</td>
<td>207</td>
</tr>
<tr>
<td>Fig.4.3</td>
<td>Boundary layer on outer wall at pre-diffuser inlet</td>
<td>208</td>
</tr>
<tr>
<td>Fig.4.4</td>
<td>Boundary layer on inner wall at pre-diffuser inlet</td>
<td>208</td>
</tr>
<tr>
<td>Fig.4.5</td>
<td>Pre-diffuser exit velocity profiles - no inlet trip</td>
<td>209</td>
</tr>
<tr>
<td>Fig.4.6</td>
<td>Pre-diffuser exit velocity profile - centre line, trip inserted</td>
<td>209</td>
</tr>
</tbody>
</table>
List of Figures

Fig. 4.7 Pre-diffuser exit boundary layers 210
Fig. 4.8 Outer annulus sidewall separation 210
Fig. 4.9 Outer annulus modifications 211
Fig. 4.10 Effect of splitter plates on outer annulus static pressure distribution 211
Fig. 4.11 Condition of sidewall boundary layers in inner annulus 212
Fig. 4.12 Comparison of inner and outer annulus static pressure distributions 213
Fig. 4.13 Measurement of pressure drop across metering plates 213
Fig. 4.14 Typical velocity histogram 214
Fig. 4.15 Bimodal velocity histogram 214
Fig. 4.16 External measurement locations 215
Fig. 4.17 Initial internal measurement locations and typical measurement grid 215
Fig. 4.18 Port exit measurement locations and typical measurement grid 216
Fig. 4.19 Final internal measurement locations 216
Fig. 5.1 Pre-diffuser inlet and exit velocity profiles 217
Fig. 5.2 Turbulence intensity at pre-diffuser inlet 218
Fig. 5.3 Pre-diffuser exit static pressure profile 218
Fig. 5.4 Normalised mean velocity at planes XI2 and XO2 219
Fig. 5.5 Circumferentially averaged velocity profiles
   Outer annulus entry (XO1) and exit (XO2) 220
Fig. 5.6 Circumferentially averaged velocity profiles
   Inner annulus entry (XI1) and exit (XI2) 220
Fig. 5.7 Coordinate system used in internal measurements 221
Fig. 5.8 Contours of normalised mean v component - Plane I37 222
Fig. 5.9 Contours of normalised mean u component - Plane I37 223
Fig. 5.10 Mean velocity vectors - Plane I37 224
Fig. 5.11 Contours of normalised mean u component - Plane I22 225
Fig. 5.12 Mean velocity vectors and contours of axial vorticity component
   - Plane I22 226
Fig. 5.13 Location of bimodal histograms - Plane I22 227
Fig. 5.14 Location of bimodal histograms - Plane I37 228
Fig. 5.15 Contours of normalised mean v component - Plane I80 229
Fig. 5.16 Contours of normalised mean u component - Plane I80 230
List of Figures

Fig. 5.17 Mean velocity vectors - Plane I80
Fig. 5.18 Location of bimodal histograms - Plane I80
Fig. 5.19 Contours of normalised mean v component - Plane I59
Fig. 5.20 Contours of normalised mean u component - Plane I59
Fig. 5.21 Mean velocity vectors - Plane I59
Fig. 5.22 Location of bimodal histograms - Plane I59
Fig. 5.23 Contours of normalised mean u component - Plane I100
Fig. 5.24 Contours of normalised mean v component - Plane I100
Fig. 5.25 Location of bimodal histograms - Plane I100
Fig. 5.26 Contours of normalised mean u component - Plane I120
Fig. 5.27 Contours of normalised mean v component - Plane I120
Fig. 5.28 Location of bimodal histograms - Plane I120
Fig. 5.29 Contours of normalised mean v component - Inner centre primary port
Fig. 5.30 Contours of normalised mean v component - Outer centre primary port
Fig. 5.31 Typical chute geometry
Fig. 5.32 Pitch angle coordinate system
Fig. 5.33 Contours of jet pitch angle - Inner centre primary port
Fig. 5.34 Contours of jet pitch angle - Outer centre primary port
Fig. 5.35 Velocity profiles in feed annuli upstream of centre primary ports
Fig. 5.36 Visualisation of flow entering centre primary ports
Fig. 5.37 Discharge coefficients of centre primary ports
Fig. 5.38 Location of bimodal histograms - Inner centre primary port
Fig. 5.39 Location of bimodal histograms - Outer centre primary port
Fig. 5.40 Bimodal v component histogram - Outer centre primary port
Fig. 5.41 Contours of normalised mean v component
  - Inner sector edge primary port
Fig. 5.42 Contours of normalised mean v component
  - Outer sector edge primary port
Fig. 5.43 Contours of jet pitch angle - Inner sector edge primary port
Fig. 5.44 Contours of jet pitch angle - Outer sector edge primary port
Fig. 5.45 Pitch angles of primary jets - Plotted against cut-off velocity
Fig. 5.46 Discharge coefficients of primary jets - Plotted against cut-off velocity
List of Figures

Fig.5.47 Velocity profiles in feed annuli upstream of centre and sector edge primary ports 254
Fig.5.48 Location of bimodal histograms - Inner sector edge primary port 255
Fig.5.49 Location of bimodal histograms - Outer sector edge primary port 255
Fig.5.50 Contours of normalised mean v component - Inner secondary port 256
Fig.5.51 Contours of normalised mean v component - Outer secondary port 256
Fig.5.52 Contours of jet pitch angle - Inner secondary port 257
Fig.5.53 Contours of jet pitch angle - Outer secondary port 257
Fig.5.54 Discharge coefficients of primary and secondary ports 258
Fig.5.55 Pitch angles of primary and secondary jets 258
Fig.5.56 Axial component of momentum - primary and secondary jets 259
Fig.5.57 Axial component of momentum per unit geometric area - primary and secondary jets 259
Fig.5.58 Radial component of momentum - primary and secondary jets 260
Fig.5.59 Radial component of momentum per unit geometric area - primary and secondary jets 260
Fig.5.60 Visualisation of flow entering secondary ports 261
Fig.5.61 Location of bimodal histograms - Inner secondary port 262
Fig.5.62 Location of bimodal histograms - Outer secondary port 262
Fig.5.63 Trajectories of centre primary jets 263
Fig.5.64 Trajectories of sector-edge primary jets 265
Fig.5.65 Trajectories of secondary jets 267
Fig.5.66 Calculation of pressure differential across jet 269
Fig.5.67 Exported streamline data and fitted polynomial curve 269
Fig.5.68 Variation in radius of curvature along the streamline 270
Fig.5.69 Estimated static pressure differential across the jet 270
Fig.5.70 Fuel injector and swirler 271
Fig.5.71 CFD prediction of flow in Phase 5 combustor with plain ports 272
Fig.5.72 CFD prediction of flow in Phase 5 combustor with chuted ports 272
Fig.5.73(a) Contours of turbulence intensity at plane OPC 273
Fig.5.73(b) Contours of turbulent kinetic energy at plane OPC 273
Fig.5.74 Anisotropy at plane OPC 274
List of Figures

Fig.5.75(a) Contours of uu normal stress at plane OPC 275
Fig.5.75(b) Contours of vv normal stress at plane OPC 275
Fig.5.76  u and v component histograms - Outer centre primary port 276
Fig.5.77 Locations of centre, secondary and sector-edge radial planes 277
Fig.5.78 Normalised turbulent kinetic energy - Centre radial plane 277
Fig.5.79 Normalised turbulent kinetic energy - Secondary radial plane 278
Fig.5.80 Normalised turbulent kinetic energy - Sector-edge radial plane 278
Fig.5.81 Bimodal v component histogram obtained at location of weak impingement between centre primary jets 279
Fig.5.82(a) Normalised uu normal stress - Secondary radial plane 280
Fig.5.82(b) Normalised vv normal stress - Secondary radial plane 280
Fig.5.83 Normalised uu normal stress - Plane I22 281
Fig.5.84 Normalised vv normal stress - Plane I22 281
Fig.5.85 Normalised ww normal stress - Plane I22 282
Fig.5.86 Normalised turbulent kinetic energy - Plane I22 282
Fig.5.87 Normalised ww normal stress - Plane I80 283
Fig.5.88 Normalised turbulent kinetic energy - Plane I80 283
Fig.5.89 Anisotropy at centre radial plane 284
Fig.5.90 Anisotropy at secondary radial plane 285
Fig.5.91 Anisotropy at sector-edge radial plane 286
Fig.5.92 Development of outer centre primary jet profile 287
Fig.5.93(a) Contours of uv shear stress at plane OPC 288
Fig.5.93(b) Contours of vw shear stress at plane OPC 288
Fig.5.94 Contours of uv shear stress at centre radial plane 289
Fig.5.95 Contours of uv shear stress at secondary radial plane 289
Fig.5.96 Contours of uv shear stress at sector-edge radial plane 290
Nomenclature

A area
C LDA calibration factor
$C_D$ coefficient of discharge
$C_p$ static pressure recovery coefficient
$f_D$ Doppler frequency
$f_N$ frequency resolution of BSA
$f_s$ BSA sampling frequency
G momentum flux
H boundary layer shape parameter
H mixing duct height
k turbulent kinetic energy
J momentum flux ratio
K button hook probe calibration factor
$m$ mass flow
N number of bursts in a sample
p static pressure
$\bar{p}$ mass weighted mean static pressure
$P_{pa}$ pseudo-static pressure
P total pressure
$\bar{P}$ mass weighted mean total pressure
q dynamic pressure
r radial co-ordinate
S swirl number
S jet spacing
$Tu$ turbulence intensity
$V_c$ velocity of cross flow
$V_j$ velocity of jet
u axial mean velocity component
$\bar{u}$ bulk average velocity
$u'$ instantaneous fluctuating velocity component
$uu, vv, ww$ Reynolds normal stresses
Nomenclature

\(uv, vw, uw\)  Reynolds shear stresses
\(v\)          radial mean velocity component
\(w\)          circumferential mean velocity component
\(x\)          axial co-ordinate
\(\alpha\)     jet pitch angle
\(\delta_{\text{bb}}\)  boundary layer thickness
\(\delta^*\)    displacement thickness
\(\varepsilon\)  rate of dissipation of turbulent kinetic energy
\(\lambda\)     stagnation pressure loss coefficient
\(\lambda\)     wavelength
\(\theta\)      angle
\(\theta\)      combustor loading parameter
\(\theta\)      momentum thickness
\(\rho\)        density
\(\Omega_x\)    axial vorticity component

subscripts
\(a\)          blue
\(c\)          cross flow
\(G\)          green
\(\text{in}\)   inlet
\(j\)          jet
\(\text{norm}\) normalised value
\(r\)          radial component
\(x\)          axial component
\(v\)          violet
\(a\)          ambient value
Introduction

1. Introduction

In its turbojet, turbofan, turboprop and turboshift forms the gas turbine engine now powers almost all new aircraft, whilst other applications include electricity generation and propulsive power for ships. Despite the variety of engines and their applications, all gas turbine engines feature the same three basic core components: a compressor, a combustor and a turbine (Fig.1.1).

The compressor provides high pressure air to the turbine, which expands the air and extracts energy to drive the shaft that drives the compressor. However, the losses that occur in the compressor and turbine prevent the system from operating without an input of energy. A combustion chamber is therefore inserted, between the compressor and turbine, within which fuel is introduced and burned. The heat thus added provides the necessary energy for the turbine to drive the compressor. However, by burning extra fuel an excess of heat energy is generated so that, after expansion by the turbine, a surplus of energy is available in the form of gas at high temperature and pressure. This gives rise to the term 'gas generator' and useful work can be extracted from this high temperature and pressure gas stream in a variety of ways.

In the simplest variant of the gas turbine engine, the turbojet (Fig.1.2), the excess energy is converted to kinetic energy by expansion through a propelling nozzle, thus producing a high velocity gas stream to propel an aircraft. The thrust due to the increased momentum imparted to the air flow through the engine is given by \( F = \dot{m}(V_j - V_a) \), where \((V_j - V_a)\) is the increase in the velocity of the air at the nozzle exit over that of the aircraft. Alternatively the nozzle may be replaced with another turbine to drive a shaft. In turboprop and turbofan (Fig.1.2) engines the turbine is used to drive a propeller or fan. This extra complexity can be justified by considering the fact that the propulsive efficiency of the engine, defined as

\[
\eta_p = \frac{2}{1 + (V_j/V_a)}
\]

can be optimised by matching the speed of the aircraft to the velocity being imparted to the air by the engine. Relative to the turbojet this gives a small increase in velocity to a large mass flow, thus producing thrust with a high propulsive efficiency, making these engines very popular for commercial aircraft. Alternatively the shaft may be used to drive the rotors of a helicopter, or to generate electricity. It can therefore be seen that there is a variety of gas turbine engine configurations, but the need for a combustion system is
Introduction

common to all. This thesis reports on an experimental investigation of the isothermal flow field in a combustion system that is typical of those used in modern turbofan engines. The investigation is introduced in this chapter. Sections 1.1 and 1.2 describe the evolution and various features of the combustion system. Sections 1.3, 1.4 and 1.5 summarise the information currently available on the flow field in the combustion system and its prediction, highlighting the features of the combustion system that must be modelled and the areas where investigation is needed to improve current knowledge of the flow field, and section 1.6 introduces the investigation with which this thesis is concerned.

1.1 Evolution of the gas turbine combustion system

Since all aircraft engines must fulfil similar objectives their combustors share the same basic geometry. Its evolution is illustrated in Fig.1.3 (from Lefebvre, 1983) and described below.

The pressure loss that arises from the addition of heat to an air flow is directly proportional to the square of the flow velocity. As the compressor outlet velocity is typically between 160 and 200m/s, burning fuel in a straight duct (Fig.1.3a) would incur a substantial pressure loss, so a diffuser must be used (Fig.1.3b) to reduce the loss to an acceptable level. Furthermore, in order to sustain combustion, that is to prevent the need for continuous ignition, the flow velocity must not exceed the flame speed and so the flame must be sheltered. This can be achieved by creating a flow reversal with a baffle, as in Fig.1.3c. However this system is still not viable, as the air/fuel ratio that must be used to yield a desirable temperature rise is too great for the mixture to be flammable. This problem is solved by the staged introduction of air through a perforated liner, or 'flame tube' (Fig.1.3d). The air required for combustion is injected through the first row of holes to produce a recirculating flow where combustion can be sustained. This region is known as the primary zone. Further air is injected downstream in the 'dilution zone' to mix with the hot combustion products and reduce the temperature to a level that is acceptable to the turbine. An 'intermediate' or 'secondary' zone may be inserted between the primary and dilution zones, to allow for the combustion of any unburned fuel and the recovery of dissociated combustion products. To enhance the recirculation in the primary zone and so improve flame stabilisation and mixing, air is usually injected through a swirler in the head of the flame tube.
Introduction

The design of the combustion system must meet a number of stringent and conflicting criteria. Perhaps the fundamental requirement is to maximise the efficiency of the combustion process by maximising combustion efficiency and minimising pressure losses. However this requirement must be met whilst meeting strict legislation limiting the emissions of pollutant combustion products such as smoke, unburned fuel, carbon monoxide (CO) and oxides of nitrogen (NOx). In addition, the temperature distribution at the combustor's exit must satisfy the demands of the downstream turbine material. Additional requirements for aircraft engine applications include reliable ignition and stable operation over the wide range of pressures encountered in the aircraft's flight envelope. Finally, weight and drag are key considerations in aircraft design. The length of the combustor must be minimised to reduce the weight of the engine, while the diameter must not be such that it increases the diameter of the engine, and thereby increases the drag.

Of the requirements discussed above, it is the need to minimise pollutant emissions that has become the main consideration in recent years with the increase in public awareness of environmental issues and the resulting introduction of increasingly stringent anti-pollution legislation. This is having a major impact on the design of the combustion system to meet both current and projected future emissions legislation. The reduction of pollutant emissions is achieved by good flame management and good aerodynamic design.

1.2 Features of the combustion system

In most applications the combustion system is arranged around the engine axis in one of three forms, as illustrated by Fig.1.4. A tubular combustor consists of a cylindrical flame tube inside a cylindrical casing, a number of combustors being mounted around the engine axis. Tubular combustors are robust and easy to design since the airflow and fuel flow can easily be matched, and such designs were often used in early engines. However the tubular combustor tends to be long and heavy and incurs a high pressure loss, with each chamber also needing its own igniter. An annular combustor, whereby an annular flame tube is mounted inside an annular casing, concentric with the engine's axis, is aerodynamically cleaner than the tubular system, thereby incurring a lower pressure loss. It is shorter and thus lighter, and fewer igniters are needed as the combustor 'lights round'
when the fuel from one injector is lit. However it is difficult to match the air flow to the fuel flow and it is inherently weaker, with a large buckling load on the outer liner, and the necessary strengthening offsets some of the weight advantage. The third type, the tuboannular system, uses cylindrical liners inside an annular casing, combining the compactness of the annular system with the advantages of the tubular system. However the design of the diffuser is difficult and more igniters are needed as it cannot light round. Although tuboannular combustors have been extensively used in the past, most modern turbofan engines employ annular combustion systems. Annular combustors will therefore be in production for some time and are of continuing interest. However, unlike the tubular combustor, the aerodynamic processes that occur within an annular combustor are not well understood at present. This thesis is therefore concerned only with the annular type combustion system.

To meet the various conflicting and stringent requirements discussed in section 1.1, most modern annular combustion systems share the same basic features. The main components of a typical combustion system are shown in Fig. 1.5 and their functions are described below.

1.2.1 The diffuser system
As already noted, the total pressure loss incurred by burning fuel in a stream of air at typical compressor outlet velocities is substantial. Hence the air must be slowed down by placing a diffuser upstream of the flame tube. Because the introduction of air to the flame tube must be staged, by introducing air through the head of the flame tube and through multiple rows of holes in the liners, the diffuser system must also distribute the air correctly. As with all engine components this must be done with the minimum possible length and pressure loss.

A common diffuser design in older engines, including many still in service, is the faired diffuser (Fig. 1.6a), which attempts to achieve a gradual reduction in velocity in a controlled manner, without separation of the flow from the diffuser's walls. Diffusion is achieved in a number of sub-diffusers and a low pressure loss is incurred. However, because of the low passage heights, the system is very sensitive to variations due to manufacturing tolerances and thermal expansion, and the system is sensitive to variations in the inlet velocity profile. Carrotte et al (1993) found that a short faired diffuser became
unstable in off-design conditions, with flow switching between the inner and outer feed annuli. An alternative to the faired diffuser is the dump diffuser system (Fig.1.6b), which was developed to counter these problems. The pre-diffuser, a short conventional diffuser whose walls project into a large dump chamber, reduces the air velocity by about 40% before the flow separates to enter the dump chamber. The flow is left to divide and flow around the liner, further diffusion occurring before the flow enters the feed annuli. Because the separation is fixed at the pre-diffuser outlet and the recirculations in the dump cavities stabilise the separated flow, the dump diffuser system is insensitive to variations in Mach number, Reynolds number and inlet velocity profile. Unlike the faired system, in which diffusion is achieved in a gradual and controlled manner by using passages with a small divergence angle to prevent separation, much of the diffusion is achieved by a free surface expansion in the dump cavity. The dump diffuser system is therefore substantially shorter than the faired diffuser system. In pressure recovery terms the performance of the dump diffuser system is comparable to the faired diffuser, however the effects of the separation and recirculation in the dump chamber and the curvature of the flow around the head of the flame tube incur a greater pressure loss. Because of the system's inherent stability, and the saving in length and weight, the dump diffuser system has been adopted in many current turbofan engines, such as the successful Rolls-Royce Trent series.

In addition to reducing the flow velocity the diffuser system must accommodate several other requirements. For example, it must provide a uniform distribution of flow into the feed annuli, from which air is supplied to various flame tube features. Bleed offtakes are often situated in the feed annuli to supply cooling air to the nozzle guide vanes (NGVs) and the first stage turbine blades. It is also common practice to bleed air from the outer dump cavity to feed air conditioning and cabin pressurisation systems. Furthermore, it is also necessary to pass structural loads, which arise from pressure loads on the NGVs and combustor, across the gas passage. This has often been achieved by designing the compressor OGVs to meet both aerodynamic and structural requirements, however the trend of increasing flame tube depth and engine pressure ratio has led to structural demands that exceed the limited load carrying capacity of the OGVs. A recent development is the use of radial struts situated in the pre-diffuser that carry the loads, downstream of aerodynamically optimised OGVs. Because the adverse pressure gradient
in the pre-diffuser would promote flow separation from a streamlined strut, the strut's thickness increases along its length to fix the separation at the blunt trailing edge. Typically one strut is used per burner sector\(^1\), often located midway between burners. In addition to these struts, a mounting pin must be used to secure the flame tube. This is often situated at the front of the flame tube, connecting the flame tube head to the outer casing to allow rearwards thermal expansion of the flame tube. As shown by Fig.1.5, the mounting pin is often situated in the wake of the fuel injector feed arm to reduce the blockage presented to the flow entering the outer feed annulus.

1.2.2 The flame tube

Of the combustor design objectives discussed in section 1.1, the flame tube geometry is principally determined by the requirement for reliable ignition and operation at all operating conditions and the need to minimise pollutant emissions. Both are regulatory requirements and must be met. The combustion efficiency is a function of the loading parameter \( \theta \) (Lefebvre, 1983), derived from

\[
\theta = \frac{P_3^{1.75} A_{\text{ref}} D_{\text{ref}}^{0.75} (T_3)^{1.59}}{m_A}
\]

where \( P_3 \) and \( T_3 \) are the temperature and pressure at combustor inlet, \( A_{\text{ref}} \) and \( D_{\text{ref}} \) are the maximum cross-sectional area and depth of the combustor, and \( m_A \) is the mass flow of air through the combustor. The relationship between combustion efficiency and loading parameter for a range of combustors is illustrated by Fig.1.7 (Lefebvre, 1983) from which it can be seen that a high loading parameter is necessary for efficient combustion. The loading parameter is reduced at high altitude due to the reduction in temperature and pressure, resulting in a decrease in combustion efficiency. The area and depth of the flame tube are therefore chosen to achieve an acceptable combustion efficiency at high altitude.

The primary and secondary zones are sized to achieve acceptable pollutant emissions characteristics. The principal factors that determine the combustor's emissions characteristics include the temperature and fuel/air ratio in the primary zone and the residence time in the primary zone. The length of the primary zone and the amount of air

\(^1\) A burner sector is defined as the smallest cyclically repeatable sector of the flame tube, the combustor being notionally divided into a number of equal sectors with one fuel injector per sector. A combustor with 24 fuel injectors, for example, contains 24 equal sectors of 15°. It is common practice to investigate the aerodynamics of only a single sector of the combustor and diffuser, making the assumption that the flow field is identical in every sector.
Introduction

allocated to the primary zone are therefore crucial. The length and temperature of the secondary zone are also important factors; the temperature, which is determined by the amount and distribution of air supplied to the secondary zone, must be sufficiently high to allow the recombination of dissociated combustion products, including carbon monoxide, and combustion of any unburned fuel, although too great a temperature would result in equilibrium concentrations of dissociated species entering the dilution zone. The length of the secondary zone must be sufficient to allow this process to be completed before the hot combustion products enter the dilution zone and the reaction is quenched.

The length of the dilution zone is determined by the requirement for an acceptable temperature distribution at turbine inlet. It must be sufficiently long to achieve the desired temperature distribution, the length being dependent on the amount of air allocated to the dilution zone and the mixing rate.

A large scale recirculation must be created in the primary zone for flame stabilisation. This is often achieved by the combined use of opposed primary jets and a swirler (Fig. 1.8). The swirler is usually an integral part of the fuel injector, and consists of a number of concentric passages that contain angled vanes to impart a rotational component to the flow. The fuel spray, injected into the swirling flow, is thus effectively distributed in the primary zone. Effective mixing of the fuel spray, fresh air and recirculated hot combustion products maintains continuous, efficient combustion and minimises the formation of NOx and soot.

The primary, secondary and dilution jets are injected into the flame tube through rows of ports in the inner and outer liners (Fig. 1.5). These ports may be plain holes, but are frequently plunged and chuted to increase their discharge coefficients and thus the penetration of the jets.

In addition to supplying the desired distribution of air to the primary, secondary and dilution zones, the designer must also allocate air to cool the liners. Fresh air is injected through slots onto the inside of the liner to produce a film of cool air that protects the liner from the heat of the combusting gas. The film is destroyed by mixing with the hot gas, so a series of slots along the liner must be used. This may require too great a proportion of the available air, so alternative methods may be used. One such method is effusion cooling, whereby cooling air enters through porous liners. This method can be
more economical than film cooling if the effusion cooling holes in the liners are correctly
designed.

In the past, the aerodynamics of the external flow field (the diffuser system,
including the pre-diffuser, dump cavity and feed annuli) and the internal flow field (the
flow field inside the combustor) have mostly been investigated separately. Sections 1.3
and 1.4 discuss the aerodynamic processes that occur in the external and internal flow
fields respectively.

1.3 Combustor external aerodynamics

The principal feature of the flow in the pre-diffuser is its deceleration and the associated
rise in static pressure. An adverse pressure gradient therefore exists, which promotes
growth of the boundary layers on the pre-diffuser walls. The degree of diffusion that can
be attempted in a given pre-diffuser length is therefore limited by the maximum pressure
gradient that can be accepted without separation of the boundary layers. Pre-diffuser
design is commonly guided by the findings of Howard et al (1967), in which a variety of
diffusers were tested to define a line of first stall (Fig. 1.9). However pre-diffusers are
often conservatively designed, using the nominal design line as a guide, due to other
detrimental effects (Carrotte, McGuirk and Stevens, 1995). Since the flame tube is often
canted outwards, it is common practice also to cant the pre-diffuser outwards. Most
turning of the flow is therefore achieved by the outer wall, and a sharp reduction in
pressure is required at the inlet to provide the necessary radial force (Fig.1.10). The
adverse pressure gradient on the outer wall is therefore much higher, resulting in a thicker
boundary layer that may become unstable (Stevens et al, 1988). This is counteracted by
the influence of the flame tube, which is usually located sufficiently close to the
pre-diffuser exit to have a beneficial effect on both pre-diffuser boundary layers. A rise in
static pressure occurs where the flow impinges with the flame tube, thus causing the
pressure to be increased at the centre of the pre-diffuser and deflecting the flow towards
the walls. This reduces the wall pressure gradient and improves the pre-diffuser exit
velocity profile by reducing the maximum velocity at the centre of the passage (Stevens et
al, 1988).

Another major influence on the performance of the pre-diffuser is the inlet
condition that is presented to it by the compressor. The compressor presents the
Introduction

pre-diffuser with a shear flow that is dominated by blade wakes, tip vortices and their interaction with annulus wall boundary layers (Carrotte, Bailey and Frodsham, 1995). The trailing edge of the outlet guide vane (OGV) row is usually placed at the pre-diffuser inlet plane. The inlet velocity profile is thus highly three-dimensional, dominated by well-defined blade wakes (Fig. 1.11). While pressure forces due to a positive pressure gradient can increase velocity gradients, in this case they are dominated by shear forces and the blade wakes decay in the pre-diffuser, although they still exist at pre-diffuser exit. In addition, if the pre-diffuser contains struts then the exit velocity profiles will contain well defined strut wakes (Barker et al, 1997).

Some residual swirl may be present if the OGVs over- or under-turn the flow leaving the rotor, a situation that will vary as the operating condition changes. Conservation of tangential momentum dictates that the swirl angle must increase if there is a decrease in radius or axial velocity. Thus Carrotte, Bailey and Frodsham (1995) found a slight increase in swirl angle at the pre-diffuser exit.

The flow is discharged from the pre-diffuser into the dump gap, generating turbulence in the process due to the highly sheared velocity profile and associated large Reynolds stresses that are generated. Depending on the circumferential location, the flow either enters the flame tube through a cowl hole or impinges with the head of the flame tube and flows around the head and into the feed annuli as shown by Fig. 1.12 (from Fishenden and Stevens, 1977). Because the flame tube head is the only solid boundary, the flow is free to adjust itself to the mass flow requirements of the flame tube and feed annuli (Klein, 1974). Since the separation from the pre-diffuser is fixed at the pre-diffuser exit, changes in the inlet velocity profile have little effect on the mass flow split or the flame tube exit temperature profile. However inlet velocity profile variations can affect the location of the flow reattachment and turbulence levels in the feed annuli (Srinivasan et al, 1990).

The size of the dump gap has a significant effect on the static pressure recovery and stagnation pressure loss of the diffuser system. Small dump gaps, because of their effect on the static pressure profile at pre-diffuser exit, increase the pre-diffuser static pressure recovery. However, the curvature of the flow around the head of the flame tube is increased by decreasing the dump gap, resulting in increased stagnation pressure losses that may outweigh the improvement made to the pre-diffuser pressure rise (Klein, 1995).
Fishenden and Stevens found that the optimum value of the dump gap ratio (the distance from the pre-diffuser exit to the flame tube head, divided by the pre-diffuser exit height) is 1.0, and this value is used in the design of many dump diffuser systems.

After impingement the flow accelerates around the flame tube head with the free shear layers bounded by stationary vortices in the dump cavities. The flow around the head therefore has a strongly biased velocity profile, with the highest velocities near the surface of the flame tube (Carrotte et al., 1993). The fluid is diffused as it is deflected back towards the axial direction and into the feed annuli, this process being assisted by centrifugal forces (Fishenden and Stevens, 1977). Further turbulence is generated in the free shear layer in the dump region (Carrotte, Denman and Wray, 1992), and by the turning process and boundary layer development around the flame tube head (Carrotte et al., 1993). This turbulence causes the OGV wakes to mix out as they pass around the head of the flame tube. As a result, they do not significantly affect the circumferential variation in stagnation pressure in the feed annuli (Barker and Carrotte, 1997).

The feed annuli are presented with a highly turbulent flow with a strongly biased radial velocity profile, the maximum velocity occurring near the surface of the flame tube. As the flow progresses along the annulus the bias may mix out (Fishenden and Stevens, 1977), however a rapid switching of the bias towards the casing may also occur (Carrotte, Bailey and Frodsham, 1995). Which effect occurs depends on the relative effects of the radial momentum component of the flow due to its deflection around the flame tube head, which causes the switching of the profile, and the turbulence in the flow, which causes the profile to mix out. The high turbulence levels also spread across the whole flow field in the feed annuli. Carrotte and Wray (1991) found turbulence intensities of 35% in the feed annuli of a typical modern combustion system. Spencer (1998) fed jets from a relatively long annular duct and observed relatively low turbulence levels in both the annulus and the jets' cores. It is considered that, when jets are fed from an annulus in which, as in Carrotte and Wray (1991), turbulence levels are high, these turbulence levels will be convected to the jets. If annulus-fed jets are to be modelled in an investigation of combustor aerodynamics, it is important that the complete diffuser system is modelled so that the feed annulus flows contain turbulence that is representative of that found in the engine.
Introduction

Circumferential variations have also been found in feed annulus velocity profiles. The flow pattern in the inner annulus is determined by the cowl geometry (Barker and Carrotte, 1997). Higher velocities are found downstream of the burner centre line because most of the flow enters the swirler from the sides. In the outer annulus, the velocity deficits in the wakes from the burner feed arms are alleviated by fluid spillage. Fluid spills from the cowl holes into the low pressure wake region, thus helping the wakes to mix out (Carrotte and Wray, 1991). It is considered that these circumferential variations may affect the feed to the flame tube ports and modelling of these features is important. Although the wakes from mounting pins and igniters may be expected to cause substantial variations in the circumferential velocity profile, these features are not present in every burner sector and their inclusion in the combustor model is not essential. Wakes from pre-diffuser struts are not significant, as they mix out rapidly (Barker et al., 1997), although their presence modifies the feed of air to the cowl holes, which also influences circumferential velocity variations in the feed annuli.

In the inner annulus, due to conservation of tangential momentum, the combined effects of the decreases in radius and axial velocity cause a significant enhancement of any swirl that is present in the flow at pre-diffuser exit. Carrotte, Bailey and Frodsham (1995) found that a 3° swirl angle at pre-diffuser exit was enhanced to a 15° angle in the inner annulus. This is likely to have an effect on the flow in the flame tube as the tangential momentum may be passed on to the jets and the discharge coefficients of the liner ports and cooling holes are likely to be affected. However there is no significant enhancement of swirl in the outer annulus, where the effect of diffusion is offset by the effect of the increase in radius.

1.4 Combustor internal aerodynamics

A complex flow field exists inside the flame tube, which is dominated by jet flows and the flow issued from the swirler, with large scale recirculations and high turbulence levels. Cooling flows are also present, but are outside the scope of this thesis and shall not be discussed here.

1.4.1 Flow issuing from the swirler

The growth, entrainment and decay of the jet issuing from the swirler are determined by the degree of swirl imparted to the flow (Lilley, 1977). In combusting conditions this
affects the size, shape and stability of the flame and the intensity of the combustion. If sufficient rotation is imparted to the flow, the static pressure in the central core becomes low enough to create a flow reversal and the flow exits the swirler in a steeply angled cone with a toroidal flow reversal at its centre (Lefebvre, 1983). Many workers (e.g. Rhode et al, 1983) have shown that the flow is characterised by this conical swirling jet with a wide recirculation at its core, as shown by Fig. 1.13, that may extend for several swirler diameters downstream.

The degree of swirl is described by the non-dimensional swirl number, $S$. This is the ratio of the axial flux of swirl momentum to the axial flux of axial momentum, multiplied by the equivalent nozzle radius. It is related approximately to the swirler geometry (Lilley, 1977) by

$$S = \frac{2}{3} \left[ \frac{1 - \left( \frac{d_h}{d} \right)^3}{1 - \left( \frac{d_h}{d} \right)^2} \right] \tan \phi \approx \frac{2}{3} \tan \phi$$

where $d$ and $d_h$ are the swirler total diameter and hub diameter, and $\phi$ is the swirler vane angle. The vortex breakdown resulting in the central recirculation occurs at a critical swirl number of 0.6, and Lilley found that at $S=0.64$ the reverse flow region had a length of 4 jet diameters and its boundary was well defined with high turbulence intensities in the recirculation zone. Lilley also states that as $S$ increased the recirculation zone grew longer, reaching a maximum at $S=1.5$, and then shortened and widened as $S$ increased past 2. Rhode et al (1983) showed that the length and diameter of the recirculation increases as the swirler vane angle increases. Escudier and Keller (1985) also observed that the recirculation zone increases in diameter as the swirl number is increased.

Sislian and Cusworth (1986) measured the turbulence in a free swirling jet with $S=0.79$. The Reynolds normal stresses showed substantial anisotropy and were significantly greater than the shear stresses. The maxima of these stresses occurred in regions with high velocity gradients - that is, near the edge of the recirculation zone and in the jet's shear layer. Ahmed et al (1992), using a swirler in a cylindrical tube to simulate a tubular combustor, also found that the turbulence is highly anisotropic. The turbulence is associated with the generation of a highly sheared flow at the swirler exit. They also demonstrated the influence of the walls, with the radial velocity decaying rapidly, the swirl component decaying near the wall due to axial pressure gradients and
friction effects exerted by the wall, and found that the axial flow did not fully recover, with some reverse flow detected at their last measurement plane. In contrast with the findings of Sislian and Cusworth and Ahmed et al, Kihm et al (1990) suggested that each jet could be considered to be close to isotropic turbulence, except at the nozzle exit, although the shear stresses showed strong anisotropy. However, this conclusion appears to be based on measurements conducted at greater axial distances from the swirler exit than the measurements of Ahmed et al and Sislian and Cusworth, and must be considered doubtful.

All of the work cited above considered the flow generated by the swirler to be two-dimensional. However there will, at least in the very near field, be wakes from the swirler vanes. Most of the past work also considered only single-passage swirlers. It is common practice to use a swirler with two or three swirl passages. Little work has yet been published on the three dimensional flow field generated by such a swirler, although this is addressed by an investigation currently being conducted at Loughborough University (Hughes, 1997). Although the characteristics of the flow field generated by an engine-representative multi-passage swirler are not fully understood, it is considered that the use of an engine-representative swirler is crucial to the accurate modelling of the internal combustor flow field.

1.4.2 Jet flows
The jet flow in the flame tube is an example of a jet issuing into a cross-stream of fluid. This so-called jet in cross flow is a common flow regime that has been the subject of substantial past research because of its wide range of applications. In addition to the gas turbine combustor, these applications include the dispersal of gases from smoke stacks and volcanoes, the use of reaction control jets on missiles and rockets, and the use of jets to produce lift for vertical/short take-off and landing (V/STOL) aircraft such as the Harrier (Margason, 1993). A complete review of the substantial amount of past research that has been published (Margason alone cites 333 references) is beyond the scope of this thesis. However, a comprehensive review of the work that is relevant to the combustor internal flow field is presented in the sub-sections below.
Introduction

1.4.2.1 Single jet in cross flow

The case of a single jet in a cross flow has been the subject of investigations by many workers, for example Andreopoulos and Rodi (1984). The main findings relate to the bending and distortion of the jet, and the complex vorticity associated with the jet, and are illustrated in Fig.1.14. As the jet enters the cross flow it creates a blockage, causing the flow upstream to decelerate and the pressure to increase, while a rarefaction occurs downstream - a situation similar to that which occurs when a solid cylinder is placed in a cross flow. The combination of the upstream increase and downstream decrease in pressure results in a pressure gradient, which causes the jet to be deflected in the cross flow direction. Furthermore, a turbulent shear layer is formed around the edge of the jet, caused by mixing of the cross flow and jet fluid, and this lower momentum fluid therefore has a more curved trajectory. As a result, the jet profile develops into a characteristic kidney shape.

As the cross flow is deflected around the jet, a horseshoe vortex is formed due to the vorticity in the approaching boundary layer. In addition, streamwise vorticity is contained in the shear layers at the sides of the jet. This fluid rolls up to form a pair of contrarotating bound vortices at the lobes of the jet, known as the contrarotating vortex pair (CVP). These enhance the kidney-like profile of the jet as it develops. Smith and Mungal (1998) found that the CVP can be asymmetric in shape, even if there are no asymmetries in the cross flow, jet feed flow or orifice.

Later work on the jet in cross flow has revealed two further vortex systems - the wake vortices and shear layer vortices. The shear layer vortices are formed as the shear layer separates from the jet orifice, with a Kelvin-Helmholtz like roll up of the flow forming vortex rings that are concentric with the jet (Fig.1.15, from Perry et al, 1993). The wake vortices originate in the cross flow boundary layer, a vortex similar to a tornado being formed in the boundary layer in the lee of the jet, whose other end is then entrained by the jet, with the vortex stretching as the jet trajectory takes it further from the wall (Fric and Roshko, 1994). Where the jet issues from a pipe protruding from the wall, the wake vortices are locked on to the vortices shed by the pipe (Smith and Mungal, 1998). It is unclear whether these wake vortices remain in the case of multiple jets, when cross flow vortices are present.
Introduction

While much of the work on the single jet in cross flow has produced valuable information on the basic features of the jet, that is the bending of the jet towards the cross flow and the distortion of its cross section, and the complex vortex systems that are present, the flow conditions of interest in the combustion system are more complex. Whereas much jet in cross flow research uses a single jet in a uniform, unconfined laminar cross flow, the cross flow in the combustion system is confined by the flame tube. The proximity to the jet exit of an opposing wall can be expected to affect the flow field, as it influences the pressure field (Crabb et al., 1981). Further complicating effects are introduced by the use of rows of jets rather than single jets, whilst opposed pairs of rows of jets are commonly used to enhance mixing performance. Due to the presence of a swirler or other jets upstream, the cross flow may also be non-uniform and highly turbulent. However, most previous work has been carried out with simplified geometries to isolate the most important features of the flow field. These more complicated cases are discussed in the following sections.

1.4.2.2 Multiple jets and opposed jets

When a row of multiple jets is injected into a confined cross flow, a further vortex pair is formed due to the fluid passing between the jets. Close to the wall this fluid moves in a lateral direction. This inward movement is assisted by the CVP. The jet entrains this fluid, which rolls up to form two cross flow vortices (Fig.1.16, from Carrotte and Stevens, 1989). In the single jet case, the CVP dominates the flow field far downstream (Kamotani and Greber, 1972) and their spacing increases as they move downstream (Krausche et al., 1978). These three pairs of vortices control the mixing of the jet and cross flow, however Andreopoulos and Rodi (1984) state that the effect of the horseshoe vortex is small compared with the effect of the CVP, while Carrotte and Stevens (1989) state that the CVP decays rapidly when multiple jets are injected into the cross flow, with the downstream flow dominated by the cross flow vortices.

The vortex systems described here and in section 1.4.2.1 were all observed in idealised geometries. The investigation of their effect on the more complex flow field that exists in the combustor must be considered in the experimental programme.

Several workers have reported studies of opposed jets in cross flows. The momentum flux ratio, J, was found to be important. Above a certain momentum flux ratio, the jets penetrate to the centre and impinge, with the flow bifurcating to cause a
back flow (Hatch et al, 1995, and Zhu et al, 1995), although the back flow represents a small portion of the total jet flow, the remainder moving in the downstream direction. Doerr et al (1995) found that the formation of the back flow is dependent on the jet spacing/diameter ratio (S/D). The impingement point moves closer to the injection plane (because the jet penetration depth increases and bending by the cross flow decreases) and mixing becomes more rapid as the momentum flux ratio is increased (Srinivasan et al, 1984). Streamlines plotted from a CFD prediction by Baker and McGuirk (1992) show the back flow forming a vortex upstream of the impingement point (Fig.1.17). Spencer (1998) also observed a recirculation upstream of the impingement of opposed jets.

Substantial turbulence generation occurs in the impingement of opposed jets. Turbulence is generated predominately in the radial direction due to the impingement of radial jets, causing the Reynolds normal stresses to be strongly anisotropic (Spencer, 1998). The turbulence generated by the impingement is transported downstream by the impinged jet fluid and upstream by the recirculation generated by the impingement.

Srinivasan et al (1984) investigated the injection of opposed jets in a rectangular duct and found that jet penetration was decreased while jet spreading increased for a given momentum flux ratio. The effective channel height is halved when equal opposed jets are used. This has led some workers (for example Stevens and Carrotte, 1990) to continue to investigate single-sided injection with a reduced duct height to simulate opposed jets. This is valid, for investigations of jet behaviour in an idealised geometry, provided that allowances are made for enhanced mixing due to impingement with the opposing wall, but of no use in investigating the complete combustor flow field.

1.4.2.3 Parametric investigations
Numerous parametric investigations have been conducted with the aim of defining empirical correlations for the penetration, trajectories and mixing of jets to aid the design of the combustion system. The most important flow field parameter is the jet-to-cross flow momentum flux ratio,

\[ J = \frac{\rho_j V_j^2}{\rho_c V_c^2} \]

with the jet penetration distance increasing with increasing momentum flux ratio (Srinivasan et al, 1982). In experiments with a row of jets injected into a confined rectangular duct, Srinivasan et al (1982) found that the jet spreading rate is increased by
increasing $J$ and the duct height/orifice diameter ratio $H/D$ (see Fig.1.18), and decreasing the orifice spacing/diameter ratio $S/D$, while the density ratio had only a second order effect. Flow area convergence was also found to increase jet penetration and spreading rate.

Holdeman and Srinivasan (1986) found that jet penetration and mixing of a row of jets in a rectangular duct are similar when $J$, $S$ and $H$ are coupled by the equation

$$C = \frac{S}{H} \sqrt{J}$$

with temperature profiles approaching an isothermal distribution in a minimum distance when $C = 2.5$. Srinivasan and White (1986) produced a correlation for jet trajectory:

$$\frac{Y}{D_j} = a_0 J^{0.12} \left( \frac{S}{D_j} \right)^{0.23} \left( \frac{H_0}{D_j} \right)^{0.07} \left( \frac{X}{D_j} \right)^{\alpha}$$

where $a_0 = 0.765 \left[ 1 + \frac{dH}{dx} \right]^{0.35}$

and $\alpha = 0.12 \left[ 1 + \frac{dH}{dx} \right]^{1.25}$

For inclined jet injection, the effective momentum flux ratio becomes

$$J_{\text{eff}} = \frac{\dot{m}_j V_j (\cos \beta)^2}{A_j \rho_e V_e^2}$$

where $\beta$ is the angle between the jet and the normal to the wall (Srinivasan et al, 1984).

Srinivasan et al (1984) also derived a formula for the equivalent duct height for unequal opposed rows of jets, based on jet area and momentum flux ratio, however Liscinsky et al (1996) found that a formula based on the parameter $C$ was more accurate:

$$\frac{H_{\text{eq(top)}}}{H} = \frac{C_{\text{top}}}{C_{\text{top}} + C_{\text{bottom}}}$$

for the top row of jets, and it follows that, for the bottom row of jets,

$$\frac{H_{\text{eq(bottom)}}}{H} = 1 - \frac{H_{\text{eq(top)}}}{H}$$

1.4.2.4 Symmetry and stability of opposed jets

Some workers have questioned the symmetry and stability of the impingement and bifurcation of opposed jets. Khan and Whitelaw (1980) found that, under certain conditions (velocity ratio of 2.25 and $S/D$ ratio of 4) the bifurcation of the jets was asymmetric. Using the same geometry, Atkinson et al (1982) found that symmetry was achieved at lower velocity ratios although the opposing jets retained their identities and weren't completely mixed. However Sivasgaram and Whitelaw (1986) found injection of opposing jets into a rectangular duct with no cross flow resulted in a symmetrical...
bifurcated flow field. A decrease in the velocity of the top row of jets resulted in an upwards deflection of the downstream flow, although doubling the S/D ratio caused a downwards deflection. Holdeman et al (1987) compared numerical predictions of mixing of opposed jets in a rectangular duct and an annular duct. While they concluded that the trajectories were similar, the data presented in their report shows a slight bias of the flow towards the inner wall although the jets do impinge at mid-height. Fig.1.19a shows streamlines plotted from a CFD prediction by McGuirk and Palma (1992) of dilution jets injected into an annular duct show that the jets bend downstream to a small inclination from the normal and impinge in the centre, bifurcating to produce a small back flow and recirculation upstream of the impingement. When the opposed jets were given opposite inclinations (Fig.1.19b) an asymmetric impingement pattern was created, the impingement occurring off the centre line. The jets bifurcated after impinging, with the downstream flow and the back flow inclined downwards and upwards respectively. Clearly geometrical and flow asymmetries affect not only the location of the impingement but also the direction of the resulting flow.

Fernandes et al (1996) observed that impinging jets periodically pinch off regions of cross flow where they meet. Pairs of vortices (called impingement vortices) were also observed. These were found to form quasi-periodically and move in the axial direction. It was proposed that the recirculation zones observed by other workers are in fact these impingement vortices. The shear layers on the upstream sides of the jets were proposed as a possible source of this vorticity. The jets were observed to merge in a very short distance downstream of the impingement. Very high transverse turbulence intensities were observed in the region of the impingement, with no increase in axial turbulence intensity.

Investigations of opposed jets representative of those found in an industrial furnace were reported by Quick et al (1993). CFD only predicted a symmetrical steady state flow when symmetry was imposed on the model, and a time dependent calculation predicted oscillations in the flow. A 2.5% difference in the velocities of the opposing jets resulted in a slight shift to one side with the main features of the flow and frequency of the oscillation essentially unchanged. LDA measurements produced similar results, however it was found that the oscillations did not have a constant period. Perchanok et al (1989) reported a similar oscillation of impinging jets in a boiler furnace. McGuirk and
Introduction

Palma (1995) recorded bimodal probability density functions in LDA measurements of impinging jets in an axi-symmetric combustor, which also indicates the instability of the impingement process. It is to be expected that the impingement of opposed jets may not be a stable process. Because any instability in the flow field has implications for the performance of the combustor, potentially causing combustion instability, noise and flame extinction, an investigation of the combustor flow field should be conducted such that instability can be identified.

1.4.2.5 Influence of feed conditions

Much of the work cited above concerned jets fed from a pipe or from a plenum. Little attention has been paid to jets fed from feed annuli. The jets have characteristics that are formed in the feed annuli - for example velocities are higher towards the centre of the port, and the axial momentum of the flow in the annulus is passed on to the jet, causing it to enter the flame tube at an angle (Spencer, 1998).

Stevens and Carrotte (1990) fed heated jets from an annulus into an annular duct with cross flow and found that many of the jets were twisted or distorted. Vortices were found in the rear half of certain jet orifices. These vortices were not seen downstream and were thought to decay rapidly as the jets stretched. A correlation existed between the distortion of the jets and the presence of vortices in the orifices. Because of this distortion of the fluid issuing from the rear of the orifice, the deflection of fluid around the jet (which leads to the formation of the CVP) was not symmetric about the centre plane, so the vortices of the CVP differed in strength and the core of the jet was distorted. As a result of this asymmetry, the cross flow vortices were also unequal (Stevens and Carrotte, 1990). Baker and McGuirk (1992) also observed similar vortical instabilities. The flow was stabilised by increasing the bleed flow (the flow past the orifice in the annulus), and vortices started to form as the bleed flow was reduced. Increasing the annulus depth was also found to eliminate the vortices.

Carrotte and Stevens (1989) found that the separation of the flow from the casing wall has a major influence on the velocity profile across the port exit plane. Spencer (1998) also found that the flow separated from the upstream lip of a chuted port. An inlet radius reduced, but did not eliminate, the separation. Several port geometries were tested and the direction of the jet core was found to be dependent mainly on the flow conditions, not the port shape. A vortex was found in the separated region inside the chute. Spencer
also found that the turbulence contained in the flow at the port exit changed with variations in the port feed conditions, and the flow history must be expected to have a major influence on the characteristics of the turbulence in the jet.

Clearly the velocity profile across the plane of the jet orifice will affect the flow field, and the origin of the velocity profile is in the feed annulus. Therefore the flame tube and annuli must be coupled for any experimental or computational investigation to be representative of true combustor conditions.

1.4.3 Combustor flow field interactions

The combustor flow field features both the swirler generated flow field and the flow field generated by the jets. These two flows can be expected to interact and modify the overall combustor flow field. Much of the past work on jet flows cited above was aimed at understanding the flow in the dilution zone to optimise turbine inlet temperature profiles. Similarly past work concerning the flow issuing from the swirler concentrated on the swirler in isolation, and little attention has been paid to the primary zone flow field in which the swirling and recirculating flows generated by the swirler interact with primary jets, or to the flame tube as a whole. The published work that has been found dealt only with the flow field in a tubular combustor.

The interaction between the primary jets and the flow field generated by the swirler is strongly dependent on two factors - the relative quantities of flow introduced through the jets and the swirler, and the location of the jets relative to the swirler. It has a considerable influence on the nature of the vortex breakdown in the swirling flow generated by the swirler. When the mass flow through the swirler is small compared with the primary jets (primary jet mass flow of 2-3 times the swirler mass flow) the back flow into the primary zone is created by a combination of the impinging primary jets and the swirl-generated recirculation. The toroidal vortex is confined within the primary zone by the primary jets and is symmetric about the centre line (Fig.1.20). The primary jets dominate the formation of the recirculation (Koutmos and McGuirk, 1989), and the swirler causes the widening of the recirculation and an increase in the back flow rate (McGuirk and Palma, 1995).

If the primary jet/cross flow momentum flux ratio is very low, the jets don't penetrate to the centre line and are swept into a spiral trajectory near to the wall by the
relatively strong swirling flow. As the momentum flux ratio is increased the contribution of the primary jets to the primary zone recirculation increases (Richards and Samuelsen, 1990). Most of the swirling fluid upstream is transported near to the flame tube walls, and little swirl is imparted to the primary back flow at the plane of the primary jets (Bicen et al, 1989). Similar trends may be observed in combusting conditions, although the velocity of the primary jets must be higher to achieve the same effect because the cross flow axial momentum flux is increased as a result of the reaction and heat release. This causes the mass flow recirculated in the primary zone to be reduced and the streamwise deflection of the primary jets to be increased (Richards and Samuelsen, 1990).

The location of the primary jets is influential. Richards and Samuelsen (1992) observed that, in combusting conditions, the velocity profile upstream of the primary jets was characteristic of a swirler-induced recirculation when the primary jets were one flame tube diameter downstream of the swirler. However the jets were close to the end of the recirculation and very little of the cold air was transported into the primary zone. When the primary jets were moved closer to the swirler the jets impinged at the centre line and fed directly into the recirculation.

The research outlined above has produced some valuable information on the nature of the flow field in a combustor. However all of this research has been conducted with combustor geometries that are not truly representative of a modern annular combustor. Much of the research cited above used simplifications of the combustor geometry to investigate isolated elements of the flow field. Few of the investigations of the flow field induced by the swirler, discussed in section 1.4.1, were conducted in the presence of jets. Similarly, many of the investigations of the jet in cross flow, discussed in section 1.4.2, were conducted in uniform cross flows, in the absence of the influence of an upstream swirler. Furthermore, many of these investigations used jets fed by a plenum or from pipes, thus neglecting the influence of external aerodynamics on the jets. Although investigations of the complete flow field in an axi-symmetric combustor, discussed above, have resulted in a good understanding of the flow interactions in that type of combustor, no such investigation has been conducted in a geometry that is representative of a modern annular combustion system and the behaviour of the flow in such a combustor is not known. There is therefore a clear need for an experimental investigation of the complete internal flow field in an annular combustion system that is typical of current design.
**Introduction**

practice and, in order to understand the interaction that may exist between the internal and external aerodynamics, this must be coupled with an investigation of the external flow field. A fully representative geometry must be used, including the use of an engine-representative fuel injector, with a feed arm and multi-passage swirler, a canted combustor with representative jets and wall cooling flows, and representative flow splits, including annulus bleed flows.

1.5 **Prediction of aerodynamic processes within the combustion system**

With the increasing improvement of computer technology in recent years, computational fluid dynamics (CFD) has become a part of the combustor design process. Ideally, geometric modifications could easily be made and tested computationally without the need for expensive experimental testing. However CFD is not yet a mature technique and this goal is still some way off.

Fluid flow is described mathematically by the Navier Stokes equations and direct solution of these equations is possible. A grid must be used to divide the solution domain into small control volumes or cells. However for a Direct Numerical Solution (DNS) of the Navier Stokes equations the grid spacing must be smaller than the smallest length scale in the flow. A large number of time steps is also needed, because the time step must be smaller than that associated with the smallest time scale. DNS is therefore best suited to the calculation of low Reynolds number flows, and for a complex flow field such as that in a combustor grid with far too many cells, and far too many time steps, for currently available computing capacity to handle would be needed.

By time-averaging the Navier Stokes equations the problem is greatly simplified. This introduces the six Reynolds stress components but also simplifies the problem by removing the time dependence from the calculation. The necessary spatial resolution is reduced to a level that a computer can handle because only mean quantities must be resolved. However a turbulence model must be used to calculate the Reynolds stresses and two such turbulence models are commonly used. The $k-\varepsilon$ model solves transport equations for the turbulent kinetic energy, $k$, and the rate of dissipation of turbulent kinetic energy, $\varepsilon$. Because this turbulence model does not directly model the Reynolds stresses, it fails to model the flow in many situations where the turbulence is not isotropic, for example the turbulence generated by the impingement of two jets has been shown to
Introduction

be strongly anisotropic (Koutmos and McGuirk, 1989). This leads to inaccuracies in the prediction. The Reynolds Stress Transport (RST) model uses transport equations for each of the six Reynolds stresses plus the rate of dissipation of turbulent kinetic energy. This model is the subject of much current work as it doesn't make the assumption of isotropy and may be much better suited to the prediction of many flows. One assumption that is made in both cases is that the flow is steady, and calculations with either of the turbulence models will fail if the flow is unstable.

Several workers have investigated the suitability of the k-ε model for predictions of the flow field in tubular combustors. The predictions have generally been found to be in reasonable agreement with experiments, but with discrepancies highest where the turbulence is strongly anisotropic. However the ability of the k-ε model to predict the flow field in an annular combustor cannot be verified at present because no experimentally determined data set exists, although discrepancies have been found between the measured and predicted combustor exit temperature profiles (Priddin, 1995). The k-ε model fails to predict accurately flow where the turbulence is anisotropic and has rapid spatial variations in the turbulence structure (e.g. Koutmos and McGuirk, 1989b).

Ahmed et al (1992), predicting the flow field in a tubular combustor with a swirler but no jets, found that the RST model performed much better than the k-ε model in the confined swirling flow. The limitations of the k-ε model suggest that the RST model may produce more accurate predictions.

The current standard practice is to model the internal and external flow fields separately, with approximated boundary conditions used for flame tube inlet flows. For example, Little and Manners (1993) and Srinivasan et al (1990) performed calculations on dump diffuser geometries, while Koutmos and McGuirk (1989b) performed calculations on an axi-symmetric combustor with boundary conditions taken from measurements. While these calculations produced reasonable predictions, they do not reproduce the coupling of the internal and external flow fields. Spencer (1998) and Bain et al (1996) demonstrated the importance of modelling the internal and external flow fields together for accurate predictions. Ideally, the complete flow field from pre-diffuser inlet to combustor exit should be modelled, however the extra computing power that is necessary to do this has prevented it. Crocker et al (1998) ran such a prediction using a k-ε model with combustion. The grid had 370,000 cells and the calculation took 48 hours. Spencer
(1998) ran a calculation of a sector of an annular combustor, with the external geometry starting at annulus entry. Experimental results were used to define the annulus entry boundary conditions and the results of a 2D swirler calculation were used to define the swirler exit boundary condition. While this is an improvement over an internal only calculation, it falls short of the ideal complete internal and external calculation. With the computing power now available to run such calculations, experimental data is needed to validate the results. Thus a further requirement exists for an experimental investigation of the internal and external flow fields in an annular combustion system, as discussed in section 1.4.3, in order to obtain a complete flow field map for comparison with CFD predictions so that aerodynamic processes that cannot be predicted may be identified and improvements made to the computational codes.

1.6 The current investigation

As discussed in sections 1.4.3 and 1.5, the behaviour of the flow field in a modern annular combustion system is not known. An internal and external flow field map is needed to obtain an understanding of the aerodynamic processes that occur in an annular combustor, in terms of the interactions that occur between the various features of the internal and external flow fields, and for the validation of CFD codes. This thesis meets this need, and reports on an experimental investigation of the isothermal flow field in a modern annular combustion system. It is important that such an experiment is conducted in isothermal conditions, so that an understanding of the aerodynamic processes, and how well such processes can be predicted, may be gained in isolation from the complications associated with combustion. However any conclusions that are drawn regarding the aerodynamic processes that occur in the isothermal flow may still be relevant to the combusting flow. Coupland and Priddin (1986) compared isothermal and combusting predictions of the flow field in a computational combustor model, and found that the flow pattern changed little in combusting conditions. Since the internal and external combustor flow field provides a huge scope for an experimental investigation, and far more measurements are possible than can be obtained in a reasonable period of time, the work was prioritised to meet the following objectives:
Introduction

- To obtain a test facility in which measurements of both the internal and external flow fields in a burner sector of a representative combustion system geometry may be obtained.
- To obtain sufficient measurements of the external flow field to verify that its behaviour is as expected.
- To obtain sufficient measurements of the internal aerodynamics so that the key features of the flow field - the jets and the flow issuing from the swirler - may be identified.
- To examine the interactions between the various features of the internal flow field, and between the external and internal aerodynamics.
- To identify any unstable processes in the flow field.
- To obtain sufficient flow field data for validation of CFD codes.

The following tasks must be performed to meet these objectives:

- An experimental facility must be developed that contains all features necessary to replicate the external and internal flow fields in a geometry that is representative of a modern annular combustion system. Although measurements are to be conducted only in isothermal conditions, any flow field features that would be present in a facility for use in combusting conditions must be replicated so that a future experiment may be conducted in combusting conditions in a facility that is aerodynamically identical to that used in this experiment.
- An appropriate instrumentation system must be used, and provision for this instrumentation made in the construction of the experimental facility, to enable all three components of mean velocity and all six Reynolds stresses to be measured in a complete burner sector of the combustion system.
- Measurements of the internal flow field must be defined and obtained to allow the identification of the main flow field features.
- Measurements of the flow entering the ports and issuing from the port exits must be obtained so that the nature of the interaction between the external and internal flow fields may be determined.
Introduction

- Sufficient measurements must be obtained to determine the trajectories of the jets, so that the interactions between opposing jets and between the jets and the flow issuing from the swirler may be observed.
- At all locations, all six Reynolds stresses must be obtained so that the nature of the turbulence in the flow field may be determined, and probability density functions (PDF's) must be obtained to reveal any instability in the flow.

The development of the experimental facility is described in Chapter 2. A new test facility was developed and optimised to provide the best possible optical access for the three component laser Doppler anemometry (LDA) system that was used for all mean and fluctuating velocity measurements. For this reason a 45° sector rig configuration (Fig. 1.21) was chosen, containing three burner sectors of 15° each, to allow full optical access through the sidewalls to the flame tube. Chapter 3 describes the instrumentation systems that were used and the development of new traversing and calibration systems to enable LDA measurements throughout the centre burner sector of the facility. Chapter 4 describes the commissioning of the test facility and the definition of the experimental programme. Great care was taken to prevent separation of the boundary layers on the sidewalls and to isolate the centre burner sector from the effects of sidewall boundary layer growth. The results of the experiment are presented and discussed in Chapter 5, and the conclusions are presented in Chapter 6.
2. Test Facility Definition

The design of the test facility was driven by two requirements:

- The main aerodynamic features of the test facility must be representative of a modern annular combustion system.
- The test facility design must allow instrumentation access to all areas of interest. Particular emphasis was placed on meeting the need to maximise instrumentation access. This requirement dictated the use of a 45° sector configuration, and detailed changes to the aerodynamic design were also necessary. A detailed test facility specification was produced, including drawings of the aerodynamic profiles of the casings and the specification of the bleed systems and all optical access windows. Perspex casings were manufactured to this specification and the flame tube sector was obtained from an existing flame tube. This chapter describes the test facility definition process, including a description of the nominal combustor geometry, the changes that were made to meet the requirements listed above, and the design of the bleed system.

2.1 Combustor geometry

2.1.1 The nominal VULCAN Phase 5 combustor geometry

The VULCAN Phase 5 combustion system is defined by a large number of engineering drawings, many of which are of interest only to the manufacturer and are of no use in defining the geometry of the test facility. The first task in the definition of the test facility geometry was to identify in these drawings the features of the combustion system that were to be modelled and to produce a specification to from which the Perspex casings could be manufactured. The Phase 5 combustor is incorporated within the Rolls-Royce Trent series of large turbofan engines and is typical of modern combustion system design practice. This combustion system was chosen because (i) it will be in production for many years and will therefore be of continuing interest, (ii) there are known problems in predicting the combustor exit temperature traverse, and (iii) an existing flame tube and fuel injectors were available that were surplus to requirements.

The VULCAN system, as shown by Fig.2.2, features a dump diffuser system with a dump gap of 41.5mm. The feed annulus skirts, linking the flame tube to the casings, dictate the aerodynamic profiles of the inner annulus and the rear portion of the outer annulus. These skirts allow for thermal expansion of the flame tube whilst maintaining the seals with the casings. Orifices in these skirts and in the baffle in the outer dump...
cavity enable the removal of air to supply cooling flows for the nozzle guide vanes and the first stage turbine blades, and to supply cabin air pressurisation systems. For a Trent 700 engine operating at take-off conditions, the inner and outer annulus bleed flows are 12.14% and 3.94% of the inlet mass flow respectively, while the outer dump cavity bleed removes 0.8% of the inlet mass flow.

The flame tube, whose liner profiles are illustrated in Fig. 2.2, is canted outwards at an angle of 9°. The flame tube incorporates two rows of ports, with opposed rows of primary and secondary ports but no dilution ports. Flame stabilisation is achieved by the combination of opposed primary jets and swirlers, and liner cooling is achieved by an angled effusion cooling system, with film cooling slots at the downstream ends of the liners.

24 fuel injectors, with swirlers, are equally spaced around the combustor, thus dividing the combustor into 15° burner sectors. The swirler design (Fig. 2.3) includes three air swirler passages. The inner air swirler uses curved vanes with a 60° exit angle. The outer and dome air swirlers use flat vanes, angled at 40° and 43° respectively. All three swirlers impart clockwise rotation to the flow. 11% of the air entering the flame tube (9.4% of the total flow through the facility) passes through the swirler.

The flame tube features 48 pairs of opposed primary and secondary ports, that is two pairs of opposed primary and secondary ports per sector. All ports are chuted. As illustrated in Fig. 2.5, the primary and secondary ports are 81 and 125 mm from the crown of the flame tube respectively. The angular distribution of the ports is shown in Fig. 2.4. The primary ports are in line with and mid-way between the fuel injectors, while the secondary ports are staggered with respect to the primary ports, being located 3.75° either side of the fuel injector centre line. The port diameters, and the estimated proportion of the air entering the flame tube through each row of ports (estimates, from a computational calculation, supplied by Boyce, 1999), are listed in Table 2.1 below.

<table>
<thead>
<tr>
<th>Port</th>
<th>Diameter (mm)</th>
<th>% of flame tube flow</th>
<th>% of total inlet flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer Primary</td>
<td>14.5</td>
<td>8</td>
<td>6.6</td>
</tr>
<tr>
<td>Outer Secondary</td>
<td>20.1</td>
<td>15.2</td>
<td>12.6</td>
</tr>
<tr>
<td>Inner Primary</td>
<td>13.1</td>
<td>6.4</td>
<td>5.3</td>
</tr>
<tr>
<td>Inner Secondary</td>
<td>16.8</td>
<td>10.8</td>
<td>9</td>
</tr>
</tbody>
</table>

Table 2.1 Liner Port Sizes
Test Facility Definition

10.2% of the inlet flow enters the flame tube through the cooling holes in the heat shield (Fig.2.5), and the remainder enters through the angled effusion liner cooling system (Fig.2.6). The angled effusion cooling system consists of a large number of small holes drilled through the liner at a shallow angle, through which air enters to form a thin film of cold air to protect the liners from the hot combusting gases. The holes have a diameter of 0.6mm and are drilled at 16.5° to the liner surface. The outer liner has 37 rows of 1139 holes each, axially and circumferentially spaced by approximately 5mm and 2.25mm. The inner liner has 43 rows of 756 holes each, axially and circumferentially spaced by approximately 4.4mm and 2.5mm.

The flame tube was designed to be attached to the outer casing by a number of mounting pins fitted to a dog-leg type fitting that protruded from the head of the flame tube. There were 12 mounting pins, one every 30°, each located mid-way between burners. As such it would not have been possible to obtain a 45° sector of the flame tube with no mounting pin fitting, and to retain the mounting pin fittings would have meant that they would have been asymmetrically located within the 45° sector. To avoid asymmetry the mounting pins were omitted and the fittings removed. An alternative mounting arrangement was devised - see section 2.1.3.

Some modifications were made to the geometry for aerodynamic reasons. The principal region of interest, that is the area in which measurements that were to be obtained to meet the objectives listed in section 1.6, is shown in Fig.2.7. The nozzle guide vanes (NGVs) do not affect the flow field in this region and, as is standard practice, were omitted. It is common practice to omit the compressor outlet guide vanes (OGVs) from CFD models because of the complexity that they add to the grid, a fine mesh being needed to resolve their wakes. The OGVs were therefore omitted to fulfil the CFD validation objective of the project. The OGVs do not have a substantial effect on the gross features of the flow field, however they do help to prevent separation in the pre-diffuser by generating extra turbulence in their boundary layers and wakes. The pre-diffuser in this combustor is a conservative design, as illustrated by Fig.2.8 - see section 1.3, and should not be seriously affected by the omission of the OGVs. The omission of the OGVs and NGVs was also dictated by instrumentation considerations. Numerous other modifications were necessary to fulfil the objectives of the project. These are discussed in the following sections.
2.1.2 Instrumentation requirements

A three component laser Doppler anemometry (LDA) system was to be used for the majority of the flow field measurements in this project. The LDA system, which is described in more detail in Chapter 3, uses six laser beams emitted in three pairs from two probes, as shown in Fig.2.9. Two pairs of beams are emitted, at 90° to each other, from the '2D' probe, with a third pair emitted from the '1D' probe, which is set at an angle to the 2D probe. All six beams are focused on a single point, the "measurement control volume". For good resolution of all three velocity components, the included angle between the probes should be 45° or greater, although angles as low as 30° can yield good results if a reduction in the included angle is necessary.

Sufficient optical access was required to permit LDA measurements in the largest possible area inside one burner sector of the flame tube. The best way to provide this access is to use a sector rig configuration, with optical access through windows in the sidewalls. Because the flow field must be affected in the sectors immediately adjacent to the sidewalls, it is necessary to use at least three burner sectors, so that at least one sector is isolated from sidewall effects. The limitations imposed by the LDA system restricted the facility to the three burner sector, 45° configuration: The LDA probes that were available for this project had a 250mm focal length (Fig.2.10). A 45° included angle reduced the effective focal length of the system to 213mm, which was further reduced by the need to incline the probes for access to certain regions. Regions of the flame tube close to the primary zone were only accessible with the probes rotated towards the heat shield, while rotation towards the flame tube exit was necessary for access to the downstream extremities of the flame tube. In addition, rotation towards the outer liner was necessary for access to the outer part of the flame tube. As a result, the clearance between the probes and the test facility was reduced and, even with a three sector configuration, it was necessary to reduce the included angle for access to some regions. The three sector configuration thus represented the best compromise between optical access requirements and the need to restrict the effects of the sidewalls.

Provision was made for LDA measurements of the external flow field, although none were conducted in the experimental programme described by this thesis (see Chapter 4), so that such measurements, if required, can be conducted in the future. The profiles of the casings prevent the use of a single window in each annulus, because it would be difficult to manufacture a window that would faithfully follow the profile of the casing.
and the windows were located to allow area traverses at annulus entry, mid-way between the primary and secondary ports, and downstream of the secondary ports. It was necessary to make some changes to the combustor geometry to achieve this.

The aerodynamic profile of the inner annulus is defined by a skirt, as shown by Fig.2.1. The replication of both the skirt and the casing would be impractical as it would require the laser beams to pass through two windows, in the skirt and the casing. This would cause extra refraction of the beams, which is undesirable, whilst also reducing the intensity of the light transmitted to the measurement control volume. The replication of both the skirt and the casing is unnecessary and the inner casing was designed with the profile of the skirt as shown by Fig.2.11.

The profile of the outer annulus was partly defined by a skirt attached to the flame tube. This skirt defined the aerodynamic profile of the outer annulus downstream of a point approximately half way along the flame tube and was made up of two parts (Fig.2.1). The skirt would block laser access to a large part of the outer annulus. The long outer part of the skirt was therefore removed and the outer casing given a profile that represented a simplification of the skirt profile, thus allowing windows to be included to permit laser access to the outer annulus. The casing was then made to fit the remaining skirt and form a seal.

The inner annulus bleed offtake consisted of a series of long slots (Fig.2.12a). Replication of these bleed slots would have caused the bleed system to block LDA access to the feed annulus downstream of the inner secondary ports, so the bleed slots were altered to allow access to this area. The slots were shortened and moved as shown by Fig.2.12b. A detailed description of the bleed system design is given in section 2.1.4. To facilitate laser access further upstream in the inner annulus it was necessary to move one of the kinks in the feed annulus casing profile (Fig.2.12b). This was not a major modification and it was not expected to adversely affect the feed annulus flow field.

Although not used in the experimental programme discussed by this thesis, provision was also made for an LDA traverse of the pre-diffuser exit. However the presence of the fuel injectors precluded an area traverse as optical access would only be possible between burners. For a single radial traverse to be valid the pre-diffuser exit velocity profile must be two-dimensional. This is a further reason for the omission of the OGVs.
Test Facility Definition

Pressure measurements were required to assess the overall pressure loss within the diffuser system and to commission the test facility. Provision was therefore made to traverse button hook probes (see section 3.2.1) at a number of locations. Details of the measurement locations can be found in Chapter 3.

A single bleed offtake in the outer dump cavity (Fig.2.1), separated from the main dump cavity by a perforated baffle, supplies air to the cabin pressurisation system. The magnitude and location of the bleed was such that it was not thought to have a significant effect on the combustor flow field, whilst its incorporation in the test facility would have compromised access to the inlet for pressure measurements. It was also considered to be likely that the ducting to remove this bleed would compromise optical access. The dump cavity bleed was therefore omitted and the upstream limit of the dump cavity was set at the baffle location.

2.1.3 Sidewall design and flame tube mounting
The sidewall design incorporated a large window (Fig.2.13) to allow laser access to the centre sector of the flame tube. A sidewall window would also be required in a combusting facility and, because of the high temperatures that are encountered inside the flame tube in combusting conditions, it would be necessary to cool the windows of a combusting facility to maintain their integrity. So that an experiment may be conducted in combusting conditions in the future, it was necessary to replicate the window cooling system to ensure aerodynamic similarity between this isothermal test facility and a future combusting facility. The window cooling system is illustrated in Fig.2.13. Air is fed to galleries across the top and sides of the window. The cooling air is then injected through 300 small slots onto each window to produce a film of cool air that protects the window from the hot gases inside the flame tube. The cooling system mass flow rate for each window is 4.35% of the inlet mass flow.

The flame tube, as discussed in section 2.1.1, could not be attached to the casings by conventional mounting pins, and was attached to the sidewalls. The flame tube was secured by a flange around its edge. This was welded to the outside of the liner and formed part of the seal between the flame tube and the casings. It was not permitted to protrude into the flame tube, where it could obstruct the laser beams.

A further requirement was the easy removal of the sidewall windows for cleaning. Fig.2.14 shows a section through the sidewall. The flame tube flange is bolted to the inner
wall. A plate, attached to the outside of the inner wall, forms the floor of the sidewall cooling system's galleries. Slots, machined into the wall, inject the cooling air onto the face of the window. The window frame sits on the wall above the slots, and is held in place by the outer wall. The outer wall was designed to be easily removed to allow the removal and cleaning of the window.

2.1.4 Turbine cooling flow simulation

In the engine, air is bled from the feed annuli to cool the NGVs and the first stage turbine blades. The simulation of these bleed flows in the experiment is necessary because they have been found to affect the flow entering the flame tube through the mixing ports (Spencer, 1998, and Baker and McGuirk, 1992) The bleed flow is usually removed from the annuli via a number of circumferentially spaced slots or holes, as shown in Fig.2.12a, and the circumferential distribution of these orifices has been retained from the original geometry, although the exact dimensions of the orifices were not. As designed, the outboard bleed offtake (Fig.2.15) consists of eleven circular holes of 12mm diameter, plus two semicircular holes adjacent to the sidewalls, the centres of all thirteen holes being equally spaced. All holes were given a 2mm radius plunging to increase their discharge coefficient. The inboard bleed offtake (Fig.2.12) consists of six slots plus two half-slots adjacent to the sidewalls which, as discussed in section 2.1.2, have been shortened and moved rearwards from their original locations to improve optical access. As with the outboard bleed holes, these slots were given a 2mm plunging to improve their discharge coefficients. Manufacturing considerations dictated that the half-slots and semicircular holes adjacent to the sidewalls were replaced by complete orifices, located adjacent to the sidewalls and angled inwards to clear the walls of the bleed ducts.

Instrumentation requirements placed strict limitations on the design of the bleed system. The ducting required to remove the bleed air must not restrict optical access to either the feed annulus or sidewall windows, and it must not restrict the movement of the LDA traversing system (described in detail in Chapter 3). The only way to achieve this is to transfer the bleed air through ducts, which run as close as possible to the test facility, and back into the main flow downstream of the flame tube exit. Thus the bleed system was driven by the pressure drop that exists between the rear of the annulus and the rig exhaust. The bleed system is shown in Fig.2.16. The height of each bleed duct was minimised to minimise any conflict with the traversing system, and the forward
extremities of the ducts were limited to prevent any conflict with the laser access windows. As shown by Fig. 2.17, the walls of the outer bleed duct were made parallel with each other. This prevented the sides of the duct from protruding too far and restricting the movement of the LDA traverse (see Chapter 3). To compensate for the local reduction in flow area, the outer wall of the duct was made flat, so the depth of the duct increases away from the centre. Metering plates were inserted in the bleed ducts to control the bleed flow rates. The bleed flows are returned to the main flow through slots just upstream of the exhaust, as shown by Fig. 2.16.

The adverse effects of the sidewalls have already been mentioned. A further consideration in the design of this bleed system was the use of the bleed system to combat the effects of sidewall boundary layer growth. This novel concept used a 'differential' bleed system, whereby splitter plates were placed in the bleed ducts (Fig. 2.16) to allow the bleed flow from the central region of each annulus to be metered separately from the regions near the sidewalls. To ensure that the design bleed flows were present in the centre burner sector, and to attempt to draw the extra 'differential' bleed from a small region close to each sidewall, the centre metering sectors were made larger than the centre burner sector by placing the splitter plates $11.25^\circ$ either side of the centre line, creating a central bleed sector across the central $22.5^\circ$ and two outer $11.25^\circ$ differential bleed sectors. However the circumferential spacing of the outer annulus bleed holes is such that splitter plates at these angles would block two of the holes, and the splitter plates were moved out to $13.125^\circ$ to create a central $26.25^\circ$ bleed sector and two outer $9.375^\circ$ differential bleed sectors.

This completes the design of the casings of the test facility. Detailed drawings were produced to enable the casings to be manufactures. These are reproduced in Figs. 2.18 to 2.21.

2.1.4.1 Design of the metering plate for the inboard bleed duct

The metering plates for the bleed ducts were designed for the differential bleed (see above) and to allow easy adjustment of the bleed flow. The metering plates were divided into three metering sectors that correspond with the three sectors of each bleed duct. The design of these metering plates is discussed in detail below.

The necessary metering hole areas could not be calculated precisely as the exact diffuser and liner pressure losses were not known, so the pressure drop that drives the
Test Facility Definition

bleed system was not known. Three further unknowns were the discharge coefficients of the annulus bleed offtakes and the bleed duct exit slots, and the stagnation pressure loss associated with the sidewall boundary layers. Each metering sector therefore has a number of metering holes, plus smaller trimming holes, rather than a single large hole, so that the bleed flows could be adjusted. Estimates of the stagnation pressures in the feed annuli, flame tube and combustor exit, and bleed offtake discharge coefficients were obtained from computational calculations (Hicks, 1997, and Boyce, 1997).

Referring to Fig.2.22, metering hole patterns for the inboard bleed duct were first calculated assuming no sidewall effects:

From Hicks (1997), the stagnation pressure drop between the inner annulus and the combustor exit, non-dimensionalised by the test facility inlet dynamic pressure, is

\[ \frac{\Delta P}{q_{in}} = 0.91 \]

This is the pressure drop that drives the inboard bleed flow, so

\[ \frac{\Delta P_{slots}}{q_{in}} + \frac{\Delta P_{throttle}}{q_{in}} + \frac{\Delta P_{exit}}{q_{in}} = 0.91 \]

The inboard bleed flow rate is 12.14% of the inlet mass flow, and the duct area upstream of the metering plate (point 1 in Fig.2.22) is 1.735 times the inlet area \( A_{in} \), so the dynamic pressure upstream of the metering plate is

\[ q_1 = \left( \frac{0.1214}{1.735} \right)^2 \times q_{in} = 0.0049q_{in} \]

The total bleed slot area is 0.636 times the inlet area, and the discharge coefficient was estimated by Boyce (1997) to be 0.677, so

\[ q_{slot} = \left( \frac{0.1214}{0.677 \times 0.636} \right)^2 \times q_{in} = 0.0795q_{in} \]

Assuming that the stagnation pressure drop across the bleed slots is the difference between the dynamic pressures in and downstream of the slots, then

\[ \frac{\Delta P_{slots}}{q_{in}} = 0.0746 \]

By a similar method, the non-dimensionalised stagnation pressure drop across the bleed duct exit slot was calculated to be 0.0669. This assumed a discharge coefficient of 0.5 and that one dynamic head of pressure is lost across the exit slot. Therefore

\[ \frac{\Delta P_{throttle}}{q_{in}} = 0.91 - 0.0746 - 0.0669 = 0.7685 \]

This value is important as the pressure drop across the metering plate is by far the greatest pressure drop in the inboard bleed system, and is therefore the means by which the bleed flow rate is controlled. To calculate the area of the metering holes, the dynamic pressures
Test Facility Definition

in the metering holes and in the duct downstream of the metering holes (point 2 in Fig.2.22) are needed. The area of the duct at point 2 is 2.579 times the inlet area, so

\[
\frac{q_2}{q_{in}} = \frac{0.1214^2}{2.579} = 0.0022
\]

\[
\frac{\Delta P_{throttle}}{q_{in}} = \frac{q_{hole}}{q_{in}} - \frac{q_2}{q_{in}} \rightarrow q_{hole} = 0.7663q_{in}
\]

Metering holes with a chamfered inlet were chosen (Hay and Spencer, 1992) because they have a high discharge coefficient that is relatively insensitive to changes in the pressure drop across the holes. Holes with a 30° chamfer angle, chamfer depth/diameter ratio of 0.16 (see Fig.2.23) and discharge coefficient of 0.94 were chosen. With the bleed duct divided into a central sector of 22.5° and two 11.25° sectors, the number of metering holes (n) must be divisible by four and the number and diameter of the holes were calculated as follows:

\[
0.1214A_{in} = C_D\pi A_p V_{hole}
\]

\[
0.1214A_{in} \sqrt{2pq_{in}} = 0.94n\pi \sqrt{2p} \times 0.7663q_{in}
\]

so

\[
nA = 0.1475A_{in}
\]

The inlet area is 7682mm², so the total metering hole area (nA) is 1133mm². A metering hole pattern with sixteen 9.5mm diameter holes was therefore chosen.

The extra hole area for the outer sector was then calculated assuming a worst case scenario in which both sidewall boundary layers separate, with all the feed annulus mass flow concentrated in the centre sector of the feed annulus and the stagnation pressure in the outer sectors therefore reduced to the level of the static pressure in the centre sector. The flow area is thus 0.207 times the inlet area, giving a dynamic pressure of 0.344q_{in}. If the stagnation pressure in the separated regions in the outer sectors is equal to the static pressure in the centre sector, then the stagnation pressure drop between the feed annulus and the combustor exit is obtained by deducting the dynamic pressure in the centre sector from the stagnation pressure drop of 0.91q_{in} that has been used above, that is

\[
\frac{\Delta P}{q_{in}} = 0.91 - 0.344 = 0.566
\]

If the differential bleed sectors are to draw the same non-dimensional mass flow as the centre sector then the pressure drops across the bleed offtake slots and the bleed duct exit slot remain the same, and

\[
\frac{\Delta P_{throttle}}{q_{in}} = 0.566 - 0.0746 - 0.0669 = 0.4245
\]

The dynamic head of pressure downstream of the metering plate (point 2 in Fig.2.22) must also be unchanged, so

\[
\frac{q_{hole}}{q_{in}} = 0.4245 - 0.0022 = 0.4223
\]
This figure was used to calculate a total metering hole area as before, resulting in a total metering hole area of 379.6\,mm^2. The existing metering holes in the differential bleed sector have a total area of 282.7\,mm^2, so the extra holes must have a total area of 96.9\,mm^2. A pattern of five holes of 5mm diameter each was chosen for each sector. To give the centre bleed sector extra hole area for extra flexibility in trimming the bleed, due to the uncertainty of the discharge coefficients and pressure drops, six 5mm diameter holes were added to the centre sector of the metering plate. The metering plate is illustrated in Fig.2.24.

2.1.4.2 Design of the metering plate for the outboard bleed duct

The outboard bleed metering plate was designed using a similar methodology as that for the inboard metering plate. However, one significant difference lay in the bleed duct geometry. As detailed in section 2.1.4 above, the duct height is not constant and the outer sidewalls are parallel with the centre line, so the mean velocity upstream of the metering plate in the outer sectors will not be equal to that in the centre sector. Sections through the outboard bleed duct are shown in Figs.2.17 and 2.25. The cross section area of the centre sector of the bleed duct upstream of the metering plate is 9609.5\,mm^2 and the area of each outer sector is 2845.4\,mm^2. The total flow through the outboard bleed is 3.94\% of the inlet flow and the flow rates in the three bleed sectors were designed to be in proportion with the included angles of the sectors.

The area of the centre bleed sector is 0.177 times the inlet area and the discharge coefficient of the bleed offtake holes was estimated by Boyce (1997) to be 0.819. The dynamic pressure in the bleed offtake holes is thus 0.0739q_{in}. One dynamic head of pressure was assumed to be lost across these holes, thus the stagnation pressure loss is 0.0739q_{in}. As the flow enters the duct normal to the casing there must be a pressure loss incurred as the flow turns to face the metering plate. However the mean dynamic head in the centre sector upstream of the metering plate was calculated to be less than 0.0004q_{in} so this turning loss was assumed to be negligible.

One dynamic head of pressure was assumed to be lost across the bleed duct exit slot. The area of the slot is 0.957A_{in} and a discharge coefficient of 0.5 was assumed for the slot. The stagnation pressure drop across the exit slot was thus calculated to be 0.0068q_{in}. The stagnation pressure drop across the bleed system is 0.991q_{in} (Hicks, 1997), so the pressure drop across the metering plate is 0.9103q_{in}. With a dynamic pressure
Test Facility Definition

downstream of the metering plate of 0.0004q_{in}, the dynamic pressure in the metering holes
is 0.9099q_{in}. Holes with an inlet chamfer of 30° and discharge coefficient of 0.94 were
used as on the inboard metering plate, resulting in a total hole area in the centre bleed
sector of 196.85mm². The chosen hole pattern has six 6.5mm diameter holes.

The metering hole pattern in the outer sectors was chosen in a similar manner,
taking the different bleed offtake hole area into account, and a hole pattern with three
5.4mm diameter holes in each outer sector was chosen.

The same technique was used to calculate the extra hole area required in the outer
sectors as in the inboard metering plate. The mass flow and velocity in the outer annulus
are somewhat lower than in the inner annulus and the extra hole area was only 1.66mm²
per outer sector. This was achieved by adding four 0.7mm diameter holes to each outer
sector. As in the inboard metering plate extra bleed holes were added to allow for
adjustment of the bleed mass flow, five 0.7mm diameter holes being added to the centre
sector. The outboard metering plate is illustrated in Fig.2.26.

2.1.5 Deviation of flame tube from nominal design

The flame tube used in the experimental facility had previously been used by Rolls-Royce
for combusting experiments, and it was expected that thermal expansion may have caused
distortion of the liners. The sectoring of the flame tube was another potential source of
distortion, with the release of internal hoop stresses causing the liners to spring outwards
(Fig.2.27). As a result, it was expected that the actual flame tube geometry may differ
from the design geometry. It was also expected that, due to manufacturing tolerances,
small errors in the locations of the primary and secondary ports may exist. A set of
detailed measurements was obtained to accurately determine the true flame tube
geometry.

Because direct measurement of the flame tube profile is extremely difficult,
measurements of the annulus depths were instead obtained. Because the casings were
machined with fixed radii, the annulus depths thus obtained could be used to define the
profile of the flame tube. Measurements were taken through annulus bleed offtakes and
primary and secondary ports to obtain annulus depths at various locations. As can be seen
in the tables below, circumferential variations in the annulus depths were observed, due to
the distortion that occurred in the sectoring of the flame tube, although these variations do
not exceed 4% in the planes of the primary and secondary ports and are thus not expected
to have a substantial effect on the external aerodynamics. However alterations to the annulus depths also resulted from the distortion of the flame tube, and at some points the annulus depth differs from its design value by up to 25%. This must have an effect on the external aerodynamics, but it was not considered to be sufficient to invalidate this project. The axial and circumferential locations of all the primary and secondary ports were also measured. All were found to be offset axially and circumferentially from their design locations, and are thus no longer symmetrically arranged in each burner sector. However the axial and circumferential misalignment of the opposed pairs of ports is not considered to be significant.

All the measurements are presented in detail below. All linear measurements are quoted in mm to one decimal place. The datum centre line ($\theta=0^\circ$) is the centre line of the centre burner. Positive angles are clockwise when viewed from upstream, i.e. looking down the combustor.

The dump gap (Fig.2.28) was measured parallel to the engine axis from the inner and outer lips of the pre-diffuser to a tangent to the flame tube head. The design values are 40.5mm (inner) and 42.3mm (outer). The dump gap was measured at four circumferential locations:

<table>
<thead>
<tr>
<th>Angle</th>
<th>-22.5°</th>
<th>-7.5°</th>
<th>7.5°</th>
<th>22.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dump gap (inner)</td>
<td>43.5</td>
<td>44</td>
<td>43.5</td>
<td>43.5</td>
</tr>
<tr>
<td>Dump gap (outer)</td>
<td>46.7</td>
<td>47.3</td>
<td>46.6</td>
<td>46.3</td>
</tr>
</tbody>
</table>

The flame tube exit sector height (Fig.2.28) was measured between the inner and outer downstream extremes of the flame tube. The design value is 78.2mm.

Measurements were taken at seven locations:

<table>
<thead>
<tr>
<th>Angle</th>
<th>-20.5°</th>
<th>-13.7°</th>
<th>-6.8°</th>
<th>0°</th>
<th>6.8°</th>
<th>13.7°</th>
<th>20.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height</td>
<td>78.6</td>
<td>78.7</td>
<td>78.4</td>
<td>78.4</td>
<td>78.3</td>
<td>78.4</td>
<td>78.3</td>
</tr>
</tbody>
</table>

The annulus height was measured through the centre of the inner and outer bleed ports, normal to the casing (Fig.2.28). The design values are 18.9mm (inner) and 12mm (outer).

<table>
<thead>
<tr>
<th>Angle</th>
<th>-15°</th>
<th>-11.25°</th>
<th>-7.5°</th>
<th>-3.75°</th>
<th>0°</th>
<th>3.75°</th>
<th>7.5°</th>
<th>11.25°</th>
<th>15°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height (inner)</td>
<td>17.2</td>
<td>-</td>
<td>17.2</td>
<td>-</td>
<td>17.4</td>
<td>-</td>
<td>16.8</td>
<td>-</td>
<td>17.2</td>
</tr>
<tr>
<td>Height (outer)</td>
<td>13.5</td>
<td>12.9</td>
<td>12.5</td>
<td>12.4</td>
<td>12.3</td>
<td>11.7</td>
<td>11.7</td>
<td>11.6</td>
<td>12.2</td>
</tr>
</tbody>
</table>
Test Facility Definition

Measurements were taken of the axial and circumferential locations of the port centres. The axial locations were measured parallel to the nominal flame tube axis (canted at 9° from the engine axis) from the crown of the flame tube (Fig.2.28). Annulus depths were measured along the port centre lines from the casing to the chute platform.

The design circumferential locations of the primary ports are ±15°, ±7.5° and 0° (Fig.2.29). The design axial locations are 81.6mm (outer) and 81.1mm (inner), and the design annulus depths are 20.6mm (outer) and 31.7mm (inner). The measured values for the outer primary ports are:

<table>
<thead>
<tr>
<th>Angle</th>
<th>-15.3°</th>
<th>-7.8°</th>
<th>0.7°</th>
<th>7.1°</th>
<th>14.5°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial location</td>
<td>80.8</td>
<td>80.3</td>
<td>80.9</td>
<td>80.7</td>
<td>78.3</td>
</tr>
<tr>
<td>Annulus depth</td>
<td>23.6</td>
<td>22</td>
<td>23.6</td>
<td>23.6</td>
<td>23.3</td>
</tr>
</tbody>
</table>

The measured values for the inner primary ports are:

<table>
<thead>
<tr>
<th>Angle</th>
<th>-15.4°</th>
<th>-8.0°</th>
<th>-0.3°</th>
<th>7.0°</th>
<th>14.3°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial location</td>
<td>78.3</td>
<td>78.4</td>
<td>78.1</td>
<td>78.1</td>
<td>77.1</td>
</tr>
<tr>
<td>Annulus depth</td>
<td>27.1</td>
<td>27.5</td>
<td>27.9</td>
<td>28.6</td>
<td>28.7</td>
</tr>
</tbody>
</table>

The design circumferential locations of the secondary ports are ±18.75°, ±11.25° and ±3.75° (Fig.2.30). The design axial locations are 127.1mm (outer) and 124.3mm (inner), and the design annulus depths are 16.4mm (outer) and 26.7mm (inner). The measured values for the outer secondary ports are:

<table>
<thead>
<tr>
<th>Angle</th>
<th>-19.1°</th>
<th>-11.5°</th>
<th>-4.2°</th>
<th>3.6°</th>
<th>10.7°</th>
<th>18.2°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial location</td>
<td>124.6</td>
<td>124.6</td>
<td>124.9</td>
<td>124.8</td>
<td>123.4</td>
<td>122.6</td>
</tr>
<tr>
<td>Annulus depth</td>
<td>20.1</td>
<td>19.7</td>
<td>19.4</td>
<td>19.2</td>
<td>19.1</td>
<td>19.3</td>
</tr>
</tbody>
</table>

The measured values for the inner secondary ports are:

<table>
<thead>
<tr>
<th>Angle</th>
<th>-18.9°</th>
<th>-11.7°</th>
<th>-4.5°</th>
<th>3.2°</th>
<th>11.1°</th>
<th>18.1°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial location</td>
<td>123.2</td>
<td>122.6</td>
<td>122.2</td>
<td>122.4</td>
<td>122.3</td>
<td>122</td>
</tr>
<tr>
<td>Annulus depth</td>
<td>21.4</td>
<td>22.2</td>
<td>22</td>
<td>21.9</td>
<td>22.3</td>
<td>22.9</td>
</tr>
</tbody>
</table>

While the flame tube exit height was relatively unchanged, changes in the profiles of the liners caused significant changes in the annulus depths. However the annulus depths do not exhibit an excessive circumferential variation, especially over the area of interest in the experimental programme, that is the central 15° sector, where the annulus depths vary by no more than 1.1mm. The alterations in the annulus depths may alter the conditions of the feed to the mixing ports, while the axial and circumferential misalignment of the
opposing jets may affect the impingement of the jets. The increased dump gap is expected
to decrease the pressure loss in the diffuser system, and the fuel injectors were moved to
be correctly located relative to the flame tube back plate and the swirler shrouds.

Although not ideal, these errors do not invalidate the investigation. The basic
aerodynamic features of the combustor are still present, although the opposing jets are not
perfectly aligned. However some misalignment of the jets may occur in production
combustors due to manufacturing tolerances, and it must also be expected that distortion
of the flame tube due to thermal stresses will occur during the lifetime of the combustor.
The data obtained in this project may still be used for CFD validation, because
sufficiently detailed measurements have been obtained to allow the development of a
representative computational model of the test facility geometry.

2.2 Test facility installation
The test facility is mounted vertically within the Airflow Lab at Loughborough University
as shown by Fig.2.31. Air is drawn from atmosphere into a large plenum above the test
facility prior to entering the inlet ducting. Having passed through the rig the air then
issues into a bottom plenum from where it passes through a throttle into a passage leading
to the fans and exhaust system. Two centrifugal fans operate in parallel to provide the
necessary pressure rise to draw air through the facility and exhaust it to atmosphere.
Details of the test facility installation and support structure are discussed in section 3.1.5.

2.2.1 Inlet and exhaust ducting
An inlet duct was designed that undergoes a transition from an 8" diameter circular
section to the test facility inlet section along its 1.15m length, with a short straight section
at the end to ensure a uniform, well defined flow into the rig. The top of the duct is flared
to ensure that the air enters the duct cleanly and is approximately 30cm proud of the floor
of the plenum to prevent the ingestion of poor quality flow from the floor of the plenum.
A gauze was placed just downstream of the flare to generate turbulence. A boundary layer
trip was installed in the inlet duct, six inches upstream of the pre-diffuser inlet, which
consists of a single piece of 0.9mm diameter wire attached around the entire perimeter of
the duct. The inlet duct is illustrated in Fig.2.32.

The exhaust duct was designed to avoid obstructing the LDA traverse system (see
section 3.1.5) and thus had to be canted inwards. There was some concern that the flow
Test Facility Definition

may separate whilst turning through an angle at the rig exit. The flow was expected to be particularly sensitive at this point as it was just downstream of the bleed duct exit slots (see section 2.1.4) and may be unstable at this point. To alleviate this problem the flow was made to accelerate in the exhaust duct by a 25% area reduction. This incurs an increased pressure loss, however it is not significant relative to the overall loss within the test facility. The exhaust duct was given a length of 425mm, as the test facility is mounted 350mm above the roof of the bottom plenum to accommodate the LDA traverse. The exhaust duct is shown in Fig.2.33.

The window cooling system in each sidewall is fed by a separate inlet duct. A rectangular section duct is used, with a flare at the inlet and a metering plate inserted at a location one third of the duct height downstream of the flare. A curved transition duct, with a reduction in area, is used to take the cooling flow from the inlet duct to the sidewall inlet section, as shown in Fig.2.34.

A test facility has therefore been developed that contains all the features that are necessary to model representative internal and external combustor flow fields. This comprises a three burner sector segment of a modern annular flame tube, mounted in Perspex casings that have representative aerodynamic profiles. Particular attention was paid to instrumentation requirements, and the facility was designed to permit measurements of substantially the whole centre burner sector of the combustor by a three component LDA system. Detailed measurements of the flame tube have been obtained to accurately determine the true flame tube geometry and, although some discrepancies were found between the true geometry and the design geometry, none were considered sufficient to jeopardise the project.
3. **Instrumentation and Experimental Procedure**

A variety of measurement techniques were used to obtain flow field data for four purposes: Operation of the test facility, commissioning of the test facility, measurement of the external flow field, and measurement of the internal flow field.

For the operation of the test facility, a fixed pitot probe and static tapping were used to monitor the inlet dynamic pressure. A thermocouple in the inlet plenum chamber was used to monitor the inlet temperature.

For the commissioning of the test facility, provision was made for extensive measurements to assess the behaviour of the flow throughout the diffuser system, with particular emphasis on the behaviour of the boundary layers on the sidewalls. Button-hook type pitot pressure probes were used, with static tappings installed at numerous locations, to measure the dynamic pressure of the flow. Provision was made to insert the button-hook probes at numerous locations throughout the diffuser system. Wool tuft probes were used to visualise the flow in the diffuser system.

Button-hook probes were also used for measurements of the external flow field. Provision was made for the measurement of the inlet velocity profile, to obtain an inlet boundary condition for CFD calculations, and for button-hook probe measurements at the pre-diffuser exit plane and in the feed annuli. A hot wire anemometer was used to obtain the inlet turbulence intensity profile for the CFD inlet boundary condition.

A three component Laser Doppler Anemometry (LDA) system was used for all internal flow field measurements. Being a non-intrusive measurement technique, notwithstanding the need to seed the flow, LDA is ideally suited to the measurement of mean velocities and turbulence quantities in the highly turbulent, highly three dimensional and complex flow field that exists inside a combustor.

This chapter describes these instrumentation systems, with particular emphasis on the LDA system because it was used to collect the majority of the measurements. Section 3.1 describes the LDA system and its traversing system, the LDA calibration and operation procedures, and the measures that were taken to maximise the accuracy of the system, with an estimate of experimental errors. Section 3.2 describes the button-hook pressure probes, the hot-wire anemometer and the wool tuft probes, and their traversing and calibration, with an estimate of experimental errors. Section 3.3 describes the data acquisition and reduction procedures.
3.1 Laser Doppler anemometry

The flow inside a combustor is highly turbulent and three dimensional, and contains regions of recirculating flow. The instrumentation used to measure mean flow field velocities and turbulence data in such a flow field must therefore be capable of resolving all three components of the mean velocity and all six Reynolds stresses, regardless of the turbulence intensity and the direction of the flow. The three-dimensionality of the flow precludes the use of pressure measurement techniques and hot wire anemometry, and hot-wire anemometers are also known to become inaccurate in turbulence intensities exceeding 30% of the mean velocity (Bradshaw, 1971). The measurement technique must also be non-intrusive, which precludes the use of pressure measurements and hot wire anemometers as the highly turbulent and recirculating flow field may be influenced by the presence of the probes. Laser Doppler anemometry (LDA) is a non-intrusive technique which meets these requirements for the measurement of velocity and turbulence data. Through the probability density function (pdf) histograms of the turbulence at each point that it produces, some information can also be derived about the nature of unsteady flows.

While detailed descriptions of the underlying principles of laser Doppler anemometry and the complex optics and processors involved are beyond the scope of this thesis, descriptions of the basic principles and systems involved are useful in understanding the operation of the LDA system as it was used for the majority of the experimental work in this project. The need to obtain good quality LDA measurements throughout the internal flow field was a major consideration in the design of the test facility (see Chapter 2). A description of the operating principles of the LDA system and the processors and software used are given here, plus a description of the alignment techniques and traversing system.

3.1.1 Introduction to Laser Doppler Anemometry

The operation of the dual beam type Laser Doppler Anemometer used in this project is described by the fringe model. If two coherent light beams with plane wave fronts - as in the Gaussian waist region of a laser beam - are focused on a single point on a surface, a pattern of constructive and destructive interference fringes is formed as shown in Fig.3.1. The laser beams must cross at a solid surface for this pattern to appear. If a small particle passes through the beams, the light and dark fringes appear alternately as the particle
moves through points of constructive and destructive interference. The particle scatters light as it moves across each fringe, the intensity of the scattered light rising and falling at a rate that is proportional to the particle's speed. The particle moves with a finite velocity and scatters light with a frequency that is slightly different from that of the incident light, an effect known as the Doppler effect, although the difference between the incident and scattered light frequencies is very small - typically less than $10^7$ times the incident light frequency - and also depends on the light collection angle (Yule, 1996). The scattered light can be collected by a detector such as a photomultiplier and converted to a frequency modulated electronic signal that has the form shown in Fig.3.2. This is known as the "Doppler burst". Its Gaussian shape is determined by the distribution of light intensity in the laser beam, the peak at the centre being caused by the maximum light intensity at the centre of the beam. The Doppler frequency ($f_D$), that is the frequency of the wave form within the burst, is determined by the fringe spacing and the particle velocity. The fringe spacing is dependent on the angle between the beams ($\theta$) and the wavelength of the incident light ($\lambda$). The particle velocity, and therefore the instantaneous velocity of the fluid carrying the particle, can be determined from the equation

$$U = f_D \times C$$  \hspace{1cm} (3.1)

where $C$ is a calibration factor which is calculated from

$$C = \frac{(\lambda/2)}{\sin(\theta/2)}$$  \hspace{1cm} (3.2)

Thus if $f_D$, $\lambda$ and $\theta$ are known the velocity component, $U$, can be determined. The direction of the measured velocity component is normal to the fringes, i.e. perpendicular to the line bisecting the two beams and in the plane of the beams (Fig.3.3).

In a simple LDA system the two beams are produced by a beam splitter, splitting a single beam into two beams of the same frequency, thus resulting in the formation of stationary interference fringes. This results in a directional ambiguity, as the system is unable to distinguish between particles moving through the fringes with a positive or negative velocity. In addition, stationary particles cannot be detected, thus rendering the system useless where very highly turbulent or reversing flows are found. This can be overcome by shifting the frequency of one beam, causing the fringe pattern to move within the measurement volume. An optical component called a Bragg cell applies a frequency shift to the incoming beam, increasing the frequency of the light by a certain amount. It should be noted that if the power of the Bragg cell is low, two separate beams
are produced (one shifted and one unshifted), so it is necessary to adjust the power of the cell to produce shifted and unshifted beams of equal intensity. With the fringes now moving within the measurement volume, a stationary particle produces a burst with a Doppler frequency equal to the frequency shift, and a particle with a negative velocity produces a Doppler burst whose frequency can be distinguished from that of a particle with a positive velocity of the same magnitude. Since the frequency shift is known its effects can be accounted for during signal processing, enabling the velocities to be obtained.

To resolve all three velocity components of a three dimensional flow three beam pairs can be used. The beam emitted by a laser typically contains light consisting of a number of different frequencies and these can be separated by refraction if passed through a prism with a sufficient optical path length. Three different coloured pairs of beams are therefore used, arranged in such a way that three independent velocity components can either be directly measured or derived from the directions of the measured velocity components. It should be noted that the beams of the three colours are emitted with different light intensities, the green, blue and violet beams having typical intensities in the ratio 7:4:1. Band pass filters are used in the receiving optics so that each of the three photomultipliers that must be used receives light of one colour only, each photomultiplier thus receiving scattered light from only one beam pair. If the three orthogonal velocity components are not directly measured, as is usually dictated by restricted optical access to the flow field, the measured components are transformed to the orthogonal components by a transformation matrix:

\[
\begin{pmatrix}
  u \\
  v \\
  w
\end{pmatrix} =
\begin{pmatrix}
  a_{11} & a_{12} & a_{13} \\
  a_{21} & a_{22} & a_{23} \\
  a_{31} & a_{32} & a_{33}
\end{pmatrix}
\begin{pmatrix}
  U_1 \\
  U_2 \\
  U_3
\end{pmatrix}
\] (3.3)

where \(U_1, U_2\) and \(U_3\) are the three measured velocities and \(u, v\) and \(w\) are the three orthogonal components. The derivation of the transformation matrix is discussed in detail in section 3.1.6 below.

To determine turbulent normal and shear stresses the data recorded from the three processors must be time correlated. In other words signals are only processed from particles that are observed and recorded by all three processors. However some individual bursts may subsequently be rejected during processing due to a poor signal to noise ratio. The arrival time and transit time (the time taken for the particle to pass through the
measurement volume, also known as the residence time) are therefore also recorded so further coincidence filtering can be done by post processing software (as described in section 3.1.7 below). This ensures that the eventual flow statistics are based only on particles that have been observed by all three processors. To record coincident bursts the accurate alignment of the three beam pairs is crucial. If the three measurement volumes are misaligned then only particles with a particular trajectory - the trajectory that takes the particle through all three measurement volumes - can produce bursts that will be recorded by the processor, resulting in an incorrect velocity measurement. This is avoided by using an accurate alignment technique. A new alignment technique, developed in the course of this project, is described in section 3.1.6 below.

3.1.2 The LDA system

The three component LDA system used in this project is illustrated in Fig.3.4. A single Coherent 5W Ar+ ion laser provides the light source, whilst the modular optics and Burst Signal Analyser (BSA) processors were supplied by Dantec.

Light emitted by the laser is initially passed into the transmitter, a unit that performs several functions. The first optical component is the Bragg cell, which applies a 40MHz frequency shift to one of the emerging beams. For the optical arrangement used in this project the three beam pairs all have calibration factors (see eqn.3.2) in excess of 3m/s per MHz (the exact value depends on the wavelength of each beam pair), enabling each processor to record velocities in the range of ±120m/s. Since the maximum inlet Mach number the facility can provide is 0.15 (which corresponds to a velocity of 51m/s), and it is unlikely that velocities significantly higher than this will be encountered anywhere in the combustor, this velocity range is more than adequate for the measurements being considered here. The Bragg cell can be adjusted to balance the intensities of the shifted and unshifted beam pairs and thereby maximise the signal to noise ratio.

The two beams then pass into a dispersion prism which splits the light into all its constituent colours. The large number of slightly diverging beams travels along a long optical path so that the three chosen beam pairs can be separated before leaving the transmitter. The three chosen colours are green (wavelength $\lambda = 514$nm), blue (λ =
488nm) and violet (\(\lambda = 476.5\text{nm}\)), these being the three components of the initial beam that have the greatest intensities.

Each of the six beams is then passed into a fibre manipulator. This device contains a prism whose lateral position and angular orientation can be manipulated by four thumbscrews to align the beam with the narrow optical fibres that transmit it to the probe. Correct alignment of the beam and fibre is crucial as improper alignment reduces the intensity of the light in the measurement volume and can cause damage to the fibre's cladding, which in turn reduces the transmission efficiency of the fibre. Similarly care must be taken to avoid twisting or excessive bending of the fibre and damage to the cladding.

The six fibres are encased within two cables which transmit the light to the two probes. The 1D probe has a single beam pair (the violet pair) and the 2D probe has two pairs, arranged so that the green and blue pairs are perpendicular to each other as shown in Fig. 3.5. The probes are fitted into beam expanders, which take the parallel beams emitted from the probes and increase their separation and diameter, in this case by a factor of four. The greater beam diameter allows the beams to be focused down to a smaller beam waist, so the measurement volume is smaller and the light intensity in the measurement volume is greater. At the front of each beam expander is a lens that focuses the beams. For the measurements reported here a focal length of 250mm was used. The green and blue beams, transmitted by the 2D probe, are automatically focused onto the same point in space. As outlined in section 3.1.1, the focal point must be accurately aligned with the focal point associated with the 1D probe. Furthermore, the test facility geometry precludes the direct measurement of the three orthogonal velocity components directly so a transformation matrix must be used. The accuracy of the transformation is optimised by optimising the arrangement of the two probes: The violet (1D) beam pair lies in the plane of the two probes, with the plane of each 2D beam pair at 45° to this plane and the included angle between the two probes set to 45° to optimise the resolution of the transformation from the measured velocity components to the chosen co-ordinate system (Carrotte and Britchford, 1994). An included angle of 45° may not be possible due to the rig geometry and, to maintain the resolution of the three orthogonal components, it should not be reduced below 30°.
The probes also contain the receiving optics and are connected to optical fibres to transmit the scattered light to the photomultipliers (Fig.3.4). Each photomultiplier is fitted with a narrow pass filter and if the 2D receiving optics are connected to the violet filter, and the 1D receiving optics connected to the green and blue filters, then the system becomes "cross-coupled". The 2D probe is now acting as the receiving optic for the light scattered from the 1D beam pair and vice versa. In the alternative "back scatter" arrangement, whereby each probe acts as the receiving optic for its own transmitted light, the measurement volume is large, due to the long measurement volume of each beam pair, and the signal to noise ratio (SNR) is poor because the intensity of light scattered at 180° to the incident light is low. The cross-coupled arrangement has a small measurement volume. The measurement volume of each beam pair is effectively shortened to the width of the measurement volume from the other probe, as shown by Fig.3.6, because the receiving optic receives light of the wavelength emitted from the other probe and thus receives light from a volume much smaller than the measurement volume of the individual beam pair. As a result the probability of receiving a burst from a particle that is not observed by all three processors is reduced and the validated data rate is increased. The intensity of light increases as the observation angle is reduced from 180°. The signal to noise ratio (SNR) is therefore improved by the use of the cross-coupled arrangement. The photomultipliers convert the varying light intensities, associated with each burst, to electronic signals that can be sampled by the three signal processors.

3.1.3 The Burst Spectrum Analyser (BSA)

The BSA performs a spectrum analysis to evaluate the Doppler frequency of each Doppler burst signal that it receives from the photomultiplier. A complete description of the electronics and the signal processing procedures used by the BSA is beyond the scope of this thesis and can be found in Dantec (1991). The brief description that follows is intended as a simple explanation of the operation of the BSA. An evaluation of the BSA was published by Tropea et al (1988). The BSA was found to be a versatile and robust instrument, and performed better than the counter, frequency tracker and photon correlator instruments to which it was compared.

The BSA takes a number (N) of samples of a burst (Fig.3.8) and then performs a Discrete Fourier Transform (DFT) to calculate the frequency spectrum of the sampled
time signal. The signal is sampled at regular intervals with a period $T_s$ and hence a sampling frequency $f_s = \frac{1}{T_s}$. The frequency resolution $f_N$ (the distance between samples) is the sampling frequency $f_s$ divided by the number of samples, $N$. The resolution is therefore improved by increasing the number of samples, however an increase in the sampling frequency would diminish the frequency resolution. The sampling frequency must be at least twice the upper frequency of the signal to avoid frequency distortions.

The BSA uses a sampling frequency 1.5 times the selected bandwidth, the maximum bandwidth being 32MHz. However the frequency range of the BSA extends to 80MHz and the sampling frequency that would be required at 80MHz, that is greater than 160MHz, would lead to a poor resolution. To overcome this the frequency spectrum is down-shifted to zero frequency and the power spectrum is calculated from this down-shifted version of the frequency spectrum.

A maximum 64 samples of the signal are used to calculate the spectrum. The position of the maximum power of the spectrum is taken to be the Doppler frequency, $f_0$, which then allows the instantaneous velocity associated with the burst to be calculated. An interpolation technique is used to improve the resolution around the maximum of the spectrum. A zero-filling technique adds $N$ zeroes to the $N$ signal samples, so the DFT is now calculated from twice as many samples without affecting the spectrum, effectively improving the resolution by a factor of two. The BSA performs a comparison of the two largest maxima of the spectrum to assess the spectrum's quality. If the ratio of these maxima is greater than 4 then the spectrum is deemed to be of good quality and is validated.

3.1.4 Seeding
The air entering the rig must be seeded to provide a sufficient number of small particles for the LDA system to operate. Because the velocity of the air flow is represented by the velocity of particles carried by the flow, the ability of the particles to faithfully follow the flow is crucial. This is of particular importance in highly turbulent flows. The ability of the particles to follow the flow is a function of particle drag (Dring and Suo, 1978), which is principally a function of the particle's diameter and the fluid velocity. A TSI six jet atomiser was used to provide particles with a mean size of diameter of 1.07μm (Bailey, 1997) using a low viscosity oil. Dring (1982) and Durst et al (1981) indicate that particles
of such a small size are necessary to ensure that they accurately follow the flow path which, in this case, is highly turbulent and, in some regions, contains swirling flows and large scale recirculations. Other workers have used similar sized particles in swirling or recirculating flows; Nejad et al (1989) used micron-sized particles in a confined swirling flow, and Stevenson et al (1984) micron-sized particles in the highly turbulent, recirculating flow behind a backward-facing step.

The particles were supplied to the test rig via a pipe, with a large number of small holes, that was placed approximately 30cm above the rig intake (Fig.3.7). In initial tests of the LDA system with this test facility, good data rates (greater than 0.5kHz) were observed throughout the centre sector of the flame tube. Thus this seeding arrangement was found to provide well dispersed seed to the whole centre sector of the flame tube. Generation of particles smaller than this is difficult and this particle size was deemed to be acceptable for an accurate resolution of the turbulence quantities of interest.

3.1.5 LDA traverse system

It was decided at an early stage that the majority of the internal flow field measurements would consist of area traverses normal to the axis of the flame tube, as shown in Fig.3.9. A cylindrical polar co-ordinate system was defined, as shown in Fig.3.9, the u, v and w velocity components describing the axial (parallel to the flame tube axis), radial (normal to the flame tube axis) and tangential components respectively. A traverse system was therefore required to move the LDA measurement volume circumferentially and radially. By moving the measurement volume along an arc that was concentric with the test rig, the relationship between the measured velocity directions and the cylindrical polar co-ordinate system remained constant. The same transformation matrix could therefore be used at every point in the traverse. While motorised traverses were used to perform area traverses inside the flame tube, axial adjustment was also necessary to move the measurement volume between traverse planes.

In addition to the circumferential, radial and axial movement of the LDA probes, rotation of the probes in the horizontal and vertical planes was necessary to maximise the LDA access area through the limited area of the sidewall window, as shown by Fig.3.10. The rotation in the vertical plane was necessary because the probes must be pointed upwards for measurements close to the flame tube heat shield, which would otherwise
block the laser beams issuing from the top probe. The rotation in the horizontal plane was necessary to point the beams towards the inner or outer liner, so that the radial extent of the traverse could be maximised. Access to a large part of the centre sector would have been restricted if this rotation were not incorporated.

A slide ring system was used to perform the circumferential traverse (Fig.3.11). This consisted of a $90^\circ$ ring segment of 1033mm mean diameter on the outside of the rig and a $180^\circ$ segment of 468mm mean diameter inside the rig. A carriage ran along each ring segment, with a beam joining the two carriages. The outer ring segment was always situated to one side of the rig, to allow measurements through the sidewall on that side, and could be moved to the other side to allow measurements through the other sidewall. Because the traverse system was mounted outside the test facility, the correct positioning of the slide rings relative to the test facility, the flatness of the slide rings and the rigidity of the mounting of both the slide rings and the test facility were crucial, so that the axial and radial locations of the measurement volume would remain constant throughout each circumferential traverse. The slide rings were bolted to a thick plywood board, which was mounted above the bottom plenum chamber. To make the board rigid and maintain its flatness, a triangular frame was constructed from a strong aluminium extrusion and bolted to the underside of the board as shown by Fig.3.12. The stiffness of the board was further improved by projecting two stiffening arms from the diagonal sides of the frame towards the corners of the board. The frame and board were supported by three legs, whose height was adjusted to ensure that the edges of the board were not in contact with the top of the plenum chamber (see Fig.2.31), thus ensuring that all of the weight was taken by the aluminium frame and none by the board, and preventing the transmission of any vibration from the plenum chamber to the test facility and traverse system. The gap between the board and the plenum chamber was sealed by a rubber gasket, and the legs were bolted to the concrete floor of the plenum chamber.

The test facility exhaust duct projected through a hole in the plywood board, together with two steel supporting rods. These rods were attached to a frame, which was connected to the main supporting frame underneath the board such that the position of the test facility could be adjusted (Fig.3.13). Further support was given by two steel posts that connected the test facility to a steel girder in the ceiling of the laboratory. The rods were adjusted so that the base of the test facility was horizontal, while the slide rings were
Instrumentation and Experimental Procedure

carefully aligned to the test facility. The slide rings were aligned to the pseudo 'engine' centre line so that they were concentric with the casings of the test facility. Thus the circumferential traverse moved the control volume at a fixed radius within the test facility. The test facility was made vertical, a spirit level being used at locations where the casings had machined surfaces deliberately incorporated which were parallel with the engine axis. The flatness of the ring segments was measured. The slide rings were found to remain flat to within 0.025mm when installed on either side of the rig. This is small compared with the diameter of the measurement volume (0.15mm), thus the positional accuracy of the traverse system is not considered to be significantly affected.

The outer ring segment had a gear machined into one face. A stepper motor was used to drive the carriages round, through a 100:1 gearbox connected to this gear. This resulted in a traverse calibration of 1428.57 motor steps per degree of rotation. At a radius of 350mm, this equates to a distance of 0.004mm for every step moved by the motor. The traverse thus had a very fine resolution.

The linear traverse, which provides the radial movement of the LDA probes, was displaced from the radius along which the measurement volume was traversed, as shown by Fig.3.14. Because of the need to rotate the probes to enable measurements throughout the flame tube, this displacement was not constant, and it was necessary to rotate the radial traverse to align the movement of the measurement volume with the radius. A rotary stage was therefore mounted on the beam between the two carriages. This rotary stage performed the necessary rotation and could be locked to prevent any deviation from its correct position during the traverse. The linear traverse was mounted on this rotary stage and inclined at 9° so as to be perpendicular to the flame tube centre line. This meant that all radial and axial velocity components would be measured in the radial and axial directions relative to the flame tube.

A tall rectangular frame was constructed from a strong, light aluminium extrusion and mounted on the linear traverse as shown by Fig.3.15. The frame was mounted so that it could rotate about its centre line to allow the probes to be pointed towards either flame tube liner. A sliding block was mounted on a vertical threaded rod between the two struts that made up the rectangle to allow vertical movement of the probes. The probes were mounted on translation stages, to allow linear adjustment of the probes, accurate to 0.01mm, for accurate alignment of the two probes, and on circular blocks for a crude
rotational adjustment of the probes. The probes were mounted at an angle on a second 
frame, and were mounted symmetrically for a constant included angle between the two 
probes. Rotation of the probes in the vertical plane, to allow them to be pointed towards 
or away from the flame tube back plate, was provided by a circular face machined into the 
sliding block.

3.1.6 LDA alignment and transformation matrix calculation
The importance of accurate alignment of the two probes and the need for a transformation 
matrix were discussed in section 3.1.1. The method that was developed for the accurate 
alignment of the probes and calculation of the transformation matrix is discussed here.

3.1.6.1 Alignment of the probes to obtain coincident measurement volumes
When aligning the LDA system it is important to account for refraction of the beams by 
the laser access window. To minimise refraction a thin (1mm) window was used. It was 
established that misalignment due to refraction did not occur during a circumferential 
traverse, however the alignment of the probes must be performed with the window 
installed at a representative distance and angle from the probes to account for refraction 
effects. Because the angle between the probes and the window was subject to substantial 
variation in both the vertical and horizontal planes, the alignment was performed with the 
probes installed on the traverse, rather than by removing them from the test facility.

A new technique was developed for the alignment of the probes. Originally a 
50μm pinhole was used that was a third of the diameter of the beams, alignment being 
performed by projecting the beams onto a screen and adjusting the positions of the probes 
until all six beams passed through the pinhole and appeared on the screen. Subsequently 
Swales et al (1993) used a technique whereby the location of each beam was measured by 
a light dependant resistor (LDR) and a smaller pinhole. They found that their technique 
improved the alignment accuracy (indicated by improved data rates) and so this method 
was adapted and developed for the LDA system used in this project.

A pinhole meter was developed with a 20μm pinhole and LDR mounted as shown 
in Fig.3.16. The pinhole was mounted on three 16mm translation stages so that it could be 
positioned to an accuracy of 10μm. The three translation stages were mounted 
orthogonally on the test facility and aligned with the test facility's co-ordinate system, as
shown by Fig.3.17, to allow accurate positioning of the pinhole in the x, r and θ directions. It was found that the resistance of the LDR, measured by a digital volt meter, varied as the beam moved across the pinhole, reaching a minimum when the pinhole was aligned with the centre of the beam. One problem was encountered, as the minimum LDR reading was dependant on the point at which the beam hit the lens of the LDR. This happened because the LDR used a filament behind the lens to measure the light, the reading depending on the point at which the beam hit the filament. This was eliminated by roughening the surface of the lens with sandpaper so that the beam, on striking the LDR lens, was evenly diffused. The minimum reading was then found to be distinctive and repeatable, with an easily obtained positional accuracy of 10μm. However it was found that the lens would burn if the laser was operated at too high a power, particularly with the more powerful beams (green and blue), and the steadiness of the reading was also found to depend on the beam intensity. Optimum laser output powers of 0.5W, 1.0W and 2.0W were found for the green, blue and violet beams respectively.

A locating plate was developed to enable the measurement volume to be accurately positioned within the test facility. A perspex plate was made to fit into the outer bleed duct exhaust slot, as shown by Fig.3.18. Slots were machined into the plate so that a vertical steel plate could be inserted with its outer face at ±3.75° from the rig centre line. The plate was dowelled and its location was found to be repeatable to within 0.05mm. As each area traverse of the centre sector was performed in two 7.5° halves, the 3.75° angle represented the centre point of an area traverse. By aligning the LDA system to obtain coincident measurement volumes at the centre of the traverse, the effects of refraction due to the circumferential movement of the probes is minimised.

Alignment to obtain coincident measurement volumes (hereafter referred to as "obtaining coincidence") required the laser access window to be positioned at the appropriate angle and distance from the pinhole meter. The measurement volume was first positioned visually on the locating plate, this being achieved by moving the circumferential traverse until two beams from a single probe were seen to converge on the plate. The measurement volume was thus deemed to be nominally located at the 3.75° angular location. The window was attached to the circumferential traverse system so that its position relative to the probes would not change as the traverse system moved, and the traverse system moved out until the beams converged on the pinhole. The system was
Instrumentation and Experimental Procedure

thus set so that the alignment of the probes could be performed with the window appropriately positioned to account for refraction effects (Fig.3.19).

Because the green and blue beams in the 2D probe were aligned to be coincident in the factory, it is only necessary to align the 1D and 2D probes. The receiving optics of each probe are also aligned with the measurement volume in the factory (which is not the case in larger probes where each beam can be steered individually). Hence by aligning the green and violet beams, coincidence of all three measurement volumes is achieved. Coincidence of the receiving optics is therefore also achieved. The direction vectors of all four violet and green beams were obtained by measuring the axial (x) and radial (r) co-ordinates of each beam at two different tangential (θ) locations - normally the two extremes of the θ translation stage, as shown by Fig.3.20(a). By a simple vector calculation, the focal points of the two probes could then be calculated, these points being the points at which the two beams from each probe crossed. Only the x and θ co-ordinates were used for this calculation - a 2D calculation was found to be adequate so the extra complication of a 3D calculation was unnecessary. The calculated focal points were checked by comparing the x and r co-ordinates of each beam pair at the calculated θ co-ordinates. The average of the two beam vectors from each probe was also calculated, these average vectors being the direction vectors of the translation stages on which the probes were mounted. The vector for each probe was thus defined in terms of a point in space (the focal point) and a direction (the average vector). A 2D vector calculation was then performed to find the x and θ co-ordinates of the coincident point, the unique point in space at which the focal points would coincide, as illustrated by Fig.3.20(b). The two focal points were then moved onto these x and θ co-ordinates by setting the pinhole to this point and moving each probe until the minimum LDR reading was reached. Discrepancies in the r co-ordinates of the two focal points were then removed by adjusting the r co-ordinate of the 2D focal point, tilting the probe by using a screw attached to the underside of the probe holder as shown by Fig.3.20(c). This method was found to be reliable and achieved probe alignment to within 30μm. The technique was rigorously tested on an earlier test facility and compared with a technique that had previously been used. Considerable improvements in validated data rates were achieved with this technique, indicating that the accuracy of the alignment was considerably improved.
3.1.6.2 Calculation of the transformation matrix

Measurement of the beams is also necessary for the calculation of the transformation matrix. As shown in Fig.3.3, each beam pair measures a velocity whose direction is perpendicular to the vector that bisects the two beams and in the plane of the beams. This is the component of the velocity of the particle in this direction. Referring to Fig.3.21(a), the unit vector \( \hat{u}_x \) that describes the direction of the velocity component \( u_x \) measured by the beam pair (subscripts G, B and V being used in place of the subscript x to denote the green, blue and violet pairs) may be described in terms of the unit vectors in the directions of the orthogonal vectors \( u \), \( v \) and \( w \) by the equation:

\[
\hat{u}_x = k_{xu}\hat{u} + k_{xv}\hat{v} + k_{xw}\hat{w}
\]

where \( k_{xu} \), \( k_{xv} \) and \( k_{xw} \) are constants whose values depend on the direction of \( u_x \). For the three colour system this matrix can be written:

\[
\begin{bmatrix}
\hat{u}_G \\
\hat{u}_B \\
\hat{u}_V
\end{bmatrix} =
\begin{bmatrix}
k_{Gu} & k_{Gv} & k_{Gw} \\
k_{Bu} & k_{Bv} & k_{Bw} \\
k_{Vu} & k_{Vv} & k_{Vw}
\end{bmatrix}
\begin{bmatrix}
\hat{u} \\
\hat{v} \\
\hat{w}
\end{bmatrix}
\]

(3.4)

However the transformation matrix that is required is the one that converts \( u_G \), \( u_B \) and \( u_V \) into \( u \), \( v \) and \( w \). This is the inverse of the above matrix.

This calculation depends on the measurement of the directions of all six beams and the calculation of the direction of the measured velocity components \( u_G \), \( u_B \) and \( u_V \). The pinhole meter and translation stages described in section 3.1.6.1 were again used for the calculation of the transformation matrix. However it was necessary to ensure that the traverse system was correctly aligned first. With the window in place, the measurement volume was moved onto the locating plate at 3.75°. This could be accurately achieved by measuring the photomultiplier output on one BSA, this output reaching a maximum when the measurement volume was located on the plate. This locating method was found to be accurate to 0.02° (0.12mm at a radius of 350mm). The measurement volume was then moved to the rig centre line and the rotary stage was adjusted so that the linear traverse was parallel with the centre line. This was repeated until no further adjustment was necessary, thus ensuring that the linear traverse would move the measurement volume along a radius. Whenever any changes were made to the orientation of the probes, the distance between the measurement volume and the linear traverse was altered. Hence the linear traverse no longer moved the measurement volume along a radius. Thus the
realignment of the rotary stage was necessary whenever changes were made to the orientation of the probes.

The circumferential traverse system then moved the measurement volume onto the pinhole. Measurements of the locations of all six beams were taken at two planes of constant $\theta$, thus providing $x$, $r$ and $\theta$ co-ordinates for two points on each beam for the calculation of the vector of that describes each beam.

The six beam vectors have been measured and the direction of each measured velocity component, $u_n$, must be calculated. Referring to Fig. 3.21(b), the vector $u_n$ must be calculated from the beam direction vectors $v_1$ and $v_2$. The mean beam direction (i.e. the direction of the bisector of the two beams) is $v_m = 0.5(v_1 + v_2)$. The vector product of $v_1$ and $v_2$, $v_n$ is the vector normal to the plane containing both beams. The direction of $u_n$ is normal to both $v_n$ and $v_m$, so $u_n = v_n \times v_m$. Care must be taken at this stage to ensure that the vectors have the correct direction. If two beams are vertically aligned and the top beam is the unshifted beam, then the velocity direction is downwards. Having calculated the direction of $u_n$, the constants in the matrix equation above (eqn. 3.4) can easily be found. The transformation matrix that converts the measured velocity components into the required orthogonal velocity components is calculated by inverting this matrix. A FORTRAN program was written to calculate the transformation matrix. This program was rigorously tested by comparing this method with matrices calculated using the earlier methods, and was found to be correct.

3.2 External flow field measurements

3.2.1 Button hook probes

A button hook pitot probe, the operation of which has been described by Carrotte et al (1993), is shown in Fig. 3.22. Relative to more conventionally shaped probes the button hook can be inserted into a test facility through a single small hole whilst the semicircular shape of the probe head allows measurements to be made at convex surfaces. The semicircular head also allows the probe to be rotated about its axis, so that at the same point in the flow pressures can be recorded with the probe facing upstream or downstream.

The stagnation pressure of the flow is obtained from measurements with the probe facing upstream. A 'pseudo-static' pressure can be obtained with the flow facing
downstream. A calibration factor can be used to derive the flow static pressure from the stagnation and pseudo-static values. To obtain the calibration factor a small wind tunnel with a venturi inlet (Fig.3.23) was used, with the probe facing upstream as shown. The flow is known to be uniform and the turbulence intensity is low at the centre of the tunnel. A static tapping on the tunnel wall allowed the true static pressure to be measured and the dynamic pressure to be derived. The button hook probe was placed in this flow facing both upstream and downstream in order to obtain the total and pseudo-static pressures.

From these measurements the calibration factor could be obtained:

\[ K = \frac{p - p_{ps}}{P - p} \]  

(3.5)

where \( P \) and \( p \) are the total and true static pressures, and \( p_{ps} \) is the pseudo-static pressure.

As the calibration factor may vary slightly with Reynolds number, the calibration was repeated at a number of flow rates with dynamic pressures in the range 6 to 150 mm H\(_2\)O that was expected to be encountered in the test facility. The calibration factor of the button hook probe that was used for all pressure measurements was 1.39, with a maximum variation over the range of dynamic pressures for which it was calibrated of \( \pm 1.4\% \).

Two types of traverse were used to position the button hook probe in the test facility. A manual traverse, shown by Fig.3.24, was used at various locations where limited space - for example due to the proximity of laser access windows, static tapping tubes or external rig features - precluded the use of the larger motorised traverse described below. The manual traverse consisted of a threaded brass shaft that could be screwed into the casing. The probe was held in the centre of the shaft and moved along the shaft by turning a threaded ring on the outside of the shaft, with a positional accuracy of 0.2 mm.

Provision was made to fit this traverse to the test facility to perform radial traverses at numerous locations throughout the diffuser system.

At three planes a motorised traverse, shown by Fig.3.25, was used. The probe was driven by a stepper motor on a lead screw and guided by a linear rail, with a positional accuracy of 0.025 mm. This traverse was used for a radial traverse on the centre line at the pre-diffuser exit plane and area traverses of the inner and outer feed annuli at planes X12 and X02 (Fig.3.26). At the latter planes the traverse was mounted on a perspex slider that moved circumferentially over a perspex block fitted in place of the laser access windows.
(Fig.3.27) to allow the probe to be traversed circumferentially. The combination of the radial and circumferential traverses thus allowed an area traverse of the centre sector of each annulus to be performed.

3.2.2 Hot wire anemometry

The constant temperature hot wire anemometer operates by maintaining a wire at a constant temperature by varying the voltage across it. As the flow passes over the wire, it is cooled by an amount that is related to the velocity of the impinging air. The cooling of the wire causes its resistance to decrease, so a greater voltage is required to maintain its temperature. The velocity of the air and the voltage across the wire are related by:

\[ E^2 - E_0^2 = BU_c^2 \]  

(3.6)

where \( E \) and \( E_0 \) are the measured voltage and the voltage at zero velocity, and \( U_c \) is the cooling velocity, and \( B \) and \( n \) are calibration constants. The frequency response of the hot wire anemometer is such that instantaneous velocity values can be obtained and turbulence quantities can be measured.

A hot wire anemometer was used to obtain the radial profile of turbulence intensity at the pre-diffuser inlet plane, although the traverse stopped short of the inner wall to avoid wire breakage. Also the hot wire is sensitive to the direction of the flow. The hot wire is placed in the flow as shown by Fig.3.28, but it should be remembered that the wire is sensitive to flow direction. The velocity component along the wire axis, \( w \), has relatively little effect on the cooling of the wire, whereas the sensitivity of the wire to the velocity components normal to the wire is much greater. For a typical hot wire the sensitivity of the effective cooling velocity to the \( u \), \( v \) and \( w \) components is approximately:

\[ U_c^2 = u^2 + 0.85v^2 + 0.03w^2 \]  

(3.7)

The measured turbulence intensity is therefore not the true turbulence intensity, but an approximation of the intensity of the turbulence in the plane normal to the wire. The inlet turbulence intensity profile presented in Chapter 5 is intended to provide data for an inlet boundary condition for CFD calculations. However it is considered that an LDA traverse should be performed in the future to provide a more complete measurement of the turbulence at the inlet.
Instrumentation and Experimental Procedure

A Dantec type 55P11 miniature single wire probe was used. The Dantec Streamline hot wire system was used for data acquisition, controlled from a personal computer (PC) by Dantec StreamWare software. A full description of this system is outside the scope of this thesis. For more information, refer to the Streamline User's Guide (Dantec, 1996).

Calibration of the hot wire was achieved by measuring the wire voltage at a range of known velocities. The calibration was performed with the wire located at the centre of the pre-diffuser inlet passage. The fixed inlet pitot probe and static tapping were used to derive the mean velocity at a number of different flow rates, and the calibration constants were calculated by the StreamWare software.

The manual traverse, as was used to traverse the button hook probe (see section 3.2.1), was used to traverse the hot wire probe.

3.2.3 Flow visualisation

A wool tuft probe was used for visualisation of the flow in the diffuser system. The probe, shown by Fig.3.29, was made from 0.9mm thick piano wire. This was flexible, allowing the probe to be bent to gain access to much of the diffuser system through the probe access holes in the casings. The wool tufts were made from cotton, and a double loop was employed as shown by Fig.3.29 to enable the tuft to move freely. The length of the tuft was 1.5cm.

The wool tuft is a crude but useful means of observing the behaviour of the flow. It provides an indication of separation, reversal, and instability in the flow, in addition to indicating the flow direction. As described in Chapter 4, it was used to assess the behaviour of the flow adjacent to the sidewalls in the feed annuli. The wool tuft enabled a picture of the separation of the boundary layers, and the resulting flow field, to be obtained quickly and easily. It also enabled the quick assessment of the effects of modifications. As described in Chapter 5, it was also used to visualise the flow entering the flame tube ports. The interpretation of these wool tuft flow visualisation results is highly subjective, as the exact location of the tuft in the annulus cannot be determined. However useful results were obtained, as discussed in Chapter 5, in a fraction of the time necessary for LDA to perform the same measurements.
3.3 Data acquisition and reduction

3.3.1 Operation of the test facility and traverse systems

The test facility was operated at a constant inlet Mach number. For reasons that are discussed in Chapter 4, a Mach number of 0.12 was chosen to optimise the performance of the LDA system. Although the behaviour of the flow in the combustion system is insensitive to changes in the inlet velocity, as discussed in Chapter 1, this inlet Mach number was maintained within a tolerance of ±0.0005 (0.4%) so that all the measured flow field data can be directly compared. Monitoring of the inlet Mach number was performed by a PC as follows.

The ambient temperature, \( t_a \), was recorded by a thermocouple in the inlet plenum chamber. The ambient pressure, \( p_a \), was read from a barometer at regular intervals and entered into the software, to allow the calculation of the air density. The inlet velocity \( V_{in} \) was derived from the dynamic pressure recorded by the fixed pitot probe and static tapping at the pre-diffuser inlet plane (Fig.3.30). The speed of sound was calculated from the measured ambient temperature to allow the Mach number to be obtained. With the centrifugal fan operating at a constant speed setting, little adjustment was necessary to maintain the inlet Mach number within its tolerance. However the Mach number was continuously monitored and adjusted when necessary by adjusting the throttle in the passage between the exit plenum chamber and the fan.

Control of the stepper motor driven traverse systems (the motorised traverse for button hook measurements and the circumferential traverse for LDA measurements) was facilitated by the PC. The PC communicated with CIL Microsystems control modules via a CIL Alpha 03 controller card. The controller card transmits commands to a series of other cards. A single channel Alpha-S card was used to transmit commands to the stepper motors. The linear traverse for LDA measurements was driven by a servo motor. This was controlled by the PC via a McLennan servo motor controller that communicated with the PC via the serial port. The servo motor provided a feedback to the McLennan controller for accurate control of the radial position of the traverse.

Acquisition of pressure and temperature data was also facilitated by the PC. Pressures from the button hook probe, inlet pitot probe and all static tappings were read by Furness pressure transducers, which supply an analogue DC output with a range of ±1.0V corresponding to pressure ranges of 100 or 500mm H\(_2\)O. The output voltages were
transmitted to the Alpha 03 card via an eight channel Alpha-A analogue input card and then digitised by an analogue to digital converter. Similarly, the signal from the thermocouple in the inlet plenum was transmitted to the Alpha 03 card via an Alpha-K card.

Acquisition of pressure data and traversing of the button hook probe were performed by a single program. This software monitored the inlet Mach number, controlled the motorised traverse and prompted the user to move the manual traverse. Pressure data was recorded in a number of half-second blocks, this number being varied according to the flow conditions that were encountered to obtain a steady average, the number of blocks having been determined by preliminary exploratory tests. Prior to recording data, the program waited for a prescribed time (the "settling time") to allow pressure fluctuations to travel through the probe and connecting tubing to the transducers and allow the reading to settle. The length of all tubing was kept to a minimum to minimise the settling time, and a period of six seconds was found to be sufficient.

This software was adapted for the hot wire and LDA traverses. In both cases, data acquisition was performed by a separate PC and software. For the hot wire traverse, the manual traverse was used and the software was only used to monitor the inlet Mach number. Data acquisition was performed by the StreamWare software supplied with the hot wire system. For LDA measurements, the traversing software was adapted according to the requirements of each traverse, with unstructured measurement grids used to optimise the measurements according to the optical access restrictions and the nature of the flow in each traverse plane. The operation of the LDA system is described below.

3.3.2 Operation of the LDA system
Before taking LDA measurements, it was necessary to locate the measurement volume at a datum point. For this purpose the location plate (described in section 3.1.6.1) had lines scribed to indicate the locations of the axial traverse planes and the flame tube centre line. It was found that when the violet measurement volume was located on the surface of the plate and moved across one of these scribed lines, the observed colour of the reflected light changed as it crossed the line. This effect was clearly visible, the colour of the light changing from violet to white. The colour change was a reliable indication that the measurement volume was located on the scribed line and was thus used to set the axial
and radial datum locations. Using this method the datum location of the measurement volume could be set to the nearest 0.02° (as discussed in section 3.1.6) circumferentially, with an axial and radial accuracy equal to the thickness of the scribed lines, estimated to be no greater than 0.25mm. With the datum location set, the locating plate was removed and the traverse proceeded.

The linear and circumferential traverses were controlled by a PC as described in section 3.3.1. At each point the operating condition was assessed and corrected if necessary. A separate PC was used to control the BSAs. LDA data acquisition and analysis were performed by the 'Burstware' software supplied with the LDA hardware.

The BSA operating parameters (photomultiplier signal gain, centre frequency, bandwidth, record interval and record length) were input on the BSA control panel and recorded by the PC. The record interval is the inverse of the frequency resolution, $f_n$, and should be set to approximately match the length of the burst in seconds. In a highly turbulent flow the burst length varies considerably and is difficult to judge. In such cases, and thus in all measurements reported here, the record interval is set to optimise the data rate and data validation rate. The data validation rate is the percentage of detected bursts that are validated by the BSA, and the data rate is the rate, in kHz, at which valid bursts are detected. These values are indicated by an LED display on the front panel of the BSA. High data rates and data validation rates indicate that the signal to noise ratio is high and the record interval is sufficiently short, relative to the burst length, to maintain the accuracy of the processor. Tropea (1988) demonstrated that the accuracy of the processor drops if the burst length is shorter than the record interval, because unnecessary signal noise enters into the frequency estimation. Tropea recommended that the maximum record length (the number of samples, maximum 64) be used at all times, as the increase in record length causes a decrease in frequency resolution and thus an increase in accuracy. Tropea found that the processor's accuracy was greatest when a record length of 64 and record interval shorter than the burst length were used.

At each point in the traverse, the signal to noise ratio (SNR) was optimised by varying the photomultiplier signal gain to maximise the data validation rate. Rates close to 100% were regularly achieved by the BSAs recording the data from the 2D probe, with lower data validation rates (70-80%) for the 1D probe because of the lower light intensity, and thus decreased SNR, of the violet beams. The processor's centre frequency and
bandwidth were chosen to ensure that all velocities at each point were recorded. Throughout the flame tube the flow was highly turbulent and a wide range of velocities were recorded at each point. A histogram of the velocity distribution at a typical point is shown in Fig.3.31. The wide range of velocities necessitated the use of a large bandwidth, typically 16MHz but occasionally increased to 32MHz. These bandwidths correspond to velocity ranges of approximately 60m/s and 120m/s respectively, the exact figures depending on the light wavelengths. A record length of 64 samples, as recommended by Tropea, was always used, resulting in record intervals of 2.667µs and 1.333µs at bandwidths of 16 and 32MHz respectively. These were found to be suitable record intervals - the fastest velocities encountered were in the jets, where the minimum mean transit time was 5µs. As lower velocities result in longer transit times, the record intervals used here were always shorter than the burst length.

The need to maintain the 16MHz bandwidth influenced the orientation of the three beam pairs. The radial velocity component in the jets is significantly larger than the axial and circumferential components. While it would be possible to measure the u and v components directly, by arranging the two beam pairs of the 2D probe so that one lies in the axial plane and the other lies in the radial plane, this is not desirable. A greater bandwidth would be required on the channel measuring the v component, due to the increased span of the v component sample. The 2D probe is rotated by 45°, as shown by Fig.3.5, so that equal parts of the v component are recorded by both channels and the same bandwidth is used on both channels.

Coincidence filtering was always used to ensure that all three BSAs always recorded velocities from the same particles. However for coincidence filtering all three record intervals must be equal. In regions of high turbulence it was occasionally found that a larger bandwidth was required on one channel. This was often caused by an instability in the flow, resulting in a bimodal velocity histogram. Fig.3.32 shows a typical bimodal histogram. The fluctuation in the flow is represented by the two peaks seen here, and the span of the histogram is increased. In extreme cases a large discrepancy exists between the velocities at which the two peaks occur, further increasing the span of the histogram. This requires a larger bandwidth of 32MHz to be used on the channel that detects the component with the greater span. As a result the record interval on that channel is reduced and the bandwidths of the other two BSAs must be increased to match
Instrumentation and experimental procedure

it. The other two BSAs are now operating away from their optimum conditions, often resulting in a decrease in the data rates. Because of other unconnected problems, discussed in Chapter 4 below, a high data rate was required at all times. As a result the operation of the BSAs at high bandwidths was undesirable, and they were always returned to the 16MHz bandwidth as soon as possible.

The validated data rate on the violet channel was always slower than on the other two, because the SNR was always lower on this channel due to the lower light intensity of the violet beams. As a result a greater number of bursts was always recorded on the blue and green channels than on the violet channel. This is because the coincidence filtering process works with the unvalidated data. Therefore all three BSAs record an equal number of unvalidated bursts, while the lower validation rate on the violet channel causes a greater number of validated bursts to be recorded by the other channels. Further coincidence filtering is performed later by the software, as discussed in section 3.3.3.3 below.

3.3.3 Data reduction and analysis

3.3.3.1 Data measured by button hook probe

Measurements of the stagnation and pseudo-static pressures were obtained by the button hook probe. The true dynamic pressure was calculated at each point, using the calibration factor $K$ as defined by eqn.3.5, from:

$$\frac{1}{2} \rho u^2 = \frac{P - P_m}{K}$$  \hspace{1cm} (3.8)

The mean velocity was thus derived from the dynamic pressure. The bulk average velocity at the pre-diffuser inlet plane was calculated from a radial button hook traverse:

$$\bar{u}_{in} = \frac{2}{r_{out} - r_{in}} \int_{r_{in}}^{r_{out}} u r dr$$  \hspace{1cm} (3.9)

The bulk average inlet velocity calculated from eqn.3.9 was used to non-dimensionalise all measured mean velocity data.

To assess the performance of the diffuser system, two performance parameters, the stagnation pressure loss coefficient ($\lambda$) and the static pressure recovery coefficient ($C_p$), were calculated from mass-weighted mean stagnation and static pressures:

$$\lambda_{a-b} = \frac{P_a - P_b}{P_a - P_s}$$  \hspace{1cm} (3.10)
Instrumentation and Experimental Procedure

\[
C_{p_{a-b}} = \frac{\bar{p}_b - \bar{p}_a}{\bar{p}_a - \bar{p}_s}
\]  
(3.11)

where 'a' and 'b' denote upstream and downstream planes respectively. For example, to calculate the performance of the pre-diffuser plane 'a' would be the pre-diffuser inlet plane and plane 'b' would be the pre-diffuser exit plane.

Integral boundary layer parameters can be defined at the pre-diffuser inlet and exit planes. Because the edge of the boundary layer is not clearly defined, the absolute thickness of the boundary layer cannot be determined. The edge velocity (U) is calculated by taking the mean velocity over several points near the apparent edge of the boundary layer, where the velocity appears to be reasonably constant. An approximation for the boundary layer thickness (\( \delta_{99} \)) is used, defined by the point where the local velocity (u) is 99% of the edge velocity.

The reduction in velocity in the boundary layer leads to a reduction in mass flow. The displacement thickness (\( \delta^* \)) is defined as the displacement of the wall that would be necessary in the inviscid case to yield an equal mass flow reduction:

\[
\delta^* = \int_{0}^{\delta_{99}} (1 - \frac{u}{U}) \, dy 
\]  
(3.12)

Similarly, a momentum thickness (\( \theta \)) can be defined as the equivalent wall displacement to obtain an equal loss of momentum to that which occurs in the boundary layer:

\[
\theta = \int_{0}^{\delta_{99}} \frac{u}{U} \left(1 - \frac{u}{U}\right) \, dy
\]  
(3.13)

The shape parameter, H, is a useful indicator of the state of the boundary layer, varying from a value between 1.3 and 1.4 in a turbulent boundary layer to 2.6 in a laminar boundary layer. It is defined as the ratio of the displacement and momentum thicknesses:

\[
H = \frac{\delta^*}{\theta}
\]  
(3.14)

3.3.3.2 Data measured by hot wire anemometry

The only flow data measured by the hot wire that is used in this project is the turbulence intensity. As discussed in section 3.2.2, the velocity \( U_c \) measured by the hot wire anemometer contains components of the three orthogonal velocities in unequal proportions, as defined by equation 3.7. The turbulence intensity is calculated by the data acquisition and reduction software and is defined as:

\[
Tu = \frac{\sqrt{u'^2 + 0.85v'^2 + 0.03w'^2}}{U_c}
\]  
(3.17)
Instrumentation and Experimental Procedure

Because it was not possible to determine the location of the hot wire inside the passage by placing it against a wall, as with the button hook probe, its location was determined by comparing the mean velocity profiles measured by the hot wire and button hook probes.

3.3.3.3 Data measured by Laser Doppler Anemometry

The reduction of the LDA data was performed by the 'Burstware' data acquisition and reduction package. This software package enables the user to specify the type of processing to be performed, plus specific algorithms and weighting factors. The reduction procedure used in this project is discussed in this section.

The software first reads the experimental data. The Doppler frequency, arrival time and transit time of each burst recorded by each BSA is stored. A separate file is used for data recorded by each BSA at each traverse point. The software converts the Doppler frequencies to velocities by using the calibration factor defined in equation 3.2. The digital binary data is converted to floating point data and stored in new data files.

Coincidence filtering of the data is then performed. The three velocity components are deemed to originate from the same particle (i.e. deemed to be coincident) only if the arrival times recorded by each channel are equal, within a specified tolerance. This tolerance, the "coincidence window", is input by the user. A coincidence window of 0.1 ms was selected. This coincidence window was found to cause the rejection of all surplus, non-coincident bursts on the green and blue channels, which recorded a greater number of bursts than the violet channel, without rejecting a significant number of coincident bursts from all three channels. The transformation matrix, calculated as described in section 3.1.6.2, is then applied to transform the coincident data to the orthogonal (u, v and w) co-ordinate system. The transformed, coincident data is then stored in a 'coincident' data file.

The coincident data is then processed to obtain the mean u, v and w velocity components, the root mean square of the fluctuating velocity components, and the uv, vw and uw Reynolds shear stress components at each traverse point. At this stage, the residence time weighting (described in detail in section 3.4.2.2) is applied to compensate for statistical bias. For example, the residence time weighted axial velocity component is calculated by:

\[
\bar{u} = \frac{\sum u_i t_i}{\sum t_i}
\]  

(3.18)
where \( u_i \) and \( t_i \) are the axial velocity component and residence time of the \( i \)th particle in the sample.

The instantaneous fluctuating velocity component is calculated by subtracting the local mean velocity from the instantaneous velocity, for example the instantaneous axial fluctuating velocity is:

\[
u' = u_i - u \tag{3.19}\]

and this data is used to calculate the six Reynolds shear stresses. For example, the \( uv \) shear stress is:

\[
u v = \frac{1}{N} \sum u' v' \tag{3.20}\]

where \( N \) is the number of bursts in the sample.

Unless otherwise stated, all mean velocity data presented in Chapter 5 is normalised by the mean velocity at the pre-diffuser inlet plane. For example, the normalised mean axial velocity component is:

\[
u_{\text{norm}} = \frac{u}{u_{\text{in}}} \tag{3.21}\]

Similarly, unless otherwise stated, all Reynolds stress data presented in Chapter 5 is normalised by the square of the mean velocity at the pre-diffuser inlet plane. For example, the normalised \( uv \) shear stress component is:

\[
u v_{\text{norm}} = \frac{\nu v}{u_{\text{in}}^2} \tag{3.22}\]

The normalisation of the data allows the use of this data set for the validation of CFD predictions that are calculated with inlet conditions that differ from that used here.

The vorticity vector can be calculated from the mean velocity data. Only the axial component of the vorticity vector has been used in this project:

\[
u = \frac{\partial U_z}{\partial y} - \frac{\partial U_y}{\partial z} \tag{3.23}\]

where \( y \) and \( z \) are the Cartesian co-ordinates defined by:

\[
y = r \cos \theta \]
\[
z = r \sin \theta \]

and \( U_y \) and \( U_z \) are the velocity components in the \( y \) and \( z \) directions.

Additional turbulence quantities are derived from the measured Reynolds stress data. The turbulence intensity is defined in terms of the local total mean velocity:

\[
u_t = 100 \times \frac{\sqrt{u'^2 + v'^2 + w'^2}}{\sqrt{u^2 + v^2 + w^2}} \tag{3.24}\]
Instrumentation and Experimental Procedure

The turbulent kinetic energy is also calculated and normalised by the square of the mean velocity at the pre-diffuser inlet plane:

\[ k = \frac{1}{2} \frac{\left( u'^2 + v'^2 + w'^2 \right)}{\bar{u}^2_{in}} \]  

(3.25)

The anisotropy parameters are defined as the ratios of the Reynolds normal stress components to the turbulent kinetic energy. For example the axial anisotropy parameter is:

\[ \frac{u'u}{k} = \frac{\bar{u}^2}{0.5\left( \bar{u}^2 + \bar{v}^2 + \bar{w}^2 \right)} \]  

(3.26)

Because the axial, radial and circumferential normal stress components are equal if the normal stresses are isotropic, the anisotropy parameters are indicators of anisotropic turbulence. If the normal stresses are not isotropic, the anisotropy parameters deviate from their isotropic value of 2/3.

Four integrated mean quantities are used to describe the mean flow through the flame tube ports. As discussed later, at the port exit measurement planes (defined in Chapter 4) it is impossible to distinguish between jet fluid and fluid entrained by the jet. It is therefore necessary to define the required mean quantities (the discharge coefficient, the jet pitch angle, and the axial and radial components of momentum) in terms of a cut-off velocity. A cut-off velocity is defined, and data from points where the cut-off velocity exceeds the mean velocity component normal to the measurement plane (the v component) is excluded from the calculation of the integrated mean quantities. Comparisons between the different flame tube ports can then be made by plotting the mean quantities against the cut-off velocity. Hence the discharge coefficients of the ports are calculated from:

\[ C_D = \frac{1}{A_{port}} \sum_{v_{cut}}^{v_{max}} \bar{A} \]  

(3.27)

where \( A_{port} \) is the geometric area of the port and \( \bar{A} \) is the area of each cell of the measurement grid. The mean jet pitch angle, \( \alpha \), is calculated from an area weighted average of the pitch angle in every grid cell where the v component exceeds the cut-off velocity:

\[ \alpha = \frac{\sum_{v_{cut}}^{v_{max}} \tan^{-1}(\frac{v}{u}) \bar{A}}{\sum_{v_{cut}}^{v_{max}} \bar{A}} \]  

(3.28)
The turbulent kinetic energy is also calculated and normalised by the square of the mean velocity at the pre-diffuser inlet plane:

\[ k = \frac{1}{2} \frac{\left(u^2 + v^2 + w^2\right)}{u_m^2} \]  

(3.25)

The anisotropy parameters are defined as the ratios of the Reynolds normal stress components to the turbulent kinetic energy. For example, the axial anisotropy parameter is:

\[ \frac{\mu_{11}}{k} = \frac{\tau_{11}}{0.5(u^2 + v^2 + w^2)} \]  

(3.26)

Because the axial, radial, and circumferential normal stress components are equal if the normal stresses are isotropic, the anisotropy parameters are indicators of anisotropic turbulence. If the normal stresses are not isotropic, the anisotropy parameters deviate from their isotropic value of 2/3.

Four integrated mean quantities are used to describe the mean flow through the flame tube ports. As discussed later, at the port exit measurement planes (defined in Chapter 4) it is impossible to distinguish between jet fluid and fluid entrained by the jet. It is therefore necessary to define the required mean quantities (the discharge coefficient, the jet pitch angle, and the axial and radial components of momentum) in terms of a cut-off velocity. A cut-off velocity is defined, and data from points where the cut-off velocity exceeds the mean velocity component normal to the measurement plane (the v component) is excluded from the calculation of the integrated mean quantities. Comparisons between the different flame tube ports can then be made by plotting the mean quantities against the cut-off velocity. Hence the discharge coefficients of the ports are calculated from:

\[ C_D = \frac{1}{A_{\text{port}}} \sum_{V_{\text{cut}}} \frac{V_{\text{max}}}{dA} \]  

(3.27)

where \( A_{\text{port}} \) is the geometric area of the port and \( dA \) is the area of each cell of the measurement grid. The mean jet pitch angle, \( \alpha \), is calculated from an area weighted average of the pitch angle in every grid cell where the v component exceeds the cut-off velocity:

\[ \alpha = \frac{\sum_{V_{\text{cut}}} \frac{\tan^{-1}(\gamma) dA}{V_{\text{cut}}}}{\sum_{V_{\text{cut}}} dA} \]  

(3.28)

2 Normally \( C_D \) is calculated from \( m = C_D \rho A_u \), however this cannot be done here. This formula calculates an effective area by “counting squares” where the velocity exceeds a chosen cut-off value, and estimates \( C_D \) by dividing this effective area by the geometric area of the port. This is valid provided that the velocity profile across the port is uniform, although experimental measurements suggest this may not be the case. However this is intended to produce comparisons between ports and not absolute \( C_D \) values. The velocity profiles of all ports measured are similar (see Chapter 5) so this method is believed to provide valid comparisons.
Axial and radial components of the momentum flux through the chute exit planes are also calculated:

\[ G_x = \sum_{\text{out}} u_i v_i \, dA \]  
\[ G_r = \sum_{\text{out}} u_i^2 \, dA \]  

(3.29)  
(3.30)

To avoid confusion with the \( uv \) shear stress, \( u_i \) and \( v_i \) are used here to denote the mean axial and radial velocity components in each measurement grid cell.

3.4 Estimate of experimental errors

3.4.1 Pressure measurements

Because the button hook probe was used for only a small proportion of this project only a limited analysis, based on that presented by Shedden (1993), is presented here.

The most significant errors in the button hook measurements are derived from three sources: the pressure transducers, the flow field turbulence and variations in the button hook calibration. In this project the pressure transducers were always zeroed to within \( \pm 0.3 \text{mV} \) of before testing. The calibration of the 100mm \( H_2O \) pressure transducer used for these measurements was accurate to \( 1 \text{mV} \), or 0.1% of the full scale deflection, thus a total uncertainty of \( \pm 0.13\% \) of the full scale deflection, or 0.13mm \( H_2O \), exists.

Ower and Pankhurst (1975) suggest that the true dynamic pressure measured by a pitot probe in high turbulence is

\[ q_{\text{meas}} = \frac{1}{2} \rho u^2 (1 + \alpha(Tu)^2) \]

where \( \alpha \) is between 1 and 5 and depends on the scale of the turbulence. Carrotte et al (1992) found turbulence intensities of 35% and attributed mass flow errors of up to 17% and 11% in the outer and inner feed annuli of a dump diffuser system to the effects of turbulence on the pressure measurements.

A button hook probe calibration of 1.39 was used for all measurements. As discussed in section 3.2.1, this calibration varied by \( \pm 1.4\% \) over the range of dynamic pressures that was encountered. Dynamic pressures between 90 and 100mm \( H_2O \) were measured in the core flow at the pre-diffuser inlet plane, and dynamic pressures between 50 and 60mm \( H_2O \) were measured at the pre-diffuser exit plane. The combined errors due to the calibrations of the pressure transducer and the button hook probe are therefore less than 2% of the measured dynamic pressure and less than 1% of the derived velocity.
3.4.2 LDA measurements

The errors associated with the use of LDA can be separated into statistical errors, caused by the use of a finite number of samples of the signal, and systematic errors associated with the instrumentation (Bailey, 1997).

3.4.2.1 Statistical errors

The mean velocity components at each point are calculated by averaging a number of discrete samples of the instantaneous velocities at that point. If it is assumed that these velocities are normally distributed, then conventional statistical methods can be used to estimate the statistical errors (Kreyszig, 1988). If \( N \) samples are used to sample a flow with a true mean of \( \mu \) and variance \( \sigma^2 \) then the calculated mean value can be said to lie within a confidence interval \( \varepsilon \) of the true mean, \( \mu \), given by

\[
\varepsilon = \frac{z \sigma}{\sqrt{N}}
\]

where \( z \) is a measure of the confidence that the calculated mean lies within \( \pm \varepsilon \) of the true mean. The value of \( z \) can be obtained for any confidence level (Kreyszig, 1988). For example, for a 99% confidence level, a value of \( z=2.576 \) is obtained. The variance \( \sigma^2 \) of the sample is the mean square of the fluctuating velocity component, so \( \sigma \) is the rms velocity component.

As will be discussed in Chapter 4, typically 10000 coincident samples were collected at each point, with typical data rates between 0.5 and 1kHz. However the samples must be statistically independent. If multiple samples of a single eddy are taken then they will make no more contribution to the accuracy of the average than a single sample would. For statistically independent samples to be obtained the velocity samples must be taken at time intervals longer than the time a single eddy takes to cross the measurement volume, however this is difficult to achieve. In practice the turbulent flow contains eddies with a wide range of length and time scales. Hence prediction of the length scales is difficult and estimates must be made of the time scales of the largest eddies, by relating known flow velocities to the characteristic lengths of certain flow field features. For example, if it is assumed that the largest eddies in the jet flows have length scales of the order of the port diameters and velocities of 40m/s, (based on the velocities observed in the jets' cores), then the time scale is approximately 0.00035 seconds, while a time scale based on the diameter of the swirler and an estimate of 10m/s for the velocity of the fluid exiting the swirler could be as high as 0.004 seconds. With a data rate of 1kHz
samples are taken every 0.001 seconds, so if the time scale of the largest eddies is 0.00035 seconds then all 10000 samples are statistically independent. However if the largest time scale is 0.004 seconds, then only 2500 samples can be assumed to be statistically independent.

At most points in the flame tube the rms velocities were found to lie in the range 3 - 9m/s. If 2500 samples are assumed to be statistically independent, and the median rms velocity of 6m/s is used, then

$$\varepsilon = \frac{2.576 \times 6}{\sqrt{2500}} = 0.31\text{m/s}$$

so there is 99% confidence that the measured mean velocities lie between ±0.31m/s of the true mean. This may be the worst case, as the largest eddies may well be smaller than those assumed here and it may be that the number of statistically independent samples taken at many points in the flow field was 10000. In this case the confidence interval is halved.

The statistical uncertainty of the measured normal stresses is calculated from the chi-squared distribution (Kreyszig, 1988). The variance of the velocity distribution, $\sigma^2$, represents the normal stress component, $u_t^2$, whose upper and lower confidence limits can be determined by statistical methods. A confidence level, $\gamma$, is assumed and the solutions of the equations

$$F(c_1) = \frac{1}{2}(1 - \gamma) \quad \text{and} \quad F(c_2) = \frac{1}{2}(1 + \gamma)$$

are obtained from tables of the chi-squared distribution. Having obtained the parameters $c_1$ and $c_2$ for the chosen confidence level, the confidence limits $k_1$ and $k_2$ are calculated:

$$k_1 = \frac{(N - 1)\sigma^2}{c_1} \quad \text{and} \quad k_2 = \frac{(N - 1)\sigma^2}{c_2}$$

The normal stress lies within the limits $k_1$ and $k_2$ with a confidence of $\gamma$. For a confidence level of 99%, with 2500 samples and an rms velocity component of 6m/s ($\sigma^2 = 36$), the normal stress lies between 33.5 and 38.8 m$^2$/s$^2$ (±7%) and the rms component lies between 5.9 and 6.1m/s (±3.5%).

### 3.4.2.2 Systematic errors

Three sources of systematic error are present in an LDA system. These are the errors associated with the use of particles to represent the air flow (known as "statistical bias", a term that, while misleading, is in widespread use and so will be used here), errors
Instrumentation and Experimental Procedure

associated with the alignment of the laser beams, and errors associated with the finite resolution of the processors.

The term "statistical bias" refers to any biasing of the calculated mean flow properties that is caused by the use of seed particles to characterise the flow. The first and most significant of these biasing effects is termed 'velocity bias'.

For measurements of the flow of air, as in this case, it is difficult to generate sufficiently large seeding concentrations for a continuous measurement to be made. Measurements made with only one particle passing through the measurement volume at a time are known as "individual realisation" measurements and the calculation of mean flow properties from such measurements is dependent on the rate of arrival of the particles in the measurement volume. In a turbulent flow any sample of the flow at a single point will contain measurements taken from particles with a range of velocities. McLaughlin and Tiederman (1973) pointed out that, in a uniformly seeded flow, particles with high velocities arrive at the measurement volume at a higher rate than those with low velocities. Any sample of a turbulent flow will therefore contain more measurements of particles with higher velocities than particles with lower velocities and an ensemble average of these velocities will be biased towards the higher velocities. This bias is known as "velocity bias".

Velocity bias has been the subject of research by a number of investigators and, while most (e.g. Stevenson et al, 1982 and Johnson et al, 1982) have agreed that velocity bias exists there has been considerable disagreement regarding its magnitude. Consensus was obtained by a special panel (Edwards, 1987) that reported what was then known about velocity bias and made recommendations for its elimination. A measure of the magnitude of the data density, the product of the validation rate $N_p$, that is the rate at which the processor measures validated bursts, and the Taylor microscale $T_\alpha$ (a time scale related to the dissipation of turbulence, described by Edwards [1987] as the time that the flow needs to change by one standard deviation), was defined. Recommendations were made for the elimination of velocity bias at different data densities.

The data rates achieved in this project were not high enough to obtain a time scale based on an autocorrelation function. The time scale that was used to estimate the statistical errors (section 3.3.2.1) was used to estimate the data density. This time scale was 0.004 seconds, and with a data rate of 1kHz the data density is 4. Edwards suggests a
definition of an intermediate data density to be \(0.05 < N_2 T_x < 5\), so the data density in this case falls within this definition.

Edwards reviews a number of processing schemes for the elimination of velocity bias. These can be divided into two groups. The processing schemes in the first group selectively sample the data to obtain either a continuous signal or a time series with statistics identical to those of the flow, which can then be averaged to calculate bias-free mean values. The "controlled processor" divides time into short, equally spaced intervals. Only the first particle measured during each time interval is stored, the remaining data being rejected, and a time series with statistics identical to those of the flow is constructed. Similarly, the "saturable detector" works by disabling the burst detector for a fixed period after each burst is recorded, thus recording data at regular intervals. The "sample and hold" scheme creates a continuous analogue signal by holding the last measurement value until a new measurement is recorded. These three processing schemes are recommended only for high data densities and are therefore considered to be unsuitable for use in this project. After initial tests of the LDA system in this test facility (see Chapter 4) it was decided that only 10,000 samples would be recorded at each traverse point, and that maintaining a high data rate was desirable. Any processing scheme that involves the rejection of data, such as the controlled processor, is therefore undesirable because it reduces the size of the sample, thus increasing statistical errors. The use of a saturable detector would reduce the data rate, and is therefore also undesirable.

The second group of processing schemes uses a weighting factor to compensate for the velocity bias. The McLaughlin and Tiedeman correction uses the inverse of the particle's speed as a weighting factor. This correction has been shown to be valid only for low turbulence intensities and is therefore unsuitable for use in this project. For intermediate data densities Edwards recommends the use of the residence time weighting scheme to correct the velocity bias. The particle transit times (also known as residence time) are used to calculate a weighted mean velocity using the equation

\[
\overline{U} = \frac{\sum U_i t_i}{\sum t_i}
\]

Buchhave (1979) found that the residence time weighting produces correct results in flows that are uniformly seeded. A third weighing scheme is the inter-arrival time weighting. This scheme uses the time between bursts, the "inter-arrival" time, as a
weighting factor. In this project the residence time weighting was applied to all data in the Burstware data processing routine.

Based on the findings of Buchhave and the recommendations of Edwards the residence time weighting scheme has been adopted. Nevertheless, turbulence intensities in the flame tube are very high and the velocity bias can be expected to be significant, and it is difficult to prove that the correction is working. However, the magnitude of the error can be assessed by comparing the statistical values obtained by evaluating the data both with and without the residence time weighting. For example, Fig.3.33 shows both the weighted and unweighted profiles of the mean v component on the centreline of one of the jets, plus the turbulence intensity. The effect of the residence time weighting clearly increases as the turbulence intensity increases. Hoesel and Rodi (1977) verified the residence time weighting in a uniformly seeded round jet, and Fuchs et al (1992) found that the residence time weighting reduced the velocity bias by 80% for turbulence intensities of around 30%. The residence time weighting scheme depends on the provision of spatially uniform seeding to the flow. This condition is believed to be met by the atomiser and by the mixing of the seed and the flow to give an even distribution, and the residence time weighting is therefore believed to be valid. The findings of Fuchs et al may be used to estimate the error that is present in residence time weighted velocity samples, and errors due to the remaining velocity bias are presented in Table 3.1 for the turbulence intensities observed in Fig.3.33.

<table>
<thead>
<tr>
<th>Turbulence intensity (%)</th>
<th>Res. time weighting (%)</th>
<th>Remaining bias (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>1</td>
<td>0.25</td>
</tr>
<tr>
<td>29</td>
<td>6.5</td>
<td>1.63</td>
</tr>
<tr>
<td>62</td>
<td>23.5</td>
<td>5.9</td>
</tr>
<tr>
<td>119</td>
<td>59.9</td>
<td>15</td>
</tr>
</tbody>
</table>

Table 3.1 Estimated remaining bias after application of residence time weighting

Therefore there is a significant bias remaining in velocity samples with very high turbulence intensities after the residence time weighting has been applied. However these very high turbulence intensities are observed where the mean velocity is low. In these cases the comparatively very large turbulent fluctuations are more significant features of
the flow than the mean velocity. At these points the remaining bias translates into a small error in the mean velocity, and is of little concern. Where the mean velocity is a more significant feature of the flow, for example in the jets, the turbulence intensity is lower and the remaining bias is small.

Durao and Whitelaw (1979) showed that the amplitude of the output signal of a photomultiplier decreases as the particle velocity increases, leading to a potential bias towards lower velocities. Edwards also states that the processor's filter settings can seriously affect the sample. No recommendations were made except that the processor settings should be set such that all velocities present in the flow are measurable. Care was taken in this project to achieve this and the centre frequency and bandwidth of each BSA were set at each point to enable all velocities in the sample at that point to be sampled. The effect of the photomultiplier bias is not considered to be significant when compared with the velocity bias that is expected in the regions of very high turbulence in the flame tube.

Angle (also known as fringe) bias can occur if a particle approaches the measurement volume at a very shallow angle to the fringes. In this case the particle does not cross enough fringes for a valid burst to be measured. This error is eliminated by the use of a Bragg cell whose frequency shift is sufficient to enable all velocities in the flow to be recorded. The 40MHz frequency shift that was used in this project was sufficient to eliminate angle bias, and the LDA system is capable of detecting and measuring particles that approach the measurement volume at any angle.

Durst et al (1993) showed that an error may occur if there is a velocity gradient across the measurement volume. However the measurement volume is very small and Durst showed that the error is significant only in regions where the velocity gradient is very high, such as the laminar sublayer of a turbulent boundary layer. The only regions where velocity gradients were observed that are considered to be large enough to bias the data are the shear layers on the lee sides of the jets. At these locations a velocity rise of 1.5m/s was observed across the measurement volume, in shear layers where the velocity rises from 10m/s to 40m/s. If the maximum error due to the velocity gradient is ±0.75m/s, then the error varies from 2% to 7.5% of the mean velocity, according to the location in the shear layer. This error is small and only occurs at a limited number of data points, and is therefore of little concern.
3.4.2.3 Alignment errors

The use of the new LDA alignment technique in this project has substantially improved the alignment accuracy. The translation stages that move the pinhole are thought to have been accurately positioned on the rig. The only significant source of error is thought to lie in the measurement of the beam vectors. The position of each point on the beam is measured with an accuracy of 10\(\mu\)m. A typical transformation matrix used in this project was:

\[
\begin{bmatrix}
0.2893 & 0.1817 & 0.7238 \\
0.9401 & -0.3204 & -0.3738 \\
0.9022 & 1.5467 & -1.5857
\end{bmatrix}
\]

To assess the error due to the 10\(\mu\)m uncertainty in the beam measurements, 10\(\mu\)m variations were introduced into the points that were used to calculate the above matrix, resulting in the following matrix:

\[
\begin{bmatrix}
0.2835 & 0.1916 & 0.7266 \\
0.9398 & -0.3201 & -0.3731 \\
0.9439 & 1.5361 & -1.5838
\end{bmatrix}
\]

The effect of these errors will vary according to the direction of the flow. For instance, if a mean velocity of 1m/s is recorded by each channel then the resolved \(u\) component calculated using the first matrix is 1.1948m/s, while a \(u\) component of 1.2017 will be calculated by using the second matrix - an error of 0.6%. This error is clearly small when compared with the potentially much larger errors that have been discussed above.

3.4.2.4 Processor resolution

Carrotte and Britchford (1994) showed that the accuracy of the resolution of each burst depends on the calibration factor of each beam pair and the record interval used by the processor. They showed that the processor resolution is

\[
\text{resolution} = \frac{c}{16RI}
\]

The calibration factor is a function of wavelength. The calibration factors used in this project were 4.028, 3.820 and 3.730 m/s per MHz for the green, blue and violet channels respectively. With a record interval of 2.667\(\mu\)s used at most points, the processor resolutions were \(\pm 0.095\), 0.09 and 0.085m/s for the green, blue and violet channels respectively. However Carrotte and Britchford found that the processor resolution is also a function of the signal to noise ratio (SNR). The SNR is lower on the blue and violet...
channels than on the green channel because of the lower intensities of those beams.
Carotte and Britchford suggested processor resolutions for the blue and violet channels,
however they used different beam expanders and their calibration factors were different.
By scaling their processor resolutions to the calibration factors used in this project,
processor resolutions of $\pm 0.11$ and $0.15\text{m/s}$ have been obtained for the blue and violet
channels.

3.4.2.5 Overall estimate of LDA errors
Due to the complex nature of the flame tube flow field there is a wide variation in mean
velocities and turbulence intensities and no estimate of LDA errors can be made that
applies to the entire flow field. The very high turbulence intensities (>100%) were found
at points with very low mean velocities (typically < 5m/s) so the 15% velocity bias may
be expected to cause an error no greater than $+0.75\text{ms}$. With an estimated statistical error
of $\pm 0.3\text{m/s}$ and processor resolution of $\pm 0.15\text{m/s}$, the mean velocities are thought to be
accurate to within $-0.5\text{m/s}$ and $+1.25\text{m/s}$. The magnitude of the velocity bias effect, after
residence time weighting, on the rms velocities is not known. However, with a statistical
error (pessimistically based on the use of only 2500 samples) of 3.5% and an alignment
error of approximately 0.6% applying to each individual velocity sample, it is felt that an
error of $\pm 5\%$ is reasonable.
4. The Experimental Programme

4.1 Test Facility Commissioning Tests

Because of the growth and possible separation of the sidewall boundary layers due to the adverse pressure gradients that exist throughout the flow field, the use of a sector rig is potentially problematic. In order to obtain data of the highest quality, an extensive test facility commissioning programme was undertaken that had three aims: To assess the behaviour of the flow in the pre-diffuser and feed annuli; to make, and assess the effects of, any modifications that were necessary to improve the behaviour of the flow; and to adjust the annulus bleed metering plates to establish the specified bleed flow rates.

4.1.1 Pre-diffuser

With no OGVs upstream of the pre-diffuser it is impossible to reproduce an engine representative inlet condition. It is important that the boundary layers in the pre-diffuser are turbulent, to reduce the tendency to separate due to the adverse pressure gradient. The adverse pressure gradient must also promote the growth of the boundary layers on the sidewalls, causing the migration of fluid towards the centre of the test facility, and it may cause their separation. While the growth of the boundary layers on the sidewalls is inevitable, and the resulting migration of fluid towards the centre of the passage must be accepted, separation of the sidewall boundary layers must be avoided. The pre-diffuser commissioning tests that are described here were conducted to establish the condition of the flow through the pre-diffuser, and an acceptable flow condition was obtained.

Radial button hook probe traverses on the rig centre line were used to compare the pre-diffuser inlet velocity profiles with and without the inlet boundary layer trip installed (Fig.4.1). As is to be expected, both boundary layers are thicker with the trip installed, with the greatest increase in thickness exhibited by the boundary layer adjacent to the outer casing. In Figs. 4.2 and 4.3 the state of the inner and outer boundary layer velocity profiles can be ascertained by a comparison with theoretical laminar (Blasius) and turbulent (1/7 Power Law) boundary layers.

The Blasius profile is a theoretical laminar boundary layer profile and an approximate form of the solution used here is:

\[
\frac{u}{U} = \frac{3}{2} \left( \frac{y}{\delta} \right) - \frac{1}{2} \left( \frac{y}{\delta} \right)^3
\]
whereas for a turbulent boundary layer an approximate profile is the $1/7$ Power Law profile:

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{1/7}$$

In both the non-tripped and tripped cases the inner boundary layers (Fig.4.2) are turbulent in nature since they approximate the $1/7$ power law profile. With a clean inlet the outer boundary layer profile (Fig.4.3) appears to be in late transition, and becomes turbulent when the trip is added.

The integral boundary layer parameters $\delta^*$, $\theta$ and $H$ (equations 3.12 - 3.14) have been calculated for both boundary layers. However, the thickness of the button hook probe caused the poor resolution of the inner boundary layer close to the wall, as shown by Fig.4.2. Comparison with the $1/7$ Power Law profile shows that, by extrapolating the velocity to zero at the wall, the velocities between the wall and the first data point are substantially underestimated. This leads to an overestimation of the displacement thickness and underestimation of the momentum thickness, resulting in the overestimation of the shape parameter. An extra point was added, as shown by Fig.4.4, to give each inner boundary layer profile a better agreement with the $1/7$ Power Law profile near to the wall, and the displacement thickness, momentum thickness and shape parameters were recalculated from these adjusted profiles. The shapes of the outer boundary layer profiles (Fig.4.3) suggest that no such adjustment is necessary for the outer boundary layers. These boundary layer profiles are summarised in the table 4.1 below:

<table>
<thead>
<tr>
<th>Boundary Layer</th>
<th>$\delta^*$ (mm)</th>
<th>$\theta$ (mm)</th>
<th>$H$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner, no trip</td>
<td>0.53</td>
<td>0.28</td>
<td>1.85</td>
</tr>
<tr>
<td>Inner, no trip (adjusted data)</td>
<td>0.46</td>
<td>0.32</td>
<td>1.42</td>
</tr>
<tr>
<td>Inner, tripped</td>
<td>0.62</td>
<td>0.36</td>
<td>1.73</td>
</tr>
<tr>
<td>Inner, tripped (adjusted data)</td>
<td>0.52</td>
<td>0.40</td>
<td>1.31</td>
</tr>
<tr>
<td>Outer, no trip</td>
<td>0.63</td>
<td>0.40</td>
<td>1.57</td>
</tr>
<tr>
<td>Outer, tripped</td>
<td>0.87</td>
<td>0.61</td>
<td>1.44</td>
</tr>
</tbody>
</table>

Table 4.1 Pre-diffuser inlet boundary layer quantities
The Experimental Programme

With no trip installed, radial button hook traverses were conducted at four locations (±13.5° and ±20.5°) on the pre-diffuser exit plane, the results of which are shown in Fig.4.5. For the shaft of the probe to be clear of the pre-diffuser, it was necessary to traverse the probe head 2mm downstream of the pre-diffuser exit plane. As a result the probe location, with respect to the pre-diffuser, could not be defined precisely. Hence the probe location in Fig.4.5 is described as "approximate".

Most of the velocity profiles appear to be smooth. However at -20.5°, 12mm from the sidewall, discontinuities are present in the profile. This indicates that the sampling time was not sufficient to obtain a good mean value - in the terms used to define the sample size for LDA data (see Chapter 3), an insufficient number of statistically independent samples was obtained at these points. This is an indication of instability in the flow, with a large length scale structure moving past the probe, caused by the separation of the boundary layer in the outer corner region. Negative velocities at both ends of the profile at -20.5°, and at the outer end of the profile at +20.5°, also indicate that the boundary layers on the sidewalls are separating. Wool tuft flow visualisation tests showed that the boundary layer on the sidewall on the negative θ side was separated, and the boundary layer on the other sidewall was unstable.

The wool tuft flow visualisation tests were repeated with the trip inserted. No separation was evident anywhere in the pre-diffuser exit plane. However the growth of the boundary layers on the sidewalls must be enhanced by the adverse pressure gradient, and the effect on the flow at the centre of the passage must be quantified. A button hook probe traverse was conducted on the centre line at the pre-diffuser exit plane (Fig.4.6). Calculations of the volume flow rate at the pre-diffuser inlet and exit planes, based on the radial velocity profiles on the centre line, revealed a 5.3% increase in mass flow at the pre-diffuser exit plane. This must be attributed to the migration of fluid towards the centre of the passage due to the growth of the boundary layers on the sidewalls.

As with the boundary layers at the inlet, the boundary layers at the pre-diffuser exit plane were compared with the 1/7 Power Law profile (Fig.4.7) and were quantified by calculating the displacement thickness, momentum thickness and shape parameter, as summarised by table 4.2:
The Experimental Programme

<table>
<thead>
<tr>
<th>Boundary Layer</th>
<th>δ* (mm)</th>
<th>θ (mm)</th>
<th>H</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner</td>
<td>1.81</td>
<td>1.15</td>
<td>1.57</td>
</tr>
<tr>
<td>Outer</td>
<td>2.61</td>
<td>1.43</td>
<td>1.82</td>
</tr>
</tbody>
</table>

Table 4.2 Pre-diffuser exit boundary layer quantities

Comparison of the boundary layer profiles of Fig. 4.7 and the 1/7 Power Law profile, and the shape parameters in Table 4.2, suggests that the boundary layers are in transition, however this is not the case - the adverse pressure gradient has altered the shape of the boundary layers by pushing fluid away from the walls, resulting in increased shape parameters, and the boundary layers are fully turbulent.

If it is assumed that the displacement thickness of both sidewall boundary layers increases from 0.5mm (the displacement thickness of the inner boundary layer at the inlet) to 2.6mm in the pre-diffuser, then the flow area at the pre-diffuser exit is effectively reduced from 11088mm$^2$ to 10917mm$^2$, a reduction of 1.5%. Thus the minimum possible increase in mass flow at the centre is 1.5%, however the true increase must be substantially greater due to the effects of the corners.

Carrotte and Wray (1991) used a pre-diffuser with the same area ratio (1.45) and the same inlet Mach number (0.14) in a four sector rig. They reported an increase in mass flow at the pre-diffuser exit of 3%. This equates to a reduction in flow area of 274mm$^2$. A similar reduction in flow area in this three sector facility would increase the mass flow measured at the pre-diffuser exit plane by 4.1%. The 5.3% seen here is thus not unreasonable. This small migration towards the centre of the sector is not considered to be sufficiently significant to compromise the objectives of the programme.

4.1.2 Outer feed annulus

As with the pre-diffuser, adverse pressure gradients exist in the feed annuli due to the diffusion that occurs when fluid enters the flame tube through effusion cooling holes and mixing ports. This must cause the boundary layers on the sidewalls to grow and may cause them to separate. Wool tuft flow visualisation was used to assess the condition of the flow in the outer feed annulus, with particular attention paid to the boundary layers on the sidewalls and the flow in the sectors adjacent to the sidewalls. The results of this flow visualisation are shown in Fig. 4.8. On both sides the boundary layer was attached.
approaching the primary ports but separation occurred at the plane of the primary ports, apparently caused by the sudden diffusion due to the removal of fluid from the annulus by the ports. On the positive \( \theta \) side there was a well-defined reversal of the flow, the wool tufts indicating that the flow moved along the sidewall in the upstream direction from as far downstream as the bleed offtakes, feeding the primary and secondary ports adjacent to the sidewall. The flow from the central region appeared to spread out and move towards the sidewall to feed the outer ports, the bleed offtakes and the reversed flow. On the negative \( \theta \) side the separation appeared to be smaller, perhaps due to the circumferential displacement of the primary ports. Because of the circumferential displacement of the ports, the "half ports" adjacent to the sidewalls are not equal. The smaller port, on the negative \( \theta \) side, must remove less fluid from the annulus, thus causing less diffusion locally and explaining the smaller separation. Unlike the separated flow on the other side, this separation appears to be unstable.

A number of modifications were investigated to eliminate the separation of the sidewall boundary layers in the outer annulus. Each modification was assessed by visualising the flow with a wool tuft. Early attempts were made at controlling the sidewall boundary layers through the variation of the differential bleed. Relative to the flow through the ports, which causes the diffusion that forces the separation, the outer bleed is insignificant and so could not affect the separation of the boundary layers. Increasing the flow through the outer sectors of the bleed duct merely drew more air from the centre of the annulus and did not affect the sidewall boundary layers.

The separation of the sidewall boundary layers was thought to be caused initially by the diffusion due to the removal of air from the feed annulus by the half primary ports adjacent to the sidewalls. Because the flow through each outer primary port is 1.3% of the total flow through the flame tube, it was felt that blocking these ports would not significantly affect the flow in the flame tube. Being situated adjacent to the sidewalls, these ports were expected to have a negligible effect on the flow in the centre sector. These ports were therefore blocked and flow visualisation results showed that the separation point of each boundary layer had been moved downstream to the plane of the secondary ports. Although this led to a significant improvement, the boundary layer separations, the migration of the central flow towards the outer sectors and the instability of one sidewall boundary layer were all still present. Blockage of the secondary ports
average improvement, the separation on the positive side being reduced in size, with the separation on the opposite side almost eliminated - the instability was still present with the boundary layer separating occasionally.

The blockage of these ports produced promising results and it was felt that the use of splitter plates in the annulus could help. The splitter plates were intended to isolate the outer sector bleed off-takes from the centre sector flow so that the differential bleed could be used to draw air down the sidewalls and the centre sector flow would not migrate towards the outer sector bleed. However it was essential that the boundary layer on the inside face (that is on the centre sector side of the plate) of each splitter plate was attached and stable along the face of the plate, otherwise the sidewall problem would be moved inboard by the splitter plates. A number of different splitter plate configurations were tried. Each plate was made from 1mm thick perspex and followed the profiles of the flame tube and casing. The length of the plate was varied to move the leading edge to different locations near the secondary ports, and the circumferential locations of the plates were varied. The best splitter plate was found to be aligned with the splitter plate in the bleed duct and its leading edge was approximately 7mm downstream of the bottom of the secondary ports. The flow between the splitter plates and the sidewalls was found to be well behaved and the boundary layer on the inside face of each splitter plate was found to be attached and stable.

While a configuration had been found that produced acceptable flow behaviour in the outer annulus, it was felt that the blockage of two secondary ports may result in an unrepresentative flow field in the flame tube, even though the blocked ports were away from the area of interest. With the secondary ports open, the flow between the splitter plates and the sidewalls was separated and unstable. A number of attempts were made to correct this. Different splitter plate configurations and locations were tried. Vortex generators were installed on one sidewall. These delayed the separation of the boundary layer, but separation was still observed in the vicinity of the secondary ports. The original splitter plate configuration (Fig.4.9) was found to be the best. While the flow between the splitter plates and the sidewalls was separated, it was isolated from the centre sector by the splitter plates. The boundary layers on the inner faces of the plates were attached and stable, and the centre sector flow was in an axial direction with no outwards migration.
Fig. 4.10 shows the circumferential static pressure distributions, obtained at the locations shown in Fig. 4.8, across the annulus both before and after modification. The static pressures are presented as the static pressure drop from ambient, normalised by the dynamic pressure at pre-diffuser inlet, $q_{in}$. The splitter plates have caused a substantial flattening of the distribution, with a variation across the centre sector of $0.02q_{in}$, compared with a variation of approximately $0.05q_{in}$ before modification.

These modifications (Fig. 4.9) have created a stable flow regime in the outer feed annulus, with the separation of the sidewalls in the vicinity of the secondary ports hidden from the centre sector by the splitter plates. Some migration of fluid towards the centre sector must occur, due to the growth of the boundary layers on the sidewalls. Because the high turbulence intensities that are expected to be present in the feed annuli would add substantial uncertainties to the mass flow measurements (Carrotte and Wray (1991) estimated that turbulence caused a 17% error in measurements in a feed annulus), and because it would be impossible to distinguish between the effects of the sidewalls and the effects of the circumferential distribution of the flow due to cowl holes, fuel injectors and ports, no attempt was made to quantify this migration. The differential bleed system was subsequently used (see section 4.1.4) to obtain an equal bleed flow rate across the whole annulus. It is considered that the effects of the sidewalls on the flow in the outer feed annulus have been sufficiently alleviated, and the objectives of the programme have not been compromised.

### 4.1.3 Inner feed annulus

Wool tuft flow visualisation (Fig. 4.11) showed that the boundary layers on both sidewalls remained attached downstream of the primary ports. The boundary layer became unstable as it passed the secondary ports and separated intermittently, but tended to remain attached. There was no spreading out of the central flow as in the outer annulus and the sidewalls did not appear to have a significant effect on the centre sector. The condition of the boundary layers on the sidewalls was much better in the inner annulus than in the outer annulus. This is attributed to the smaller ports and greater bleed flow in the inner annulus, which results in less diffusion when fluid is removed by the ports.

The sensitivity of the sidewall boundary layers in the inner feed annulus to the bleed flow was assessed by varying the flow through the outer bleed sectors. It was found
that the boundary layers were destabilised as the bleed flow was reduced, with separation occurring downstream of the half primary ports adjacent to the sidewalls. The half primary ports were therefore blocked. Wool tuft flow visualisation showed that the sidewall boundary layers now remained attached, although they displayed unstable tendencies but did not separate, and no migration of the centre sector flow was observed. With the differential bleed system (see section 4.1.4) set to obtain an equal bleed flow rate across the whole annulus, the sidewall boundary layers were observed to be intermittently unstable in the vicinity of the secondary ports, but remained attached. The instability was restricted to the small region between the secondary ports and the sidewalls.

Circumferential static pressure distributions across the inner and outer annuli are compared in Fig.4.12. The normalised static pressure varies by less than $0.02q_{in}$, a smaller variation than that observed in the modified outer annulus. With a minor local instability between the secondary ports and the sidewalls, the condition of the flow in the inner feed annulus is not perfect, but it is thought to be sufficient to meet the programme objectives.

4.1.4 Annulus bleed flows

The pressure drops across each sector of the bleed duct metering plates were measured so that the mass flow could be adjusted to achieve the design mass flow splits of 3.93% and 12.14% of the inlet mass flow through the outer and inner bleeds respectively. Pressure drops were measured using static tappings upstream and downstream of the plate in each sector (Fig.4.13). The measured pressure drop was assumed to equal the dynamic pressure in the metering holes, from which the velocity and therefore the mass flow could be calculated. To verify this assumption the dynamic pressure in a number of metering holes was measured by measuring the total pressure in the centre of the hole, using a pitot probe, referenced against the static pressure downstream of the plate. The static pressure drop was found to be an accurate reflection of the dynamic pressure in the holes.

The holes in the metering plates were designed to have a discharge coefficient of 0.94, and this value was assumed to be correct. The velocity of the flow through the plate was calculated from the pressure drop, therefore the mass flow through each sector could be calculated and trimming holes blocked or opened to adjust the mass flow. The differential bleed facility proved to be useful at this point, as the pressure drops across the
The Experimental Programme

outer sectors and the centre sector of each metering plate were not equal. In the outer annulus, this was caused by the pressure loss due to the flow separation in the regions between the splitter plates and sidewalls, while a smaller discrepancy existed in the inner annulus where a pressure loss was incurred due to the growth of the boundary layers on the sidewalls. The three sectors of each plate were adjusted independently to achieve the mass flow splits summarised in table 4.3 below:

<table>
<thead>
<tr>
<th></th>
<th>Centre sector</th>
<th>Outer sector (left)</th>
<th>Outer sector (right)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner annulus</td>
<td>12.96%</td>
<td>13.07%</td>
<td>12.97%</td>
</tr>
<tr>
<td>Outer annulus</td>
<td>3.98%</td>
<td>3.91%</td>
<td>3.89%</td>
</tr>
</tbody>
</table>

Table 4.3 Annulus bleed flow splits

4.2 LDA system tests
Before using the LDA system for measurements inside the flame tube it was necessary to test the system on the rig and to test the seeder. These tests were needed to ensure that all velocities in the flame tube would be measurable and to ensure that the seeder was suited to measurements of this flow field.

Two seeders were available: a Dantec evaporation/condensation type seeder, which heats the oil and condenses it to form a mist of oil particles; and a TSI six-jet atomiser. Previous experience of both seeders showed that the TSI seeder is more reliable, produces greater quantities of seed and produces smaller seed particles, however the particles are given an electrostatic charge by the seeder that can cause them to be attracted to surfaces within the rig and thus contaminate the windows. The greatest contamination occurs when a separated flow reattaches to the surface.

The chief criterion by which the suitability of each seeder was assessed was the time taken for the laser access windows to be contaminated by the oil. Both seeders were tested at various seed flow rates. Both were found to contaminate the laser access windows in the sidewalls as seed particles passing through the swirler closest to the sidewall were thrown outwards by the swirler and onto the window. This happened very quickly with the Dantec seeder, a fine mist being sprayed onto the window within a very
short time. The laser beams were refracted by the oil on the window, thus preventing all six beams from crossing at the same point and thus halting the collection of data. Seed produced by the TSI seeder tended to coagulate in the swirler, forming a film of oil on the surface of the swirler, before being thrown towards the window in large droplets. When one droplet hits the window at a point where a beam passed through it, that beam is refracted so that the six beams do not cross at the same point. At this point the collection of data is instantly halted, but all data collected previously is of good quality. Good LDA data rates close to the swirler indicated that sufficient quantities of good quality seed enter the flame tube through the swirler, and the coagulation only occurs on the surface of the swirler. The large particles thrown outwards by the swirler were typically of approximately 1mm diameter, and were thus larger than the LDA measurement control volume. These particles were thus not included in the validated data as the BSAs reject oversized particles. The TSI seeder was found to be more appropriate for use with this rig as these droplets took some time to form, thus allowing time for LDA measurements to be taken before the window became too badly contaminated. This was found to happen in 15 to 30 minutes, depending on the area of the window that the beams passed through - the top of the window generally became contaminated faster than other areas - although there was an element of luck involved, as a droplet may hit window at its intersection with a laser beam at any time.

A further problem was found in the heating of the room as the test progressed. The fan controllers, situated in the same room, emitted heat so that the room temperature rose by up to 5° C over a three hour period. This caused a loss in light intensity of up to 50% in the beams emitted from the LDA probes. This is believed to be caused by the thermal expansion of the bench on which the laser and transmitter box are mounted, resulting in misalignment of the beams and fibres in the fibre manipulators. The output power loss causes the light intensity in the measurement volume to be reduced, resulting in a drop in the signal to noise ratio and data rates. Data rates were observed to drop slowly over time until they became so slow that LDA measurements became impractical. In addition, operation of the LDA system with the beams and fibres misaligned is to be avoided as the misalignment may cause the laser beam to burn the cladding of the fibres, resulting in a reduction in the fibre's transmission efficiency. For safety reasons, the LDA system and the test facility were controlled from outside the room in which the facility and the laser
The Experimental Programme

were situated, and the room temperature and output power were found to recover slightly when the doors of this room were left open while the laser access windows were cleaned. As the windows were cleaned up to fifteen times during some traverses the duration of each test could be extended to four or five hours.

Because of the restrictions imposed by the regular contamination of the windows and the temperature rise it was necessary to restrict the number of samples taken at each point to 10,000 and to maximise the validated data rates at all times so that measurements could be taken at a reasonable number of points in each traverse. It was estimated that the statistical uncertainty, due to the sample size, in the mean velocity measurements is ±0.31 m/s, and the uncertainty in the measurements of the rms components is ±3.5% (see Chapter 3). While a larger sample size would ideally be used, the accuracy of the data obtained in this project is considered to be reasonable.

Exploratory LDA measurements were conducted inside the flame tube to establish the optimum BSA settings and to check that all the velocities could be measured. The flow was found to be highly turbulent throughout the flame tube. A typical velocity histogram is shown in Fig. 4.14. This illustrates the distribution of the 10,000 velocity samples at a single point. Because the turbulence intensity is high a wide range of velocities is sampled, giving the histogram a large span. The bandwidth of the BSA must be large enough to measure all velocities at each point, i.e. larger than the span of the histogram. A bandwidth of 16 MHz was found to produce good validated data rates throughout much of the flame tube, however at many points the span of one or more of the three velocity samples dictated an increase to 32 MHz. As discussed in section 3.2.7 above, this can result in a poor validated data rate and is therefore undesirable. Because the span of the histogram is a function of the turbulence intensity, it can be reduced by reducing the mean velocity. Reducing the inlet Mach number to 0.12 was found to reduce the span of the velocity histograms so that, at most points in the flame tube, all velocities could be measured with a span of 16 MHz.

Some histograms obtained in the exploratory tests also revealed that the flow exhibits a bimodal distribution at some locations inside the flame tube. One such histogram is illustrated in Fig. 4.15. This histogram has two distinct peaks, indicating that the flow at this point is switching between two turbulent states. This causes a distortion of the data calculated from the LDA samples at such points. The calculated mean velocity
The Experimental Programme

will be a value that falls somewhere between the two peak values, and thus does not adequately describe the flow at this point, while averaged turbulence data such as the normal stresses will be influenced by the broadening of the histogram. This broadening could affect the operation of the BSA. If the bimodal velocity distribution causes the span to be increased so that the bandwidth must be increased the validated data rate will be reduced. It was felt that, if the histogram is bimodal, the bandwidth of the BSA should be increased only if a significant number of samples would be lost if the 16MHz bandwidth were maintained. If a small number of samples is lost from one end of the histogram then the mean velocity will not be significantly affected. The turbulence data will be affected, however if a small number of samples is lost then the effect is likely to be insignificant relative to the effect of the broadening of the histogram.

4.3 The experimental programme

The experimental programme was defined to meet the objectives discussed in section 1.6. To meet these objectives, the measurements discussed below are required.

Much is already known about the behaviour of the flow in feed annuli. So that the experimental programme could concentrate on the internal flow field, the external measurements were reduced to the minimum necessary to ensure that the behaviour of the external flow is satisfactory and that boundary conditions were available for CFD calculations. Radial button hook probe measurements of the pre-diffuser inlet and exit velocity profiles were used to obtain an inlet condition for CFD calculations and to assess the performance of the pre-diffuser. Area traverses of planes downstream of the secondary ports in the feed annuli were performed, using button hook probes, to assess the performance of the feed annuli. The only subsequent external measurements were radial button hook probe traverses at annulus entry, and wool tuft experiments to visualise the flow entering the flame tube ports, performed at a later stage to assess the coupling of the internal and external flows. These measurements are listed in Table 4.4 below and their locations are illustrated in Fig.4.16.
The Experimental Programme

<table>
<thead>
<tr>
<th>Location</th>
<th>Instrumentation</th>
<th>Traverse type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>XP1</td>
<td>button hook</td>
<td>radial</td>
<td>Pre-diffuser inlet</td>
</tr>
<tr>
<td>XP2</td>
<td>button hook</td>
<td>radial</td>
<td>Pre-diffuser exit</td>
</tr>
<tr>
<td>XO1</td>
<td>button hook</td>
<td>radial - 3 locations (0°, 3.75°, 7.5°)</td>
<td>Outer annulus entry</td>
</tr>
<tr>
<td>XO2</td>
<td>button hook</td>
<td>area - centre sector</td>
<td>Outer annulus</td>
</tr>
<tr>
<td>XI1</td>
<td>button hook</td>
<td>radial - 3 locations (0°, 3.75°, 7.5°)</td>
<td>Inner annulus entry</td>
</tr>
<tr>
<td>XI2</td>
<td>button hook</td>
<td>area - centre sector</td>
<td>Inner annulus</td>
</tr>
</tbody>
</table>

Table 4.4 External measurements

The LDA system was used for all measurements inside the flame tube. To obtain a coarse overall map of the flow field, the initial set of measurements comprised area traverses of the centre sector at four planes of constant axial (x) location within the flame tube. These four planes chosen to investigate the four most important flow field features: the primary jets, the secondary jets, the recirculating flow field in the secondary zone and the recirculation and swirling jet in the primary zone. The first four measurement planes were thus located upstream of the primary ports, in the plane of the primary ports, between the primary and secondary ports and in the plane of the secondary ports. Measurement locations were separated by 1.25° circumferentially and 10mm radially. The orientation of the LDA probes was optimised to obtain measurements in the maximum possible area of each plane. At some points, particularly on the centre line, the radial extent of some planes was limited by the interference of the sidewalls and port chutes with the laser beams. Because the whole centre sector cannot be accessed from one side of the rig, each traverse was performed in two halves. The first set of internal LDA measurements is described in Table 4.5 below and illustrated in Fig.4.17.
The Experimental Programme

<table>
<thead>
<tr>
<th>Plane</th>
<th>Axial location</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>122</td>
<td>22mm from heat shield</td>
<td>Primary zone - 20mm upstream of primary ports</td>
</tr>
<tr>
<td>137</td>
<td>37mm from heat shield</td>
<td>Primary ports plane</td>
</tr>
<tr>
<td>159</td>
<td>59mm from heat shield</td>
<td>Secondary zone</td>
</tr>
<tr>
<td>180</td>
<td>80mm from heat shield</td>
<td>Secondary ports plane</td>
</tr>
</tbody>
</table>

Table 4.5 First set of internal measurements

The subsequent internal measurements did not follow a strictly defined programme, but were chosen to clarify the nature of certain flow field features identified from the first set of measurements, as discussed in detail in Chapter 5. The second set of measurements comprised area traverses across the exit planes of six ports. These were the two centre primary ports, the two opposed primary ports at the edge of the centre sector on the positive θ side, and the two opposed secondary ports on the positive θ side. Measurements in each plane were taken at a constant radius located close to the port chute exit. However it was necessary to locate the measurement volume to a known datum, the flame tube centre line, and because of the distortion of the flame tube the precise location of the port chutes was not known. All these measurements were taken at radial locations less than 2mm from the bottom edge of the chutes. The axial and circumferential spacing of data points were varied throughout each plane, being determined according to the position of the measurement volume within each jet, with minimum axial and circumferential spacings of 1mm and 0.1° used to resolve the jets' shear layers, which could easily be identified during the traverse. The second set of internal LDA measurements is listed in Table 4.6 below and illustrated in Fig.4.18.
The Experimental Programme

<table>
<thead>
<tr>
<th>Plane</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPE</td>
<td>Inner primary port, edge of sector</td>
</tr>
<tr>
<td>IPC</td>
<td>Inner primary port, centre of sector</td>
</tr>
<tr>
<td>OPE</td>
<td>Outer primary port, edge of sector</td>
</tr>
<tr>
<td>OPC</td>
<td>Outer primary port, centre of sector</td>
</tr>
<tr>
<td>IS</td>
<td>Inner secondary port</td>
</tr>
<tr>
<td>OS</td>
<td>Outer secondary port</td>
</tr>
</tbody>
</table>

Table 4.6 Second set of internal measurements

The final set of internal measurements comprised a mixture of area traverses and radial traverses. Area traverses were performed in two planes downstream of the secondary ports to assess the development of the flow field. Because of time constraints only half of the centre sector was traversed. Radial traverses were conducted at selected axial locations and at 0°, -3.75° and -7.5° to determine the trajectories of the jets. Although these angles were not on the jet centre lines, due to the displacement of the ports, they were found to be well within the jets and were chosen because they corresponded with circumferential locations that had been used in the initial set of internal measurements. The orientation of the LDA probes was optimised to obtain measurements over the greatest possible part of each traverse plane, although, as with the area traverses, the radial extent of each traverse was limited by the interference of the sidewalls and port chutes with the laser beams. The final set of internal measurements is listed in Table 4.7 below and illustrated in Fig.4.19.
<table>
<thead>
<tr>
<th>Plane</th>
<th>Axial location</th>
<th>Traverse type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>I100</td>
<td>100mm from heat shield</td>
<td>area</td>
<td>½ sector, 20mm from secondaries</td>
</tr>
<tr>
<td>I120</td>
<td>120mm from heat shield</td>
<td>area</td>
<td>½ sector, 40mm from secondaries</td>
</tr>
<tr>
<td>I27</td>
<td>27mm from heat shield</td>
<td>radial: 0°</td>
<td>Centre primary jets</td>
</tr>
<tr>
<td>I32</td>
<td>32mm from heat shield</td>
<td>radial: 0°</td>
<td>Centre primary jets</td>
</tr>
<tr>
<td>I42</td>
<td>42mm from heat shield</td>
<td>radial: 0°, -7.5°</td>
<td>Centre and edge primary jets</td>
</tr>
<tr>
<td>I47</td>
<td>47mm from heat shield</td>
<td>radial: 0°, -7.5°</td>
<td>Centre and edge primary jets</td>
</tr>
<tr>
<td>I52</td>
<td>52mm from heat shield</td>
<td>radial: 0°, -7.5°</td>
<td>Centre and edge primary jets</td>
</tr>
<tr>
<td>I64</td>
<td>64mm from heat shield</td>
<td>radial: 0°, -3.75°, -7.5°</td>
<td>Primary and secondary jets</td>
</tr>
<tr>
<td>I69</td>
<td>69mm from heat shield</td>
<td>radial: -3.75°, -7.5°</td>
<td>Edge primary and secondary jets</td>
</tr>
<tr>
<td>I74</td>
<td>74mm from heat shield</td>
<td>radial: -3.75°</td>
<td>Secondary jets</td>
</tr>
<tr>
<td>I85</td>
<td>85mm from heat shield</td>
<td>radial: -3.75°</td>
<td>Secondary jets</td>
</tr>
<tr>
<td>I90</td>
<td>90mm from heat shield</td>
<td>radial: -3.75°</td>
<td>Secondary jets</td>
</tr>
<tr>
<td>I95</td>
<td>95mm from heat shield</td>
<td>radial: -3.75°</td>
<td>Secondary jets</td>
</tr>
<tr>
<td>I105</td>
<td>105mm from heat shield</td>
<td>radial: -3.75°</td>
<td>Secondary jets</td>
</tr>
<tr>
<td>I113</td>
<td>113mm from heat shield</td>
<td>radial: -3.75°</td>
<td>Secondary jets</td>
</tr>
</tbody>
</table>

Table 4.7 Final internal measurements
5. Results and Discussion

5.1 External flow field

Fig. 5.1a shows the radial velocity profiles measured at planes XP1 and XP2, the pre-diffuser inlet and exit planes. A non-dimensional velocity is shown, normalised by the maximum velocity at the inlet, plotted against the percentage passage height, with 0% and 100% passage heights corresponding to the inner and outer walls respectively. Both profiles are reasonably flat in the central regions, with the lower free stream velocities at the exit plane demonstrating the diffusion achieved by the pre-diffuser. Note that, as discussed in section 4.1.1, the mean velocity at the pre-diffuser exit plane is 5.3% too high due to the migration of fluid towards the centre of the passage caused by the growth of the boundary layers on the sidewalls. The same data is presented in Fig. 5.1b, with the velocity normalised by the free stream value in each plane. This highlights the relative boundary layer growth between the inlet and exit planes of the pre-diffuser. This is to be expected since the adverse pressure gradient in the pre-diffuser will promote growth of the boundary layers. The displacement thickness, momentum thickness and shape parameter of each boundary layer are listed in Table 5.1 - see also section 4.1.1.

<table>
<thead>
<tr>
<th>Boundary Layer</th>
<th>δ* (mm)</th>
<th>θ (mm)</th>
<th>H</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet, inner wall</td>
<td>0.6</td>
<td>0.4</td>
<td>1.7</td>
</tr>
<tr>
<td>Inlet, outer wall</td>
<td>0.9</td>
<td>0.6</td>
<td>1.4</td>
</tr>
<tr>
<td>Exit, inner wall</td>
<td>1.8</td>
<td>1.2</td>
<td>1.6</td>
</tr>
<tr>
<td>Exit, outer wall</td>
<td>2.6</td>
<td>1.4</td>
<td>1.8</td>
</tr>
</tbody>
</table>

Table 5.1 Pre-diffuser boundary layer parameters

Fig. 5.2 shows a radial profile of turbulence intensity at the pre-diffuser inlet, plane XP1. As discussed in Chapter 3, this turbulence intensity is the root mean square of the velocity measured by a single hot wire probe, normalised by the local mean velocity measured by the hot wire probe, and thus contains velocity components in the radial, circumferential and axial directions. Because there is no rotor or OGV blade row upstream of the inlet, the turbulence intensity is low across the centre of the passage, and rises in the turbulent boundary layers, with a peak of 11% in the outer boundary layer. As discussed in Chapter 3, the hot wire probe could not be allowed to approach the inner wall for fear of breakage. As a result the inner boundary layer could not be measured, however a distinct increase in
Results and Discussion

turbulence intensity can be seen in the inner 15% of the passage, caused by the turbulent boundary layer.

The static pressure profile at the pre-diffuser exit plane is presented in Fig. 5.3. The static pressure is presented in the form of a non-dimensional static pressure rise coefficient, \( p_n \), which is normalised by the dynamic pressure at pre-diffuser exit:

\[
p_n = \frac{p_{exit} - p_{in}}{q_{exit}}
\]

The variation in \( C_p \) across the pre-diffuser exit plane is small. This indicates that the influence of the flame tube on the flow in the pre-diffuser is not significant. Given the increase in the dump gap from its design value (see section 2.1.5), it is to be expected that the influence of the flame tube should be reduced as the distance between the pre-diffuser and flame tube has increased. The mean \( C_p \) value, that is the mean non-dimensional static pressure rise in the pre-diffuser, is 0.42. A pre-diffuser stagnation pressure loss coefficient, \( \lambda \), of 0.01 was calculated using eqn. 3.10.

Fig. 5.4 shows contours of non-dimensional velocity at planes XI2 and XO2, downstream of the secondary ports in the inner and outer feed annuli respectively. The velocities observed in the inner annulus are much greater than those observed in the outer annulus, the mean velocity in the outer annulus being 56% of that in the inner annulus. This is due to differences in the areas and flow splits of the feed annuli. At these planes, the area of the outer annulus is 32% greater than the area of the inner annulus, while the mass flow through plane XO2 is estimated to be 68% of the mass flow through plane XI2 due to the removal of a greater bleed flow from the inner annulus. The differences observed here are therefore to be expected. Circumferentially averaged velocity profiles at these planes are presented, together with circumferentially averaged velocity profiles at the annulus inlet planes (XI1 and XO1, 23mm and 20mm from the flame tube back plate respectively), in Figs. 5.5 and 5.6. All velocities presented here are normalised by the mean velocity at pre-diffuser inlet. A substantial circumferential variation in the velocity can be seen in the outer annulus, with a velocity deficit in the centre indicating that a wake is present due to the fuel injector feed arm situated upstream. There is no such central deficit in the inner annulus, where the velocity contours peak close to the centre of the sector in a pattern that suggests a cyclic, once per sector variation. This is caused by the circumferential distribution of the cowl hole, which has been shown previously (Barker and Carrotte, 1997) to dominate the circumferential variation in the flow in the
inner annulus, causing peak velocities to occur in line with the centre of the cowl hole. Further circumferential variations are caused by the removal of air from the annuli by the two rows of ports situated upstream of these planes. The contour patterns in both annuli are asymmetric, being shifted slightly to the left. The flame tube ports are displaced in the same direction and by a similar degree. This may be the cause of the circumferential asymmetry in the feed annulus flows.

In common with the findings of Carrotte et al (1993) the velocity profile close to the inlet of each annulus is strongly biased, displaying a peak close to the surface of the flame tube, due to the acceleration of the flow around the head of the flame tube. As has been observed by several workers (e.g. Fishenden and Stevens, 1977, and Carrotte, Bailey and Frodsham, 1995), these profiles have been substantially mixed out by the end of the annuli, although the peaks in the profiles are still present and have moved towards the casings.

The mean stagnation and static pressures at the inlet and at planes XO2 and XI2 have been used to calculate the stagnation pressure loss coefficient, $\lambda$, and static pressure rise coefficient, $C_p$, between the pre-diffuser inlet and each annulus exit plane. The stagnation pressure loss coefficients are 0.30 and 0.265 in the outer and inner annuli respectively, while the outer and inner annulus $C_p$ values are 0.633 and 0.552. Carrotte et al (1993) measured stagnation pressure loss coefficients for a similar geometry of 0.318 and 0.302 for the outer and inner annuli respectively, and static pressure rise coefficients (between pre-diffuser inlet and annulus inlet) of 0.469 and 0.454. The pressure losses measured here appear to be relatively low, while the static pressure rise appears to be high. It is thought that the low stagnation pressure loss coefficients arise from the increase in the dump gap from its design value, while the difference between the values recorded in the inner and outer annuli arises from the different mass flow splits in the feed annuli.

The evidence presented here indicates that there is nothing remarkable to be seen in the behaviour of the external flow. It was therefore the experimental programme should concentrate in the internal flow field.

5.2 Internal flow field

This section presents a detailed examination of the flow field inside the flame tube, with three sub-sections dealing with the initial area traverses, the measurements at the primary
and secondary chute exits, and the measurements of the trajectories of the primary and secondary jets. The initial area traverses produced coarse measurements of the flow in the primary and secondary zones, the results of which are discussed in section 5.2.1. Information on the development of the flow downstream of the secondary zone was obtained from subsequent area traverses, the results of which are also discussed in section 5.2.1. In section 5.2.2, the results of measurements of the flow exiting the primary and secondary ports are presented and discussed. Comparisons between ports are made and used to explain some of the results presented in section 5.2.1. Detailed measurements and analysis of the trajectories of the primary and secondary jets are presented in section 5.2.3.

The mean velocity data presented here was all obtained by Laser Doppler Anemometry (LDA). A right handed co-ordinate system is used here, as illustrated by Fig.5.7. Axial (u) velocity components are parallel to the axis of the flame tube, that is 9° from the engine axis, and positive in the downstream direction. Radial (v) components are normal to the flame tube axis, parallel to the back plate, and positive outwards. Tangential (w) components are normal to the axial and radial components.

5.2.1. Area traverse results

5.2.1.1 Primary zone

Contours of the mean radial (v) velocity component at plane 137 are presented in Fig.5.8. This plane is situated 37mm from the back plate of the flame tube, close to the centre of the primary ports. Only one primary jet, the outer central primary jet, is evident here, indicated by the contours of relatively large negative radial velocities projecting from the outer edge of the plot. The lack of any corresponding positive v component contours on or near the centre line suggests that the inner central primary jet is absent from this plane. The negative v velocities observed on the centre line across the entire plot suggest that the outer jet penetrates across most of the depth of the flame tube to a point close to the inner port. The spreading of the contours in the inner half of the plot indicates that this jet spreads out and entrains surrounding fluid in the process. A small region of strong negative v component, visible in the top left hand corner of the plot, is attributed to the sector-edge primary jet at this location, and smaller v velocities in the top right hand corner may also indicate the presence of a sector-edge primary jet. There is little evidence
Results and Discussion

of the inner sector-edge primary jets except a very small positive v component in the bottom left hand corner. To summarise, at this plane the dominant feature is the outer central primary jet and, surprisingly, there is little indication of the presence of the other primary jets.

Contours of the mean axial (u) velocity component at plane 137 are presented in Fig.5.9. A large region of negative u velocities is present in the centre of the flame tube, situated in a location that corresponds to the dominant primary jet seen in Fig.5.8. This negative velocity region therefore represents a back flow, moving towards the primary zone and the swirler. It appears that the outer central primary jet is moving upstream into the primary zone. However contours of small positive u velocities can be seen in the portion of the jet closest to the outer central primary port. This suggests that the jet enters the flame tube with a shallow downstream trajectory, or that the downstream component is imparted by the crossflow, and that the process by which the jet is turned to the upstream direction takes place within the flame tube. The size of the region of negative u components is greater than the apparent size of the primary jet. A region of reverse flow is expected to exist downstream of the swirler, and it appears that the jet enters this region and is thus drawn upstream, contributing higher negative velocities to the reverse flow. These higher negative velocities are found away from the fuel injector centre line, being shifted towards the inner liner, due to the radial momentum of the jet. The shape of the negative u contours is similar to the characteristic kidney shape observed by most workers investigating jets in cross flows. Meanwhile positive (downstream) u components can be seen outside this central region, in the corners of the plot.

The presence of a single, dominant outer central primary jet is confirmed by the mean velocity vectors in Fig.5.10, which reflect the v and w components. These show the jet penetrating across much of the flame tube's depth and spreading out, as indicated by Fig.5.8. Entrainment of surrounding fluid by the jet is also apparent in the smaller velocity vectors either side of the jet. It can be seen that the jet is not moving in a purely radial direction and has a small circumferential (w) velocity component, which may have been imparted to the jet by fluid introduced through the swirler. The vectors in the top left hand corner region of the plot have a significant w component with the same direction as the w component of the jet vectors. However there is little evidence in this plane of an overall swirling flow pattern associated with the flow issuing from the swirler.
Results and Discussion

A single large vector in the top left hand corner of Fig.5.10, plus a smaller vector with the same direction, shows that a sector-edge primary jet is present at this location. There are no significant vectors to indicate the presence of the other four primary jets.

Contours of the mean u velocity component at plane I22 are shown in Fig.5.11. This plane is situated in the primary zone, 15 mm upstream of plane I37. The flow patterns that can be seen here are largely typical of the flow field generated by a swirler, a circular central reverse flow region surrounded by flow issuing from the swirler and moving in a downstream direction. However the flow field differs substantially from the conventional swirler generated flow field. The reverse flow region is evidently not the symmetrical toroidal recirculation observed by Koutrmos and McGuirk (1989). There is a distinct bias towards the inner liner and the maximum negative u velocities appear to be in locations that correspond with the location of the maximum negative u velocities at plane I37 (Fig.5.9). The size of this reverse flow region is similar to that seen at plane I37, indicating that the reverse flow observed at that plane is a continuation of that seen here and is created by the swirler. A kidney shape, which is characteristic of a jet in cross flow, can also be observed, suggesting that a contrarotating vortex pair may be present that is associated with the outer central primary jet. The kidney pattern is not symmetric about the centre line. This suggests that the contrarotating vortex pair, if present, is asymmetric. Smith and Mungal (1998) observed an asymmetric contrarotating vortex pair in a jet in a symmetric cross flow, despite their experimental apparatus being, within experimental tolerances, symmetric. In this experiment the cross flow approaching the jet is swirling and must therefore be asymmetric, so asymmetry in the vortices must be expected.

The positive u velocities surrounding the reverse flow region must be the source of the positive u velocities seen in the corners of plane I37 (Fig.5.9). These velocities may be sufficient to cause the sector-edge primary jets to be rapidly deflected downstream, thus explaining the fact that they cannot be seen at plane I37.

Mean velocity vectors at plane I22 (Fig.5.12a) show negative v components in much of the lower half of the plot. Vector patterns at locations corresponding to the lobes of the kidney pattern of Fig.5.11 indicate the possible presence of a contrarotating vortex pair. Contours of the \( \Omega_x \) component of vorticity at this plane (Fig.5.12b) display peaks close to the locations of these possible vortices. However the resolution of the measurement grid is not sufficient to resolve these vortices accurately. The radial
Results and Discussion

velocities, the kidney shape, the asymmetry and the possible contrarotating vortex pair indicate that there is a single primary jet entering the reverse flow, instead of the two impinging primary jets that would normally be expected. The evidence from both planes I22 and I37 show conclusively that this is the outer central primary jet.

Although plane I22 is close to the swirler there is no evidence in Fig.5.12 of swirl occurring consistently throughout the plane. However the top left hand corner region of Fig.5.12 exhibits swirl components of the same direction and similar magnitude to those seen at plane I37 (Fig.5.10) and with the clockwise rotation that is expected to be imparted to the flow by the swirler. This shows the influence of the swirler, however the jet appears to suppress the swirl component elsewhere in this plane. More measurements are needed to assess the behaviour of the flow issuing from the swirler.

Some bimodal velocity histograms were found in both planes I22 and I37. The locations of these histograms are marked in Fig.5.13 and 5.14. It is noteworthy that in the majority of cases only the v component histograms were found to be bimodal. This indicates that, at these locations, there is a low frequency unsteadiness in the flow and this is associated with the momentum component in the radial (v) direction. The occurrence of these bimodal histograms must therefore be linked to the primary jets. In plane I37 there are few bimodal histograms located in or near the jet. It is unlikely that, if this low-frequency unsteadiness was a feature of the bulk jet flow, bimodal histograms would be observed in the core of the jet, because the velocity profile is flat. However bimodal histograms would be observed in the shear layers surrounding the jet core. The lack of bimodal histograms in the shear layers therefore indicates that the jet is reasonably steady. There is a significant cluster of bimodal histograms close to the centre line near the inner extreme of the plot. This suggests that some unsteady process is occurring near the inner wall. This could be caused by the proximity of the outer and inner central jets, suggesting that there may be some interaction between the two jets other than the conventional impingement that is expected to occur between two opposed jets. It should be noted that the time dependence in the flow that is indicated by the bimodal histograms suggests that the use of CFD methods that use the Reynolds Averaged Navier Stokes equations, whereby the time dependence is removed from the CFD model, is not appropriate for the prediction of the combustor flow field, and that time dependent methods such as Large
Results and Discussion

Eddy Simulation (LES) should instead be used. As noted in Chapter 1, instability in the flow field also affects the performance of the combustor.

The bimodal histograms at plane 122 were mostly found in or near the shear layer between the reverse flow region and the flow issuing directly from the swirler. This is not particularly remarkable as unsteadiness may be expected in such shear layers.

It was expected that the two opposed central primary jets would impinge and contribute to the reverse flow in the primary zone. However this is clearly not suggested by the experimental data. The outer central primary jet dominates the primary zone flow field and there is no evidence of the presence of the inner central primary jet at plane 122 or 137. Given that nothing unexpected was observed in the behaviour of the external flow, it would be reasonable to expect the inner central primary jet to have similar momentum to the outer jet and, since the outer jet has sufficient momentum to penetrate across most of the flame tube, it would be reasonable to expect the two jets to impinge. Some mechanism is therefore present that causes the inner central primary jet to be deflected downstream of plane 137 while the outer jet is deflected into the primary zone. The absence, in Figs.5.13 and 5.14, of bimodal histograms in the outer central primary jet suggests that the mechanism by which this jet is deflected is stable. However bimodal histograms indicate that there is a region of low frequency unsteadiness in the flow that may be influenced by one or both central primary jets. These processes were unknown at this stage, and further investigation was deemed to be necessary, the results of which will be discussed in sections 5.2.2 and 5.2.3. It appears that the cross flow causes the sector-edge primary jets to be deflected downstream of plane 137, although whether they impinge cannot be determined from the measurements presented so far. The results of further investigation of these jets are also discussed in sections 5.2.2 and 5.2.3.

5.2.1.2 Secondary zone

Measurements were obtained at two planes in the secondary zone: Plane 180, which is close to the centre of the secondary ports, and plane 159, which is 22mm downstream of plane 137 and 21mm upstream of 180. The flow at plane 159 was observed to be influenced by both sets of jets. Thus the measurements obtained at plane 180 are discussed first to assist in the discussion of the measurements obtained at plane 159.
Results and Discussion

Fig. 5.15 shows contours of mean radial ($v$) velocity components at plane $I_{180}$. The contours of positive $v$ emanating from the inner secondary ports, and of negative $v$ emanating from the outer secondary ports, clearly represent the secondary jets. The rapid deceleration of all four jets towards the centre of the flame tube suggests that the opposed jets impinge.

Contours of mean axial ($u$) component at plane $I_{180}$ (Fig. 5.16) show that a back flow exists in the central part of the region covered by this plot. Maximum back flow velocities occur at two points, corresponding to the locations of the jet impingements indicated by Fig. 5.15. Positive $u$ components are indicated in areas where Fig. 5.15 indicated the presence of secondary jets. The trajectories of the jets appear to take them in a downstream direction, which is to be expected of any jet in a cross flow and is typical of a jet being supplied from an annulus since the jets have an initial downstream axial momentum component due to the velocity of the approaching fluid in the feed annuli, and they impinge downstream of plane $I_{180}$ to create a back flow. The small negative $u$ velocities in the backflow suggest that the impinged jets split or "bifurcate" (Fig. 1.10) to create an upstream/downstream flow, the bulk of the flow going downstream. Similar bifurcations have been reported previously, for example Hatch et al (1995) and Baker and McGuirk (1992).

The jets can be seen in the mean velocity vectors of Fig. 5.17. Circumferential ($w$) velocity components in the vectors surrounding the impingement points also indicate that some of the fluid moving upstream from the impinged jets spreads circumferentially, particularly towards the outer central portion of the sector.

Locations where bimodal velocity histograms were found in plane $I_{180}$ are marked in Fig. 5.18. Although a few bimodal histograms were found in the shear layers of the jets, the bulk secondary jet flow appears to be steady. Significant clusters of bimodal histograms were found in the back flow regions. As in the primary zone, only the $v$ component histograms were found to be bimodal at these points, suggesting that there is a radial fluctuation in the backflow or a radial fluctuation in the location of the impingement point. The dominant momentum component here must be the radial component, due to the dominance of the secondary jets at this plane, and the bimodal histograms observed here indicate that the impingement of opposing secondary jets is an unsteady process due to the high radial momentum components of the opposing jets.
Figs. 5.19 and 5.20 show the contours of mean v and u velocity components respectively at plane 159. In the outer left and right hand portions of the sector, approximately aligned with the sector-edge primary ports, regions of negative (radially inwards) v components can be seen. Opposing these, in the inner half of the sector, are similar regions of positive (radially outwards) v components. Fig. 5.20 shows that these flows have positive u components and are therefore moving in the downstream direction. These are therefore opposed jets and must be the sector-edge primary jets that were absent from plane 137. The mean velocity vectors (Fig. 5.21) corresponding to these points have the appearance of opposing jet flows, and they appear to be impinging in the vicinity of this plane. V components and vectors in corresponding parts of plane 180 (Figs. 5.15 and 5.17) show radial velocities with opposite directions, suggesting that the jets spread radially outwards after impingement, while the u components (Fig. 5.16) show that they continue to move in the downstream direction. The jets appear to be subject to very little circumferential deflection. They also suppress any circumferentially outwards movement of the back flow that was observed at plane 180, which explains why the circumferential movement of the back flow observed in Fig. 5.17 was predominantly towards the centre of the sector.

Some of the fluid from the impinged jets appears in Figs. 5.17 and 5.21 to move circumferentially inwards and then radially towards the inner liner, approximately in line with the secondary ports. It is thought that this fluid may be contributing to a recirculation that is caused by the back flow from the impingement of the secondary jets.

A region of flow moving radially outwards and downstream can be seen on the centre line in the inner half of the sector at plane 159. This appears to be the upstream inner central primary jet that has been deflected downstream. Two small regions of flow moving radially outwards can be seen on either side of the centre line in the outer half of the sector. These flows have a small negative u component and are located in areas that suggest that they may represent recirculating flow from the downstream secondary jets.

The locations where bimodal velocity histograms were found in plane 159 are marked in Fig. 5.22. At all but three of these points only the v component histograms were found to be bimodal, which implies that the flow at these points is fluctuating in a radial direction. The main source of radial momentum is the jet flows, so the occurrence of these bimodal histograms must be linked to the primary and secondary jets that appear to
Results and Discussion
dominate the flow at this plane. A large cluster of these points is located in the outer
central portion of the sector where the flow appears to be part of the recirculation caused
by the impingement of the secondary jets. Similarly, clusters on either side of the centre
line in the inner half of the sector are located in regions of the flow that are thought to be
recirculating. This implies that the recirculations due to the impingement of the secondary
jets are not steady and fluctuate radially. The back flow at plane 180, which was created
by the impingement of the secondary jets and is a part of this recirculation process, was
shown in Fig.5.18 to exhibit a similar low frequency unsteadiness. Further bimodal
velocity histograms are located in the sector-edge primary jets, near to their impingement.
This indicates that the impingement of these jets is also unsteady and suggests that the
impingement point is not fixed. Because of the apparent dominance of impinged jet flows
in the secondary zone, a greater number of bimodal histograms was observed in plane 159
than elsewhere, and low frequency unsteadiness is a feature of a substantial proportion of
the flow at this plane.

The sector-edge primary jets therefore have a downstream trajectory due to their
initial injection angles, which were not known at this stage, and deflection by the
crossflow that is present in the outer extremes of the sector in the primary zone due to the
flow injected through the swirler, which can be seen in Fig.5.11. These jets impinge, but
the absence of any corresponding back flow at plane 137 (Fig.5.9) indicates that they do
not bifurcate, and bimodal histograms suggest that the impingement is unstable. The
impinged jets move downstream and remain at the edges of the sector, although some jet
fluid spreads circumferentially inwards to contribute to recirculations in the secondary
zone. The inner central primary jet has been deflected downstream and appears in the
secondary zone. The secondary jets impinge at a point downstream of plane 180 and
create a back flow and upstream recirculations. Bimodal histograms suggest that the
impingement and recirculations are unstable. Further investigations were deemed
necessary to fully determine the behaviour of the jets, the results of which are discussed in
sections 5.2.2 and 5.2.3.

5.2.1.3 Flow development downstream of the secondary ports
LDA measurements were conducted at planes 1100 and 1120, to assess the development of
the internal flow downstream of plane 180. These planes are 20mm and 40mm, or one and
Results and Discussion

two outer secondary port diameters, downstream of plane 180 respectively. Due to time constraints, measurements were conducted in only half of the centre sector because of the similarity of the flow pattern between both halves of the sector measured at plane 180.

Contours of mean $u$ velocity components at plane 1100 are shown in Fig. 5.23 and indicate that the $u$ components in this plane are mostly positive. The core flow at the centre of the sector appears to be the product of the impingement of the secondary jets, confirming that they bifurcate, as was indicated by the results at plane 180. Circumferential spreading of the jet after impingement is indicated by the presence of positive $u$ components close to the centre line. Sector-edge primary jet fluid is represented by the region of positive $u$ components at the edge of the sector.

Two regions of small negative $u$ velocities can be seen near to the inner and outer extremes of the sector and in line with the secondary ports. These regions are in the lee of the jets and are close to the jets. Plane 1100 is less than 10mm downstream of the bottom of the outer secondary chutes and, since the secondary jets were shown in section 5.2.1.2 to have a downstream trajectory, the regions of negative $u$ velocity shown here may be much closer to the jets. These regions appear to represent fluid that is being entrained by the jets. Consistent with this are the $v$ components at these locations (Fig. 5.24) which show the flow moving radially away from the liners. The contours at these points may therefore represent cross flow fluid that moves around the jets or chutes, near to the liners, and is entrained and drawn back upstream in the lee side of the jets. This flow pattern is typical of that observed in the formation of vortices behind a jet in a cross flow by, for example, Carrotte and Stevens (1989).

Fig. 5.25 shows the locations of bimodal velocity histograms. A cluster of these points is located in the secondary jet flow downstream of the impingement region. As at the other planes, only the $v$ component histograms were found to be bimodal and this is further evidence that the secondary jet impingement is unstable.

One final noteworthy observation from the measurements at plane 1100 is that the radial velocity components (Fig. 5.24) are much higher in the outer half of the plane than the inner half. Also the jet impingement appears to be closer to the inner liner than the outer liner. Further investigation was needed to explain this, the results of which are discussed in sections 5.2.2 and 5.2.3.
Results and Discussion

Contours of mean $u$ components at plane I120, 20mm downstream of plane I100, are shown in Fig.5.26. All the flow measured at this plane is moving in the downstream direction, and the flow appears to have mixed considerably in the 20mm from the upstream plane I100. Radial ($v$) components (Fig.5.27) are largely negative, exhibiting a continuing trend for the flow to move towards the inner liner, although the $v$ components observed here are smaller than those observed at the upstream plane, I100. Again, further investigation was needed to explain this. Bimodal velocity histograms were found at only five locations (Fig.5.28), and all were $v$ component histograms. This suggests that there is still some radial fluctuation present, which is associated with the upstream impingement, but its effects reduce with increasing downstream location.

5.2.2 Port exit measurements

The results reported in section 5.2.1 exhibit some unexpected features that required further investigation. The almost complete absence of all but one primary jet from plane I37 in particular was unexpected, and the behaviour of the two central primary jets was a considerable departure from the jet trajectories observed by a number of workers in axi-symmetric combustors. To attempt to explain these findings measurements of the jets were obtained in the vicinity of the port exits. The results of area traverses across the chute exit planes (Fig.4.18) are presented and discussed in this section.

5.2.2.1 Central primary jets

Mean radial velocity component contours at the inner and outer centre primary port exit planes (IPC and OPC) are shown in Figs.5.29 and 5.30, with the superimposed vectors showing the $u$ and $w$ components. Both are plotted to the same scale.

The inner jet (Fig.5.29) has a highly distorted profile. A jet fed from a pipe or a plenum might be expected to have a circular profile, although Perry et al (1993) have shown that the fluid at the leading edge of a jet fed from a pipe into a cross flow can sense the presence of the cross flow and is subject to a small degree of distortion inside the pipe. This jet, which is fed from an annulus, has a highly distorted profile, suggesting that nearly all of the flow enters through the back of the chute. While the high velocity core of the outer jet (Fig.5.30) has a similarly distorted shape to that of the inner jet, much less distortion is apparent and its profile occupies a greater proportion of the port area. As shown in Fig.4.18 the chute exit measurement planes are not parallel to the chute exits,
Results and Discussion

causi ng the upstream side of the jets to be exposed to the cross flow. However the cross
flow vectors on the upstream side of both jets are similar in magnitude, whilst the jet
velocities are similar in magnitude, which suggests that differences between the two jets
would not be seen if the jets' distortion were caused by the cross flow. Also the jet to
cross flow velocity ratio is relatively large (around 8, defined in terms of the cross flow
velocity, measured upstream of the jet in plane IPC, of 5m/s, and the jet core velocity of
40m/s), so it seems unlikely that the cross flow could cause this distortion of the jet
profile at such an early stage in the jet development.

The vectors in both figures show a movement of cross flow fluid around the lee of
the jet, with some of this fluid being drawn upstream towards the jet. Although no vortex
like structures can be observed in either figure, these vector patterns indicate that cross
flow vortices, as observed by Carrotte and Stevens (1989), are formed.

The inner jet is deflected downstream of the chute exit. The profile of Fig.5.29 is
misleading in this respect, as are all the chute exit profiles presented here, as small
portions of the flow seen at the bottom and sides of the jet are attributed to fluid entering
the flame tube through the chute cooling slots that are located on the lee side of each
chute as shown in Fig.5.31. Therefore the downwards deflection, which appears to be
approximately 1.4mm, is likely to be only 0.4mm when the 1mm deep cooling slot is
accounted for. To assess whether a jet deflection of 0.4mm could be caused by the cross
flow, the correlation of Srinivasan and White (1986) was used to define the jet centre line
trajectory:

\[
\frac{Y}{D_j} = a_0 J^{0.12} \left( \frac{S}{D_j} \right)^{0.23} \left( \frac{H}{D_j} \right)^{0.57} \left( \frac{X}{D_j} \right) \alpha
\]

\[
a_0 = 0.765 \left[ 1 + \frac{dH}{dX} \right]^{-0.55}
\]

\[
\alpha = 0.12 \left[ 1 + \frac{dH}{dX} \right]^{1.25}
\]

where X and Y are the axial and radial distances from the jet injection point, S is the jet
spacing, \(D_j\) is the jet diameter, \(H_0\) is the flame tube depth at the jet injection plane, and \(J\) is
the jet to cross flow momentum flux ratio. As the axial variation of the depth of the flame
tube is small around the jet injection plane, \(dH/dX\) is negligible and can be discounted.
The momentum flux ratio \(J\), which in an incompressible, isothermal flow is effectively
the square of the jet to cross flow velocity ratio, is 64. If the relevant geometric quantities
Results and Discussion

are applies then the radial distance, \( Y \), at which the cross flow deflects the jet by a distance \( X \) of 0.4mm is therefore:

\[
\frac{Y}{13.1} = 0.765 \times 64^{0.12} \left( \frac{38.4}{13.1} \right)^{0.23} \left( \frac{102}{13.1} \right)^{0.57} \left( \frac{0.4}{13.1} \right)^{0.12} \rightarrow Y = 44.8\text{mm}
\]

Hence the jet to cross flow momentum flux ratio is too great for the cross flow to cause the observed deflection of the jet in the short distance \(( \leq 2\text{mm})\) between the chute exit and the measurement plane, and the downwards pitch of the jet indicated by the axial components of the vectors in the core of the jet must be a feature of the flow through the chute and not a result of deflection by the cross flow. The combustor is a practical application of the jet in cross flow type flow regime and, unlike the work of Srinivasan and White and many others who have used normal jet injection fed by pipes or plenums, the jets are fed from feed annuli. They thus have an initial pitch angle, due to the momentum that is retained from the axial flow in the annulus, which is not normal to the hole unlike the normal jet injection that is assumed in the Srinivasan and White correlation.

However the jet deflection can be explained in terms of the pitch angle of the core flow. Contours of jet pitch angles are shown in Figs.5.33 and 5.34 for the inner and outer centre primary jets respectively. These angles are defined as shown in Fig.5.32, a 90° pitch angle defining a vector normal to the chute exit measurement plane, i.e. radially towards the centre of the flame tube, while a 0° pitch angle describes a vector pointing axially downstream. It can be seen that most pitch angles in the core of the inner jet are between 70° and 80°, i.e. subtending an angle between 10° and 20° relative to the centre line. If the jet's pitch angle is 75° then its axial movement in a radial distance of 2mm is:

\[
\Delta x = 2 \tan (90 - 75) = 0.536\text{mm}
\]

Hence the axial deflection of the jet can be explained by the pitch angle of the velocity vectors in the jet's core.

In contrast to the inner jet, the outer jet profile appears to occupy a much larger area (Fig.5.30). Whilst this is partly explained by the 22.5% greater area of the outer port, the discrepancies between the two jets are not in proportion with the port diameters. While the outer jet has a similarly shaped core of high velocity flow, there is a larger region of lower velocity flow. Fluid with a velocity greater than 30% of the inlet mean velocity occupies only 60.5% of the port area in the inner jet profile, while the equivalent portion of the outer jet profile is 94% of the outer port area. The core velocities and peak
Results and Discussion

velocities of the opposed jets are similar, the peak velocities differing by only 2.3 m/s (5% of the mean velocity at pre-diffuser inlet), which indicates that the pressure drops across the two ports are similar. Also the outer jet pitch angles (Fig. 5.33) are higher, being between 80° and 90°, than those found in the inner jet, resulting in less axial deflection. As there is no substantial difference between the cross flows approaching the two jets these differences must be explained in terms of the flow properties of the jets and their feed from the annuli.

The velocity profile of the jet at the port exit is strongly dependent on the condition of the flow supplied to the port (Lefebvre, 1983). Other workers have previously established that the jet velocity profile is dependent on the feed annulus flow. Spencer (1998) found that the flow separates from the upstream lip of a chuted port, and that the jet is caused to enter the flame tube at an angle because of the axial momentum of the flow that is supplied to the port. Carrotte and Stevens (1989) reported that the "deflection of flow and its separation from the casing wall... has a major influence on the velocity profile across [the port] exit plane". Comparisons of the two jet profiles and the chute cross sections shows that, in both cases, there is clearly a separation from the upstream lip of the chute, as described by Spencer. The axial momentum of the approaching flow in the feed annuli must cause the flow in the chute to move towards the back wall of the chute, resulting in the location of the core flow in the bottom half of each jet. Hence the differences between the two jets must be explained in terms of differences in the conditions of the flow fed to the two jets.

Fig. 5.35 shows the velocity profiles measured at planes XII and XO1, upstream of the centre primary ports in the inner and outer feed annuli. The peak velocity levels are similar, however away from the flame tube the velocities in the inner annulus are significantly higher than in the outer annulus. While a similar mass flow, approximately 40% of the total inlet flow, enters each annulus, and the annulus heights at these planes are similar, the area of the outer annulus at plane XO1 is 57% greater than the area of the inner annulus at plane XI1, due to the difference in radius. Hence the mean velocity is higher in the inner annulus than in the outer annulus. However the effect that this difference has on the flow entering the ports must depend on the part of the annulus from which each port draws air. If, for instance, the ports are fed by flow that is close to the flame tube then the difference in the mean velocities of the two feed flows is lower than
Results and Discussion

that seen over the full depth of the annuli. The flow immediately adjacent to the flame tube in the inner annulus is slower than that in the outer annulus, and the mean velocities in the two annuli over the first six to eight mm from the wall cannot be vastly different. If the ports are fed from this portion of the flow then it seems unlikely that the axial momentum of the flow fed to each port would be sufficiently different to explain the differences seen at the chute exit planes.

Wool tuft flow visualisation tests were performed to examine the manner in which the two centre primary jets are fed. The results of these tests are shown in Fig.5.36. There are two clear differences in the manner in which the two ports are fed. The inner jet is fed from a much wider area than the outer jet. It is perhaps more significant that the outer jet is fed from a much deeper portion of the annulus than the inner jet. The outer port takes air from approximately half the depth of the annulus, while the inner port takes air from a much shallower depth, estimated to be only 20% of the annulus height. The reason for these different feed conditions is not clear, however it is thought to be related to the circumferential distribution of the secondary ports and bleed offtakes. In the outer annulus, it was estimated (Boyce, 1999) that 43% of the flow passing the primary ports enters the secondary ports, and 13% is removed by the bleed ducts, and in the inner annulus, 29% enters the secondary ports and 42% enters the bleed ducts. All remaining air passes through the effusion cooling holes in the liners. If it is assumed that, as the ports draw air from a region close to the flame tube, the bleed offtakes draw air from a region close to the casing, then the greater bleed flow in the inner annulus must restrict the depth from which the primary ports can draw air, thus increasing the circumferential width of the area from which they are fed. In the outer annulus, the secondary ports have a greater influence and, because of their circumferential location, must draw air between the primary ports. This, coupled with the lesser influence of the outer bleed flow, must cause the outer primary ports to draw air from a narrower, deeper region.

At the plane of the primary ports, the outer annulus has a 31% greater area than the inner annulus. Estimated flow splits (Boyce, 1999) are summarised in table 5.2. The mass flow rates approaching the two rows of primary ports are similar, so the mean velocity of the air approaching the outer primary ports must be lower. It should be noted that the discrepancy between the mass flow rates at annulus entry approaching the primary ports is due to the removal of air from the annulus by effusion cooling holes.
Results and Discussion

upstream of the primary ports. The lower momentum and deeper feed area allow the air entering the outer port to turn towards it more easily. There is therefore a larger region of separated flow in the inner chute and the inner jet will have more axial momentum because the jet fluid undergoes less turning than that of the outer jet.

<table>
<thead>
<tr>
<th>Flow split (percentage of pre-diffuser inlet flow)</th>
<th>Outer annulus</th>
<th>Inner annulus</th>
</tr>
</thead>
<tbody>
<tr>
<td>Annulus entry</td>
<td>40.5</td>
<td>40</td>
</tr>
<tr>
<td>Flow approaching primary ports</td>
<td>35.8</td>
<td>36.4</td>
</tr>
<tr>
<td>Flow entering primary ports</td>
<td>6.6</td>
<td>5.3</td>
</tr>
<tr>
<td>Flow passing primary ports</td>
<td>29.2</td>
<td>31.1</td>
</tr>
<tr>
<td>Flow approaching secondary ports</td>
<td>26.3</td>
<td>29.1</td>
</tr>
<tr>
<td>Flow entering secondary ports</td>
<td>12.6</td>
<td>9</td>
</tr>
<tr>
<td>Flow passing secondary ports</td>
<td>13.7</td>
<td>20.1</td>
</tr>
</tbody>
</table>

Table 5.2 Feed annulus flow splits

Lefebvre (1983) relates the port discharge coefficient to the hole and annulus flow rates, the hole and annulus areas and the dynamic pressures in the port and annulus by:

$$C_D = \frac{\alpha}{A_r \sqrt{K}}$$

where $\alpha$ is the ratio of the hole mass flow rate to the annulus mass flow rate, $A_r$ is the ratio of the hole area to the annulus area, and $K$ is the ratio of the jet dynamic pressure to the annulus dynamic pressure. Based on the flow splits in table 5.2, $C_D$ values of 0.625 and 0.575 have been calculated for the outer and inner ports respectively. Although the formula above applies to a plain hole, and the true discharge coefficients must be higher due to the use of chuted ports with radiused inlets, these figures confirm that the different feed conditions and the different port and annulus geometries of the opposed jets must affect the discharge coefficients and thus the velocity profiles seen at the port exit planes.

Direct measurement of the discharge coefficients of the jets from these measurements is impossible. The contour plots of Figs.5.29 and 5.30 both include entrained flow, indicated by the contours outside the shear layers of the jets, and some mixing must take place between the entrained fluid and the jet fluid, particularly on the upstream side of the jet, which has been exposed to the cross flow. However attempts were made to quantify the jets by using a variable cut-off velocity. By discarding all
Results and Discussion

velocities below the cut-off, mean jet quantities such as mean velocity, total jet area, mean pitch angle and discharge coefficient can be estimated for a given cut-off velocity. In Fig.5.37, the calculated discharge coefficient is plotted against the cut-off velocity for the inner and outer centre primary ports. Clearly the higher $C_D$ values at lower cut-off levels are wrong, with discharge coefficients greater than 1 for the outer jet. Whatever value is correct, it can be seen from the figure that the discharge coefficient of the outer port remains substantially higher than that of the inner port at all cut-off levels. Lefebvre links the discharge coefficient to the initial jet pitch angle $\theta$:

$$\sin^2 \theta = \frac{C_D}{C_{D,n}}$$

where $C_{D,n}$ is the value of $C_D$ calculated from:

$$C_D = \frac{1.65(K-1)}{\left[4K^2 - K(2 - \alpha)^2\right]^{0.5}}$$

as $K$ tends to infinity. For the inner jet the $C_D$ value of 0.78 corresponds with a kink in the curve of Fig.5.37, which could indicate a cut-off velocity below which a significant quantity of entrained flow is included in the $C_D$ calculation and could thus be an indication of the true $C_D$ value, although this is a very subjective interpretation. For this $C_D$ value the formula above produces a pitch angle of 76.5°, which is consistent with the data presented in Fig.5.33. No such interpretation can be made from Fig.5.37 for the outer jet, however it is clear that the discharge coefficient of the outer jet is substantially higher. According to the formula above, this must result in a higher pitch angle, which is apparent in the pitch angle distribution of Fig.5.34, with pitch angles between 80° and 90° observed across the core of the jet.

Bimodal velocity histograms were obtained at a number of locations in planes IPC and OPC. These locations are marked on Fig.5.38 (inner jet) and 5.39 (outer jet). In both cases the bimodal histograms were encountered only in the jet shear layers and in the entrained fluid surrounding the jets. The majority of the bimodal histograms occurred across the upstream sides of the jets, with very few elsewhere, and at most points only bimodal v component histograms were found, although bimodal histograms were found in both the u and v components at some points. Few bimodal histograms were observed in the shear layers on the downstream sides of the jets. This shows that the instability indicated by the bimodal histograms does not occur throughout the jets. It is possible that these bimodal histograms indicate the presence of shear layer vortices, as observed by
Results and Discussion

Perry et al (1993). These vortices are formed from the roll up of the shear layer, vortex rings being released periodically. No description of the measurement of such vortices by LDA has been found in the open literature, however it is considered that the movement of a vortex through the measurement volume would result in bimodal histograms in at least two velocity components. For example, if a vortex exists in the x-y plane and moves through the measurement volume, the histograms would reveal disturbances in samples of both the u and v components. It is also considered that bimodal histograms would be observed around the complete periphery of the jet. A typical bimodal histogram is shown in Fig. 5.40, taken at a point in plane OPC that is level with plane I37 and on the jet centre line. The two peaks of the histogram occur approximately at -32 m/s and -12 m/s, indicating a fluctuation between high velocity jet fluid and lower velocity fluid that may be either entrained fluid or jet shear layer fluid. This suggests that the streamlines on the upstream side of the jet are moving axially through the measurement volume so that samples of high velocity jet fluid and entrained or shear layer fluid are taken alternately.

This movement could be caused by the separation that occurs at the leading edge of the chute (Fig. 5.36). Because the chute is radiused the separation point is not fixed and may be unstable. It is likely that a vortex exists in the separated region inside the chute (Spencer, 1998), and that this vortex will tend to draw fluid from the upstream side of the jet. The instability of the separation point will cause this vortex to be unstable, thus causing the instability in the streamlines on the upstream side of the jet. The fluid on the downstream sides of the jet was attached inside the chute and the separation point was fixed at the chute exit and is therefore stable. Hence the instability only occurs on the upstream side of the jet.

Distinct differences between the two centre primary jets have been found, which result in the inner primary jet emerging from its chute at a shallower angle to the liner than the outer jet. The source of these differences is in the manner in which the two jets are fed, in particular the area of the feed annulus from which each jet extracts air. This appears to be caused by differences in the proportions of the flow through each annulus that are extracted by the primary and secondary ports and the bleed system, causing the opposed ports to draw air from different parts of the annulus. This is strong evidence of the coupling between the external and internal flow fields, and shows the importance of modelling the external and internal aerodynamics together. Also the jet profiles observed
here do not necessarily explain the jet behaviour that was observed in section 5.2.1.1 above. Both jets emerge from their chutes with pitch angles that point the jets in a downstream direction and they should therefore meet at some point. It has been shown that they do not, and further investigations were conducted to explain this, the results of which are discussed in section 5.2.3.

5.2.2.2 Sector-edge primary jets

Mean radial velocity component contours at the inner and outer sector-edge primary port exit planes (IPE and OPE) are shown in Figs.5.41 and 5.42, with superimposed vectors showing the u and w components. Both display the axially distorted profile that was found in the centre primary jets, however the distortion of the outer sector-edge primary jet profile is similar to that of the inner jet. While both exhibit less distortion than the inner centre primary jet profile (Fig.5.29) they are still distorted more than the outer centre primary jet profile of Fig.5.30. Fluid with a velocity greater than 30% of the inlet mean velocity occupies 75% of the port area in the inner jet profile and 78% of the port area in the outer jet profile, the equivalent figures for the inner and outer centre jets being 60.5% and 94%. The jets are separating from the leading edges of the chutes and entering the flame tube through the rear of the ports, in the same manner as the centre primary jets. A deflection downstream of the chute exits, as seen in the inner centre primary jet, is apparent in both jets, and the pitch angles of both jets (Figs.5.43 and 5.44) are generally similar, being mostly in the range 70° to 80°. It is noteworthy that neither the velocity profile (Fig.5.41) nor the pitch angle profile (Fig.5.43) of the inner jet is symmetric about the centre line of the port. The greatest asymmetry occurs in the shear layer on the upstream side of the jet. Substantial asymmetry is evident in the cross flow approaching the jet, this asymmetry being caused by the swirl in the flow, which must be the cause of the asymmetry in the jet shear layer. Substantial mixing of the jet and cross flow must occur in this region, and the velocity and pitch angle profiles are thus influenced by the swirl in the approaching cross flow. Far less asymmetry is evident in the cross flow approaching the outer jet (Fig.5.42), which exhibits much less asymmetry in its velocity (Fig.5.42) and pitch angle (Fig.5.44) profiles.

Because it is impossible to distinguish between jet fluid and entrained fluid, the cut-off method discussed in section 5.2.2.1 must be used to estimate any mean quantities.
Results and Discussion

Fig. 5.45 shows the mean jet pitch angles, calculated using this method, plotted against the cut-off velocity for the sector-edge primary jets and, for comparison, the centre primary jets. At low cut-off velocities the pitch angles are reduced, because entrained fluid, which has a lower radial velocity, is included in the calculation. At higher cut-off velocities only fluid with a high radial velocity component is included in the calculation, so points that correspond to these higher cut-off velocities can be said to represent the jet fluid. It can be seen that the discrepancy between the pitch angles of the sector-edge primary jets is small compared with that seen in the centre primary jets. Fig. 5.46 shows that the discharge coefficients of the two sector-edge ports are similar, which was also indicated by the similarity between the jet areas calculated above. The jet area figures also suggest that the discharge coefficients of the sector-edge ports are lower than that of the outer centre port, and higher than that of the inner centre port, which is confirmed by Fig. 5.46.

Wool tuft flow visualisation showed that the sector-edge ports in each annulus are fed in a similar manner to the centre ports, the outer ports being fed from a narrower but deeper area than the inner ports. However the exact depth of these feed areas could not be estimated accurately, and small differences may exist in the feed to the sector-edge and centre ports that were not apparent from these subjective observations. Fig. 5.47 shows the velocity profiles measured at planes XI and XO1, upstream of the sector-edge ports in the feed annuli, with the profiles upstream of the centre ports included for comparison. The profiles upstream of the two outer ports have peaks at similar velocities, but the peak in the velocity profile upstream of the sector-edge port is approximately 2mm further from the wall and velocities across most of the annulus are higher upstream of the sector edge port. The mean velocity over the inner half of the annulus, that is the region from which the outer ports extract air, is 10% lower upstream of the centre port than the sector-edge port. The fluid entering the centre port must therefore have less axial momentum and thus a greater ability to turn towards the axis of the port. In addition, by extracting air from a region of higher velocities, the sector-edge port is likely to extract its air from a shallower region of the annulus, thus reducing the ability of the air to turn towards the axis of the port. The distortion of the jet profile at the outer sector-edge port exit plane is thus considered to be greater than that of the outer centre port because of the shallower depth of the port feed area and the higher velocity of the fluid fed to the port.
Results and Discussion

The velocity profiles in the inner annulus (Fig. 5.47) display greater discrepancies, the profile upstream of the sector-edge port having a lower peak velocity. The mean velocity over the approximate region from which the ports are fed is 5% higher upstream of the centre port than the sector-edge port. The fluid entering the inner sector-edge port must therefore be slower than that entering the inner centre port, and must have less axial momentum and thus a greater ability to turn towards the axis of the jet. This explains the reduced axial distortion of the inner sector-edge jet compared with the inner centre jet.

As discussed in section 1.3, it is to be expected that circumferential variations in the feed annulus velocity profiles should be seen. The flow pattern in the inner annulus is determined by the cowl geometry, with higher velocities downstream of the burner centre line (as observed in Fig. 5.47) because most of the flow entering the swirler is drawn from its sides.

Velocities upstream of the outer centre port are similar in magnitude to those upstream of the outer sector-edge port, although the peak in the profile is located further outboard in the sector-edge profile. The burner feed arm wake causes a 7.5% velocity deficit upstream of the centre port, while the mean velocity in the inner annulus is 16% higher upstream of the centre port than the sector-edge port. The lack of a large velocity deficit, despite the presence of the burner feed arm wake, upstream of the centre port is due to the spillage of fluid from the cowl hole into the wake behind the burner feed arm. Carrotte and Wray (1991) found that fluid spills out from the cowl hole into the low pressure wake region behind the burner feed arm, helping to reduce the velocity deficit associated with the wake.

The sector-edge primary jets display a far greater degree of similarity to each other than the centre primary jets do. This is due to the feed to the jets from the annuli, and it is a further demonstration of the influence of the feed annuli on the flow inside the flame tube. Like the centre jets, the sector-edge jets emerge from their chutes with pitch angles that suggest that they will move downstream of the primary zone. Unlike the outer centre primary jet, neither of the sector-edge jets was visible at plane 137 (Fig. 5.8). This is to be expected; the u components illustrated in Figs. 5.9 and 5.11 show that there is a cross flow with a positive axial velocity component across most of the depth of the flame tube at the circumferential location of the sector-edge ports. The behaviour of jets in cross flows is well known and these jets can be expected to be deflected in the downstream direction by
Results and Discussion

this cross flow. The results presented in section 5.2.1.2 above suggest that this is happening and that the jets impinge. The results of further investigations to confirm the trajectories of these jets are presented and discussed in section 5.2.3.

As with the centre primary jets, bimodal velocity histograms were obtained at a number of locations in planes IPE and OPE. The locations where these histograms were obtained are marked on Figs. 5.48 and 5.49. Like the centre primary ports, in both cases the majority of the bimodal histograms occurred across the top of the jets and at most points only the \( v \) component histograms were found to be bimodal. This suggests that the source of the instability indicated by these bimodal histograms is the same as the source of the instability seen in the centre jets, that is an unsteadiness in the separation from the leading edge of the chute caused by the radius on the chute inlet.

5.2.2.3 Secondary jets

Contours of mean radial velocity at the inner and outer secondary port exit planes (IS and OS) are presented in Figs. 5.50 and 5.51, with vectors representing the \( u \) and \( w \) components. Figs. 5.52 and 5.53 show contours of pitch angle.

The velocity profiles exhibit similar distortions to those seen in the primary jets, the jet fluid in both the inner (Fig. 5.50) and outer (Fig. 5.51) jets being concentrated at the back of the port, distorting the velocity profiles. As with the other ports, this suggests that the velocity profiles at the jet exit planes are strongly dependent on the flow conditions in the feed annuli. Also, a group of \( u-w \) vectors is present in the bottom left side of the outer jet (Fig. 5.51) that indicates the possible presence of a vortex. This appears to be similar to the vortices seen by Stevens and Carrotte (1990) and by Baker and McGuirk (1992). Such vortices are further evidence of the strong influence of feed annulus conditions on the velocity profile of the emerging jet.

The velocity profile of the outer jet exhibits slightly less distortion than the inner jet. Fluid with a velocity greater than 30% of the inlet mean velocity occupies 74% of the port area in the inner jet profile and 86% of the port area in the outer jet profile, the equivalent figures for the inner and outer centre primary jets being 60.5% and 94%. Port discharge coefficients were calculated in terms of a cut-off velocity, as described in section 5.2.2.1, and are plotted against cut-off velocity in Fig. 5.54. The primary jets are included for comparison. The discharge coefficients of the two secondary ports are more
Results and Discussion

comparable to each other compared with the two centre primary ports, but do not exhibit the same degree of similarity as the sector-edge primary ports. This suggests that there are closer similarities between the feed conditions of the two secondary ports than the two centre primary ports, but that significant differences still exist.

Contours of jet pitch angles show substantial differences between the two jets. Pitch angles in the inner jet (Fig. 5.52) vary between 80° and 90°, while pitch angles in the outer jet (Fig. 5.53) show a greater variation (70° to 90°) across the jet, with a large portion of fluid with pitch angles between 70° and 80° to one side of the centre line. Stevens and Carrotte found that the presence of a vortex at the jet exit plane could cause a substantial blockage, resulting a redistribution of the jet fluid. In Fig. 5.51 a vortex appears to be present on the left hand side of the jet, while vectors on the right hand side of the jet have larger axial components and significant tangential components. These vectors coincide with the region of lower pitch angles in Fig. 5.53. It appears that jet fluid may be caused to accelerate around the blockage caused by this vortex, causing the increased u and w velocity components seen on the other side of the jet.

Mean jet pitch angles were calculated using the cut-off velocity method, and are plotted against cut-off velocity in Fig. 5.55. The centre primary jets are included for comparison. There is a substantial difference between the pitch angles of the two secondary jets, although this difference is not as large as that seen between the two centre primary jets. As observed in section 5.2.2.2, the difference between the pitch angles of the two sector-edge primary jets is comparatively small.

A further means of quantification of the jets is the calculation of the axial component of momentum, that is the component of momentum in the x direction. The cut-off method was used to calculate the axial momentum using equation 3.29. Axial momentum is plotted against cut-off velocity in Fig. 5.56 for the secondary jets and, for comparison, the primary jets. The v components used in the calculation of axial momentum in the outer jets were reversed, as the v components of the outer jets are negative, so that positive axial momentum components could be calculated for all jets for easier comparison. The axial momentum of the outer secondary jet is very much greater than that of the inner secondary jet, while the inner centre primary jet has a larger axial momentum component than the outer centre primary jet. Similarities in the pitch angles and discharge coefficients of the sector-edge primary jets are reflected in the similarity
Results and Discussion

between their axial momentum components. However the momentum presented here is a function of the area of each jet, which is a function of the geometric area of each port. Because the geometric areas of the opposed ports are different, the outer secondary port having an area 1.43 times the area of the inner secondary port, it is perhaps more meaningful to compare values of axial momentum per unit effective area. The axial momentum values of Fig.5.56 were divided by the geometric areas of the respective ports and the results are presented in Fig.5.57. The difference between the two secondary jets is now similar to the difference between the two centre primary jets, with a smaller difference evident between the sector-edge primary jets. For the radial momentum component (Fig.5.58) the outer jets all have greater values than the inner jets due to the larger port sizes in the outer liner. When divided by the geometric areas of the ports (Fig.5.59) the differences between the primary jets, as observed in the axial momentum components of Fig.5.57, are reversed. The inner primary jets have greater components of axial momentum per unit geometric area than their outer counterparts. The outer secondary jet has greater axial and radial momentum per unit area than the inner secondary jet.

Fig.5.60 shows the results of wool tuft visualisation of the flow entering the secondary ports from the feed annuli. The outer secondary jet measurements were taken at the exit plane of the port that appears on the left of this figure. The outer secondary ports draw fluid from a wider area than the outer centre primary port (Fig.5.36) and from most of the depth of the feed annulus. Each port is also fed asymmetrically. Fluid is drawn from the area in between ports downstream of the centre primary port, while some flow at the edge of the sector passes the secondary ports. This must be caused by the upstream primary ports. More flow is captured by the centre primary port than by each sector-edge primary port, so there must be more local diffusion caused by the centre primary port and the velocity of the flow downstream of the centre primary port must be lower. This must cause the asymmetry seen in the vectors of Fig.5.51 and pitch angle contours of Fig.5.53, where the fluid on the right hand side of the jet has a lower pitch angle and higher u and w velocity components. The flow entering the right hand side of the port was observed to have a larger axial component, which corresponds with the region of higher u components seen on the right hand side of Fig.5.51 and the region of lower pitch angles in Fig.5.53.

The flow entering from the other side of the port was observed to have a greater tangential
component. This must cause a tangential movement of fluid from left to right, which corresponds with the direction of the $w$ components seen in Fig. 5.51.

Some fluid was observed entering the outer secondary ports from downstream. Baker and McGuirk attributed the formation of a vortex in a port to a breakdown of the flow at the rear of the port when the proportion of the annulus flow that passes the port is low, allowing air to enter the port from the sides of the port or from downstream of the port. Flow entering from either side of the port impinges, resulting in the formation of the vortex. The combination of larger secondary ports and lower bleed flow rates in the outer annulus results in a much higher proportion of the flow entering the secondary ports, compared with that passing downstream, relative to the conditions in the inner annulus. For example, 48% of the flow in the outer annulus approaching the secondary ports is estimated to enter the ports (see Table 5.2), while only 31% of the flow approaching the inner secondary ports is estimated to enter those ports. While the flow rates approaching the secondary ports in both annuli are similar (estimated to be 26.3% and 29.1% of the inlet flow rate in the outer and inner annuli respectively), the outer annulus has a 32% greater area at this plane. As a result the mean velocity of the flow approaching the inner secondary ports is estimated to be 46% higher than that approaching the outer secondary ports. The flow in the outer annulus thus has less axial momentum and can turn towards the port more easily. More flow can therefore enter the outer secondary ports from their sides. Hence vortex formation is more likely to occur in the outer secondary port than in the inner port, and a vortex is observed in the flow exiting the outer port but not the inner port.

The inner secondary ports appear to draw fluid symmetrically. The contours and vectors of Fig. 5.50 show that the flow emerging from the port is reasonably symmetrical, which is consistent with this observation of the flow into the port. The flow is also captured from a greater depth than the inner centre primary port. This, in addition to the effect of the lower approach velocity discussed above, explains the reduced distortion of the velocity profile and higher discharge coefficient.

As with the other jets, bimodal velocity histograms were found at both planes IS and OS. Their locations are marked in Figs. 5.61 and 5.62. Again, most of the bimodal histograms occur on the upstream sides of the jets and at most points only the $v$ component histograms were found to be bimodal. This indicates that the same instability
Results and Discussion

occurs in the flow entering the secondary ports as occurred in the primary ports. While very few of the bimodal histograms found in the inner secondary chute exit plane (Fig.5.61) occurred around the sides of the jet, more were seen around the sides of the outer secondary jet (Fig.5.62). These did not occur symmetrically, more appearing on the right of the figure than on the left. A comparison with Fig.5.51 shows that most of these points were outside the jet, in the cross flow. The cross flow approaching this jet has a greater swirl component than that approaching the inner jet (Fig.5.50) and the flow around the sides of the outer jet is clearly asymmetric. The source of the instability indicated at the sides of the outer secondary jet must therefore lie in the interaction between the cross flow and the jet.

Like the primary jets, the secondary jets emerge from their chutes with a velocity profile that is strongly influenced by the flow conditions in the feed annuli, providing further evidence of the importance of the coupling between the external and internal aerodynamics. The proportion of the approaching flow that is supplied to the port is dictated by the port size and the bleed flow rate. These factors thus dictate the area of the annulus from which air is extracted to supply the port, and thus affect both the discharge coefficient of the port and the pitch angle of the emerging jet. The velocity of the air in the annulus, which is influenced by the area of the annulus and thus sensitive to the difference in radius between the inner and outer annuli, affects the ability of the air to turn towards the port and thus also affects the discharge coefficient of the port and the pitch angle of the emerging jet. Because the diameters of the opposing ports, and the annulus areas and bleed flow rates, differ substantially, the two opposed secondary jets that have been measured exhibit significant differences. Differences in the pitch angles and axial momentum components of the secondary jets (factors that determine the initial trajectories of the jets) are comparable to those exhibited between the centre primary jets. However it has been shown that, like the sector-edge primary jets, the secondary jets impinge in the manner that is expected of opposed jets in a cross flow, despite their differences. While feed conditions determine the initial state of the secondary and sector-edge primary jets, their further development must be determined by flow conditions in the flame tube.

Like the secondary jets, each centre primary jet emerges from the port with an initial downstream trajectory, but the centre primary jets develop in a very different
Results and Discussion

manner. The evidence presented in section 5.2.1 showed that the outer centre primary jet is transported upstream into the primary zone, while the inner centre primary jet is deflected downstream. Because both jets emerge from their ports with positive axial momentum, the mechanism that causes the outer jet to be transported upstream must overcome this momentum. This suggests that the lower initial axial momentum of the outer jet is the factor that dictates that this jet, and not its inner counterpart, is transported upstream. The source of this different flow development must lie in the different axial momentum components exhibited by the two jets at their chute exit planes, these differences being caused by different feed conditions. However, these differences are similar to those seen in the secondary jets. If the development of the jets is influenced by feed conditions alone, then the centre primary jets and the secondary jets must be expected to develop in a similar manner. Some mechanism must therefore exist that makes the development of these jets downstream of the chute exit planes sensitive to the differences caused by different feed conditions.

The only physical feature that is thought may have such an effect is the swirler. A key feature of the flow induced by a swirler is the recirculating flow inside the conical swirling jet. It is proposed that the centre primary jets are exposed to the low pressure in this recirculation and that this low pressure must act to draw the jets upstream, into the primary zone. To do so, the low pressure must overcome the axial momentum of the jets. Because the centre primary jets have significantly different axial momentum components (Figs. 5.56 and 5.57), the jet with the lower axial momentum will be drawn upstream more readily than the other jet. If the jet that is drawn upstream first has sufficient radial momentum to penetrate across the whole depth of the flame tube, then the other jet will no longer be exposed to a low pressure upstream and will now be exposed to a pressure field due to the jet that is drawn upstream, which will act on the other jet to force it downstream.

This is a difficult hypothesis to prove, because a pressure probe must be used to obtain measurements of the pressure field to which the jets are exposed, and reliable pressure measurements cannot be obtained in the primary zone. This is due to the very high turbulence levels, the reversing flow, and the fact that the presence of any pressure probe in such a flow would inevitably influence the flow field and thus the pressure that it is intended to measure. However clarification of the jet trajectories will allow an estimate
to be made of the pressure drop that is necessary. Further measurements of the trajectories of the centre and sector-edge primary jets, and the secondary jets, were made and their results are discussed in section 5.2.3 below.

5.2.3 Jet trajectory measurements

A number of radial traverses were conducted to provide more detailed information on the trajectories of three pairs of opposed jets. Measurements were conducted at the 0°, -3.75° and -7.5° circumferential locations. Although these are not the exact locations of the port centre lines, the results presented in section 5.2.1 indicated that they would be sufficiently close to the centre of the jets to produce useful results. The choice of these locations proved advantageous because data was already available at planes I22, I37, I59, I80, I100 and I120.

The circumferential locations chosen do not coincide with the locations of the jets whose exit plane measurements are presented in section 5.2.2. The 0° traverses produced measurements of the same centre primary jets, however the sector-edge primary jets that were measured at -7.5° and the secondary jets that were measured at -3.75° are on the opposite side of the sector to those presented above. However the results presented in section 5.2.1 suggest that sufficient symmetry exists in the flow field for the results presented here to be applicable to jets on both sides of the sector.

The radial traverses were conducted at planes separated by 5mm, as shown in Fig.4.19, with a 10mm radial separation between points, to obtain data at axial locations in between those measured previously. This provided detailed information on the trajectories of the jets, presented in Figs.5.63, 5.64 and 5.65. Locations where bimodal velocity histograms were encountered are marked. To complete the plots the velocities at the chute exit planes were added. For the reasons given above, the chute exit velocity vectors used in the sector-edge primary and secondary jet trajectory plots (Figs.5.64 and 5.65) were measured in jets on the opposite side of the sector, but it was felt that their use would be valid because of the degree of symmetry that has been seen in the flow field.

The trajectories of Figs.5.63 to 5.65 indicate that the chute exit velocity vectors used in all three plots are consistent with the other internal measurements presented here. The new measurements also appear to be consistent with the measurements obtained previously at
Results and Discussion

the initial axial measurement planes (see section 5.2.1), indicating that these measurements are repeatable.

The streamlines presented here were calculated by the 'Tecplot' plotting software using only the u and v components and therefore treat the flow as two-dimensional. Obviously the internal flow field is highly three-dimensional, but the w components in the jets are sufficiently small, compared with the u and v components, for the streamlines to give a reasonable indication of the jet trajectory. To confirm this, the w components are presented in Figs.5.63(b), 5.64(b) and 5.65(b), normalised by the resultant of the u and v components at each point, i.e. the normalised w component = \( \frac{w}{\sqrt{u^2 + v^2}} \). The w components are small in the core regions of the jets, which determine their trajectories. This confirms the assumption that the two-dimensional representation of the jet trajectory is valid.

The centre primary jet trajectories illustrated by Fig.5.63(a) confirm the observation of section 5.2.1 that the outer centre primary jet is deflected upstream into the primary zone from an initial shallow downstream trajectory. The streamlines show that the jet retains the trajectory with which it emerged from the chute for a short distance, approximately one port diameter, before being gradually deflected upstream. The curvature of the streamlines gradually increases, reaching a maximum at approximately mid-height. The absence of any significant impingement between the two jets indicates that this deflection is caused by some mechanism other than the interaction of the two jets. The outer centre primary jet's streamlines are deflected in the downstream direction as it nears the inner liner. A significant cross flow is thought to exist upstream of this region (see Fig.5.11) due to the flow field generated by the swirler, which must be the cause of this downstream deflection.

The initial development of the inner centre primary jet appears not to be affected by the outer jet. Like the outer jet, its initial trajectory is determined by the pitch angle of the flow emerging from the chute (see section 5.2.2.1) and is very similar to that of the inner sector-edge jet of Fig.5.64(a) up to the central region of the flame tube, where the sector-edge jets impinge and the centre jets do not. The streamlines suggest that the inner centre jet is deflected in the downstream direction as it passes the centre of the flame tube, presumably as it encounters an increase in stagnation pressure on its upstream side due to the presence of the outer jet. This indicates that there is a very weak interaction between
the inner jet and the lee side of the outer jet. In section 5.2.1.2 it was speculated that fluid on the centre line in the inner half of the sector at plane 159, shown by Figs.5.19 and 5.20 to be moving radially outwards and downstream, was the inner centre jet, which was thought to have been deflected downwards by the outer jet. Fig.5.63(c) shows that this is not the case and the flow seen in this region of plane 159 is entrained fluid in the wake of the jet. The flow moving radially outwards in the outer central region of plane 159 (Fig.5.21) can now be attributed to fluid from the inner centre primary jet.

The streamlines suggest that a recirculation exists between the two jets in the inner half of the flame tube. This is questionable, the plotting package having constructed this from u and v velocity components in a region in which the w component becomes more significant (see Fig.5.63(b)), however the nature of the flow field surrounding the recirculation suggests that it is possible. It is in a region where opposing streamlines pass close to each other, and some streamlines from the outer jet appear to be turned by the crossflow such that they could contribute to a recirculation.

Bimodal velocity histograms were found at the locations marked on Fig.5.63(a). They were encountered at locations associated with the interaction between the two jets, that is in the recirculation, in between the jets and in the deflected portion of the inner jet. Situated on the lee side of the jets, these bimodal histograms indicate a different source of instability from those observed on the upstream sides of the jets in the measurements at the chute exit planes (see section 5.2.2). The absence of bimodal histograms from the upstream sides of the jets further inside the flame tube indicates that the instability observed at the chute exit planes is not a feature of the internal flow, the instability having been broken down by turbulent mixing. Only v component histograms were found to be bimodal. This indicates that an unstable interaction between the opposed jets causes radial instabilities to occur at these locations. This suggests that the location of the recirculation is not steady but fluctuates radially, and that the interaction between the two jets, whereby the inner jet is deflected downstream, is not a stable process.

The sector-edge primary jet trajectories of Fig.5.64(a) confirm that these jets behave in the manner expected of a pair of opposed jets in a cross flow, impinging close to mid-height with most of the jet fluid moving downstream. As indicated by the results presented in sections 5.2.1.1 and 5.2.1.2, the sector-edge jets' trajectories take them downstream to impinge between planes 137 and 159. The pitch angles throughout both jets...
Results and Discussion

appear to be largely determined by the jets' initial pitch angles, with little deflection due to the cross flow, because the ratio of the jet and cross flow velocities is large. The trajectories of these jets are therefore principally determined by the flow in the feed annuli, which determines the initial jet pitch angles.

After impingement, most of the jet fluid moves downstream and spreads out towards the liners. The relatively small w components seen in Fig.5.64(b) and in the vectors at plane 180 (Fig.5.64(c)) show that the jet fluid is not subject to any significant circumferential deflection after impingement, and this jet fluid was still discernible at plane 1100 (see section 5.2.1.3). This radial spreading of the jet fluid occurs at the axial location of the secondary jets, and it is considered that the presence of the secondary jets prevents the sector-edge primary jet fluid from moving circumferentially and may constrict it, forcing it to spread radially outwards. An outwards deflection of the flow can be seen immediately downstream of the impingement point. A small back flow exists, with an inwards deflection, which appears to be quickly deflected towards a downstream trajectory by the cross flow. A vortex may be formed at this point, however the data density is too coarse to confirm this. The radial deflection of the jet fluid after impingement is a result of the difference between the pitch angles of the two jets. As shown by Fig.5.45, the outer jet has a greater pitch angle than the inner jet. A similar result was achieved by Sivasgaram and Whitelaw (1988), who conducted experiments on unequal opposed jets with no cross flow, with a back wall close to the jets to ensure that the impinged jet fluid moved predominantly in a single direction. Their jets impinged with most of the fluid directed in the downstream direction and a small back flow moving towards the back wall. With one jet introduced normal to the wall, and its opponent inclined by 15% from the normal in the downstream direction, the fluid moving downstream after impingement was deflected radially away from the inclined jet and the back flow was deflected radially towards the inclined jet. The deflection angle of the fluid moving downstream was 35°, much greater than that observed here, however the difference between the pitch angles of the two jets (15°) is much greater than the difference between the pitch angles of the sector-edge primary jets of approximately 3°. Similar deflections can be observed in the impinged secondary jet fluid (Fig.5.65(a)). In this case the pitch angle of the outer jet is approximately 5° less than that of the inner jet (see Fig.5.55), resulting in an inwards deflection of the fluid moving downstream and an
outwards deflection of the back flow. The larger difference in pitch angles results in a greater deflection angle. As discussed in section 5.2.2, the different pitch angles observed in pairs of opposed jets result from different flow conditions in the feed annuli. While the secondary and sector-edge primary jets do not display the same sensitivity to differing feed conditions as the centre primary jets, the development of the jet flow after impingement thus exhibits sensitivity to feed conditions in the form of a radial deflection that appears to be dependent on the relative initial pitch angles of the opposed jets.

Locations where bimodal velocity histograms were encountered are marked on Fig.5.64(a). One was found on the upstream side of each jet, close to the chute exit, with another in the entrained fluid on the upstream side of the inner jet. These locations correspond with those found at the jet exit planes (see section 5.2.2.2). The absence of bimodal histograms from the upstream sides of the jets further inside the flame tube indicates that the instability observed at the chute exit planes is broken down by turbulent mixing. Most of the bimodal histograms were found close to the impingement point, and in the jet fluid downstream of this point. As with the other jets this indicates that the impingement process is unstable. Only $v$ component histograms were found to be bimodal, due to the dominance of the radial momentum of the jets on the flow in this region. This suggests that the instability is a radial fluctuation of the impingement point, which causes radial instabilities in the jet fluid downstream of the impingement. Bimodal histograms were also found in the entrained flow downstream of the jets, suggesting that the entrainment process is affected by the instability in the impingement. Other workers have also found instabilities in strongly impinging jets. Perchanok et al (1989) and Quick et al (1993) both found oscillations at the impingement of opposed jets in industrial furnaces. It is therefore to be expected that the impingement of opposed jets may not be a stable process, which is further confirmed by the data presented here.

Similar behaviour is exhibited by the secondary jets whose trajectories are shown in Fig.5.65(a). In this case there is a greater back flow, which appears to set up a double recirculation in the secondary zone upstream of the jets. This is questionable, the plotting package having constructed these vortices from $u$ and $v$ velocity components in a region in which the $w$ component becomes more significant (see Fig.5.65(b)). It should be noted though that other workers (e.g. Spencer, 1998) have observed similar recirculations upstream of jet impingements. Prior to impingement very little curvature of the jet
Results and Discussion

Streamlines can be observed, indicating that the effect of the cross flow on the trajectories of the two jets is minimal, the trajectories being mainly determined by the initial pitch angles of the jets. As these pitch angles are dependent on the nature of the flow in the feed annuli, the trajectories of the secondary jets, as with the other jets described above, must be dependent on external flow conditions.

Unlike the sector-edge primary jets, the back flow has an outwards deflection, while the impinged jet fluid moving downstream has an inwards deflection. There also appears to be no radial spreading of the impinged jet fluid, although measurements were obtained in a much shorter distance downstream of the impingement point at this plane than at the sector-edge primary jets' plane. The deflection of the fluid after impingement is in line with the findings of Sivasgaram and Whitelaw and the sector-edge primary jets, as in this case it is the outer jet that has the greater inclination towards the downstream direction.

As discussed in section 5.2.1.2, the back flow from the impinged secondary jets spreads circumferentially towards the centre of the sector, resulting in negative u components in the vectors on the centre line at plane 180, as shown by Fig.5.63(a). The vectors of Fig.5.63(a) suggest that an impingement occurs between this back flow and the inner centre primary jet in the outer half of the sector, restricting the downstream movement of the jet and diverting it onto a radial trajectory. Downstream of the centre primary jets, at planes 1100 and 1120, Fig.5.63(a) exhibits vectors with significant positive (downstream) u components. This is thought to be fluid that has spread circumferentially inwards from the impinged secondary jets.

The vectors downstream of the two secondary jets in Fig.5.65(c) indicate that fluid has been entrained in the jet wakes. This confirms the speculation in section 5.2.1.3 that the negative u components seen in Fig.5.23 represented entrained fluid. It was also noted in section 5.2.1.3 that the v components seen in Fig.5.24 were of greater magnitude in the outer half of the sector than in the inner half, and that the impingement point appeared to be closer to the inner liner than the outer liner. It can be seen from Fig.5.65(c) that plane 1100 is closer to the outer jet than the inner jet, so that the entrained fluid measured in the outer part of the sector was much closer to the jet and therefore had greater velocities. Fig.5.65(a) also shows that the impingement point is closer to the inner liner than the outer liner, being approximately 3mm below mid-height. This was exaggerated in
Results and Discussion

Fig. 5.24 by the inwards deflection of the jet fluid after impingement, which causes the impingement point to appear even closer to the inner liner at plane I100. It is thought that the greater radial momentum component (Fig. 5.59) of the outer jet causes the impingement to occur closer to the inner liner. Since the difference in the radial momentum components is a product of the differences in the jets' feed conditions, this is further evidence of the sensitivity of the internal flow field to external flow conditions.

The locations where bimodal velocity histograms were found are also shown. As at the other planes only \( v \) component histograms were found to be bimodal at these points. In common with the sector-edge primary jets, these bimodal histograms were mostly found around the impingement points. This suggests that, as with the sector-edge primary jets, the impingement is an unstable process, with the location of the impingement point fluctuating radially.

5.2.3.1 Analysis of outer centre primary jet trajectory

The hypothesis that was introduced in section 5.2.2.3 to explain the deflection of the outer centre primary jet is difficult to prove. It was proposed that the centre primary jets are exposed to the low pressure in the recirculation downstream of the swirler and that this low pressure must act to draw the jets upstream, into the primary zone. However, to do so the low pressure must overcome the downstream axial momentum of the jets. Because the centre primary jets have significantly different axial momentum components, the jet with the lower axial momentum will be drawn upstream more readily than the other jet. If the jet that is drawn upstream first has sufficient radial momentum to penetrate across the whole depth of the flame tube, then the other jet will no longer be exposed to a low pressure upstream and will now be exposed to a pressure field due to the jet that is drawn upstream. This will act to force the jet downstream. Hence the outer centre primary jet, which has the lower axial momentum component, is drawn upstream and the inner jet is forced downstream. The measurement of the static pressure in the recirculation downstream of the swirler requires the use of a pressure probe. Pressure probes are not suited to the measurement of swirling flows as the flow is sensitive to perturbations caused by the presence of the probe (Gouldin et al., 1985). Hughes (1999) used both LDA and five hole probe measurement techniques in the flow field downstream of a large scale swirler model and found that, when measurements were made at an axial location 0.1
swirler diameters downstream of the swirler exit plane, the presence of the pressure probe caused the start of the recirculation to be moved downstream of the probe. The LDA measured negative axial velocities at the centre line at locations where the five hole probe measured positive velocities. Pressure probes are also highly sensitive to the direction of the flow, which is extremely three-dimensional in this case.

However it is possible to broadly estimate the pressure drop that is necessary to deflect the jet. The jet streamlines of Fig.5.63(a) clearly show that the axial momentum component of the outer centre primary jet at the chute exit plane causes it to move downstream initially, but it is turned upstream before it reaches the centre of the flame tube. By calculating the radius of curvature at every point on the streamline, the pressure gradient that must exist to provide the centripetal acceleration of the fluid can be estimated. This can then be used, in conjunction with an estimation of the pressure drop in the recirculation created by the swirler, to confirm that the swirler creates a pressure gradient of sufficient magnitude to cause the upstream deflection of the jet.

This pressure force acting on the jet fluid, as shown by Fig.5.66, is a function of the radius and the velocity component normal to the radius of curvature of the streamline (i.e. tangential to the streamline), and the rate of change of static pressure with radius is given by the formula:

\[
\frac{dp}{dR} = \frac{\rho V^2}{R}
\]

where \( V \) is the velocity component tangential to the streamline and \( R \) is the local radius of curvature of the streamline. So if the radius of curvature of each streamline and the tangential velocity component at each point are known, the pressure differential across the jet at each point can be estimated by assuming a suitable value of the increase in radius across the jet, \( dR \).

The streamlines of Fig.5.63 were calculated by the plotting package from a sufficient number of vectors for their trajectories to be regarded as reliable. The outer centre primary jet streamlines were then exported to a spreadsheet and plotted individually in the form shown in Fig.5.67. The example shown here was taken from the streamline at the centre of the outer centre primary jet shown in Fig.5.63. The data points marked in Fig.5.67 are interpolated points calculated by the plotting package to define the streamline, and thus do not define the streamline with absolute accuracy. As shown by Fig.5.63, the plotting package used over thirty velocity vectors, throughout the jet, to
calculate the streamlines. Comparison of the calculated streamlines and the measured velocity vectors suggests that these streamlines are reasonably accurate. The interpolated data points of Fig.5.67 are thus considered to be sufficiently accurate for the estimation of the magnitude of the pressure gradient across the jet. The curve fitting facility of the spreadsheet package was used to fit a sixth order (the maximum available) polynomial to the interpolated data set. The polynomial curve's equation is shown in the figure with the correlation coefficient (R² value) of 0.9997, which indicates that the curve fits the interpolated data with reasonable accuracy.

The polynomial equation that describes the trajectory of the streamline was then used to obtain the radius of curvature throughout the streamline, by using the following formula, obtained from Stroud (1987):

\[ R = \left[ 1 + \left( \frac{dy}{dx} \right)^2 \right]^{\frac{1}{2}} \]

The radius of curvature was thus calculated throughout the jet using the approximate trajectory equations calculated for each streamline. The radius of curvature is plotted alongside the streamline of Fig.5.67 in Fig.5.68. The radius of curvature tends to infinity where the streamline is flat, as is to be expected, and reduces to a minimum value of 23mm where the curvature of the streamline is greatest.

A dR value, representing the increase in radius of curvature across the jet, must be chosen. The depth of the jet at its chute exit plane (Fig.5.30) is 10mm. The streamlines of Fig.5.63 indicate that this is approximately equal to the depth of the jet at its maximum curvature. Thus 10mm was chosen as the dR value to be used to obtain a broad estimation of the pressure differential across the jet. The estimated static pressure differential is plotted alongside the streamline in Fig.5.69. Although the velocity decays along the streamline, the decrease in the radius of curvature has a much greater effect and is dominant in the determination of the pressure differential. The maximum pressure differential therefore occurs at the same location as the minimum radius of curvature. This estimation procedure was repeated for all streamlines in the jet and the maximum pressure differential was found to be the 124N/m², or approximately 12.6mm H₂O, found in the streamline illustrated here. This result is sensitive to the choice of the dR value. For example, the use of a dR value of 14.5mm, based on the diameter of the port (which the
streamlines of Fig. 5.63 indicate is greater than the increase in radius of curvature across
the jet), would yield a maximum pressure differential of 18.3 mm H₂O. For comparison,
the pressure drop across the outer liner, based on the dynamic pressure in the core of the
outer centre primary jet at the chute exit plane, is estimated to be 100 mm H₂O. Thus the
estimated pressure differential across the jet, that is the pressure differential that causes
the jet to be drawn upstream, is small compared with the pressure drop across the liner.
This is significant because the pressure drop across the swirler, and thus the pressure drop
inside the recirculation, may be expected to be of a similar magnitude.

While it is known that a low static pressure must be present in the recirculation
created by the swirler, the data available here is not sufficient to calculate its magnitude.
However an estimation of the magnitude of the pressure gradient that exists at the
swirler's exit can be made. Within the swirler, the fluid is made to rotate by angled vanes
(Fig. 5.70). A radial pressure gradient must therefore exist to balance the centrifugal force
due to the rotation of the fluid (Chigier and Beer, 1964). This pressure gradient can be
obtained from the equation
\[
\frac{dp}{dr} = \frac{\rho w^2}{r}
\]
so if the radial distribution of the tangential velocity component, \( w \), is known this
equation can be integrated with respect to the swirler radius, \( r \), to obtain the pressure
differential between the hub of the swirler and its shroud. However the radial distribution
of the tangential velocity component is not known, but it can be approximated at the exit
plane of the swirl vanes if it is assumed that a uniform axial flow enters the swirler. The
condition of the flow entering the swirler is also unknown, and the results of Barker and
Carrote (1997b) show that it is not uniform, however this method is thought sufficient to
provide a good approximation of the magnitude of the pressure differential that must
exist.

In each of the three swirler passages angled vanes impart swirl to the flow. These
vanes all have constant exit angles across the radius and the velocity distribution created
by the vanes can be assumed to be similar to that created by constant angle nozzle blades
in a turbine. The velocity distribution in constant nozzle angle flow (Cohen et al, 1987) is
given by
\[
w_r \sin^2 \alpha = c
\]
Results and Discussion

In this case \( r \) is the radial distance from the centre of the swirler, \( \alpha \) is the vane exit angle and \( c \) is a constant. Because \( \alpha \) is constant, the tangential velocity component \( (w) \) is directly proportional to the axial component \( (u) \) and, since \( w = u \tan \alpha \),

\[
\frac{u \tan (\alpha) r \sin \alpha}{\alpha} = c
\]

Since 11% of the fluid entering the flame tube enters through the swirler, the mean axial velocity entering each swirl vane row was calculated to be 12.9 m/s. Because the exit area of each vane row is the same as the inlet area, this also corresponds to the mean axial velocity at the exit of each vane row. Due to the growth of the boundary layers on the vanes and the passage walls the effective area must be reduced, but because this estimation is only intended to produce an approximation of the magnitude of the pressure differential this has been neglected.

If the mean velocity at the vane exit plane is known then the value of the constant \( c \) can be calculated. The mean velocity of 12.9 m/s must equal the area-weighted average velocity across the passage. Therefore

\[
\frac{2\pi}{A} \int_{r_{in}}^{r_{out}} u \, dr = 12.9
\]

The vane angle in the inner air swirler is 60°, so the radial distributions of the axial \( (u) \) and tangential \( (w) \) velocity components are:

\[
w = cr^{-0.75}
\]

and

\[
u = \frac{cr^{-0.75}}{\sqrt{3}}
\]

Thus the mean axial velocity at the vane exit plane was calculated to be 28.71 cm/s, which equates to the mean velocity of 12.9 m/s, so the \( c \) value of the inner swirler is 0.449. Now the radial distribution of \( w \) is known and the pressure differential between the hub and shroud of the inner swirler passage can be calculated:

\[
\Delta p = \int_{r_{in}}^{r_{out}} \frac{\rho w^2}{r} \, dr = 0.449^2 \times 1.225 \int_{r_{in}}^{r_{out}} \frac{r^{-1.5}}{r} \, dr = 1059 \text{N/m}^2
\]

The equivalent values for the outer and dome swirlers, whose vane angles are 40° and 43° respectively, were calculated to be 24.7 N/m² and 34.5 N/m² respectively. These values are much less than that calculated for the inner swirler because the vane angles and radius ratios of these swirler passages are smaller. The total pressure differential between the hub of the inner swirler and the shroud of the dome swirler is therefore 1118.2 N/m², or 114 mm H₂O.
Results and Discussion

This pressure differential is equivalent to the pressure difference that would exist between the fluid at the centre and outer edge of the swirler were the swirler exit plane to be situated at the vane exit plane. As can be seen in Fig. 5.70, the swirler passages contract substantially before the exit plane, and the radii of the outer and dome passages reduces substantially. Because the axial flux of tangential momentum must be conserved, this reduction in radius will cause the tangential velocities in the outer and dome passages to be increased. It can therefore be expected that their contributions to the pressure differential will also be increased.

The pressure differential that has been calculated above is therefore a reasonable, but conservative, estimate of the pressure difference that must exist between the swirler centre and outer edge, at the swirler exit plane. Downstream of this plane the fluid is no longer constrained and the swirling jet flow moves radially outwards to form a cone with a recirculation at the centre. Thus the equilibrium between the pressure force and centrifugal force no longer exists, and the pressure differential decreases away from the swirler (Chigier and Beer, 1964). The static pressure must increase between the swirler and the plane of the primary ports, thus affecting the pressure differential across the jet, but the magnitude of this increase is not known. However the above estimation indicated that the pressure differential at the swirler exit plane is much greater than that required to draw the outer centre primary jet upstream, and the pressure differential that exists at the plane of the primary ports due to the recirculation induced by the swirler may be expected to be of at least the same order of magnitude as that required to cause the deflection of the jet. Thus the swirler is capable of producing the pressure differential that is necessary to cause the upstream deflection of the outer centre primary jet. The experimental results are insufficient to provide a conclusive proof that this is the cause of the deflection, however it has been shown that it is feasible.

The measurements of the jets at the chute exit planes (see section 5.2.2) revealed that similar momentum differences exist between the opposed secondary ports and the opposed centre primary ports. These results highlighted the influence of the flow conditions in the feed annuli on the internal flow field, however the development of the centre primary jets has been shown to differ substantially from the development of the secondary jets. While the secondary jets exhibit some sensitivity to their differing feed conditions, their feed conditions determine only the pitch angles of the jets and the
Results and Discussion

direction of the jet fluid after impingement. The sensitivity of the centre primary jets has been substantially enhanced, resulting in a flow pattern in the primary zone that differs completely from that which was expected. The influence of the swirler is crucial in determining the primary zone flow pattern. It could be argued that the differences in the feed annulus flow conditions are a result of the distortion of the flame tube, which caused the annulus depths to be substantially altered, and that the unusual primary jet behaviour that has been seen here may not occur in an engine. However the CFD predictions of Spencer (1998) provide further evidence of the importance of the feed conditions and the influence of the swirler, and show that this result is not merely an anomaly caused by the altered feed annulus flow fields.

Spencer modelled a sector of a Phase 5 annular combustor from feed annulus entry to combustor exit. This is a similar geometry to that used in this project, but the flame tube employs conventional cooling rings instead of effusion cooling and the annulus geometry is different. Therefore it can be expected that the feed to the primary and secondary ports will be different. Experimentally determined velocity profiles were used as inlet velocity profiles at the annulus entry plane and a swirler exit velocity profile was also used as an inlet boundary condition. Flame tubes with two different port types were modelled - plain holes and chuted ports. Details of the turbulence model and mesh can be found in Spencer (1998). The results are reproduced in Figs. 5.71 and 5.72.

In the CFD prediction for the flame tube with plain ports (Fig. 5.71) the jets do not impinge. Comparison with Fig. 5.72 shows that the use of chutes increases the initial pitch angles of the jets, thus increasing their radial momentum components and penetration depth. The outer jet is deflected into the primary zone, the deflection occurring at a radial location that is directly downstream of the swirler. The opposing jets do not impinge, and the inner jet moves downstream. Significant w components, not shown in the figure, show that the inner jet also moves out of this plane, while the outer jet remains in the plane. An asymmetric primary zone flow field is generated. As in the experiment reported in this thesis, there is no apparent mechanism by which this deflection could take place other than the suction caused by the swirler.

The swirler has a similar effect on the flow field in the flame tube with chuted ports (Fig. 5.72). In this case the opposing jets penetrate further, and with greater pitch angles, and impinge. However the outer jet is clearly subject to some deflection towards
Results and Discussion

the primary zone, although to a lesser degree than that seen experimentally in this project, and the primary zone flow field is again asymmetric. The outer jet vectors indicate that the deflection occurs prior to impingement and the swirler appears to be the cause. In addition, streamlines in the feed annuli indicate differences between the feed conditions of the two jets. In both cases, the inner port draws air from a shallower depth than the outer port. As a result the initial pitch angle of the outer jet is greater than that of the inner jet. These CFD results also demonstrate that the effect seen here is not a result of the distortion of the flame tube, and that it is not unique to the geometry of this test facility.

The mechanism by which the outer centre primary jet is deflected explains the structure of the jet cross section observed in the primary zone (see section 5.2.1). In Fig.5.11 the cross section of the jet at plane I22 appears to have the kidney shape that is a classic feature of a jet in cross flow, while the vectors of Fig.5.12 suggest that a contrarotating vortex pair is present in the lobes of the kidney-shaped jet. These features would normally be present on the lee side of a jet that has been deflected by a cross flow. The cross flow causes the deflection of low momentum fluid in the jet's shear layer around the side of the jet to form the kidney shape. The shear layer contains streamwise vorticity and its roll-up leads to the formation of the contrarotating vortices.

In this case the jet has not been deflected downstream by a cross flow but upstream due to a low pressure. The lobes of the kidney-shaped jet and the apparent vortices are situated on the upstream side of the jet. The low pressure that has caused the deflection of the jet must be expected to cause the low-momentum fluid in the shear layer to be deflected more rapidly upstream than the core fluid, in the same way that a cross flow causes the shear layer fluid to be deflected towards the lee side of a jet. The kidney-shaped cross section of the jet would therefore be reversed, with the lobes situated on the upstream side as in Fig.5.11. The formation of the contrarotating vortices would be similarly affected, with vortices situated on the upstream side of the jet as suggested by Fig.5.12. However, as shown by the vectors upstream of the jet in Fig.5.30, the outer centre primary jet is initially exposed to a cross-flow which originates in the fluid entering the flame tube through the swirler. As a result it may be expected that the cross flow should initially cause a conventional distortion of the jet's structure, with the contrarotating vortices situated on the lee side of the jet. The data presented here is insufficient for a complete evaluation of the vorticity present in the flow field, and this
apparent reversal of the conventional structure of the jet in cross flow suggests that further investigation would be worthwhile.

5.3 Turbulence in the internal flow field

In addition to the mean velocity flow field previously discussed, sufficient data was obtained to evaluate the six Reynolds stresses at each measurement location. For convenience a complete set of turbulence data can be found in Appendix A. The turbulence that is generated inside the combustor is of interest because good mixing of fuel, air and combustion products is dependent on the generation of high turbulence levels, although it must be noted that, as with the mean flow field, the turbulence observed here is likely to differ substantially from that observed in the combusting flow field. The generation of turbulence in the combustor is dependent on certain features of the mean flow field, for example the impingement of the secondary jets, and the unexpected behaviour of the centre primary jets may be expected to affect the turbulence levels that are observed in the primary and secondary zones. The turbulence data obtained at several planes inside the flame tube is presented and discussed in this section. As with the mean velocity measurements presented in section 5.2, the locations where bimodal histograms were observed are marked by circles where appropriate.

While both the velocity and turbulence fields at the port exit planes differ in detail between ports, the turbulence levels and patterns are similar in all six jets. Thus turbulence data at only one port exit plane, the outer centre primary port exit plane (OPC), is presented here. The turbulence intensities found at plane OPC, are presented in Fig.5.73. For comparison of turbulence levels with local mean velocity levels, the turbulence intensities presented in Fig.5.73a are normalised at each point by the total mean velocity at that point, using equation 3.24. However this is not suitable at every point as the turbulence intensity calculated in this manner must be infinite where the mean velocity is zero. Hence turbulence intensities greater than 100% are observed outside the jet in Fig.5.73a. For a better indication of the turbulence level at each point, and for comparison with other locations, the turbulent kinetic energy is presented in Fig.5.73b, normalised by the square of the mean velocity at pre-diffuser inlet (equation 3.25).
Results and Discussion

The core of the jet contains turbulence whose intensity is greater than 30% throughout. Higher turbulence intensities are present in the shear layer on the upstream side of the jet, where fluctuations in the mean velocity were indicated by bimodal velocity histograms whose widening must increase the apparent turbulence intensity. Some turbulence generation may also occur here due to the impingement of the crossflow and the jet.

The isotropy of the turbulence can be determined from the anisotropy parameters defined in equation 3.26. The anisotropy of the turbulence throughout the jet at plane OPC is clearly illustrated by the uu/k and vv/k data presented in Fig.5.74. The uu and vv normal stresses are presented in Fig.5.75. Across much of the jet, the uu normal stress is significantly greater than the vv normal stress. A large region of raised uu normal stresses occurs near the downstream edge of the port. Spencer (1998) observed raised turbulence intensities in a similar region, and attributed them to the jet's flow history. The fluid in this region has been subjected to considerable curvature, turning from an axial direction in the annulus to an almost radial direction at the port exit plane. Instability due to the strong curvature of the streamline causes the enhancement of the turbulence in the plane normal to the streamline (Bradshaw, 1973) and thus raises the uu normal stress.

The vv normal stress is greater in the shear layer on the upstream side of the jet. In addition to the raised turbulence levels that are to be expected in a free shear layer, the vv normal stress is exaggerated by the bimodal v component histograms encountered at the locations marked by circles in Figs.5.73 to 5.75. The u and v component histograms at a point in the shear layer, marked in Fig.5.73, are presented in Fig.5.76. The v component histogram is clearly bimodal, consisting of two unequal superimposed gaussian distributions, with mean velocities of approximately -32m/s and -12m/s. This bimodal histogram indicates that the flow at this point is unstable, fluctuating between these two mean values, and the inequality of the histograms suggests that the flow has a mean velocity of -32m/s for the greater proportion of the sample. However it must be noted that the frequency of the fluctuation may be too great for the histogram to be adequately resolved in the sampling time used at this point. Spectral analysis failed to reveal any distinct frequencies at this point.

To establish whether the apparent anisotropy in the shear layer is caused by the instability indicated by the bimodal histogram, the root mean square of the larger
component of the bimodal histogram was compared with the root mean square of the u component histogram. An approximation of the larger component of the bimodal histogram was constructed by removing the smaller component. To replace it and complete the approximated histogram, the mirror image of the data on the opposite side of the histogram was added, as shown by the red part of Fig.5.76. This approximate histogram thus represents a sample of the radial velocity component with the instability removed. The root mean squares of the approximated and bimodal v component histograms, and the u component histogram, were calculated to be 8.5m/s, 12m/s and 8.8m/s respectively. Thus the raised vv normal stress and the anisotropy of the turbulence in the shear layer on the upstream side of the jet appear to be caused by the widening of the v component histograms due to the bimodal flow in the shear layer.

Within the core region the high turbulence intensities, which are in excess of 30%, and the distribution of turbulence between the three normal stresses are in contrast with the measurements of Spencer (1998). In a plain primary jet, Spencer found that the turbulence intensity across the central portion of the jet was relatively low, being around 5%, and the turbulence was isotropic. Spencer’s jets were fed from a feed annulus that contained turbulence of relatively low intensity, with a turbulence intensity at inlet of 5%. However, the flow in the feed annuli of an annular combustor with a dump diffuser is known to contain turbulence with a much greater intensity. Carrotte and Wray (1991) found a turbulence intensity of 35% in the feed annulus of a combustor of similar geometry to that used in this project. Carrotte, Denman and Wray (1992) also found high turbulence intensities in feed annuli. The majority of this turbulence is generated by the unstable nature of the flow as it is deflected around the head of the flame tube. Carrotte et al observed that the high turbulence levels spread across the whole flow field in the feed annuli. The flow turning around the flame tube head was omitted from Spencer’s geometry, with the jets being supplied from a relatively long annular passage. Although the limited external flow field measurements in this project preclude any definite conclusions regarding turbulence levels in the feed annuli, it must be expected that the same turbulence generation processes were present and that the turbulence in the feed annuli is the cause of the high turbulence levels seen at the chute exit planes. Such high turbulence levels must impact on jet mixing and are a further illustration of the
Results and Discussion

importance of modelling the internal and external aerodynamics together in both experimental and computational investigations.

In addition to the port exit plane, turbulence data is also presented for three radial measurement planes. The centre, secondary and sector edge planes are at $0^\circ$, $-3.75^\circ$ and $-7.5^\circ$ respectively, as shown by Fig. 5.77. Figures 5.78, 5.79 and 5.80 present the turbulent kinetic energy, normalised by the square of the mean pre-diffuser inlet velocity, at each plane. The locations where bimodal velocity histograms were observed are marked by circles. At each plane it is the jet fluid that has the highest turbulence levels. It is also to be expected that the shear between the jets and the cross flow will cause the generation of turbulence, in addition to that already contained in the jets when they leave the ports.

At the centre plane (Fig.5.78) a small peak in turbulent energy can be seen at the location of the weak impingement between the inner primary jet and the underside of the outer jet. However at this point the v component velocity histogram, presented in Fig.5.81, is highly bimodal. The raised levels of the apparent turbulence at this point and at other locations where bimodal histograms were observed are not indications of increased turbulence levels, but are caused by the increased rms of the v component due to the widening of the histogram due to radial fluctuations in the flow.

As expected, at the secondary plane (Fig.5.79) the impingement of the secondary jets causes a substantial increase in turbulent energy. Fig.5.82 shows that there is a considerable increase in the radial (vv) normal stress due to impingement, but not in the axial (uu) normal stress, the radial normal stress being more than twice the axial normal stress. This is caused by turbulence generation in the radial direction due to the strong impingement of the two predominately radial jets, and it is a result that has been seen previously by other workers such as Spencer (1998). Bimodal v component histograms were observed at the impingement of the jets, due to the instability of the impingement process (see section 5.2.3), which will exaggerate the apparent vv normal stress, however a genuine increase in this stress exists here due to the impingement of the jets.

Spencer's data, with jets in a cross flow representing a row of impinging primary jets, showed that the turbulence was transported downstream by the impinged jet fluid and also upstream by the recirculation that was set up by the impinged jets. Fig.5.79 shows that this occurs in these secondary jets, with the back flow causing a rise in turbulent energy in a portion of the secondary zone upstream of the impingement, and a
Results and Discussion

rise in turbulent energy downstream of the impingement. If the conventional primary zone
flow field, where the jets impinge and their back flow dominates the recirculation in the
primary zone, existed here, then a significant rise in turbulent kinetic energy would be
seen in the primary zone in Fig.5.78. This isn't evident, in fact the turbulence levels in the
primary zone are low at all three radial planes. Hence it can be seen that the primary jet
trajectories observed in section 5.2 have significantly affected the turbulence levels in the
primary zone.

Figs.5.83 to 5.86 present the three normal stresses and the turbulent kinetic energy
at plane 122. Comparison of turbulent energy levels with those found elsewhere in the
flow field confirms that the turbulence levels are relatively low throughout the primary
zone. The peaks of all three normal stresses occur at a location that suggests that the
greatest turbulence at this plane is associated with the outer centre primary jet. Peaks also
occur in the axial (uu) and circumferential (ww) normal stresses at the edge of the sector.
These may be caused by the impingement of fluid from the swirler with fluid from the
swirlers in the adjacent sectors, although there is not enough information on the flow
imparted by the swirler available here to confirm this.

Figs.5.78 and 5.80 (centre and sector-edge radial planes) show peaks in turbulent
energy in line with the secondary jets, the peak in the centre plane covering a much larger
area. It has been shown previously that some of the impinged secondary jet fluid is
deflected in the circumferential direction, particularly towards the centre of the sector.
These peaks are caused by the circumferential transport of turbulence from the impinged
secondary jets. The circumferential (ww) normal stress at plane 180 (Fig.5.87) shows a
peak at the centre of the sector that corresponds with that in Fig.5.78, with high levels at
the edge of the sector. These high turbulence levels can also be seen in the turbulent
kinetic energy at plane 180 in Fig.5.88.

In contrast with the secondary jets the sector-edge primary jets do not display an
increase in turbulent kinetic energy at their impingement (see Fig.5.80) although
turbulence levels are generally higher downstream of the jets than upstream. The
impingement of these jets is clearly weaker than the impingement of the secondary jets.
This was previously indicated by the presence of only a very small back flow upstream of
their impingement.

143
Results and Discussion

Figs. 5.89 to 5.91 present the uu/k and vv/k values at the three radial planes to illustrate the anisotropy of the turbulence. While portions of the secondary plane show reasonable isotropy, elsewhere much of the flow field contains highly anisotropic turbulence. This suggests that the k-ε turbulence model may not be appropriate for computational calculations of the combustor.

While the behaviour of the jet flows in the flame tube determines the internal turbulence field, the jets themselves are affected by turbulent shear forces. Fig. 5.92 shows the development of the profile of the outer centre primary jet over the first 30mm from the port exit plane. While the fluid at the core of the jet is pressure-driven, shear stresses have a substantial effect. The profile mixes out, with the core velocity reducing as the core fluid mixes with fluid from the shear layers, and edge velocities increasing as the shear layers mix with entrained fluid. The uv and vw shear stresses at the outer centre primary port exit plane are presented in Fig. 5.93. Large uv shear stresses are observed in the shear layers on the upstream and downstream sides of the jet. The highest uv shear stresses coincide with locations where bimodal v component histograms were observed. The uv shear stress will be enhanced by the bimodal nature of the sample at these points, and the fluctuation in the flow that is indicated by these bimodal histograms will enhance the mixing rate. Large vw shear stresses are observed in the shear layers at the sides of the jet. The uv shear stress profile at the centre radial plane (Fig. 5.94) shows that these concentrations of large uv shear stresses persist up to mid-height. Profiles of the uv shear stress at the secondary and sector-edge radial planes (Figs. 5.95 and 5.96) show that large uv shear stress concentrations exist at similar locations in the secondary and sector-edge primary jets.

5.4 Closure

While the four impinging opposed jets that were measured in this project showed some sensitivity to their feed conditions, they impinged as expected although the behaviour of the jet fluid showed some sensitivity to the initial conditions of the jets. In contrast, the primary jets in line with the swirler have been shown to be highly sensitive to their feed conditions, the differences in the initial conditions of the two jets causing a gross change in behaviour whereby one jet is deflected upstream and the other is deflected downstream. This dramatically increased sensitivity has been shown to be caused by the presence of
Results and Discussion

the upstream swirler. This demonstrates the influence of the feed annuli on the internal flow field. This effect is not a product of the distortion of the flame tube that has been used here and therefore may occur in any combustor geometry where the primary jets are located such that they may be exposed to the recirculation generated by the swirler. It therefore also follows that the feed annuli and flame tube flow fields should be modelled together if experimental or computational investigations are to produce results that truly reflect the flow field in the combustor.

This modified primary jet behaviour has caused a severe modification to the flow field in the primary zone. The outer centre primary jet dominates the flow in the inner half of the primary zone. In the outer half, two small recirculations exist on either side of the centre line, one of which appears to be much weaker than the other. In addition the turbulence levels in the primary zone are substantially lower than those found elsewhere in the flow field, due to the lack of impingement between the centre primary jets. It is therefore to be expected that the mixing processes that take place in the primary zone will be affected, although it is difficult to predict the behaviour of the reacting flow field from isothermal flow field results. However Coupland and Priddin (1986) found that the flow patterns in isothermal and combusting flow fields differed little, and the potential effect on the temperature distribution at the turbine inlet, the efficiency of the combustor and the pollutant emissions of the engine should not be underestimated. It is therefore important that the combusting flow field is investigated, and the results of this investigation and any investigation of the combusting flow field should be taken into consideration when designing future combustion chambers.

Bimodal velocity histograms were observed at numerous locations throughout the internal flow field. The instability in the flow that is indicated by these bimodal histograms implies that the k-ε and Reynolds Stress Transport CFD methods are not appropriate for the prediction of the combustor flow field and that time dependent CFD methods such as Large Eddy Simulation should instead be used. The instability in the flow also potentially affects the performance of the combustor, and its effect on the combustion process and the combusting flow field should be investigated.
6. Conclusions and Recommendations for Further Work

6.1 Conclusions
A detailed investigation of the flow field in an annular combustor has been conducted. A new test facility was designed and constructed, comprising a three burner sector segment of an annular combustor that is typical of current design practice. The test facility simulated the flow field from pre-diffuser inlet to the combustor exit. A considerable effort was made to design and manufacture a facility to produce an accurate simulation of the isothermal flow field within a gas turbine combustor. Provision was also made for high quality, extensive measurements throughout the flow field. To fulfil these objectives, all major flow field features were present, including representative annulus bleed flows. A novel differential bleed system and modifications to the feed annuli were incorporated to minimise the effects of the sidewall boundary layers on the flow in the centre burner sector. The test facility was designed to allow optical access for internal and external measurements of the flow field in the centre burner sector using a three component Laser Doppler Anemometry (LDA) system. A specialised traverse system was designed to enable LDA measurements throughout the centre burner sector. New alignment and calibration techniques were developed to optimise the accuracy of the 3D LDA system.

Measurements of three orthogonal mean velocity components and all six Reynolds stresses were obtained throughout a burner sector of the combustor. A set of velocity and turbulence data has been obtained that is sufficiently extensive for use as a benchmark data set for CFD validation. In addition the following conclusions have been drawn from analysis of this data:

• A strong coupling was found to exist between the external and internal flows. Differences in the geometries and flow splits in the inner and outer feed annuli caused the opposed ports to draw air from different regions of the annuli and with different velocities. These different feed conditions resulted in significant differences between the opposed jets inside the flame tube. Measurements of the jets at the port exit planes revealed substantial variations between the jets in terms of the ports' discharge coefficients and the initial pitch angles and axial and radial momentum components of the jets. The turbulence levels observed in the jets were also affected by the external
Conclusions and Recommendations for Further Work

Aerodynamics. Turbulence intensities in excess of 30% were measured in the jets, due to the generation of turbulence in the diffuser system.

- The general behaviour of the secondary jets and the primary jets at the edges of the sector was as expected, the jets impinging with the bulk of the jet fluid moving towards the combustor exit after impingement. However, differences between the opposed jets, due to differences in their feed conditions, affected the location of the impingement and the trajectory of the jet fluid after impingement. The impingement process was found to be unstable, with localised fluctuations in the flow in the region of the impingement point and downstream of the impingement point.

- The primary jets at the centre of the sector, which are in line with the swirler, are highly sensitive to their feed conditions. While the differences in the initial conditions of the two jets were similar to those observed in the secondary jets, the centre primary jets exhibited a gross change in behaviour, whereby one jet was deflected upstream and the other was deflected downstream. The low pressure in the recirculation generated by the swirler upstream of the jets dramatically increased their sensitivity to their feed conditions, resulting in this unexpected behaviour. A similar deviation from the expected flow field may occur in any combustor geometry where the primary jets are located such that they may be exposed to the recirculation generated by the swirler.

- The modified behaviour of the primary jets severely changed the flow field in the primary zone, which was dominated by a single primary jet. Turbulence levels in the primary zone were substantially lower than those observed elsewhere in the flow field, because of the lack of impingement between the centre primary jets. The mixing processes that occur in the primary zone must therefore be affected and the potential effect on the temperature distribution at the turbine inlet, the efficiency of the combustor and the pollutant emissions of the engine should not be underestimated. It is therefore important that the results of this investigation should be considered when designing future combustion systems. In particular, it should not be assumed that the conventional primary zone flow pattern, with impinging primary jets and a symmetrical recirculation, exists, and the effect of the external aerodynamics must be considered.
Conclusions and Recommendations for Further Work

- These results demonstrate the importance of the coupling between the internal and external flow fields. While other workers have previously observed a link between the external and internal aerodynamics, none have observed an effect as severe as that seen in the behaviour of the centre primary jets. It is therefore crucial that the internal and external flow fields should be modelled together if experimental or computational investigations are to produce correct results.

- These results also have implications for the turbulence modelling methods that are used for CFD predictions of the combustor flow field. The anisotropy of the turbulence in the internal flow field demonstrates that the k-ε turbulence model not appropriate for predictions of the Reynolds stresses. However the instability that was observed at several locations, particularly that associated with the impingement of the jets, shows that time dependent CFD methods, such as Large Eddy Simulation, should be used, and methods that rely on the Reynolds Averaged Navier Stokes Equations, such as the k-ε model and the Reynolds Stress Transport model, are not appropriate.

6.2 Recommendations

There is substantial scope for further work to complement the results reported here. The internal measurements should be repeated in combusting conditions to determine whether the modified primary zone flow field observed here is affected by heat release due to combustion, and to determine how it affects the turbine inlet temperature profile, combustor efficiency and emissions characteristics.

A CFD prediction of the combined internal and external flow field should be obtained for comparison with the measurements presented here. This would assist in the development of computational methods for the prediction of combustor flow fields. It would also highlight any areas where further measurements must be obtained to complete the CFD validation data set.

Although a turbulence intensity profile has been obtained at the pre-diffuser inlet plane, it was not possible to perform a complete radial traverse because the hot wire probe could not approach the inner wall. A 3D LDA traverse should therefore be conducted at this location to obtain a better CFD inlet boundary condition. In addition to the
Conclusions and Recommendations for Further Work

Completion of the radial traverse, this would provide more detailed information on the turbulence at the inlet. Detailed measurements should be conducted in the feed annuli to obtain more information on the extraction of air from the annuli by the ports.

There is also scope for further internal measurements to be obtained. Little information has been obtained regarding the flow entering the flame tube through the swirler. It is not known whether the flow leaving the swirler is axi-symmetric. This information would be of interest for CFD validation purposes. Little swirl was observed in the measurements in the primary zone. Further measurements should be conducted to explain this. No observations have been made of flow downstream of the primary zone that could be attributed to fluid that entered the flame tube through the swirler. The measurements at the planes used in this project should be completed by obtaining measurements close to the liners to determine whether this fluid remains in this region. This would also reveal how the liner cooling flow interacts with the other flows in the flame tube.

Vortex structures are a common feature of jets in cross flows, however few vortices were detected in this project. Measurements of jet cross sections should be obtained to examine the development of the jets and detect any vortex structures that are present and how they develop. Information on whether the vortices break down, whether they are symmetric, which vortices are dominant and how they affect the mixing processes that occur in the flame tube would be of interest.

Important questions that should be asked in the design of future combustion systems are whether the desired primary zone flow pattern, with the primary jets contributing equally to a symmetric toroidal vortex, can be achieved, or should it be assumed that it cannot be achieved, with the combustor designed for an asymmetric flow, as observed here? A parametric investigation of the effect of the factors that may affect the jets' trajectories, such as port size and shape, the flow rate through the ports and swirler, bleed flow rates, annulus height/area etc., should be conducted to answer these questions.
References


Boyce, R., 1999, Private Communication


Bradshaw, P., 1973, "Effects of Streamline Curvature on Turbulent Flow", AGARD AG-169


150
References


151
References


References


Hicks, R., 1997, Private Communication


Hughes, N.J., 1999, Private Communication


Klein, A., Katheder, K., and Rohlfis, M., 1974, "Experimental Investigation of the Performance of Short Annular Combusotor Dump Diffusers", Proc. of the 2nd ISABE (Int. symposium on air breathing engines), Paper No.23


153
References


Margason, R.J., 1993, "Fifty Years of Jet in Cross Flow Research", AGARD Meeting on Computational and Experimental Assessment of Jets in Cross Flow, AGARD CP-534, Paper No.1


Quick, J.W., Gartshore, I.S., and Salcudean, M., 1993, "The Interaction of Opposing Jets", Ninth Symposium on Turbulent Shear Flows, Kyoto, Japan, pp.6-4-1 to 6-4-6


References


Figures
Fig. 1.1 Gas generator

(a) Turbojet engine cycle

(b) Turbofan engine

Fig. 1.2 Application of gas generator in aircraft engines
Fig. 1.3 Evolution of the combustion system, from Lefebvre
Fig. 1.4 Alternative forms of the combustion system
(a) tubular; (b) tubo-annular; (c) annular, from Lefebvre (1983)
Fig. 1.5 Features of a typical combustion system

Fig. 1.6 Alternative diffuser systems
(a) faired diffuser; (b) dump diffuser, from Fishenden and Stevens (1977)
Fig. 1.7 Dependence of combustion efficiency on combustor loading ($\theta$) from Lefebvre (1983)

$$\theta = p_3^{1.75} A_{\text{ref}}^{D_3^{0.75}} \exp \left( \frac{T_3}{300} \right) / \dot{m}_A$$

Fig. 1.8 Primary zone with opposed jets and swirler from Lefebvre (1983)
2.0.-------------------
L5
1.0 UNSTABLE
Line of first stall
Pre-diffuser design line
Phase 5 pre-diffuser
STABLE

Fig. 1.9 Pre-diffuser stability chart

outer wall turns flow through greater angle, causing increase in pressure gradient

Fig. 1.10 Effect of pre-diffuser cant angle on outer wall pressure gradient from Stevens, Wray and Price (1988)
Fig. 1.11 Three dimensional pre-diffuser inlet flow due to OGV wake from Carrotte, Bailey and Frodsham (1995)

Fig. 1.12 Flow around head of flame tube from Fishenden and Stevens (1977)
Fig. 1.13 Recirculation in a swirling flow field from Lefebvre (1983)

Fig. 1.14 Vortex systems associated with a single jet in cross flow from Margason (1993)
Fig.1.15 Shear layer vortices from Perry, Kelso and Lim (1993)

Fig.1.16 Vortex systems associated with multiple jets in cross flow from Carrotte and Stevens (1989)
jets impinge and bifurcate
back flow and recirculation upstream of impingement
main flow downstream of impingement

Fig. 1.17 Impingement of strongly opposed jets in a cross flow - typical of primary jets
from Baker and McGuirk (1992)

Fig. 1.18 Definition of variables in the jet trajectory correlation of Srinivasan and White (1986)
limited back flow and small recirculation

(a) equal opposed jets

(b) opposed jets with unequal inclines

Fig. 1.19 Impingement of weakly opposed jets - typical of secondary jets from McGuirk and Palma (1992)
primary jets contribute to toroidal vortex, length of vortex limited by jets

Fig. 1.20 Flow in tubular combustor from Koutmos and McGuirk (1989)

3 burner sector segment of flame tube

Sidewalls

Fig. 1.21 Sector rig configuration
Fig.2.1 VULCAN Phase 5 combustion system

Fig.2.2 VULCAN Flame Tube
Fig. 2.3 Fuel Injector and Swirler
Courtesy of Parker Hannifin Corporation

Fig. 2.4 Circumferential distribution of flame tube ports
Outer starter film (0.036m.)

Shroud cooling (0.007m.)

Heatshield cooling (0.034m.)

Inner starter film (0.025m.)

**Fig.2.5 Heat shield cooling flows**

Air enters through holes

Emerging jets form cooling film

Angled effusion cooling holes drilled at 16.5° to flame tube surface

**Fig.2.6 Angled effusion cooling**

**Fig.2.7 Principal regions of interest**
1. UNSTABLE
Line of first stall

STABLE

Fig. 2.8 Pre-diffuser stability chart

All six beams focused on single point ("measurement control volume")

1D PROBE

single pair of laser beams

included angle

two pairs of laser beams

250mm focal length

2D PROBE

Fig. 2.9 3D LDA probe arrangement
Rotation of probes in vertical plane necessary for access to region close to heat shield.

Rotation of probes in horizontal plane necessary for access to outer region of flame tube to prevent blockage of beams by liners.

Fig. 2.10 Rotation of probes to maximise access to centre sector.
Fig. 2.11 Omission of skirts from test facility geometry
Fig. 2.12 Modifications to inboard bleed offtakes and casing profile
Flow from inlet duct

Cooling air flows through slots onto window

Fig. 2.13 Sidewall window cooling system

Outer casing
Flame tube
Window cooling slots
Window
Window frame
Cooling system gallery
Inner casing

Fig. 2.14 Section through sidewall
13 equally spaced bleed holes

(a) Plan view of bleed holes

(b) Section through one hole

**Fig. 2.15 Outboard bleed offtakes**

13 Plunged Circular Bleed Holes

Flow path

Annular Bleed Exit Slots

Exhaust

7 Plunged Bleed Slots

Flow path

**Fig. 2.16 Bleed System**

Flat outer bleed duct wall

Location of splitter plates

Outer casing

Parallel bleed duct side walls

**Fig. 2.17 Section through outer bleed duct**
Fig. 2.18 Test Facility General Arrangement
Fig. 2.2.1 Bleed System Specification
Fig. 2.22 Inboard bleed system

\[ \Delta P = 0.0669 q_1 \]
\[ m = 0.1214 m_n \]
\[ \Delta P_m = 0.0746 q_n \]
\[ q_m = 0.0049 q_n \]
\[ q_{mm} = 0.0795 q_n \]
\[ \Delta P_{mm} = 0.7685 q_n \]

Fig. 2.23 Orifice with chamfered inlet
(see Hay and Spencer, 1992)

Fig. 2.24 Inboard metering plate

\[ 16 \text{ holes} \quad 5 \text{mm diameter} \]
\[ 16 \text{ holes} \quad 9.5 \text{mm diameter} \]
Fig. 2.25 Outboard bleed system

Fig. 2.26 Outboard metering plate

Fig. 2.27 Distortion of sectored flame tube
Fig. 2.28 Test facility geometry measurements

Fig. 2.29 Circumferential location of primary ports
Fig. 2.30 Circumferential location of secondary ports

Fig. 2.31 Test facility installation
Fig. 2.32 Inlet duct

Fig. 2.33 Exhaust duct
Fig. 2.34 Window cooling air inlet
Direction of velocity component, $u_x$

Fig. 3.1 Laser beams crossing to form interference fringes

Fig. 3.2 Doppler signal

Fig. 3.3 Direction of measured velocity component
Fig. 3.4 Three component LDA system

All six beams focused on single point ("measurement control volume")

Fig. 3.5 LDA Optical arrangement

250mm focal length
Backscatter arrangement - receiving optics receive scattered light from whole measurement volume

Cross coupled arrangement - receiving optics receive scattered light only from intersection of measurement volumes so effective measurement volume is smaller

Fig.3.6 Backscatter and cross coupled arrangements

Fig.3.7 Seed delivery apparatus
(a) Sampled time function
(b) Sampled frequency function calculated using DFT

**Fig.3.8 Sampling of the burst by the BSA**

**Fig.3.9 Cylindrical polar co-ordinate system and typical area traverse**
Rotation of probes in vertical plane necessary for access to region close to heat shield.

Rotation of probes in horizontal plane necessary for access to outer region of flame tube to prevent blockage of beams by liners.

Fig. 3.10 Rotation of probes to maximise access to centre sector
Fig. 3.11 Slide ring system for circumferential traverse

Fig. 3.12 Frame supporting test facility and traverse
Steel posts connected to girder in ceiling

Inlet duct

Steel supporting rods

Plywood board

Exhaust duct

Horizontal adjustment by movement of steel supporting rods

Aluminium beam connects supporting rods to support frame

Fig. 3.13 Test facility support structure
Required radial traverse

Linear traverse

Direction of movement not parallel to required traverse direction

Direction of movement parallel to required traverse direction

Rotary stage added for rotation of linear traverse

Fig. 3.14 Addition of rotary stage to LDA probe traverse
Fig. 3.15 Support, rotation and movement of LDA probes
Light dependent resistor

Fig. 3.16 Pinhole meter

Translation stages provide 16mm vertical, radial and tangential movement with 10µm accuracy.

Translation stages are aligned to ensure the pinhole moves in the vertical, radial and tangential directions.

Fig. 3.17 Probe alignment apparatus
Etched lines represent locations of traverse planes and flame tube centre line.

Steel locating plate

Perspex plate sits in outer bleed exhaust slot CNC machined for exact fit.

Fig.3.18 Measurement volume locating plate

Window attached to traverse to account for refraction in alignment of probes.

Test facility

Measurement volume focused on pinhole.

Fig.3.19 Location of probes and window for alignment of probes
x and r co-ordinates measured at each point to calculate beam vectors

Average vector = direction of travel of probe

Location of focal point calculated from beam vectors

(a) Measurement of beams

Probes adjusted to achieve coincidence

Probe vectors used to calculate location of coincident point

(b) Alignment of probes

Screw adjusted to tilt probe

(c) Adjustment of 2D probe to equalise r co-ordinates

Fig.3.20 Probe alignment procedure
(a) Velocity measurement by a single beam pair

\[ \mathbf{u}_x = \mathbf{v}_n \times \mathbf{v}_m \]

beams vector \( \mathbf{v}_2 \)
mean direction vector \( \mathbf{v}_m \)
beam vector \( \mathbf{v}_1 \)

(b) Calculation of velocity vector \( \mathbf{u}_x \) from beam measurements

Fig.3.21 Calculation of the transformation matrix
Upstream facing measurement

Downstream facing measurement
(probe rotated through 180°)

\( \Theta_{\text{Tip}} = 0.8\text{mm} \)

\( \Theta_{\text{Stem}} = 2.7\text{mm} \)

Calibration factor \( K = \frac{(P - p_{ps})}{(P - p)} \)

Fig. 3.22 Button hook probe operation from Carrotte et al (1993)

Exit Pipe

Working section

Venturi inlet

Fig. 3.23 Button hook probe calibration from Shedden (1993)
Traverse direction

Scale marked on rod and screw
Probe secured to sliding block

Threaded rod

Probe is moved along threaded rod by turning this screw

Casing

Fig.3.24 Manual traverse for button hook and hot wire probes

Stepper motor

Threaded rod

Probe

Fig.3.25 Motorised traverse for button hook probe

Lead screw

Guide rail

Probe shaft

Fig.3.26 Pressure measurement locations

(blue lines denote traverse planes; crosses denote locations of extra probe access points and static tappings)
Traverse direction

Fixed block

Casing

Slider moves across fixed block to traverse probe circumferentially

Fig. 3.27 Circumferential traverse of button hook probe

Fig. 3.28 Hot wire probe

Fig. 3.29 Wool tuft
Fig. 3.30 Location of inlet pitot and static tapping

Fig. 3.31 Typical velocity histogram
Fig.3.32 Bimodal velocity histogram

Fig.3.33 Effect of residence time weighting
Fig. 4.1 Pre-diffuser inlet velocity profiles

Fig. 4.2 Boundary layer on inner wall at pre-diffuser inlet
Fig. 4.3 Boundary layer on outer wall at pre-diffuser inlet

Fig. 4.4 Boundary layer on inner wall at pre-diffuser inlet
Fig. 4.5 Pre-diffuser exit velocity profiles
no inlet trip

Fig. 4.6 Pre-diffuser exit velocity profile
centre line, trip inserted
Fig. 4.7 Pre-diffuser exit boundary layers

Fig. 4.8 Outer annulus sidewall separation
Fig. 4.9 Outer annulus modifications

Fig. 4.10 Effect of splitter plates on outer annulus static pressure distribution
Fig. 4.11 Condition of sidewall boundary layers in inner annulus
Fig. 4.12 Comparison of inner and outer annulus static pressure distributions

Fig. 4.13 Measurement of pressure drop across metering plates
Fig. 4.14 Typical velocity histogram

Fig. 4.15 Bimodal velocity histogram
Fig. 4.16 External measurement locations

Fig. 4.17 Initial internal measurement locations and typical measurement grid
circles denote measurement locations (note unstructured grid)

Fig. 4.18 Port exit measurement locations and typical measurement grid

Fig. 4.19 Final internal measurement locations
Fig. 5.1 Pre-diffuser inlet and exit velocity profiles
Fig. 5.2 Turbulence intensity at pre-diffuser inlet

Fig. 5.3 Pre-diffuser exit static pressure profile
Fig. 5.4 Normalised mean velocity at planes XI2 and X02 (feed annuli - downstream of secondary ports)
Fig. 5.5 Circumferentially averaged velocity profiles
Outer annulus entry (X01) and exit (X02)

Fig. 5.6 Circumferentially averaged velocity profiles
Inner annulus entry (Xi1) and exit (Xi2)
Fig. 5.7 Coordinate system used in internal measurements
Fig. 5.8 Contours of normalised mean v component
Plane I37 - primary ports plane
Fig. 5.9 Contours of normalised mean u component  
Plane I37 - primary ports plane
Fig. 5.10 Mean velocity vectors
Plane I37 - primary ports plane
Fig. 5.11 Contours of normalised mean u component
Plane I22 - 15mm upstream of primary ports plane
Fig. 5.12 Mean velocity vectors and contours of axial vorticity component
Plane I22 - 15mm upstream of primary ports plane
Fig. 5.13 Location of bimodal histograms
Plane I22 - 15mm upstream of primary ports plane
Fig. 5.14 Location of bimodal histograms
Plane I37 - primary ports plane
Fig. 5.15 Contours of normalised mean v component
Plane 180 - secondary ports plane
Fig. 5.16 Contours of normalised mean u component
Plane 180 - secondary ports plane
Fig. 5.17 Mean velocity vectors
Plane 180 - secondary ports plane

231
Fig. 5.18 Location of bimodal histograms
Plane 180 - secondary ports plane
Fig. 5.19 Contours of normalised mean v component
Plane I59 - 22mm upstream of secondary ports plane

233
Fig. 5.20 Contours of normalised mean u component
Plane I59 - 22mm upstream of secondary ports plane
Fig. 5.21 Mean velocity vectors
Plane I59 - 22mm upstream of secondary ports plane
Fig. 5.22 Location of bimodal histograms
Plane I59 - 22mm upstream of secondary ports plane
Fig. 5.23 Contours of normalised mean u component
Plane I100 - 20mm downstream of secondary ports plane
Fig. 5.24 Contours of normalised mean v component
Plane I100 - 20mm downstream of secondary ports plane
Fig. 5.25 Location of bimodal histograms
Plane I100 - 20mm downstream of secondary ports plane
Fig. 5.26 Contours of normalised mean u component
Plane I120 - 40mm downstream of secondary ports plane
Fig. 5.27 Contours of normalised mean v component
Plane I120 - 40mm downstream of secondary ports plane
Fig. 5.28 Location of bimodal histograms
Plane I120 - 40mm downstream of secondary ports plane
Fig. 5.29 Contours of normalised mean v component
Inner centre primary port

Fig. 5.30 Contours of normalised mean v component
Outer centre primary port
Fig. 5.31 Typical chute geometry

Fig. 5.32 Pitch angle coordinate system
Fig. 5.33 Contours of jet pitch angle
Inner centre primary port

Fig. 5.34 Contours of jet pitch angle
Outer centre primary port
Fig. 5.35 Velocity profiles in feed annuli upstream of centre primary ports (planes XI1 and XO1)
Port fed from narrow portion of approaching flow

Port fed from wider portion of approaching flow

Outer port fed from depth of approx. ½ the annulus, allowing flow to turn into port

Separation from leading edge of chute

Larger separation from leading edge of chute

Inner port fed from much shallower depth (approx. 5mm) allowing much less turning

Inner port

Fig.5.36 Visualisation of flow entering centre primary ports
Fig. 5.37 Discharge coefficients of centre primary ports plotted against cut-off velocity
Fig. 5.38 Location of bimodal histograms
Inner centre primary port

Fig. 5.39 Location of bimodal histograms
Outer centre primary port
Fig. 5.40 Bimodal v component histogram
Outer centre primary port
Fig. 5.41 Contours of normalised mean v component
Inner sector edge primary port

Fig. 5.42 Contours of normalised mean v component
Outer sector edge primary port
Fig. 5.43 Contours of jet pitch angle
Inner sector edge primary port

Fig. 5.44 Contours of jet pitch angle
Outer sector edge primary port
Fig. 5.45 Pitch angles of primary jets
Plotted against cut-off velocity

Fig. 5.46 Discharge coefficients of primary jets
Plotted against cut-off velocity
Fig. 5.47 Velocity profiles in feed annuli upstream of centre and sector edge primary ports (planes XI1 and XO1)
Fig. 5.48 Location of bimodal histograms
Inner sector edge primary port

Fig. 5.49 Location of bimodal histograms
Outer sector edge primary port
Fig. 5.50 Contours of normalised mean v component
Inner secondary port

Fig. 5.51 Contours of normalised mean v component
Outer secondary port
Fig. 5.52 Contours of jet pitch angle
Inner secondary port

Fig. 5.53 Contours of jet pitch angle
Outer secondary port
Fig. 5.54 Discharge coefficients of primary and secondary ports
Plotted against cut-off velocity

Fig. 5.55 Pitch angles of primary and secondary jets
Plotted against cut-off velocity
Fig. 5.56 Axial component of momentum primary and secondary jets

Fig. 5.57 Axial component of momentum per unit geometric area primary and secondary jets
Fig. 5.58 Radial component of momentum primary and secondary jets

Fig. 5.59 Radial component of momentum per unit geometric area primary and secondary jets
Outer ports
- Flow fed from wider area than primary
- Flow fed from most of annulus depth
- Flow separates from leading edge of chute
- Some flow from downstream into back of chute

Inner ports
- Flow fed from wide area
- Flow fed from greater depth than primary
- Flow separates from leading edge of chute
- No flow from downstream into back of chute

Fig. 5.60 Visualisation of flow entering secondary ports
Fig. 5.61 Location of bimodal histograms
Inner secondary port

Fig. 5.62 Location of bimodal histograms
Outer secondary port
Fig.5.63(a) Trajectories of centre primary jets with locations of bimodal histograms

Fig.5.63(b) Trajectories of centre primary jets and contours of normalised $w$ component
Fluid from inner centre primary jet

Contours of normalised mean v component at plane I59
(Fig. 5.19)

Fig. 5.63(c) Comparison between trajectories of centre primary jets and earlier measurements
Fig. 5.64(a) Trajectories of sector-edge primary jets with locations of bimodal histograms

Fig. 5.64(b) Trajectories of sector-edge primary jets and contours of normalised w component
small back flow

outwards deflection of impinged jet fluid due to lower pitch angle of inner jet

sector-edge jets exhibit little circumferential deflection

Mean velocity vectors at plane I80 (Fig.5.17)

Fig.5.64(c) Comparison between trajectories of sector-edge primary jets and earlier measurements
Fig. 5.65(a) Trajectories of secondary jets with locations of bimodal histograms

Fig. 5.65(b) Trajectories of secondary jets and contours of normalised w component
Contours of normalised mean velocities at plane I100

Fig. 5.65(c) Comparison between trajectories of secondary jets and earlier measurements
radius of curvature, $R$

pressure differential across jet, $dp$

\[ dp = \frac{p}{R} \frac{V^2}{dR} \]

Fig. 5.66 Calculation of pressure differential across jet

\[ y = -3E+06x^4 + 518765x^3 - 28712x^2 + 701.32x^1 - 7.645x^0 - 0.0166 \times 0.0028 \\ R^2 = 0.9997 \]

upstream

downstream

radial distance from outer chute exit plane (m)

Fig. 5.67 Exported streamline data and fitted polynomial curve
Fig. 5.68 Variation in radius of curvature along the streamline

Fig. 5.69 Estimated static pressure differential across the jet
Fig. 5.70 Fuel injector and swirler
Courtesy of Parker Hannifin Corporation
Fig. 5.71 CFD prediction of flow in Phase 5 combustor with plain ports (Spencer, 1998)

Fig. 5.72 CFD prediction of flow in Phase 5 combustor with chuted ports (Spencer, 1998)
Fig. 5.73(a) Contours of turbulence intensity at plane OPC normalised by local mean velocity

Fig. 5.73(b) Contours of turbulent kinetic energy at plane OPC normalised by square of inlet mean velocity
Fig. 5.74 Anisotropy at plane OPC
Fig.5.75(a) Contours of \( uu \) normal stress at plane OPC normalised by square of inlet mean velocity

Fig.5.75(b) Contours of \( vv \) normal stress at plane OPC normalised by square of inlet mean velocity
mirror image of actual data used to complete approx. histogram

\[ \text{rms} = 8.5 \text{m/s} \]

\[ \text{rms} = 12 \text{m/s} \]

histogram obtained here

Fig. 5.76 u and v component histograms
Outer centre primary port
Fig. 5.77 Locations of centre, secondary and sector-edge radial planes

Fig. 5.78 Normalised turbulent kinetic energy
Centre radial plane
Fig. 5.79 Normalised turbulent kinetic energy
Secondary radial plane

Fig. 5.80 Normalised turbulent kinetic energy
Sector-edge radial plane
Fig. 5.81 Bimodal v component histogram obtained at location of weak impingement between centre primary jets
Fig. 5.82(a) Normalised $u_u$ normal stress
Secondary radial plane

Fig. 5.82(b) Normalised $v_v$ normal stress
Secondary radial plane
Fig. 5.83 Normalised uu normal stress  
Plane l22 - 15mm upstream of primary ports

Fig. 5.84 Normalised vv normal stress  
Plane l22 - 15mm upstream of primary ports
Fig. 5.85 Normalised $ww$ normal stress
Plane I22 - 15mm upstream of primary ports

Fig. 5.86 Normalised turbulent kinetic energy
Plane I22 - 15mm upstream of primary ports
Fig. 5.87 Normalised $ww$ normal stress
Plane I80 - secondary ports plane

Fig. 5.88 Normalised turbulent kinetic energy
Plane I80 - secondary ports plane
Fig. 5.89 Anisotropy at centre radial plane

284
Fig. 5.90 Anisotropy at secondary radial plane

285
Fig. 5.91 Anisotropy at sector-edge radial plane
Fig. 5.92 Development of outer centre primary jet profile
Fig. 5.93(a) Contours of $uv$ shear stress at plane OPC normalised by square of inlet mean velocity.

Fig. 5.93(b) Contours of $vw$ shear stress at plane OPC normalised by square of inlet mean velocity.
Fig. 5.94 Contours of uv shear stress at centre radial plane normalised by square of inlet mean velocity

Fig. 5.95 Contours of uv shear stress at secondary radial plane normalised by square of inlet mean velocity
Fig. 5.96 Contours of uv shear stress at sector-edge radial plane normalised by square of inlet mean velocity
Appendix

For CFD validation purposes, all turbulence data collected inside the flame tube is presented in the form of normalised Reynolds normal stresses, turbulent kinetic energy and Reynolds shear stresses in this appendix. Note that, as in the main text, locations where bimodal histograms were observed are marked by circles.

<table>
<thead>
<tr>
<th>Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plane</td>
<td></td>
</tr>
<tr>
<td>I22</td>
<td>292</td>
</tr>
<tr>
<td>I37</td>
<td>296</td>
</tr>
<tr>
<td>I59</td>
<td>300</td>
</tr>
<tr>
<td>I80</td>
<td>304</td>
</tr>
<tr>
<td>I100</td>
<td>308</td>
</tr>
<tr>
<td>I120</td>
<td>310</td>
</tr>
<tr>
<td>Centre radial plane</td>
<td>312</td>
</tr>
<tr>
<td>Sector-edge radial plane</td>
<td>316</td>
</tr>
<tr>
<td>Secondary radial plane</td>
<td>320</td>
</tr>
<tr>
<td>Outer centre primary port</td>
<td>324</td>
</tr>
<tr>
<td>Outer sector-edge primary port</td>
<td>328</td>
</tr>
<tr>
<td>Outer secondary port</td>
<td>332</td>
</tr>
<tr>
<td>Inner centre primary port</td>
<td>336</td>
</tr>
<tr>
<td>Inner sector-edge primary port</td>
<td>340</td>
</tr>
<tr>
<td>Inner secondary port</td>
<td>344</td>
</tr>
</tbody>
</table>
Normalised uu normal stress at plane 122

Normalised vv normal stress at plane 122
Normalised wall-normal stress at plane 122

Normalised turbulent kinetic energy, $k$, at plane 122
Normalised uv shear stress at plane 122

Normalised vw shear stress at plane 122
Normalised uw shear stress at plane I22
Normalised uu normal stress at plane l37

Normalised vv normal stress at plane l37
Normalised wall normal stress at plane 137

Turbulent kinetic energy, k, at plane 137
Normalised uw shear stress at plane 137
Normalised $uu$ normal stress at plane 159

Normalised $vv$ normal stress at plane 159
Normalised $w w$ normal stress at plane 159

Turbulent kinetic energy, $k$, at plane 159
Normalised uv shear stress at plane 159

Normalised vw shear stress at plane 159
Normalised uw shear stress at plane 159
Normalised uu normal stress at plane 180

Normalised vv normal stress at plane 180
Normalised \( uv \) shear stress at plane 180

Normalised \( vw \) shear stress at plane 180

306
Normalised uw shear stress at plane 180
Normalised normal stresses
and turbulent kinetic energy at plane I100

\[ uu \]

\[ vv \]

\[ ww \]

\[ k \]
Normalised shear stresses at plane I100

- u\(v\)
- v\(w\)
- u\(w\)
Normalised normal stresses and turbulent kinetic energy at plane 120
Normalised shear stresses
at plane l120

uv

vw

uw

0.006
0.005
0.004
0.003
0.002
0.001
0
-0.001
-0.002
-0.003
-0.004
-0.005
-0.006
-0.007
-0.008
-0.009
-0.01

0.0035
0.003
0.0025
0.002
0.0015
0.001
0.0005
0
-0.0005
-0.001
-0.0015
-0.002
-0.0025
-0.003

0.0015
0.00125
0.001
0.00075
0.0005
0.00025
0
-0.00025
-0.0005
-0.00075
-0.001
-0.00125
-0.0015
-0.00175
-0.002
-0.00225
-0.0025

311
normalised uu normal stress
centre radial plane

normalised vv normal stress
centre radial plane
normalised \( \tau_{ww} \) normal stress
centre radial plane

normalised \( \tau_{uv} \) shear stress
centre radial plane
normalised vw shear stress
centre radial plane

normalised uw shear stress
centre radial plane
Normalised turbulent kinetic energy
centre radial plane
normalised uu normal stress sector-edge radial plane

normalised vv normal stress sector-edge radial plane
normalised \( \omega \omega \) normal stress
sector-edge radial plane

normalised \( u \omega \) shear stress
sector-edge radial plane
normalised vw shear stress
sector-edge radial plane

normalised uw shear stress
sector-edge radial plane
Normalised turbulent kinetic energy
Sector-edge radial plane
normalised uu normal stress secondary radial plane

normalised vv normal stress secondary radial plane
normalised \( \vec{w} \) normal stress
secondary radial plane

normalised \( \vec{u} \vec{v} \) shear stress
secondary radial plane
normalised \( \nu \nu \) shear stress
secondary radial plane

normalised \( u \nu \) shear stress
secondary radial plane
Normalised turbulent kinetic energy
secondary radial plane
Distance from port centre line (mm)

Normalised \( u_u \) normal stress
Outer centre primary port exit plane

Distance from port centre line (mm)

Normalised \( v_v \) normal stress
Outer centre primary port exit plane
Normalised $ww$ normal stress
Outer centre primary port exit plane

Normalised turbulent kinetic energy
Outer centre primary port exit plane
Normalised uv shear stress
Outer centre primary port exit plane

Normalised vw shear stress
Outer centre primary port exit plane
Distance from port centre line (mm)

Normalised uw shear stress
Outer centre primary port exit plane
Distance from port centre line (mm)

Normalised $uu$ normal stress
Outer sector-edge primary port exit plane

Distance from port centre line (mm)

Normalised $vv$ normal stress
Outer sector-edge primary port exit plane
Distance from port centre line (mm)

Normalised \( \nu \nu \) normal stress
Outer sector-edge primary port exit plane

Normalised turbulent kinetic energy
Outer sector-edge primary port exit plane
Normalised $uv$ shear stress
Outer centre primary port exit plane

Normalised $vw$ shear stress
Outer centre primary port exit plane
Normalised uw shear stress
Outer sector-edge primary port exit plane
Normalised $uu$ normal stress
Outer secondary port exit plane

Normalised $vv$ normal stress
Outer secondary port exit plane
Distance from port centre line (mm)

Normalised $\xi$ normal stress
Outer secondary port exit plane

Distance from port centre line (mm)

Normalised turbulent kinetic energy
Outer secondary port exit plane
Normalised \( \tau_\text{uw} \) shear stress
Outer secondary port exit plane
Normalised uu normal stress
Inner centre primary port exit plane

Normalised vv normal stress
Inner centre primary port exit plane
Normalised uv shear stress
Inner centre primary port exit plane

Normalised vv shear stress
Inner centre primary port exit plane
Normalised uw shear stress
Inner centre primary port exit plane
Distance from port centre line (mm)

Normalised $uu$ normal stress
Inner sector-edge primary port exit plane

Distance from port centre line (mm)

Normalised $vv$ normal stress
Inner sector-edge primary port exit plane
Normalised uv shear stress
Inner sector-edge primary port exit plane

Normalised vw shear stress
Inner sector-edge primary port exit plane
Normalised uw shear stress
Inner sector-edge primary port exit plane
Distance from port centre line (mm)

Normalised \( \mathbf{u}_u \) normal stress
Inner secondary port exit plane

Distance from port centre line (mm)

Normalised \( \mathbf{v}_v \) normal stress
Inner secondary port exit plane
Distance from port centre line (mm)

Normalised $\sigma_{ww}$ normal stress
Inner secondary port exit plane

Distance from port centre line (mm)

Normalised turbulent kinetic energy
Inner secondary port exit plane
Distance from port centre line (mm)

Normalised $uv$ shear stress
Inner secondary port exit plane

Distance from port centre line (mm)

Normalised $vw$ shear stress
Inner secondary port exit plane
Normalised uw shear stress
Inner secondary port exit plane